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. If such modifications are required, please contact the nearest Technical Center or Technology Center.

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.
SAFETY PRECAUTIONS

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.

<table>
<thead>
<tr>
<th>Sample program</th>
<th>Machines of INTEGREX e-Series</th>
<th>Turning machines</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1000M3</td>
<td>The milling spindle rotates at 1000 min⁻¹.</td>
<td>The turning spindle rotates at 1000 min⁻¹.</td>
</tr>
<tr>
<td>S1000M203</td>
<td>The turning spindle rotates at 1000 min⁻¹.</td>
<td>The milling spindle rotates at 1000 min⁻¹.</td>
</tr>
</tbody>
</table>
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.

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

**WARNING**
- 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.

**CAUTION**
- 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.
## CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1  CONTROLLED AXES</td>
<td>1-1</td>
</tr>
<tr>
<td>1-1 Coordinate Words and Controlled Axes</td>
<td>1-1</td>
</tr>
<tr>
<td>2  UNITS OF PROGRAM DATA INPUT</td>
<td>2-1</td>
</tr>
<tr>
<td>2-1 Units of Program Data Input</td>
<td>2-1</td>
</tr>
<tr>
<td>2-2 Units of Data Setting</td>
<td>2-1</td>
</tr>
<tr>
<td>2-3 Ten-Fold Program Data</td>
<td>2-1</td>
</tr>
<tr>
<td>3  DATA FORMATS</td>
<td>3-1</td>
</tr>
<tr>
<td>3-1 Tape Codes</td>
<td>3-1</td>
</tr>
<tr>
<td>3-2 Program Formats</td>
<td>3-5</td>
</tr>
<tr>
<td>3-3 Tape Data Storage Format</td>
<td>3-6</td>
</tr>
<tr>
<td>3-4 Optional Block Skip</td>
<td>3-6</td>
</tr>
<tr>
<td>3-5 Program Number, Sequence Number and Block Number: O, N</td>
<td>3-7</td>
</tr>
<tr>
<td>3-6 Parity-H/V</td>
<td>3-8</td>
</tr>
<tr>
<td>3-7 List of G-Codes</td>
<td>3-10</td>
</tr>
<tr>
<td>4  BUFFER REGISTERS</td>
<td>4-1</td>
</tr>
<tr>
<td>4-1 Input Buffer</td>
<td>4-1</td>
</tr>
<tr>
<td>4-2 Preread Buffer</td>
<td>4-2</td>
</tr>
<tr>
<td>5  POSITION PROGRAMMING</td>
<td>5-1</td>
</tr>
<tr>
<td>5-1 Dimensional Data Input Method</td>
<td>5-1</td>
</tr>
<tr>
<td>5-1-1 Absolute/Incremental data input: G90/G91</td>
<td>5-1</td>
</tr>
</tbody>
</table>
5-2 Inch/Metric Selection: G20/G21........................................................................5-3
5-3 Decimal Point Input ......................................................................................5-4
5-4 Polar Coordinate Input ON/OFF: G16/G15.....................................................5-7
5-5 Selection between Diameter and Radius Data Input: G10.9 ..........................5-8

6 INTERPOLATION FUNCTIONS ............................................................................6-1

6-1 Positioning Command: G00..............................................................................6-1
6-2 One-Way Positioning: G60 .............................................................................6-4
6-3 Linear Interpolation Command: G01...............................................................6-5
6-4 Circular Interpolation Commands: G02, G03.................................................6-7
6-5 Radius Designated Circular Interpolation Commands: G02, G03 ...............6-10
6-6 Helical Interpolation: G17, G18, G19 and G02, G03 .....................................6-12
6-7 Spiral Interpolation: G2.1, G3.1 (Option) ......................................................6-14
6-8 Plane Selection Commands: G17, G18, G19................................................6-22
   6-8-1 Outline .......................................................................................................6-22
   6-8-2 Plane selection methods.................................................................6-22
6-9 Polar Coordinate Interpolation ON/OFF: G12.1/G13.1 .............................6-23
6-10 Virtual-Axis Interpolation: G07.....................................................................6-27
6-11 Spline Interpolation: G06.1 (Option) ..........................................................6-28
6-12 Modal Spline Interpolation: G61.2 (Option) .................................................6-39
6-13 NURBS Interpolation: G06.2 (Option) ..........................................................6-40
6-14 Cylindrical Interpolation Command: G07.1 ...............................................6-47
6-15  Threading ......................................................................................................................... 6-55

6-15-1  Constant lead threading: G32/G33 ................................................................................. 6-55
6-15-2  Inch threading: G32/G33 .............................................................................................. 6-58
6-15-3  Continuous threading .................................................................................................. 6-59
6-15-4  Variable lead threading: G34 ...................................................................................... 6-60
6-15-5  Threading with C-axis interpolation: G01.1 ............................................................... 6-61
6-15-6  Automatic correction of threading start position (for overriding in a threading cycle) (Option) .............................................................................................................. 6-63

7  FEED FUNCTIONS ................................................................................................................. 7-1

7-1  Rapid Traverse Rates ......................................................................................................... 7-1
7-2  Cutting Feed Rates ............................................................................................................ 7-1
7-3  Synchronous/Asynchronous Feed: G95/G94 ..................................................................... 7-1
7-4  Selecting a Feed Rate and Effects on Each Control Axis .............................................. 7-3
7-5  Threading Leads ................................................................................................................. 7-6
7-6  Automatic Acceleration/Deceleration ............................................................................. 7-7
7-7  Speed Clamp ..................................................................................................................... 7-7
7-8  Exact-Stop Check Command: G09 ................................................................................... 7-8
7-9  Exact-Stop Check Mode Command: G61 ......................................................................... 7-11
7-10 Automatic Corner Override Command: G62 ............................................................... 7-11
7-11 Tapping Mode Command: G63 ....................................................................................... 7-16
7-12 Cutting Mode Command: G64 ........................................................................................ 7-16
7-13 Geometry Compensation/Accuracy Coefficient: G61.1/,K ........................................ 7-17
7-13-1 Geometry compensation function: G61.1 ................................................................. 7-17
7-13-2 Accuracy coefficient (K) ................................................................. 7-18
7-14 Inverse Time Feed: G93 (Option) ..................................................... 7-19

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 Constant Surface Speed Control ON/OFF: G96/G97 ...................... 10-1
10-3 Spindle Clamp Speed Setting: G92 .................................................. 10-3

11 TOOL FUNCTIONS ............................................................................. 11-1
11-1 Tool Function [for ATC Systems] ....................................................... 11-1
11-2 Tool Function [4-Digit T-Code for Turret-Indexing Systems] ......... 11-1
11-4 Tool Function [8-Digit T-Code] ....................................................... 11-2

12 TOOL OFFSET FUNCTIONS ............................................................. 12-1
12-1 Tool Offset ....................................................................................... 12-1
12-2 Tool Length Offset/Cancellation: G43, G44, or T-code/G49 .......... 12-4
12-3 Tool Length Offset in Tool-Axis Direction: G43.1 (Option) .......... 12-13
12-4 Tool Position Offset: G45 to G48 ................................................... 12-20
12-5  Tool Radius Compensation: G40, G41, G42 .................................................. 12-26

12-5-1 Overview ........................................................................................................ 12-26

12-5-2 Tool radius compensation .......................................................................... 12-26

12-5-3 Tool radius compensation using other commands .................................... 12-35

12-5-4 Corner movement ....................................................................................... 12-42

12-5-5 Interruptions during tool radius compensation ............................................ 12-42

12-5-6 Nose-R compensation ............................................................................... 12-44

12-5-7 General precautions on tool radius compensation ..................................... 12-45

12-5-8 Offset number updating during compensation mode .................................. 12-46

12-5-9 Excessive cutting due to tool radius compensation ..................................... 12-48

12-5-10 Interference check ..................................................................................... 12-50

12-6  Three-Dimensional Tool Radius Compensation (Option) .......................... 12-57

12-6-1 Function description ................................................................................... 12-57

12-6-2 Programming methods ............................................................................... 12-58

12-6-3 Correlations to other functions .................................................................. 12-62

12-6-4 Miscellaneous notes on three-dimensional tool radius compensation ....... 12-62

12-7  Programmed Data Setting: G10 ................................................................. 12-63

12-8  Tool Offsetting Based on MAZATROL Tool Data ....................................... 12-72

12-8-1 Selection parameters .................................................................................. 12-72

12-8-2 Tool length offsetting ................................................................................ 12-73

12-8-3 Tool radius compensation ......................................................................... 12-75

12-8-4 Tool data update (during automatic operation) ........................................ 12-76

12-9  Shaping Function (Option) ................................................................. 12-77

12-9-1 Overview .................................................................................................. 12-77
12-9-2 Programming format ........................................................................................................ 12-78
12-9-3 Detailed description ........................................................................................................ 12-78
12-9-4 Remarks ........................................................................................................................ 12-85
12-9-5 Compatibility with the other functions ........................................................................... 12-86
12-9-6 Sample program .............................................................................................................. 12-87

13 PROGRAM SUPPORT FUNCTIONS .............................................................................. 13-1

13-1 Fixed Cycles ...................................................................................................................... 13-1
13-1-1 Outline .......................................................................................................................... 13-1
13-1-2 Fixed-cycle machining data format ............................................................................... 13-2
13-1-3 G71.1 [Chamfering cutter CW] ..................................................................................... 13-5
13-1-4 G72.1 [Chamfering cutter CCW] ................................................................................... 13-6
13-1-5 G73 [High-speed deep-hole drilling] ............................................................................. 13-7
13-1-6 G74 [Reverse tapping] .................................................................................................. 13-8
13-1-7 G75 [Boring] ................................................................................................................ 13-9
13-1-8 G76 [Boring] ................................................................................................................. 13-10
13-1-9 G77 [Back spot facing] .................................................................................................. 13-11
13-1-10 G78 [Boring] ............................................................................................................... 13-12
13-1-12 G81 [Spot drilling] ........................................................................................................ 13-13
13-1-13 G82 [Drilling] .............................................................................................................. 13-14
13-1-14 G83 [Deep-hole drilling] .............................................................................................. 13-15
13-1-15 G84 [Tapping] ............................................................................................................. 13-16
13-1-16 G85 [Reaming] ............................................................................................................ 13-17
13-1-17 G86 [Boring] ................................................................................................................. 13-17
13-1-18 G87 [Back boring] ....................................................................................................... 13-18
13-1-19 G88 [Boring] ................................................................. 13-19
13-1-20 G89 [Boring] ................................................................. 13-19
13-1-21 Synchronous tapping [Option] ................................. 13-20
13-2 Hole Machining Pattern Cycles: G34.1/G35/G36/G37.1 .......... 13-24
  13-2-1 Overview ................................................................. 13-24
  13-2-2 Holes on a circle: G34.1 ........................................ 13-25
  13-2-3 Holes on a line: G35 .............................................. 13-26
  13-2-4 Holes on an arc: G36 .............................................. 13-27
  13-2-5 Holes on a grid: G37.1 ............................................ 13-28
13-3 Hole Machining Fixed Cycles: G80, G283 to G289 ............ 13-29
  13-3-1 Outline ................................................................. 13-29
  13-3-2 Face/Outside deep hole drilling cycle: G283/G287 .......... 13-32
  13-3-3 Face/Outside tapping cycle: G284/G288 ..................... 13-33
  13-3-4 Face/Outside boring cycle: G285/G289 ....................... 13-33
  13-3-5 Face/Outside synchronous tapping cycle: G284.2/G288.2 . 13-34
  13-3-6 Hole machining fixed cycle cancel: G80 ....................... 13-35
  13-3-7 General notes on the hole machining fixed cycles .......... 13-36
  13-3-8 Sample programs with fixed cycles for hole machining ... 13-37
13-4 Initial Point and R-Point Level Return: G98 and G99 ......... 13-38
13-5 Fixed Cycles for Turning ................................................. 13-39
  13-5-1 Longitudinal turning cycle: G290 ............................. 13-40
  13-5-2 Threading cycle: G292 ............................................ 13-42
  13-5-3 Transverse turning cycle: G294 ................................. 13-44
13-6  Compound Fixed Cycles for Turning .............................................................. 13-46

13-6-1 Longitudinal roughing cycle: G271 ............................................................. 13-47

13-6-2 Transverse roughing cycle: G272 ............................................................... 13-53

13-6-3 Contour-parallel roughing cycle: G273 ....................................................... 13-56

13-6-4 Finishing cycle: G270 .................................................................................. 13-60

13-6-5 Longitudinal cut-off cycle: G274 ................................................................. 13-61

13-6-6 Transverse cut-off cycle: G275 ................................................................. 13-64

13-6-7 Compound threading cycle: G276 ............................................................... 13-67

13-6-8 General notes on the compound fixed cycles G270 to G276 ..................... 13-74

13-7  Workpiece Coordinate Setting in a Fixed-Cycle Mode .............................. 13-77

13-8  Scaling ON/OFF: G51/G50........................................................................... 13-78

13-9  Mirror Image ON/OFF: G51.1/G50.1 ............................................................ 13-91

13-10 Subprogram Control: M98, M99 ................................................................. 13-92

13-11 Figure Rotation: M98 (Option) ................................................................. 13-100

13-12 End Processing: M02, M30, M998, M999.................................................. 13-105

13-13 Chamfering and Corner Rounding at Arbitrary Angle Corner ................. 13-107

13-13-1 Chamfering at arbitrary angle corner: , C_ ............................................. 13-107

13-13-2 Rounding at arbitrary angle corner: , R_ .................................................. 13-108

13-14 Linear Angle Commands ............................................................................ 13-109

13-15 Macro Call Function: G65, G66, G66.1, G67.............................................. 13-110

13-15-1 User macros ............................................................................................ 13-110

13-15-2 Macro call instructions .............................................................................. 13-111

13-15-3 Variables .................................................................................................. 13-120

13-15-4 Types of variables ................................................................................... 13-122
13-15-5 Arithmetic operation commands ................................................................. 13-150
13-15-6 Control commands ....................................................................................... 13-154
13-15-7 External output commands (Output via RS-232C) ......................................... 13-158
13-15-8 External output command (Output onto the hard disk) ................................. 13-160
13-15-9 Precautions ............................................................................................... 13-162
13-15-10 Specific examples of programming using user macros ............................ 13-164

13-16 Geometric Commands ................................................................................. 13-168

14 COORDINATE SYSTEM SETTING FUNCTIONS ........................................... 14-1

14-1 Coordinate System Setting Function: G92 ...................................................... 14-1
14-2 Selection of Workpiece Coordinate System: G54 to G59 ............................... 14-4
14-3 Additional Workpiece Coordinate System Setting and Selection: G54.1 (Option) .......................................................... 14-5

14-4 Workpiece Coordinate System Shift ............................................................... 14-11
14-5 Change of Workpiece Coordinate System by Program Command .............. 14-11
14-6 Selection of Machine Coordinate System: G53 ............................................. 14-12
14-7 Selection of Local Coordinate System: G52 .................................................. 14-13
14-8 Ram Spindle Offset ON/OFF: G52.1/G52.2 ................................................... 14-14
14-9 Reading/Writing of MAZATROL Program Basic Coordinates ....................... 14-22

14-9-1 Calling a macroprogram (for data writing) .................................................. 14-22
14-9-2 Data reading ................................................................................................ 14-22
14-9-3 Rewriting ..................................................................................................... 14-23

14-10 Automatic Return to Reference Point (Zero Point): G28, G29 ..................... 14-24
14-11 Return to Second Reference Point (Zero Point): G30 ................................. 14-26
14-12 Return to Reference Point Check Command: G27...............................14-28
14-13 Three-Dimensional Coordinate Conversion ON/OFF: G68/G69.............14-29
14-14 Workpiece Coordinate System Rotation..................................................14-35

15 MEASUREMENT SUPPORT FUNCTIONS........................................... 15-1
15-1 Skip Function: G31 ..................................................................................15-1
15-1-1 Function description..................................................................................15-1
15-1-2 Amount of coasting in the execution of a G31 block..............................15-3
15-1-3 Skip coordinate reading error .................................................................15-4

16 PROTECTIVE FUNCTIONS............................................................. 16-1
16-1 Pre-move Stroke Check ON/OFF: G22/G23 ..............................................16-1

17 MEASUREMENT MACROS ............................................................... 17-1

18 TWO-SYSTEM CONTROL FUNCTION ............................................ 18-1
18-1 Two-Process Control by One Program: G109 ..........................................18-1
18-2 Specifying/Cancelling Cross Machining Control Axis: G110/G111............18-2
18-3 M, S, T, B Output Function to Counterpart: G112 ......................................18-7

19 COMPOUND MACHINING FUNCTIONS........................................... 19-1
19-1 Programming for Compound Machining..................................................19-1
19-2 Waiting Command: M950 to M997, P1 to P99999999..............................19-2
19-3 Balanced Cutting ......................................................................................19-4
19-4 Milling with the Lower Turret....................................................................19-6
19-5 Compound Machining Patterns .................................................................19-8
20 POLYGONAL MACHINING AND HOBBING (OPTION) ......................... 20-1
  20-1 Polygonal Machining ON/OFF: G51.2/G50.2 .............................. 20-1
21 TORNADO TAPPING (G130) ......................................................... 21-1
22 HIGH-SPEED MACHINING MODE FEATURE (OPTION) ................. 22-1
23 HIGH-SPEED SMOOTHING CONTROL FUNCTION (OPTION) ......... 23-1
  23-1 Programming Format ............................................................. 23-2
  23-2 Commands Available in the High-Speed Smoothing Control Mode .......... 23-2
  23-3 Additional Functions in the High-Speed Smoothing Control Mode .......... 23-3
  23-4 Related Parameters ............................................................... 23-4
  23-5 Remarks ............................................................................. 23-4
  23-6 Related Alarms ................................................................. 23-4
24 FUNCTION FOR SELECTING THE CUTTING CONDITIONS .......... 24-1
25 AUTOMATIC TOOL LENGTH MEASUREMENT: G37 ..................... 25-1
26 DYNAMIC OFFSETTING: M173, M174 (Option) ......................... 26-1
27 DYNAMIC OFFSETTING II: G54.2P0, G54.2P1 - G54.2P8 .......... 27-1
28 FUNCTION OF ESTIMATING THE MOMENT OF INERTIA FOR SPINDLE AND SERVO AXIS (OPTION) ................................. 28-1
  28-1 Estimating the Moment of Inertia for Setting Parameters: G297 ........ 28-1
  28-2 Setting Parameters Related to the Moment of Inertia: G298 .......... 28-3
29  FIVE-AXIS MACHINING FUNCTION ................................................. 29-1

29-1  Tool Tip Point Control for Five-Axis Machining (Option) .......................... 29-1
  29-1-1  Function outline ......................................................................................... 29-1
  29-1-2  Detailed description .................................................................................... 29-2
  29-1-3  Relationship to other functions ................................................................... 29-17
  29-1-4  Related parameters .................................................................................... 29-25

29-2  Inclined-Plane Machining: G68.2, G53.1 (Option) ..................................... 29-28
  29-2-1  Function outline ........................................................................................ 29-28
  29-2-2  Restrictions ................................................................................................ 29-34
  29-2-3  Related alarms ........................................................................................... 29-39

29-3  Tool Radius Compensation for Five-Axis Machining (Option) .................. 29-40
  29-3-1  Function outline ........................................................................................ 29-40
  29-3-2  Function description ................................................................................... 29-40
  29-3-3  Operation of tool radius compensation for five-axis machining ............... 29-41
  29-3-4  Method of computing the offset vector ....................................................... 29-42
  29-3-5  Relationship to other functions ................................................................... 29-44
  29-3-6  Related alarms ........................................................................................... 29-46

30  PROGRAM EXAMPLES ............................................................................. 30-1

31  EIA/ISO PROGRAM DISPLAY ............................................................ 31-1

31-1  Procedures for Constructing an EIA/ISO Program ...................................... 31-1

31-2  Editing Function of EIA/ISO PROGRAM Display ........................................ 31-2
  31-2-1  General ....................................................................................................... 31-2
  31-2-2  Operation procedure .................................................................................. 31-2
31-3 Macro-Instruction Input .......................................................................................... 31-8

31-4 Division of Display (Split Screen) ....................................................................... 31-9

31-5 Editing Programs Stored in External Memory Areas .......................................... 31-12
1 CONTROlLED AXES

1-1 Coordinate Words and Controlled Axes

The NC unit can control up to a maximum of sixteen axes. The direction of machining can be designated by a predetermined coordinate word beginning with an alphabetic character.

For vertical machines

For horizontal machines with work rests

V and , are used to control respectively No. 1 and No. 2 work rest movement.
CONTROLLED AXES

For horizontal machines with a lower turret

C1-axis rotation

C1-axis coordinates

For horizontal machines with an NC tailstock

C-axis rotation

C-axis coordinates

For horizontal machines with a workpiece handling device

C-axis rotation

C-axis coordinates
1. Programming examples

A. For work rest movement axes

G1,V100.F1000  "For a No. 2 work rest movement (positioning) at F1000.

In the indication window concerned the No. 1 and No. 2 work rest axes are designated with their suffixes: V1 and V2.

B. For the C-axis

M200  "Selection of C-axis control mode
G0C90.  "For a C-axis positioning (at rapid traverse) to 90°.

C. For the U-axis

M300  "Selection of U-axis control mode
G0U90.  "For a U-axis positioning (at rapid traverse) to 90°.
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.

<table>
<thead>
<tr>
<th>Linear axis</th>
<th>Rotational axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Units of program data input</td>
<td>Units of data setting</td>
</tr>
<tr>
<td>Metric system</td>
<td>Inch system</td>
</tr>
<tr>
<td>Units of data setting</td>
<td>0.0001 mm</td>
</tr>
</tbody>
</table>

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 micron. There may be cases that a machining program which has been set in units of one micron is to be used with a numerical control unit based on 0.1 micron 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.

<table>
<thead>
<tr>
<th>Controlled axis</th>
<th>Program command</th>
<th>Moving distance when program commands are executed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear axis</td>
<td>X1 (Y1 / Z1)</td>
<td>F91 bit 0 = 0</td>
</tr>
<tr>
<td>Rotational axis</td>
<td>B1</td>
<td>0.0001&quot;</td>
</tr>
</tbody>
</table>
- 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**

[Diagram showing EIA code (RS-244-A) and ISO code (R-840) tape codes with explanations of various codes and channel numbers.]
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.

```
Word

   Alphabet (address)

   Numeral

   Word configuration
```

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>
<thead>
<tr>
<th>Item</th>
<th>Metric command</th>
<th>Inch command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program No.</td>
<td>O8</td>
<td></td>
</tr>
<tr>
<td>Sequence No.</td>
<td>N5</td>
<td></td>
</tr>
<tr>
<td>Preparatory function</td>
<td>G3 or G21</td>
<td></td>
</tr>
<tr>
<td>Moving axis</td>
<td>0.0001 mm (deg.), 0.00001 in.</td>
<td>X+54 Y+54 Z+54 α+54 X+45 Y+45 Z+45 α+45</td>
</tr>
<tr>
<td>Auxiliary axis</td>
<td>0.0001 mm (deg.), 0.00001 in.</td>
<td>I+54 J+54 K+54 I+45 J+45 K+45</td>
</tr>
<tr>
<td>Dwell</td>
<td>0.001 mm (rev.), 0.0001 in.</td>
<td>X54 P8 U54</td>
</tr>
<tr>
<td>Feed</td>
<td>0.0001 mm (deg.), 0.00001 in.</td>
<td>F54 (per minute) F33 (per revolution) F45 (per minute) F24 (per revolution)</td>
</tr>
<tr>
<td>Fixed cycle</td>
<td>0.0001 mm (deg.), 0.00001 in.</td>
<td>R+54 Q54 P8 L4 R+45 Q45 P8 L4</td>
</tr>
<tr>
<td>Tool offset</td>
<td>T1 or T2</td>
<td></td>
</tr>
<tr>
<td>Miscellaneous function</td>
<td>M3 × 4</td>
<td></td>
</tr>
<tr>
<td>Spindle function</td>
<td>S5</td>
<td></td>
</tr>
<tr>
<td>Tool function</td>
<td>T4 or T6</td>
<td></td>
</tr>
<tr>
<td>No. 2 miscellaneous function</td>
<td>B8, A8 or C8</td>
<td></td>
</tr>
<tr>
<td>Subprogram</td>
<td>P4 Q5 L4</td>
<td></td>
</tr>
<tr>
<td>Variables number</td>
<td>#5</td>
<td></td>
</tr>
</tbody>
</table>
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 ($\alpha$) denotes additional axis address. +44 will be used when $\alpha$ 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.

<table>
<thead>
<tr>
<th>NC input machining program</th>
<th>NC MONITOR display</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Program No.</td>
</tr>
<tr>
<td>G1234 (DEMO. PROG);</td>
<td>1234</td>
</tr>
<tr>
<td>N100 G00 X120. Z100.;</td>
<td>1234</td>
</tr>
<tr>
<td>G94 S1000;</td>
<td>1234</td>
</tr>
<tr>
<td>N102 G271 P210 Q220 10.2 K0.2 D0.5 F600;</td>
<td>1234</td>
</tr>
<tr>
<td>N200 G94 S1200 F300;</td>
<td>1234</td>
</tr>
<tr>
<td>N210 G01 X0 Z95.;</td>
<td>1234</td>
</tr>
<tr>
<td>G01 X20.;</td>
<td>1234</td>
</tr>
<tr>
<td>X55.;</td>
<td>1234</td>
</tr>
<tr>
<td>X80. Z40. I15.;</td>
<td>1234</td>
</tr>
<tr>
<td>X100.;</td>
<td>1234</td>
</tr>
<tr>
<td>Z30.;</td>
<td>1234</td>
</tr>
<tr>
<td>G20;</td>
<td>1234</td>
</tr>
<tr>
<td>G230 G00 X120. Z150.;</td>
<td>1234</td>
</tr>
<tr>
<td>M02;</td>
<td>1234</td>
</tr>
</tbody>
</table>
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

<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
<th>Group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00 01</td>
<td>01</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01 01</td>
<td>01</td>
</tr>
<tr>
<td>Threading with C-axis interpolation</td>
<td>G01.1 01</td>
<td></td>
</tr>
<tr>
<td>Circular interpolation (CW)</td>
<td>G02 01</td>
<td></td>
</tr>
<tr>
<td>Circular interpolation (CCW)</td>
<td>G03 01</td>
<td></td>
</tr>
<tr>
<td>Spiral interpolation (CW)</td>
<td>G02.1 01</td>
<td></td>
</tr>
<tr>
<td>Spiral interpolation (CCW)</td>
<td>G03.1 01</td>
<td></td>
</tr>
<tr>
<td>Dwell</td>
<td>G04 00</td>
<td></td>
</tr>
<tr>
<td>High-speed machining mode</td>
<td>G05 00</td>
<td></td>
</tr>
<tr>
<td>Fine spline interpolation</td>
<td>G06.1 01</td>
<td></td>
</tr>
<tr>
<td>NURBS interpolation</td>
<td>G06.2 01</td>
<td></td>
</tr>
<tr>
<td>Virtual-axis interpolation</td>
<td>G07 00</td>
<td></td>
</tr>
<tr>
<td>Cylindrical interpolation</td>
<td>G07.1 00</td>
<td></td>
</tr>
<tr>
<td>Exact-stop</td>
<td>G09 00</td>
<td></td>
</tr>
<tr>
<td>Data setting mode ON</td>
<td>G10 00</td>
<td></td>
</tr>
<tr>
<td>Command address OFF</td>
<td>G10.1 00</td>
<td></td>
</tr>
<tr>
<td>Data setting mode OFF</td>
<td>G11 00</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate interpolation ON</td>
<td>G12.1 26</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>G13.1 26</td>
<td></td>
</tr>
<tr>
<td>XY-plane selection</td>
<td>G17 02</td>
<td></td>
</tr>
<tr>
<td>ZX-plane selection</td>
<td>G18 02</td>
<td></td>
</tr>
<tr>
<td>YZ-plane selection</td>
<td>G19 02</td>
<td></td>
</tr>
<tr>
<td>Inch command</td>
<td>G20 06</td>
<td></td>
</tr>
<tr>
<td>Metric command</td>
<td>G21 06</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check ON</td>
<td>G22 04</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23 04</td>
<td></td>
</tr>
<tr>
<td>Reference point check</td>
<td>G27 00</td>
<td></td>
</tr>
<tr>
<td>Reference point return</td>
<td>G28 00</td>
<td></td>
</tr>
<tr>
<td>Return from reference point</td>
<td>G29 00</td>
<td></td>
</tr>
<tr>
<td>Return to 2nd, 3rd and 4th reference points</td>
<td>G30 00</td>
<td></td>
</tr>
<tr>
<td>Skip function</td>
<td>G31 00</td>
<td></td>
</tr>
<tr>
<td>Multi-step skip 1</td>
<td>G31.1 00</td>
<td></td>
</tr>
<tr>
<td>Multi-step skip 2</td>
<td>G31.2 00</td>
<td></td>
</tr>
<tr>
<td>Multi-step skip 3</td>
<td>G31.3 00</td>
<td></td>
</tr>
<tr>
<td>Thread cutting (straight, taper)</td>
<td>G32/G33 01</td>
<td></td>
</tr>
<tr>
<td>Variable lead thread cutting</td>
<td>G34 01</td>
<td></td>
</tr>
<tr>
<td>Hole machining pattern cycle (on a circle)</td>
<td>G34.1 00</td>
<td></td>
</tr>
<tr>
<td>Hole machining pattern cycle (on a line)</td>
<td>G35 00</td>
<td></td>
</tr>
<tr>
<td>Hole machining pattern cycle (on an arc)</td>
<td>G36 00</td>
<td></td>
</tr>
<tr>
<td>Hole machining pattern cycle (on a grid)</td>
<td>G37 00</td>
<td></td>
</tr>
<tr>
<td>Automatic tool length measurement</td>
<td>G37 00</td>
<td></td>
</tr>
<tr>
<td>Vector selection for tool radius compensation</td>
<td>G38 00</td>
<td></td>
</tr>
<tr>
<td>Corner arc for tool radius compensation</td>
<td>G39 00</td>
<td></td>
</tr>
<tr>
<td>Nose R/Tool radius compensation OFF</td>
<td>G40 07</td>
<td></td>
</tr>
<tr>
<td>Nose R/Tool radius compensation (left)</td>
<td>G41 07</td>
<td></td>
</tr>
<tr>
<td>3-D tool radius compensation (left)</td>
<td>G41.2 07</td>
<td></td>
</tr>
<tr>
<td>Function</td>
<td>G-code</td>
<td>Group</td>
</tr>
<tr>
<td>--------------------------------------------------------</td>
<td>--------</td>
<td>-------</td>
</tr>
<tr>
<td>3-D tool radius compensation (left)</td>
<td>G41.5</td>
<td>07</td>
</tr>
<tr>
<td>Nose R/Tool radius compensation (right)</td>
<td>G42</td>
<td>07</td>
</tr>
<tr>
<td>3-D tool radius compensation (right)</td>
<td>G42.2</td>
<td>07</td>
</tr>
<tr>
<td>3-D tool radius compensation (right)</td>
<td>G42.5</td>
<td>07</td>
</tr>
<tr>
<td>Tool length offset (+)</td>
<td>G43</td>
<td>08</td>
</tr>
<tr>
<td>Tool length offset in tool-axis direction</td>
<td>G43.1</td>
<td>08</td>
</tr>
<tr>
<td>Tool tip point control (Type 1) ON</td>
<td>G43.4</td>
<td>08</td>
</tr>
<tr>
<td>Tool tip point control (Type 2) ON</td>
<td>G43.5</td>
<td>08</td>
</tr>
<tr>
<td>Tool length offset (–)</td>
<td>G44</td>
<td>08</td>
</tr>
<tr>
<td>Tool position offset, extension</td>
<td>G45</td>
<td>00</td>
</tr>
<tr>
<td>Tool position offset, reduction</td>
<td>G46</td>
<td>00</td>
</tr>
<tr>
<td>Tool position offset, double extension</td>
<td>G47</td>
<td>00</td>
</tr>
<tr>
<td>Tool position offset, double reduction</td>
<td>G48</td>
<td>00</td>
</tr>
<tr>
<td>Tool position offset OFF</td>
<td>▲G49</td>
<td>08</td>
</tr>
<tr>
<td>Coordinate system setting/Spindle clamp speed setting</td>
<td>G92</td>
<td>00</td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>▲G50</td>
<td>11</td>
</tr>
<tr>
<td>Scaling ON</td>
<td>G51</td>
<td>11</td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>▲G50.1</td>
<td>19</td>
</tr>
<tr>
<td>Mirror image ON</td>
<td>G51.1</td>
<td>19</td>
</tr>
<tr>
<td>Polygonal machining mode OFF</td>
<td>▲G50.2</td>
<td>23</td>
</tr>
<tr>
<td>Polygonal machining mode ON</td>
<td>G51.2</td>
<td>23</td>
</tr>
<tr>
<td>Local coordinate system setting</td>
<td>G52</td>
<td>00</td>
</tr>
<tr>
<td>Machine coordinate system selection</td>
<td>G53</td>
<td>00</td>
</tr>
<tr>
<td>Tool-axis direction control</td>
<td>G53.1</td>
<td>00</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 1</td>
<td>▲G54</td>
<td>12</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 2</td>
<td>G55</td>
<td>12</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 3</td>
<td>G56</td>
<td>12</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 4</td>
<td>G57</td>
<td>12</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 5</td>
<td>G58</td>
<td>12</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 6</td>
<td>G59</td>
<td>12</td>
</tr>
<tr>
<td>Selection of additional workpiece coordinate systems</td>
<td>G54.1</td>
<td>12</td>
</tr>
<tr>
<td>Selection of fixture offset</td>
<td>G54.2</td>
<td>23</td>
</tr>
<tr>
<td>One-way positioning</td>
<td>G60</td>
<td>00</td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td>13</td>
</tr>
<tr>
<td>High-accuracy mode (Geometry compensation)</td>
<td>G61.1</td>
<td>13</td>
</tr>
<tr>
<td>Modal spline interpolation</td>
<td>G61.2</td>
<td>13</td>
</tr>
<tr>
<td>Automatic corner override</td>
<td>G62</td>
<td>13</td>
</tr>
<tr>
<td>Tapping mode</td>
<td>G63</td>
<td>13</td>
</tr>
<tr>
<td>Cutting mode</td>
<td>▲G64</td>
<td>13</td>
</tr>
<tr>
<td>User macro single call</td>
<td>G65</td>
<td>00</td>
</tr>
<tr>
<td>User macro modal call A</td>
<td>G66</td>
<td>14</td>
</tr>
<tr>
<td>User macro modal call B</td>
<td>G66.1</td>
<td>14</td>
</tr>
<tr>
<td>User macro modal call OFF</td>
<td>▲G67</td>
<td>14</td>
</tr>
<tr>
<td>Programmed coordinate rotation ON</td>
<td>G68</td>
<td>16</td>
</tr>
<tr>
<td>Programmed coordinate rotation OFF</td>
<td>G69</td>
<td>16</td>
</tr>
<tr>
<td>3-D coordinate conversion ON</td>
<td>G68</td>
<td>16</td>
</tr>
<tr>
<td>3-D coordinate conversion OFF</td>
<td>▲G69</td>
<td>16</td>
</tr>
<tr>
<td>Inclined-plane machining ON</td>
<td>G68.2</td>
<td>16</td>
</tr>
<tr>
<td>Inclined-plane machining OFF</td>
<td>G69.5</td>
<td>16</td>
</tr>
</tbody>
</table>
### Function G-code Group

<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
<th>Group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finishing cycle</td>
<td>G270</td>
<td>09</td>
</tr>
<tr>
<td>Longitudinal roughing cycle</td>
<td>G271</td>
<td>09</td>
</tr>
<tr>
<td>Transverse roughing cycle</td>
<td>G272</td>
<td>09</td>
</tr>
<tr>
<td>Contour-parallel roughing cycle</td>
<td>G273</td>
<td>09</td>
</tr>
<tr>
<td>Longitudinal cut-off cycle</td>
<td>G274</td>
<td>09</td>
</tr>
<tr>
<td>Transverse cut-off cycle</td>
<td>G275</td>
<td>09</td>
</tr>
<tr>
<td>Compound thread-cutting cycle</td>
<td>G276</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle OFF</td>
<td>▲G80</td>
<td>09</td>
</tr>
<tr>
<td>Face drilling cycle</td>
<td>G283</td>
<td>09</td>
</tr>
<tr>
<td>Face tapping cycle</td>
<td>G284</td>
<td>09</td>
</tr>
<tr>
<td>Face synchronous tapping cycle</td>
<td>G284.2</td>
<td>09</td>
</tr>
<tr>
<td>Face boring cycle</td>
<td>G285</td>
<td>09</td>
</tr>
<tr>
<td>Outside drilling cycle</td>
<td>G287</td>
<td>09</td>
</tr>
<tr>
<td>Outside tapping cycle</td>
<td>G288</td>
<td>09</td>
</tr>
<tr>
<td>Outside synchronous tapping cycle</td>
<td>G288.2</td>
<td>09</td>
</tr>
<tr>
<td>Outside boring cycle</td>
<td>G289</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle A (Longitudinal turning cycle)</td>
<td>G290</td>
<td>09</td>
</tr>
<tr>
<td>Threading cycle</td>
<td>G292</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle B (Transverse turning cycle)</td>
<td>G294</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Chamfering cutter 1, CW)</td>
<td>G71.1</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Chamfering cutter 2, CCW)</td>
<td>G72.1</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (High-speed deep-hole drilling)</td>
<td>G73</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Reverse tapping)</td>
<td>G74</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 1)</td>
<td>G75</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 2)</td>
<td>G76</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Back spot facing)</td>
<td>G77</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 3)</td>
<td>G78</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 4)</td>
<td>G79</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Spot drilling)</td>
<td>G81</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Drilling)</td>
<td>G82</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Pecking)</td>
<td>G82.2</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Deep-hole drilling)</td>
<td>G83</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Tapping)</td>
<td>G84</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Synchronous tapping)</td>
<td>G84.2</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Synchronous reverse tapping)</td>
<td>G84.3</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Reaming)</td>
<td>G85</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 5)</td>
<td>G86</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Back boring)</td>
<td>G87</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 6)</td>
<td>G88</td>
<td>09</td>
</tr>
<tr>
<td>Fixed cycle (Boring 7)</td>
<td>G89</td>
<td>09</td>
</tr>
<tr>
<td>Absolute data input</td>
<td>■G90</td>
<td>03</td>
</tr>
<tr>
<td>Incremental data input</td>
<td>■G91</td>
<td>03</td>
</tr>
<tr>
<td>Workpiece coordinate system rotation</td>
<td>G92.5</td>
<td>00</td>
</tr>
<tr>
<td>Inverse time feed</td>
<td>G93</td>
<td>05</td>
</tr>
<tr>
<td>Constant surface speed control ON</td>
<td>■G96</td>
<td>17</td>
</tr>
<tr>
<td>Constant surface speed control OFF</td>
<td>■G97</td>
<td>17</td>
</tr>
<tr>
<td>Feed per minute (asynchronous)</td>
<td>■G94</td>
<td>05</td>
</tr>
<tr>
<td>Feed per revolution (synchronous)</td>
<td>■G95</td>
<td>05</td>
</tr>
<tr>
<td>Function</td>
<td>G-code</td>
<td>Group</td>
</tr>
<tr>
<td>--------------------------------------------------------</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>Initial point level return in fixed cycles</td>
<td>▲G98</td>
<td>10</td>
</tr>
<tr>
<td>R-point level return in fixed cycles</td>
<td>G99</td>
<td>10</td>
</tr>
<tr>
<td>Single program multi-system control</td>
<td>G109</td>
<td>00</td>
</tr>
<tr>
<td>Cross machining control ON</td>
<td>G110</td>
<td>20</td>
</tr>
<tr>
<td>Cross machining control OFF</td>
<td>G111</td>
<td>20</td>
</tr>
<tr>
<td>M, S, T, B output to opposite system</td>
<td>G112</td>
<td>00</td>
</tr>
<tr>
<td>Hob milling mode OFF</td>
<td>G113</td>
<td>23</td>
</tr>
<tr>
<td>Hob milling mode ON</td>
<td>G114.3</td>
<td>23</td>
</tr>
<tr>
<td>Polar coordinate input ON</td>
<td>G16</td>
<td>18</td>
</tr>
<tr>
<td>Polar coordinate input OFF</td>
<td>G15</td>
<td>18</td>
</tr>
<tr>
<td>Selection between diameter and radius data input</td>
<td>G10.9</td>
<td></td>
</tr>
<tr>
<td>Tornado cycle</td>
<td>G130</td>
<td></td>
</tr>
<tr>
<td>Measurement macro</td>
<td>G136</td>
<td></td>
</tr>
<tr>
<td>Compensation macro</td>
<td>G137</td>
<td></td>
</tr>
</tbody>
</table>

**Notes:**

1. The codes marked with ▲ are selected in each group when the power is turned ON or executing reset for initializing modal.

2. The codes marked with ■ are able to be selected by a parameter as an initial modal which is to become valid when the power is turned ON or executing reset for initializing modal. Changeover of inch/metric system, however, can be made valid only by turning the power ON.

3. G-codes of group 00 are those which are not modal, and they are valid only for commanded blocks.

4. If a G-code not given in the G-code list is commanded, an alarm is displayed. And if a G-code without corresponding option is commanded, an alarm is displayed (808 MIS-SET G CODE).

5. If G-codes belong to different groups each other, any G-code can be commanded 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 commanded in the same block, a G-code commanded last is valid.
- NOTE -
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 $248 \times 4$ 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 $248 \times 5$ 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.

Note: One block of data is stored in one buffer.
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 nose radius compensation, however, maximal five 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) Xx1 Yy1 Zz1 αα₁ (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.

   \[
   \text{N1 G90G00X0 Y0}
   \]

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

   \[
   \text{N2 G91G01X200. Y50. F100}
   \]

   \[
   \text{N2 G90G01X200. Y50. F100}
   \]

   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.

\[(G90) \quad N3 \ X100. \ Y100.\]

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

\[(G91) \quad N3 \ X-100. \ Y50.\]

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.

\[N4 \ \text{G90X300. G91Y100.}\]

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 type)

<table>
<thead>
<tr>
<th>Axis</th>
<th>Example</th>
<th>Initial Inch (parameter) OFF</th>
<th>Initial Inch (parameter) ON</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>G21</td>
<td>G20</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G21</td>
<td>G20</td>
</tr>
<tr>
<td>X</td>
<td>X100</td>
<td>0.0100 mm</td>
<td>0.0254 mm</td>
</tr>
<tr>
<td>Y</td>
<td>Y100</td>
<td>0.0100 mm</td>
<td>0.0254 mm</td>
</tr>
<tr>
<td>Z</td>
<td>Z100</td>
<td>0.0100 mm</td>
<td>0.0254 mm</td>
</tr>
<tr>
<td>B</td>
<td>B100</td>
<td>0.0100 deg</td>
<td>0.0100 deg</td>
</tr>
</tbody>
</table>

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 ÷ 25.4 ≈ 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 Yy1 Zz1
   G20 G92 Xx2 Yy2 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

1. 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.

2. 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.

<table>
<thead>
<tr>
<th>Command</th>
<th>Command unit × 10</th>
<th>Type I</th>
<th>Type II</th>
</tr>
</thead>
<tbody>
<tr>
<td>x1</td>
<td>OFF</td>
<td>0.0001 (mm, inches, deg)</td>
<td>1.0000 (mm, inches, deg)</td>
</tr>
<tr>
<td></td>
<td>ON</td>
<td>0.0010 (mm, inches, deg)</td>
<td>1.0000 (mm, inches, deg)</td>
</tr>
</tbody>
</table>

3. 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.

4. The number of effective digits for each type of decimal-point command is as follows:

<table>
<thead>
<tr>
<th></th>
<th>Move command (Linear)</th>
<th>Move command (Rotational)</th>
<th>Feed rate</th>
<th>Dwell</th>
</tr>
</thead>
<tbody>
<tr>
<td>Integral part</td>
<td>Decimal part</td>
<td>Integral part</td>
<td>Decimal part</td>
<td>Integral part</td>
</tr>
<tr>
<td>mm</td>
<td>0. - 9999.</td>
<td>.00000 - .99999</td>
<td>0. - 9999.</td>
<td>.00000 - .99999</td>
</tr>
<tr>
<td>inch</td>
<td>0. - 9999.</td>
<td>.0000000 - .99999</td>
<td>0. - 99999.</td>
<td>.00000 - .99999</td>
</tr>
</tbody>
</table>

5. Decimal-point commands are also valid for definition of variables data used in subprograms.

6. 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.

7. 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

<table>
<thead>
<tr>
<th>Command category</th>
<th>For 1 = 1 μ</th>
<th>For 1 = 0.1 μ</th>
<th>1 = 1 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0X123.45</td>
<td>X123.450 mm</td>
<td>X123.450 mm</td>
<td>X123.450 mm</td>
</tr>
<tr>
<td>(With the decimal point always given as the millimeter point)</td>
<td>X123.450 mm</td>
<td>X123.450 mm</td>
<td>X123.450 mm</td>
</tr>
<tr>
<td>G0X12345</td>
<td>X12.345 mm*</td>
<td>X1.2345 mm**</td>
<td>X12345.000 mm***</td>
</tr>
<tr>
<td>#111=123</td>
<td>X123.000 mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#112=5.55</td>
<td>Y5.550 mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#113=#111+#112</td>
<td></td>
<td>#113 = 128.550</td>
<td></td>
</tr>
<tr>
<td>(ADD)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#114=#111−#112</td>
<td>#114 = 117.450</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(SUBTRACT)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#115=#111×#112</td>
<td>#115 = 682.650</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(MULTIPLY)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#116=#111/#112</td>
<td>#116 = 22.162</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(DIVIDE)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#117=#112/#111</td>
<td>#117 = 0.045</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* The least significant digit is given in 1 micron.

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

*** The least significant digit is given in 1 mm.
### B. Validity of decimal point for each address

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

**Note:** The decimal point is valid in all the arguments for a user macroprogram.
5-4 Polar Coordinate Input ON/OFF: G16/G15

1. Function and purpose
   The end point of interpolation can be designated with polar coordinates (radius and angle). Polar coordinate input is available only in the mode of polar coordinate interpolation.

2. Programming format
   G16............. Polar coordinate input ON (G-code group No. 18)
   G15............. Polar coordinate input OFF (G-code group No. 18)

3. Detailed description
   Even in the mode of polar coordinate input, positional commands for the axes that have no relation to the polar coordinate interpolation are available as ordinary commands.
   In the mode of polar coordinate input, the length must always be designated in radius values, regardless of the modal state for radius/diameter data input. This also applies to the axes that have no relation to the polar coordinate interpolation.
   The last modal state for radius/diameter data input before the G16 command will be restored automatically by the cancel command G15.

4. Sample program
   G91G17XC; ..................Plane selection (for XC-plane)
   G12.1; ......................Polar coordinate interpolation ON
   G16; .........................Polar coordinate input ON
   G90 G01 X50.C30.F100;
   G02 X50.C60.R50;
   G15; .........................Polar coordinate input OFF
   G13.1; ......................Polar coordinate interpolation OFF

5. Remarks
   1. Enter polar coordinates with respect to the plane of polar coordinate interpolation. Select the appropriate plane beforehand for polar coordinate interpolation.
   2. Positive values (+) for angle data refer to measurement in the counterclockwise direction on the plane of polar coordinate interpolation.
   3. Use address R to designate the radius for circular interpolation (G02 or G03).
   4. If the G16 command is given without selecting the mode of polar coordinate interpolation (by G12.1), an alarm will occur.
   5. If the polar coordinate interpolation mode is cancelled (by G13.1) during polar coordinate input, the mode of polar coordinate input will be cancelled together with the mode of polar coordinate interpolation.
   6. G15 and G16 must be given in an independent block. That is, the block of G15 or G16 must not contain any other G-codes or addresses with the exception of N and P.
7. The following G-codes are available during polar coordinate input. An alarm will occur if any G-code other than these is specified.

**Available G-codes**

- **G00** Positioning
- **G01** Linear interpolation
- **G02** Circular interpolation (CW)
- **G03** Circular interpolation (CCW)
- **G04** Dwell
- **G09** Exact-stop check
- **G13.1** Polar coordinate interpolation OFF
- **G15** Polar coordinate input OFF
- **G40-G42** Tool radius compensation
- **G61** Exact-stop mode
- **G64** Cutting mode
- **G65** User macro single call
- **G66** User macro modal call A
- **G66.1** User macro modal call B
- **G67** User macro modal call OFF
- **G71.1-G89** Fixed cycles for hole machining
- **G283-G289** Fixed cycles for hole machining
- **G94** Asynchronous feed

### 5-5 Selection between Diameter and Radius Data Input: G10.9

1. **Function and purpose**
   
The G10.9 command allows changeover between diameter data input and radius data input, facilitating the creation of the turning section in a compound machining program.

2. **Programming format**
   
   ```
   G10.9 Ax_
   ```
   
   - **Ax**: Address of the axis for which diameter or radius data input is to be specified.
   - Numerical value = 0: Radius data input
   - Numerical value = 1: Diameter data input

3. **Remarks**
   
   1. Give the G10.9 command in a single-command block. Otherwise it may be ignored.
   2. If the G10.9 command is not followed by an axis address, the alarm 807 ILLEGAL FORMAT is caused. Also, the alarm 806 ILLEGAL ADDRESS is caused if a rotational axis is specified in the G10.9 command.
   3. Do not assign a decimal point to the numerical value that follows the axis address. Moreover, assigning a value other than 0 and 1 results in the alarm 809 ILLEGAL NUMBER INPUT.
   4. The G10.9 command only changes the method of programming the positional data for the particular axis. It does not affect various external data such as parameters, workpiece origin data, tool data, and tool offset data.
   5. Irrespective of whether the absolute programming (G90) or the incremental programming (G91) is currently modal, designate the position in diameter values for the axis for which diameter data input has been selected.
4. **Relationship to other G-codes**

Diameter data input applies in general to the positional data of the specified axis.

1. **For positioning (G00), linear interpolation (G01) and coordinate system setting (G92)**
   Designate the position in diameter values for the specified axis.

2. **For circular interpolation (G02/G03)**
   Only the position of the ending point is to be designated in a diameter value for the specified axis. The center, or radius, of the arc must always be designated in radius values (with I, K, or R). The example below refers to a turning program with the X-axis specified as the axis in question. The values with X and I denote the diameter data of the ending point and the radius data of the arc center (incremental to the starting point), respectively, for the X-axis.

   **Absolute programming:**  
   G90 G02 X120.Z70.I50.F200

   **Incremental programming:**  
   G91 G02 X100.Z-30.I50.F200

3. **For fixed cycle of turning**
   Designate the position in diameter values for the specified axis. The amount of taper (for turning fixed cycle) as well as the depth of cut and the finishing allowance (for compound cycle of turning), however, must always be designated in radius values.

4. **For threading (G32/G33, G34, G1.1)**
   Designate the position of the ending point in diameter values for the specified axis. The lead, however, must always be designated in radius values (with F or E).
6 INTERPOLATION FUNCTIONS

6-1 Positioning Command: 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

\[ G00 \ Xx \ Zz \ \alpha_\alpha ; \quad (\alpha \text{ denotes an additional axis, that is, B-, C- or Y-axis}) \]

Where \( x, z \) and \( \alpha \) denote a coordinate.
The command addresses are valid for all additional axis.

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 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-codes of command group 09 (for fixed cycles) are canceled by the G00 command (set to G80).

4. The tool path can be made either linear or polygonal 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 rapid feed rate of each axis.
   - Polygonal line path
     The tool is positioned according to the separate rapid feed rate of each axis.

5. When no number following G address, this is treated as G00.
4. Sample programs

Example:

```
TEP012

Jaw

Chuck

Workpiece

Starting point (+100, +150)

Ending point (+180, +300)

+Z

+X

(Unit: mm)

The diagram above is for
G90 G00 X100. Z150.; Absolute data command, or
```

5. Remarks

1. 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 you set a rapid feed rate of 9600 mm/min for both X- and Z-axes and make the program:

```
G90 G00 Z–300. X400.;
```

then the tool will move as shown in the diagram below.

```
F91 bit 6 = 0

Ending point

400

300

Starting point

X-axis effective feedrate: 6400 mm/min

Z-axis effective feedrate: 9600 mm/min

(Unit: mm)

TEP013
```

For inch-specification machines, the rapid feed rate of the C-axis is limited to 89 rpm (32000/360) even if item C of parameter M1 is set to a value greater than 32000.
2. If bit 6 of user parameter \textbf{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 you set a rapid feed rate of 9600 mm/min for both X- and Z-axes and make the program:

\begin{verbatim}
G90 G00 Z-300. X400.;
\end{verbatim}

then the tool will move as shown in the diagram 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 (Td) has passed.

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

- Linear acceleration/linear deceleration \( Td = Ts + a \)
- Exponential acceleration/linear deceleration \( Td = 2 \times Ts + a \)
- Exponential acceleration/exponential deceleration \( Td = 2 \times Ts + a \)

(Where Ts is the acceleration time constant, \( a = 0 \) to 14 msec)

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 Zz αα;  (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.

   ![Diagram of G60 movement]( MEP018 )

   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 Command: G01

1. Function and purpose

Command G01 involves use of both a coordinate word and a feed rate command. This command moves (interpolates) linearly a tool from the current position to the ending point specified by a coordinate word, at the feed rate specified by address F. The feed rate specified by address F, however, acts as the linear velocity relative to the direction of movement of the tool center.

2. Programming format

G01 Xx Zz αα Ff;  (α: Additional axis)

where x, z and α each denote a coordinate.

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 G32 of command group 01 is given. Thus, a coordinate word will only need be given if the next command is also G01, that is, if the feed rate for the next block remains the same. A programming error will result if an F-code command is not 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-codes of command group 09 (for fixed cycles) are canceled by G01 (set to G80).
4. Sample program

Example 1: Taper turning

```
G90 G01 X50.0 Z20.0 F300;
```

Example 2: Program for moving the tool at a cutting feed rate of 300 mm/min via the route of P₀→P₁→P₂→P₃→P₁ (where the sections P₀→P₁ and P₄→P₀ form a positioning route for the tool):

```
G90 G00 X200.0 Z40.0;
P₀ → P₁
G01 X100.0 Z90.0 F300;
P₁ → P₂
Z160.0;
P₂ → P₃
X140.0 Z220.0;
P₃ → P₄
G00 X240.0 Z230.0;
P₄ → P₀
```
6-4 Circular Interpolation Commands: G02, G03

1. Function and purpose

Commands G02 and G03 feed the tool along an arc.

2. Programming format

G02 (G03) Xx Yy Ii Jj Ff
where G02/G03 : Circular movement direction CW/CCW
x, y : Coordinates of ending point
i, j : Position of arc center
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 position 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 position 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 group 01 is given. The direction of circular movement is determined by G02/G03.

G02: CW (Clockwise)
G03: CCW (Counterclockwise)
Interpolation of an arc that spans multiple quadrants can be defined with one block. To perform circular interpolation, the following conditions are required:
- Plane selection ......................... XY-, ZX-, or YZ-plane
- Rotational direction ....................... CW (G02) or CCW (G03)
- Arc ending point coordinates ...... Given with addresses X, Y and Z.
- Arc center position ....................... Given with addresses I, J and K (Incremental data input)
- Feed rate ................................. Given with address F.

4. Sample programs

**Example 1:** Complete-circle command

![Diagram of complete-circle command](image)

G02 J50. F500

**Example 2:** Three-quarter circle command

![Diagram of three-quarter circle command](image)

G91 G02 X50. Y50. J50. F500

**Remark:** If the coordinates of the ending point are all omitted or set to the same position as the starting point, designating the center using addresses I, J and K will result in an arc of 360 degrees (complete circle).
Note: Either below will result if the starting-point radius and the ending-point radius are not the same.

- If error $\Delta R$ is larger than the parameter data, a program error will occur at the starting point of the arc (Alarm 817 INCORRECT ARC DATA).

![](image1.png)

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

![](image2.png)

Error $\Delta R$ can be set in user parameter F19. The examples shown above assume that excessively large parameter data is given to facilitate your understanding.
6-5  Radius Designated Circular Interpolation Commands: 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 position \((I, J, K)\).

2. Programming format

\[
\text{G02 (G03) } \text{Xx Yy Rr Ff}
\]
where
\[
\begin{align*}
\text{x} & : \text{X-axis coordinate of the ending point} \\
\text{y} & : \text{Y-axis coordinate of the ending point} \\
\text{r} & : \text{Radius of the arc} \\
\text{f} & : \text{Feed rate}
\end{align*}
\]

3. Detailed description

The arc center is present on the bisector perpendicular to the segment which connects the starting point and the ending point. The crossing point of the bisector 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 a semi-circle will be generated if \( R \) is a negative value.

To use the radius-designated circular 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.

Alarm 818 MISSING CENTER (NO DATA) will result if the requirement is not met.

If radius data and arc center data \((I, J, K)\) are both set in the same block, then 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.

Select a plane for the radius-designated circular interpolation in the same manner as for center-specification method.
4. Sample programs

- G02 X\(_x_1\) Y\(_y_1\) R\(_r_1\) F\(_f_1\)  
  XY-plane, radius-designated arc

- G03 Z\(_z_1\) X\(_x_1\) R\(_r_1\) F\(_f_1\)  
  ZX-plane, radius-designated arc

- G02 X\(_x_1\) Y\(_y_1\) J\(_j_1\) R\(_r_1\) F\(_f_1\)  
  (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 I\(_i_1\) J\(_j_1\) R\(_r_1\) F\(_f_1\)  
  XY-plane, center-designated arc  
  (Radius-specification is invalid for complete circle)
6-6 Helical Interpolation: G17, G18, G19 and 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

\[
\text{G17 G02 } Xx_1 \ Yy_1 \ Zz_1 \ Ii_1 \ Ji_1 \ Pp_1 \ Ff_1; \quad (G03)
\]

or

\[
\text{G17 G02 } Xx_2 \ Yy_2 \ Zz_2 \ Rr_2 \ Pp_2 \ Ff_2; \quad (G03)
\]

3. Detailed description

\[
\ell = \frac{z_1}{(2\pi \cdot \frac{\theta_1}{\theta} + \theta)} \quad \text{for helical interpolation, movement designation is additionally required for one to two linear axes not forming the plane for circular interpolation.}
\]

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

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 Z0 Y0;
   G92 X0 Z0 Y0;
   ```

   **Example 2:**
   ```
   G91 G28 X0 Z0 Y0;
   G92 X0 Z0 Y0;
   ```
6-7  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.

![Diagram showing normal circular interpolation and spiral interpolation](image)

2. Programming format

- **G17** G2.1 (or G3.1) Xp_ Yp_ I_ J_ (α_) F_ P_
- **G18** G2.1 (or G3.1) Zp_ Xp_ K_ I_ (α_) F_ P_
- **G19** G2.1 (or G3.1) Yp_ Zp_ J_ K_ (α_) 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: When the following program is executed, the feed rates for each of the points will be as shown in the diagram below.

```
G28 X0 Y0
G00 Y-200.
G17 G3.1 X-100. Y0 I-150. J0 F3000 P2
M30
```
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 \((X = 0, Y = 0)\) and absolute data input of the arc ending point \((X = 0, Y = -15.0)\).

```
G91 G28 Z0  
G80 G40  
T001T000M06  
G54.1 P40  
G90 G94 G00 X0 Y-45.0  
G43 Z30.0 H01  
Z3.0  
S1500 M03  
G01 Z-1.0 F150  
G2.1 X0 Y-15.0 I0 J45.0 F450 P2  
G00 Z3.0  
M05  
Z30.0  
M30
```

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:

\[
\frac{\text{Ending point's radius}}{\text{Starting point's radius}} \times \text{Command value of the rate of feed.}
\]

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

\[
\left(\frac{15.0}{45.0}\right) \times 450 = 150 \text{ mm/min}
\]
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
T001T000M06
G54.1 P40
G90 G94 G00 X0 Y-70.0
G43 Z30.0 H01
S1500 M03
Z3.0
G01 Z-1.0 F150
G2.1 X0 Y1.0 I0 J70.0 F450
X0 Y-70.0 I0 J-1.0
G00 Z3.0
M05
Z30.0
M30
```

- G91 G28 Z0: Zero point return on the Z-axis
- G80 G40: Fixed-cycle cancellation
- T001T000M06: Tool change
- G54.1 P40: Coordinate system setting
- G90 G94 G00 X0 Y-70.0: Approach in the XY-plane to the starting point (0, −70.0)
- G43 Z30.0 H01: Positioning on the Z-axis to the initial point
- S1500 M03: Normal rotation of the spindle
- Z3.0: Infeed on the Z-axis
- G01 Z-1.0 F150: Command for the left-hand half curve
- G2.1 X0 Y1.0 I0 J70.0 F450: Command for the right-hand half curve
- X0 Y-70.0 I0 J-1.0: Return on the Z-axis
- G00 Z3.0: Spindle stop
- M05: 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:

\[
\begin{align*}
G3.1 & \ Y130, \ J100, \ F1000 \ \ldots \ \text{(Right half)} \\
& a+b \quad b \\
G3.1 & \ Y-130, \ J-30 \ \ldots \ \text{(Left half)} \\
& -a-b \quad -a \\
\end{align*}
\]

- \(a = 30\). (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−i₁−i₂ Y₀ ZZ₁ I−i₁ J₀ Ff₁ (Infeed block, half-circle)
G03 X₀ Y₀ ZZ₂ Ii₂ J₀ Pp₂ (Threading block, complete circle)
G3.1 Xl₂+l₃ Y₀ ZZ₃ I₃ J₀ (Upward-cutting block, half-circle)

* The number of pitches, p₂, in the threading block is given by dividing the stroke z₂ by the pitch ℓ. Note that the value p₂ 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 \(p_1\).

\[ G3.1 \ X \ x_1 \ Y \ y_1 \ Z \ z_1 \ I \ i_1 \ J \ j_1 \ P \ p_1 \ F \ f_1 \]

The amount of taper \(t\) and the pitch \(\ell\) are calculated as follows:

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

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

\[ \ell = \frac{z_1}{(2\pi \cdot \pi_1 + \theta) / 2\pi} \]

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 \( z_1/p_1 \).

\[
\begin{align*}
G2.1 & \quad X-x_1 \quad Y0 \quad Zz_1 \quad I-x_1 \quad Pp_1 \quad Ff_1 \\
x_1 & : \text{Radius of the base} \\
z_1 & : \text{Height} \\
p_1 & : \text{Number of pitches} \\
f_1 & : \text{Feed rate}
\end{align*}
\]

Note: Use the TRACE display to check the tool path during spiral interpolation.
6-8  Plane Selection Commands: G17, G18, G19

6-8-1  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
   - Nose radius compensation
   - Polar coordinate interpolation
   - Chamfering (or thread run-out)
   - Positioning for fixed cycle operation
   - Corner rounding/chamfering

2.  Programming format

   G17; (XY-plane selection)
   G18; (ZX-plane selection)
   G19; (YZ-plane selection)

   X, Y, and Z denote respective coordinate axes or their corresponding parallel axes.

   ![Diagram of G17, G18, and G19 planes](image)

6-8-2  Plane selection methods

   Plane selection by parameter setting is explained in this section.

   1.  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.

   ![Diagram of plane selection methods](image)
2. 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)

3. 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:** The plane that is automatically selected upon power on or resetting depends on the settings of bits 0 and 1 of parameter F92 as follows:

<table>
<thead>
<tr>
<th>bit 1</th>
<th>bit 0</th>
<th>Plane selected</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>G17 plane</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>G18 plane</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>G19 plane</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>G18 plane</td>
</tr>
</tbody>
</table>

**Note 2:** In turning mode, G17 or G19 plane selection is impossible and in milling mode, G18 plane selection respectively. If such selection were attempted, alarm would be caused.

**Note 3:** 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.

### 6-9 Polar Coordinate Interpolation ON/OFF: G12.1/G13.1

1. **Function and purpose**
   
   This function is available for face helical grooving or cam shaft grinding. It 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 26).

   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. The block of G12.1 must be preceded by a command of selecting the appropriate plane (G17UH); otherwise an alarm will be caused (1802 ILLEGAL STARTUP CONDITION G12.1). When turning on the power and resetting, the polar coordinate interpolation cancel mode (G13.1) is provided.
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 inch as with the 1st axis of the plane (command by the axis address of the linear axis), and not in degrees. And whether designation is given by the diameter or by the radius is not determined by the 1st axis of the plane, but the designation is the same as the rotational axis.

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 nose radius compensation can also be made for the program command, and the polar coordinate interpolation is given to the path after the nose radius compensation. However, the polar coordinate interpolation mode (G12.1, G13.1) cannot be changed during the nose radius compensation mode (G41, G42). G12.1 and G13.1 must be commanded in G40 mode (Nose 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 inch/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 G94;
N004 G91 G28 X0 Z0;
N008 M200;
N010 T001 T000 M06;
N020 G90 G00 X100.0 Z10.0 C0.0;
G91
N030 G17 XC;
G12.1;
N040 G42;
N050 G90 G01 X50.0 F500;
N060 C10.0;
N070 G03 X-50.0 C10.0 I-50.0;
N080 G01 C-10.0;
N090 G03 X50.0 C-10.0 R50.0;
N100 G01 C0.0;
N110 G00 X100.0;
N120 G40;
N130 G13.1;
N140 M202;
```

- Positioning to the start point
- Selection of the XC-plane
- Polar coordinate interpolation start
- Shape program
  (Program with rectangular coordinate values on the XC-plane)
- Polar coordinate interpolation cancel
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 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 XpYp-plane.
   - Command is given by J and K taking the linear axis as the Y-axis of YpZp-plane.
   - Command is given by K and I taking the linear axis as the Z-axis of ZpXp-plane.
   The circular radius can also be designated by R command.

3. G-codes capable of command during G12.1 mode are G04, G65, G66, G67, G00, G01, G02, G03, G98, G99, G40, G41 and G42.

4. Move command of an axis other than those on the selected plane during G12.1 mode is executed independently of the polar coordinate interpolation.

5. 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.

6. 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.

7. Program restart cannot be made for a block during G12.1 mode.
6-10 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 $\alpha$0 \hspace{1em} To set a virtual axis
: \hspace{1em} To interpolate with the virtual axis
G07 $\alpha$1 \hspace{1em} To cancel the virtual axis

3. Detailed description

1. Only helical or spiral interpolation can be used for the virtual-axis interpolation.
2. In the program section from G07$\alpha$0 to G07$\alpha$1, 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.
3. The virtual axis is valid only for automatic operation; it is invalid for manual operation.
4. Protective functions, such as interlock, stored stroke limit, etc., are valid even for the virtual axis.
5. 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 \hspace{1em} Sets the Y-axis as a virtual axis.
G17G2.1X0Y–5.I0J–10.Z40.P2F50 \hspace{1em} Sine interpolation on the XZ-plane
G07 Y1 \hspace{1em} Resets the Y-axis to an actual axis.
6-11 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 Xx Yy

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:

N100 G00 X_Y_ P1
N200 G06.1 X_Y_ P2
N201 X_Y_ P3
N202 X_Y_ P4
N203 X_Y_ P5
... 
N290 X_Y_ Pn
N300 G01 X_Y_ Pn+1

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 Pn, 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:

N100 G01 X_Y_ P1
N200 G06.1 X_Y_ P2
N300 G01 X_Y_ P3

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

As to the sequence of points $P_1, P_2, P_3, \ldots, P_n$ in a spline interpolation mode, when the angle $\theta$ made by two continuous vectors $P_{i-1}P_i$ and $P_iP_{i+1}$ 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 $P_n$, 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

![Example diagram](image)

Fig. 6-3 Spline cancel depending on angle
When there are more than one point where $\theta_i > F101$, such points are treated as corners to divide the point group and multiple spline curves are created for respective sections.

![Multiple-cornered spline curve depending on angle](image)

When any two corner points (where $\theta_i > F101$) 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:** $F101 < 90$ (deg)

In the following program (shown in Fig. 6-5), the angle of the Y-directional pick feed to the XZ-plane (of spline interpolation) is always 90°. If $F101$ 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 $F101$, it is required to specify G-codes parenthesized in the program below to change the mode of interpolation.

```
N100  G00  X_Y_Z_ P1
N200  G06.1 X_Z_ P2
N210          X_Z_ P3
          ...
N300          X_Z_ P4
N330          (G01) Y_  P4+1
N320          (G06.1) X_Z_ P4+2
          ...
N400          X_Z_ P5
N410          (G01) Y_  P5+1
N420          (G06.1) X_Z_ P5+2
          ...
N700          X_Z_ Pn
N710  G01  ...
```

![Linear interpolation for pick feed in spline-interpolation mode](image)
2. When the movement distance of a block exceeds the spline-cancel distance

**Spline-cancel distance \( \cdots \cdots \cdots \) Parameter \( \text{F100} \)**

As to the sequence of points \( P_1, P_2, P_3, \ldots, P_n \) in a spline interpolation mode, when the length \( P_iP_{i+1} \) of the vector \( P_iP_{i+1} \) is longer than \( \text{F100} \), 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 \( P_iP_{i+1} \) in general.

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

![Fig. 6-6 Spline cancel depending on movement distance of a block](image)

When there are more than one block where \( P_iP_{i+1} > \text{F100} \), 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](image)
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 $P_3P_4$ 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.

![Diagram showing spline curve with significant chord error](image-url)

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](image)

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)](image)

**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.

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.

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

$$l_{i-1} > F103 \times 2$$
$$l_i \leq F103$$
$$l_{i+1} > F103 \times 2$$

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.
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.

![Diagram of fairing at the starting point of a spline curve](image1)

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 \(n\)th block and as a result, the \(n\)th block is deleted.

![Diagram of fairing at the ending point of a spline curve](image2)

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.

![Change of acceleration depending on curvature](image)

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.

![Feed-rate limitation for spline interpolation](image)

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

$$\Delta V = \frac{G1bF}{G1bL} \text{ (mm/min)}$$
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 $P_1P_2 \ldots P_n$ 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 $P_0'P_1'P_2' \ldots P_n'P_{n+1}'$ that is offset by the radius compensation value $r$ compared with the original polygonal line $P_0P_1P_2 \ldots P_nP_{n+1}$.

   2) Next, a point $P_i''$ where $\overrightarrow{P_iP_i''} = r$ on the vector $\overrightarrow{P_iP_i'}$ is determined for all the pass points $P_i$ ($i = 2, 3, \ldots n-1$) 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 $P_1'P_2''P_3'' \ldots P_{n-1}''P_n'$ 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](image)

   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_   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.
6-12 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.

<table>
<thead>
<tr>
<th></th>
<th>G61.2</th>
<th>G61.1</th>
<th>G64</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0</td>
<td>G61.1 + G0</td>
<td>G61.1 + G0</td>
<td>G0</td>
</tr>
<tr>
<td>G1</td>
<td>G61.1 + G06.1</td>
<td>G61.1 + G1</td>
<td>G1</td>
</tr>
<tr>
<td>G06.1</td>
<td>G61.1 + G06.1</td>
<td>G61.1 + G06.1</td>
<td>G06.1</td>
</tr>
</tbody>
</table>

4. Sample program

```
N099 G61.2
N100 G00 X_Y_Z_ P1
N200 G01 X_Y_Z_ P2
N201 X_Y_Z_ P3
N202 X_Y_Z_ P4
N203 G00 X_Y_Z_ P5
N204 G01 X_Y_Z_ P6

N099 G61.2
N205 X_Y_Z_ P7
N206 X_Y_Z_ P8
N290 G01 X_Y_Z_ Pn
N300 G00 X_Y_Z_ Pn+1
```

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.
6-13 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:

\[
P(t) = \sum_{i=0}^{n} \frac{\sum_{j=0}^{m} N_{i,j}(t) w_i P_i}{\sum_{i=0}^{n} w_i} \quad (x_{m-1} \leq t \leq x_{n+1})
\]

\[
N_{i,k}(t) = \begin{cases} 
1 & (x_i \leq t \leq x_{i+1}) \\
0 & (t < x_i, x_{i+1} < t)
\end{cases}
\]

\[
N_{i,k}(t) = \frac{(t-x_i) N_{i,k-1}(t)}{x_{i+k} - x_i} + \frac{(x_{i+k} - t) N_{i+1,k-1}(t)}{x_{i+k} - x_{i+1}}
\]

- “Pi” and “wi” 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.
- “xi” denotes a knot (xi ≤ xi+1), and an array of knots [x₀ x₁ x₂ ... xₙ₋₁] is referred to as the knot vector.
- A variation in parameter “t” from xₙ₋₁ to xₙ₊₁ produces NURBS curve P(t).
- Nᵢₖ(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

\[\text{G6.2[P]} \quad K_{-X_{-Y_{-Z_{-}}}}[R_{-}][F_{-}] \quad \leftarrow \text{NURBS interpolation ON}\]

\[K_{-X_{-Y_{-Z_{-}}}}[R_{-}]
K_{-X_{-Y_{-Z_{-}}}}[R_{-}]
K_{-X_{-Y_{-Z_{-}}}}[R_{-}]
\]

\[P : \text{Rank (omissible)} \]
\[X, Y, Z : \text{Coordinates of the control point} \]
\[R : \text{Weight on the control point (omissible)} \]
\[K : \text{Knot} \]
\[F : \text{Speed of interpolation (omissible)} \]

\[\leftarrow \text{NURBS interpolation OFF}\]
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 \( P_6 \).

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: \[0.0 \ 1.0 \ 2.0 \ 3.0 \ 4.0 \ 5.0 \ 6.0 \ 7.0 \ 8.0 \ 9.0\]

The starting point of the curve differs from the first control point.

The final point of the curve differs from the last control point.

**Example 2:**
- Rank: 4
- Number of control points: 6
- Knot vector:
  \[
  [0.0 \ 0.0 \ 0.0 \ 0.0 \ 1.0 \ 2.0 \ 3.0 \ 3.0 \ 3.0 \ 3.0] \\
  ^1 \quad ^2
  \]

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

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

The curve starts from the first control point.

The curve ends in the last control point.
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.

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>before G6.2</th>
<th>with G6.2</th>
<th>after G6.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>G-codes of group 00</td>
<td>all</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>G-codes of group 01</td>
<td>all</td>
<td>O</td>
<td>O (*)</td>
<td>x</td>
</tr>
<tr>
<td>G-codes of group 02</td>
<td>G17</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G18</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G19</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G-codes of group 04</td>
<td>G22</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G23</td>
<td>O</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>G-codes of group 05</td>
<td>G93</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G94</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G95</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G-codes of group 06</td>
<td>G20</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G21</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G-codes of group 07</td>
<td>G40</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G41</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G42</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>G-codes of group 09</td>
<td>G80</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>the others</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>G-codes of group 12</td>
<td>G64 - G69</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G-codes of group 13</td>
<td>G61.1</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G61.2</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G61</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G62</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G63</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G64</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>G-codes of group 14</td>
<td>G66</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G66.1</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G66.2</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G67</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>G-codes of group 16</td>
<td>G68</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G69</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>High-speed machining mode</td>
<td>G5P0</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td></td>
<td>G5P2</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Feed function</td>
<td>F</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td>Auxiliary function</td>
<td>MSTB</td>
<td>O</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>

(*) 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.

<table>
<thead>
<tr>
<th>Instruction</th>
<th>before G6.2</th>
<th>with G6.2</th>
<th>after G6.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optional block skip</td>
<td>O</td>
<td>O</td>
<td>×</td>
</tr>
<tr>
<td>Control Out/In</td>
<td>O</td>
<td>O</td>
<td>×</td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Function</th>
<th>before G6.2</th>
<th>with G6.2</th>
<th>after G6.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single-block operation</td>
<td>O</td>
<td>×</td>
<td>O (Note)</td>
</tr>
<tr>
<td>Feed hold</td>
<td>O</td>
<td>×</td>
<td>O</td>
</tr>
<tr>
<td>Reset</td>
<td>O</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Program stop</td>
<td>O</td>
<td>×</td>
<td>×</td>
</tr>
<tr>
<td>Optional stop</td>
<td>O</td>
<td>×</td>
<td>×</td>
</tr>
<tr>
<td>Manual interruption</td>
<td>O</td>
<td>×</td>
<td>×</td>
</tr>
<tr>
<td>(Pulse feed and MDI)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Restart</td>
<td>O</td>
<td>×</td>
<td>×</td>
</tr>
<tr>
<td>Comparison stop</td>
<td>O</td>
<td>×</td>
<td>×</td>
</tr>
</tbody>
</table>

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: \( P_0 \ P_1 \ P_2 \ P_3 \ P_4 \ P_5 \ P_6 \)

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. ............. P_6
G6.2 P4 X0 Y100.R1.K0... P_5
X10.Y100.R1.K0........ P_1
X10.Y60.R1.K0......... P_2
X60.Y50.R1.K0......... P_3
X80.Y60.R1.K1......... P_4
X100.Y40.R1.K2........ P_5
X100.Y0 R1.K3......... P_6
K4.
K4.
K4.
K4.
G01 X120............... P_7
```

Fig. 6-21  NURBS interpolation and linear interpolation
9. Related alarms

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

<table>
<thead>
<tr>
<th>Alarm No.</th>
<th>Alarm message</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
<tbody>
<tr>
<td>806</td>
<td>ILLEGAL ADDRESS</td>
<td>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.</td>
<td>Clear the inadequate address.</td>
</tr>
<tr>
<td>807</td>
<td>ILLEGAL FORMAT</td>
<td>1. The modal condition is not appropriate to set G6.2. 2. A block in the G6.2 mode is set without knot (K). 3. The number of blocks with the same knot in succession does not reach the rank.</td>
<td>1. Satisfy the modal condition with reference to item 7-A. 2. Do not fail to set a knot in each block in the G6.2 mode. 3. Set an appropriate knot vector with reference to example 2 given in item 6.</td>
</tr>
<tr>
<td>809</td>
<td>ILLEGAL NUMBER INPUT</td>
<td>1. The number of digits exceeds the specification of axis commands (X, Y or Z). 2. The rank (P) is not admissible. 3. The value of a knot is not admissible. 4. The knot vector is not monotonic increasing.</td>
<td>1. Specify the axis command within eight digits. 2. Set 2, 3 or 4 at address P. 3. Set a value in a range of 0.0001 to 99.9999. 4. Check the blocks for a decreasing knot.</td>
</tr>
<tr>
<td>816</td>
<td>FEEDRATE ZERO</td>
<td>The feed rate (F-code) has not yet been designated.</td>
<td>Set an F-code before or in the same block as the G6.2 code.</td>
</tr>
<tr>
<td>936</td>
<td>OPTION NOT FOUND</td>
<td>The system is not equipped with the optional function of the NURBS interpolation.</td>
<td>Purchase and install the optional function.</td>
</tr>
<tr>
<td>955</td>
<td>START AND END POINT NOT AGREE</td>
<td>The axis coordinates designated in the block of G6.2 do not correspond to the final point of the previous block.</td>
<td>Designate in the first block of the NURBS interpolation the same position as the final point of the previous block.</td>
</tr>
<tr>
<td>956</td>
<td>RESTART OPERATION NOT ALLOWED</td>
<td>The designated restart block falls within the mode of G6.2.</td>
<td>Restart operation is not allowed from the midst of the NURBS interpolation.</td>
</tr>
<tr>
<td>957</td>
<td>MANUAL INTERRUPT NOT ALLOWED</td>
<td>An interruption by pulse handle or MDI operation is commanded in the midst of the G6.2 mode.</td>
<td>Manual interruption is not allowed in the midst of the NURBS interpolation.</td>
</tr>
</tbody>
</table>
**6-14 Cylindrical Interpolation Command: 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**
   - G07.1 Ax,r Cylindrical interpolation mode ON
   - G07.1 Ax,0 Cylindrical interpolation mode OFF
   
   \(
   \begin{align*}
   Ax & : \text{Address of the rotational axis} \\
   r & : \text{Radius of the cylinder (of the groove bottom)}
   \end{align*}
   \)

3. **Detailed description**
   - The above preparatory function (G-code) must be given in a single-command block.
   - Enter a precise value for the radius of the cylinder \((r)\), which is used for the internal calculation of the dimensions and the rate of feed in the developed plane.
   - Enter a positive value for the radius of the cylinder \((r)\).
   - In the mode of cylindrical interpolation the radius of the cylinder \((r)\) cannot be otherwise modified than to zero. That is, the modification must be done after canceling the mode temporarily.

The figure above refers to a machine tool with an indexable spindle head. To perform machining on the surface of the cylinder, use the spindle head being positioned at 90° on the B-axis and perform a three-dimensional coordinate conversion for the corresponding rotation around the Y-axis. 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 (at the cylinder’s radial position of “\(r\)”) and then select the mode of a cylindrical interpolation with simultaneous control of the X- and C-axes.
To use the cylindrical interpolation on such a machine as shown above, 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 (at the cylinder’s radial position of “r”) and then select the mode of a cylindrical interpolation with simultaneous control of the Y- and B-axes.

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>
<thead>
<tr>
<th>G-code group</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>G-code group 1</td>
<td>G0/G1 (Positioning or Linear interpolation only)</td>
</tr>
<tr>
<td>G-code group 2</td>
<td>G17/G18/G19 (XY/ZX/YZ plane) [unconditional]</td>
</tr>
<tr>
<td>G-code group 3</td>
<td>G90/G91 (Absolute/Incremental programming) [unconditional]</td>
</tr>
<tr>
<td>G-code group 4</td>
<td>G22/G23 (Stroke check ON/OFF) [unconditional]</td>
</tr>
<tr>
<td>G-code group 5</td>
<td>G93/G94/G95 (Inverse time/Asynchronous/Synchronous feed) [unconditional]</td>
</tr>
<tr>
<td>G-code group 6</td>
<td>G20/G21 (Inch/Metric data input) [unconditional]</td>
</tr>
<tr>
<td>G-code group 7</td>
<td>G40 (Tool radius or Nose R compensation OFF) (Note 1)</td>
</tr>
<tr>
<td>G-code group 8</td>
<td>G43/G49 (Tool length offset ON/OFF) [unconditional]</td>
</tr>
<tr>
<td>G-code group 9</td>
<td>G80 (Fixed cycle OFF)</td>
</tr>
<tr>
<td>G-code group 10</td>
<td>Invalid (Return level selection [initial or R-point] is only valid for fixed cycles.)</td>
</tr>
<tr>
<td>G-code group 11</td>
<td>G50 (Scaling OFF)</td>
</tr>
<tr>
<td>G-code group 12</td>
<td>G54 to G59, G54.1 (Standard/Additional workpiece coordinate system) [unconditional]</td>
</tr>
<tr>
<td>G-code group 13</td>
<td>G64 (Cutting mode)</td>
</tr>
<tr>
<td>G-code group 14</td>
<td>G67 (User macro modal call OFF)</td>
</tr>
<tr>
<td>G-code group 15</td>
<td>G40.1 (Shaping OFF)</td>
</tr>
<tr>
<td>G-code group 16</td>
<td>G68/G69 (Programmed coordinates rotation ON/OFF) [unconditional]</td>
</tr>
<tr>
<td>G-code group 17</td>
<td>G97 (Constant peripheral speed control OFF)</td>
</tr>
<tr>
<td>G-code group 18</td>
<td>G50.1/G51.1 (Mirror image ON/OFF) [unconditional] (Note 2)</td>
</tr>
<tr>
<td>G-code group 23</td>
<td>G50.2 (Polygonal machining OFF)</td>
</tr>
<tr>
<td>G-code group 23</td>
<td>G113 (Hob milling OFF)</td>
</tr>
<tr>
<td>G-code group 26</td>
<td>G13.1 (Polar coordinate interpolation OFF)</td>
</tr>
<tr>
<td>Others</td>
<td>G5P0 (High-speed machining OFF)</td>
</tr>
<tr>
<td>Others</td>
<td>G7.1A.0 (Cylindrical interpolation OFF)</td>
</tr>
</tbody>
</table>

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: 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°).

B. Commands in the cylindrical interpolation mode

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

<table>
<thead>
<tr>
<th>G-code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0</td>
<td>Rapid positioning</td>
</tr>
<tr>
<td>G1, G2, G3</td>
<td>Linear and circular interpolation</td>
</tr>
<tr>
<td>G4</td>
<td>Dwell</td>
</tr>
<tr>
<td>G9</td>
<td>Exact-stop check</td>
</tr>
<tr>
<td>G17, G18, G19</td>
<td>Plane selection</td>
</tr>
<tr>
<td>G40, G41, G42</td>
<td>Tool radius compensation</td>
</tr>
<tr>
<td>G90/G91</td>
<td>Absolute/Incremental programming</td>
</tr>
</tbody>
</table>

Note 1: To give, in the cylindrical interpolation mode, a motion command whose execution requires a half turn (180°) of the workpiece or more, use the incremental programming method (G91), or set multiple blocks of absolute programming (G90) to describe the desired contour. The above-mentioned command will always be executed on the short cut route of rotation for the target angular position if it is given in the G90 mode.

Note 2: After the mode selection by a block of G7.1, do not fail to give a plane selection command of the following format (with the plane selection parameters preset as appropriate) in order to specify the cylindrical interpolation plane determined by the corresponding linear and rotational axes:

- G17 BbYy For an interpolation with the Y-axis (linear) and B-axis (rotational).
- G18 AaXx For an interpolation with the X-axis (linear) and A-axis (rotational).
- G19 CcZz For an interpolation with the Z-axis (linear) and C-axis (rotational).
- G17 XxCc With the X-axis (linear) and C-axis (rot.) after coordinate conversion.

2. In the mode of cylindrical interpolation the rate of feed refers to a resultant speed in the plane onto which the surface of the cylinder is developed.

Example: The speed on each component axis is calculated for a block of G1XxCcFf as follows:

\[
F_x = \frac{x}{\sqrt{x^2 + \left(\frac{2\pi c}{360}\right)^2}} \times f
\]

\[
F_c = \frac{2\pi c}{360} \times f
\]
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.

C. Remarks

1. 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.

   <Positioning error according to the angle and radius>

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

3. 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.

4. 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 NONMODAL] menu function which skips the preceding blocks.

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

D. Operation examples with manual interruption

1. With “manual absolute” ON

2. With “manual absolute” OFF
E. Sample program for cam grooving

N01 G54 G90 G0 X0 Y0 C0
N02 Z0 B90.
N03 M3 S800
N04 G68 X0 Y0 Z0 I0 J1 K0 R90.
N05 G1 Z-5. F2000
N06 G7.1 C63.662
N07 G17 G1 G41 X0 C0 D1
N08 G3 C45. R30.
N09 G2 C135. R60.
N10 G1 X50. C180.
N11 G3 C270. R60.
N12 G1 X0. C315.
N14 G1 G40
N15 G1 G42 X0. C360. D1
N16 G3 C45. R30.
N17 G2 C135. R60.
N18 G1 X50. C180.
N19 G3 C270. R60.
N20 G1 X0. C315.
N22 G1 G40
N23 G7.1 C0
N24 M30
F. Sample program for the use of mirror image function

Main program
N01 G54 G0 G17 G90 X0 Y0 B0;
N02 Z0 S800 M3;
N03 M98 P2000;
N04 G55 G0 G17 G90 X0 Y0 B0;
N05 Z0 S800 M3;
N06 G51.180; ............... Mirror image ON
N07 M98 P2000;
N08 G50.180; ............... 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;

<Workpiece origin data>

<table>
<thead>
<tr>
<th>G54</th>
<th>G55</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0.</td>
</tr>
<tr>
<td>Y</td>
<td>0.</td>
</tr>
<tr>
<td>Z</td>
<td>0.</td>
</tr>
<tr>
<td>B</td>
<td>225.</td>
</tr>
</tbody>
</table>
G. Sample program for the use of circular interpolation commands

```plaintext
N01 G90 G1 Y47.5 F1000;
N02 G7.1 B63.662;
N03 G17 G1 Y47.5 B45.;
N04 G2 B135.R62.5;
N05 G1 B180.;
N06 G3 B270.I50.J37.5;
```

Radius of groove bottom = 63.662 mm

\[ r = \sqrt{(47.5 - 10)^2 + 50^2} = 62.5 \text{ mm} \]
6-15 Threading

6-15-1 Constant lead threading: G32/G33

1. Function and purpose
   The G32 or G33 command controls the feedrate of the tool in synchronization with the spindle rotation and so this enables both the straight and scrolled thread cutting of constant leads and the continuous thread cutting.

2. Programming format
   G32/G33 Zz Xx Ff; (Normal lead thread cutting commands)
   Where Zz, Xx: Thread ending point addresses and coordinates
   Ff: Lead of long axis (axis of which moving distance is the longest) direction
   G32/G33 Zz Xx Ee; (Precision lead threading commands)
   Where Zz, Xx: Thread ending point addresses and coordinates
   Ee: Lead of long axis (axis of which moving distance is the longest) direction
3. Detailed description

1. The E command is also used for the number of threads in inch threading, and whether the thread number or precision lead is to be designated can be selected by parameter setting. (Bit 7 of address F91 is set to 0 for precision lead designation.)

2. The lead in the long axis direction is commanded for the taper thread lead.

![Diagram of Tapered thread section](TEP028.png)

When a < 45° lead is in Z-axis direction.
When a > 45° lead is in X-axis direction.
When a = 45° lead can be in either Z- or X-axis direction.

Refer to Section 7-5 for details of lead setting range.

**Note:** It is not possible to designate a lead where the feed rate as converted into per-minute feed exceeds the maximum cutting feed rate.

3. The constant surface speed control function should not be used here.

4. The spindle speed should be kept constant throughout from the roughing until the finishing.

5. If the feed hold function is employed to stop the feed during thread cutting, the thread height will lose their shape. For this reason, feed hold does not function during thread cutting.

   If the feed hold button is pressed during threading, block stop will result at the ending point of the block following the block in which threading is completed (no longer in G32 mode).

6. An M0 block (for programmed stop) immediately following a threading block will not cause a stop until completion of the next, non-threading motion block (no longer in G32 mode) so as not to deform the thread.

7. The converted cutting feed rate is compared with the cutting feed clamp rate when threading starts, and if it is found to exceed the clamp rate, an alarm will result. (See the Note in item 2 above.)

8. In order to protect the lead during threading, a converted cutting feed rate may sometimes exceed the cutting feed clamp rate.

9. An illegal lead is produced at the start and at the end of the thread cutting because of servo system delay and other factors.

   Therefore, it is necessary to command a thread length obtained by adding the illegal lead lengths $\delta_1$ and $\delta_2$ to the required thread length.

10. The spindle speed is subject to the following restriction:

   $1 \leq R \leq \text{Maximum feed rate/Thread lead}$

   where $R$: Spindle speed (rpm) $\leq$ Permissible speed of encoder (rpm)

   Thread lead $= \text{mm or inches}$

   Maximum feed rate $= \text{mm/min or inch/min}$ (this is subject to the restrictions imposed by the machine specifications).
11. During threading, use or disuses of dry run can be specified by setting parameter F111 bit 1.

12. Synchronous feed applies for the threading commands even with an asynchronous feed mode (G94).

13. Spindle override is valid even during threading. But the override value will not be changed during threading.

14. When a threading command is programmed during tool nose R compensation, the compensation is temporarily cancelled and the threading is executed.

15. When the mode is switched to another automatic operation mode while G32 is executed, the following block which does not contain a threading command is first executed and then the automatic operation stops.

16. When the mode is switched to manual operation mode while G32 is executed, the following block which does not contain a threading command is first executed and then the automatic operation stops. In the case of the single block operation, the following block which does not contain a threading command is first executed and then the automatic operation stops.

17. The threading command waits for the single rotation synchronization signal of the rotary encoder and starts movement.
   With this NC unit, however, movement starts without waiting for this signal when another system issues a threading command during threading by one system. Therefore, threading commands should not be issued by a multiple number of systems.

4. Sample programs

G90 G32/G33 X90.0 Z40.0 E12.34567; ..........Absolute data command
G91 G32/G33 X70.0 Z-50.0 E12.34567; ..........Incremental data command
6-15-2 Inch threading: G32/G33

1. Function and purpose

If the number of threads per inch in the long axis direction is designated in the G32 or G33 command, the feed of the tool will be controlled to synchronize with the spindle rotation. That is, constant lead straight threading, taper threading and continuous threading can be performed.

2. Programming format

G32/G33 Zz Xx Ee;

Where Zz, Xx: Thread ending point addresses and coordinates

Ee: Number of threads per inch in direction of long axis (axis of which the moving distance is the longest)

(Decimal point command can also be assigned.)

3. Detailed description

1. The number of threads in the long axis direction is assigned as the number of threads per inch.

2. The E code is also used to assign the precision lead length, and whether the thread number or precision lead length is to be designated can be selected by parameter setting (allowed by parameter F91 bit 7).

3. The E command value should be set within the lead value range when converted to the lead.

4. See Subsection 6-15-1 on “Constant lead threading” for further details.
4. Sample programs

![Diagram of a threading process](image1)

G90 G32/G33 X90.0 Z40.0 E12.0; .............Absolute data command
G91 G32/G33 X70.0 Z–50.0 E12.0; .............Incremental data command

6-15-3 Continuous threading

Continuous threading is possible by designating threading commands continuously. In this way, it is possible to cut special threads whose lead or shape changes.

![Diagram of continuous threading](image2)
6-15-4 Variable lead threading: G34

1. Function and purpose
   Variable lead threading is possible by commanding the increase or decrease of a lead per screw rotation.

2. Programming format
   G34 Xx Zz Ff/Ee Kk;
   It is the same as the case of straight and taper threading of G32 (described in Subsections 6-15-1 and -2) except an address K.
   A value commanded with K gives the increase or decrease of a lead per screw rotation.
   Values which K can take are as follows:
   Metric input: ±0.00001 to ±999.9999 mm/rev
   Inch input: ±0.000001 to ±99.9999 in./rev

3. Notes
   1. As a result of the increase or decrease of a lead, when exceeding the range of the command value of screw lead or when cutting feed gets excessively high, the feed rate is clamped at rapid feed rate.
   2. “Feed hold during threading” function is invalid for G34.
6-15-5 Threading with C-axis interpolation: G01.1

1. Function and purpose

The G01.1 command in the milling mode enables a simultaneous interpolation on the C-axis and the X- and/or the Z-axis for straight, tapered or scrolled thread cutting of constant leads.

2. Programming format

G01.1 Zz Xx Ff Ss;

Where Zz, Xx: Thread ending point addresses and coordinates (mm or in.)
Ff: Lead of long axis (axis of which moving distance is the longest) direction
Ss: Rotational speed of C-axis (rpm)

Set parameter F111 bit 3 to select the direction of C-axis rotation:
F111 bit 3 = 0 : Normal rotation of C-axis
= 1 : Reversed rotation of C-axis

3. Detailed description

1. For tapered thread cutting, specify the lead in the long axis direction.

2. Range of specification of lead (address F)
   - For data input in mm : 0.0001 to 500.0000 mm
   - For data input in in. : 00.00001 to 9.99999 in.

3. Specification range of rotational speed (address S)
   1 \leq S \leq \text{Max. speed of C-axis rotation}
   - The maximum speed of C-axis rotation (1/360 of value “C” of parameter M3) depends on the respective machine model.
   - Do not create a program nor operate the overriding keys in such a manner that the maximum speed of C-axis rotation should be exceeded.

4. During execution of G01.1 command, it is possible, indeed, but not advisable at all to apply feed hold or to change the override value for fear of deformation of the thread.

5. The speed of C-axis rotation should be kept constant throughout from roughing till finishing.

6. The number of C-axis revolutions for execution of one G01.1 command must not exceed 298.
4. Sample programs

G94 G97;
G91 G28 X0 Z0;
T001T000M06;
G92 X300.Z100.;
M200;
G90 G00 X100.Z2.C0.;
G91 G01.1 Z-100.F2.S400;(*)
G00 X10.;
Z100.C0.;
X-11.;
G01.1 Z-100.F2.S400;(*)
G00 X11.;
Z100.C0.;
G00 X-12.;
G01.1 Z-100.F2.S400;(*)
G00 X12.;
Z100.;
G28 X0 Z0.;
M202;
M30;

(*) Command for threading with C-axis control, 2 mm lead and 400 rpm
6-15-6  Automatic correction of threading start position (for overriding in a threading cycle) (Option)

1. Function and purpose

The phase of the spindle is automatically corrected at the start of each threading pass to prevent the threading position from deviating even when the spindle override value is updated in the middle of a threading cycle.

The use of this option allows the thread cutting conditions to be changed even in the flow of a threading cycle.

2. Related G-codes

The automatic correction function is applicable to the following G-codes of threading:

<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thread cutting (straight, taper)</td>
<td>G32/G33</td>
</tr>
<tr>
<td>Turning fixed cycle for threading</td>
<td>G292</td>
</tr>
<tr>
<td>Compound fixed cycle for threading</td>
<td>G276</td>
</tr>
</tbody>
</table>

Note: Variable-lead threading (G34), or continuous threading for different-pitch sections, requires continuous or transitional acceleration between blocks, as well as different distances of acceleration. The automatic correction function cannot guarantee correct thread forming for a speed overriding in the middle of these threading cycles.
3. Detailed description

1. Even in the middle of a threading pass, operating the turning/milling spindle speed overriding keys immediately changes the speed indication in percentages, indeed, but the actual speed will not accordingly change till completion of the threading block (or a series of the threading blocks in the case of “continuous threading”).

2. The function for automatic correction of threading start position does not include corresponding adjustment of the acceleration distance for threading. To use an overriding value above 100%, therefore, specify in the machining program such an acceleration distance as to allow for the maximum spindle speed.

3. As for the end of thread, the length of the upward cutting path on the workpiece will become shorter, or greater, for a spindle override value below, or above, 100%.

4. Notes

1. This function is not valid for a threading by simultaneous cutting with both turrets.
2. This function is not valid for a threading by synchronization of both turning spindles.
3. This function is only valid for a longitudinal threading (by cutting feed on the Z-axis).
4. After changing the spindle override value the execution of a threading block should not be started until spindle rotation has been stabilized; otherwise the starting section will only be cut to an incomplete thread.
5. Do not allow a threading block to be executed with the spindle override value set to 0%; otherwise the machine operation will be stopped at the beginning of that block.
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, G32/G33, and G34.

Example:

<table>
<thead>
<tr>
<th>Asynchronous feed</th>
<th>Feed rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>G01 X100. Z100. F200*;</td>
<td>200.0 mm/min</td>
</tr>
<tr>
<td>G01 X100. Z100. F123.4;</td>
<td>123.4 mm/min</td>
</tr>
<tr>
<td>G01 X100. Z100. F56.789;</td>
<td>56.789 mm/min</td>
</tr>
</tbody>
</table>

* It means the same if F200. or F200.000 is set instead 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, G32/G33, or G34) that is read firstly after power-on.

7-3 Synchronous/Asynchronous Feed: G95/G94

1. Function and purpose

Command G95 allows a feed rate per revolution to be set using an F-code. To use this command, a rotational encoder must be mounted on the spindle.

2. Programming format

G94: Feed per minute (/min) [Asynchronous feed]
G95: Feed per revolution (/rev) [Synchronous feed]

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

<table>
<thead>
<tr>
<th>Input in mm</th>
<th>G94F_ (Feed per minute)</th>
<th>G95F_ (Feed per revolution)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 to 240000 mm/min (F1 to F240000)</td>
<td>0.0001 to 500.0000 mm/rev (F1 to F5000000)</td>
<td></td>
</tr>
<tr>
<td>0.01 to 9600.00 in./min (F1 to F960000)</td>
<td>0.000001 to 9.999999 in./rev (F1 to F9999999)</td>
<td></td>
</tr>
</tbody>
</table>

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

\[ FC = F \times N \times OVR \] (Expression 1)

where
- \( FC \): Effective feed rate (mm/min or inches/min)
- \( F \): Designated feed rate (mm/rev or inches/rev)
- \( N \): Spindle speed (rpm)
- \( OVR \): 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. An effective feed rate that is expressed in a feed rate per minute (mm/min or inches/min) is displayed as \texttt{FEED} on the \texttt{POSITION} display.

2. If the effective feed rate is larger than the cutting feed clamping speed, that clamping speed will become valid.

3. In the dry run mode, feed will become asynchronous and the machine will operate at an externally preset feed rate (mm/min or inches/min).

4. According to the setting of bit 1 of parameter \texttt{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 Z-axes) are to be controlled using a feed rate of \( f \):

![Diagram showing linear axes control](image)

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:

\[
\text{X-axis feed rate} = f \times \frac{x}{\sqrt{x^2 + z^2}}
\]

\[
\text{Z-axis feed rate} = f \times \frac{z}{\sqrt{x^2 + z^2}}
\]

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) \( fc \) is calculated by:

\[
f_c = f \times \frac{\pi \cdot f}{180}
\]

Hence, the feed rate to be programmed for the required value \( fc \) is:

\[
f = fc \times \frac{180}{\pi \cdot f}
\]

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 Z-axes) are to be controlled at a feed rate of \( f \) using the circular interpolation function:

In this case, the X- and Z-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 (A, B, C or U) 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:

\[
\begin{align*}
\text{X-axis feed rate (linear velocity), } f_x &= f \times \frac{x}{\sqrt{x^2 + c^2}} \quad \text{[1]} \\
\text{C-axis feed rate (angular velocity), } \omega &= f \times \frac{c}{\sqrt{x^2 + c^2}} \quad \text{[2]} \\
\text{Linear velocity } f_c \text{ that relates to C-axis control is expressed as:} \\
&= \omega \cdot \frac{\pi \cdot r}{180} \quad \text{[3]} \\
\text{If the velocity in the moving direction of the tool at starting point } P_1 \text{ is taken as } f_t, \text{ and its } X- \text{ and } Y-axis components as } f_{tx} \text{ and } f_{ty} \text{ respectively, then one can express } f_{tx} \text{ and } f_{ty} \text{ as follows:} \\
&= -r \sin \left( \frac{\pi}{180} \cdot \theta \right) \times \frac{\pi}{180} \omega + f_x \quad \text{[4]} \\
&= -r \cos \left( \frac{\pi}{180} \cdot \theta \right) \times \frac{\pi}{180} \omega \quad \text{[5]} \\
\end{align*}
\]

where \( r \) denotes the distance (in millimeters) from the rotational center to the tool, and \( \theta \) 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:

\[ ft = \sqrt{f_{x}^2 + f_{y}^2} \]

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

\[ \ldots \ldots \text{[6]} \]

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

\[ f = \frac{ft \times \frac{\pi}{180}}{\sqrt{x^2 + c^2}} \]

\[ \ldots \ldots \text{[7]} \]

In expression [6], “ft” is the velocity at starting point \( P_1 \) and thus the value of \( ft \) changes with that of \( \theta \) 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 \( \theta \).

7-5 Threading Leads

The thread lead in the threading mode (G32/G33, G34, G276 or G292) can be designated using a seven-digit value preceded by address \( F \) or eight-digit value preceded by address \( E \).

The thread lead command range is 0.0001 to 999.9999 mm/rev (\( F \) with 7 digits) or 0.0001 to 999.99999 mm/rev (\( E \)8-digit) (with unit of data setting of microns).

### Thread cutting (metric input)

<table>
<thead>
<tr>
<th>Unit of program data input</th>
<th>0.0001 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command address</td>
<td></td>
</tr>
<tr>
<td>F (mm/rev)</td>
<td></td>
</tr>
<tr>
<td>E (mm/rev)</td>
<td></td>
</tr>
<tr>
<td>E (Number of threads per inch)</td>
<td></td>
</tr>
<tr>
<td>Smallest input capacity</td>
<td>1 (=0.0001)</td>
</tr>
<tr>
<td></td>
<td>(1.=1.0000)</td>
</tr>
<tr>
<td>Range of command data</td>
<td>0.0001 to 500.0000</td>
</tr>
</tbody>
</table>

### Thread cutting (inch input)

<table>
<thead>
<tr>
<th>Unit of program data input</th>
<th>0.000001 inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command address</td>
<td></td>
</tr>
<tr>
<td>F (in./rev)</td>
<td></td>
</tr>
<tr>
<td>E (in./rev)</td>
<td></td>
</tr>
<tr>
<td>E (Number of threads per inch)</td>
<td></td>
</tr>
<tr>
<td>Smallest input capacity</td>
<td>1 (=0.000001)</td>
</tr>
<tr>
<td></td>
<td>(1.=1.000000)</td>
</tr>
<tr>
<td>Range of command data</td>
<td>0.000001 to 9.999999</td>
</tr>
</tbody>
</table>
7-6 **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 msec steps within a range from 1 to 500 msec. 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 msec steps within a range from 1 to 500 msec. (Normally, the same time constant is set for each axis.)

![Diagram](image)

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.

7-7 **Speed Clamp**

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 clamping is not applied to synchronous feed and threading.
7-8  Exact-Stop Check Command: 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    Z100.000 ;

The solid line indicates a feedrate pattern with the G09 available. The dotted line indicates a feedrate pattern without the G09.

Fig. 7-1  Validity of exact-stop check
4. Detailed description

A. Continuous cutting feed commands

![Diagram of Continuous Cutting Feed Commands](TEP039)

Fig. 7-2  Continuous cutting feed commands

B. Cutting feed commands with in-position status check

![Diagram of Block-to-Block Connection](TEP040)

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

In Fig. 7-2 and 7-3 above,

- **Ts**: Cutting feed acceleration/deceleration time constant
- **Lc**: In-position width

As shown in Fig. 7-3, in-position width Lc 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

\[ Td = Ts + (0 \text{ to } 14 \text{ms}) \]

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.
### 7-9 Exact-Stop Check Mode Command: 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;

### 7-10 Automatic Corner Override Command: 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 overriden 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](image-url)
<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 Ci.

<Setting parameters>
Set the following parameters as user parameters:
- **E22**: Override 0 to 100 (%)
- **F21**: Inner-corner maximum angle θ 0 to 180 (deg)
- **F22**: Deceleration zone Ci data 0 to 99999.999 (mm) or to 3937.000 (inches)

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

### B. Automatic corner rounding

![Diagram of automatic corner rounding](TEP046)

<Operation>
For inner corner cutting with automatic corner rounding, override will be effected as set in parameter through the deceleration zone Ci 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 $E22$ through deceleration zone $Ci$.

- Line-to-circular (outside offsetting) corner

The feed rate is automatically overridden with the preset value by the parameter $E22$ through deceleration zone $Ci$.

- Arc(internal compensation)-to-line corner

The feed rate is automatically overridden with the preset value by the parameter $E22$ through deceleration zone $Ci$.

Note: Data of deceleration zone $Ci$ at which automatic overriding occurs represents the length of the arc for a circular interpolation command.
The feed rate is automatically overridden with the preset value by the parameter \textbf{E22} through deceleration zone \( C_i \).

5. Correlations to other command functions

<table>
<thead>
<tr>
<th>Function</th>
<th>Override at corners</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cutting feedrate override</td>
<td>Automatic corner override is applied after cutting feed override.</td>
</tr>
<tr>
<td>Override cancel</td>
<td>Automatic corner override is not cancelled by override cancel.</td>
</tr>
<tr>
<td>Feed rate clamp</td>
<td>Valid (for the feed rate after automatic corner override)</td>
</tr>
<tr>
<td>Dry run</td>
<td>Automatic corner override is invalid.</td>
</tr>
<tr>
<td>Synchronous feed</td>
<td>A synchronous feed rate is automatically corner-overridden.</td>
</tr>
<tr>
<td>Skip (G31)</td>
<td>During tool radius compensation, G31 will result in a program error.</td>
</tr>
<tr>
<td>Machine lock</td>
<td>Valid</td>
</tr>
<tr>
<td>G00</td>
<td>Invalid</td>
</tr>
<tr>
<td>G01</td>
<td>Valid</td>
</tr>
<tr>
<td>G02, G03</td>
<td>Valid</td>
</tr>
</tbody>
</table>

6. Precautions

1. 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.

2. 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.
7-11 Tapping Mode Command: 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 geometry compensation command G61.1, exact-stop check mode command G61, automatic corner override command G62 or cutting mode command G64.

2. **Programming format**

   G63 ;

7-12 Cutting Mode Command: 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/cessation the machine between cutting feed blocks.

   The G64 command mode is cleared by geometry compensation command G61.1, exact-stop check mode command G61, 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 ;
7-13  Geometry Compensation/Accuracy Coefficient: G61.1/K

7-13-1  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 mode (G61), automatic corner override (G62), tapping mode (G63) and cutting mode (G64).

   The geometry compensation function is composed of the following five functions:
   1. Pre-interpolation acceleration/deceleration
   2. Feedforward control
   3. Precise vector compensation
   4. Optimum feed control (Optimum deceleration at corners, Optimum acceleration control)
   5. Feed clamping for circular interpolation

   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 G90 G0X100.Z100.
   G61.1G91G01F2000-selected geometry compensation function
   G64-cancellation of 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), Constant surface speed control, Threading.
   4. The pre-interpolation acceleration/deceleration is effective from the block of G61.1 onward.
   5. An alarm (due to overload or the like) may be caused as a result of the geometry compensation. In such a case reduce the rate of feed so as to prevent the alarm.
7-13-2 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_Z_,K30    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.
   N300 X_Z_           
   N400 ...
   
   <Example 2>
   
   N001 G61.1
   N200 G2I-10.,K30    Deceleration to 70% occurs for this block only.
   N300 G1X10.,K0      The accuracy coefficient is canceled from this block onward.
   N400 ...

4. Remarks
   - 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.
7-14 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 shape 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 rate of feed at the point of cutting (along the programmed contour).

Setting of command code G93 specifies the inverse time assignment mode.
In the G93 mode, the reciprocal of the machining time for the block of cutting feed (G01, G02 or G03) is to be assigned using an F-code. The setting range of data with address F is from 0.001 to 99999.999.

The rate of feed for the corresponding block is internally calculated from the length of the programmed contour and the value of the F-code.

- For linear interpolation (G01)

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

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

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

- For circular interpolation (G02 or G03)

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

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

[Arc radius] : mm (for metric system) or inches (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 G94 (feed per minute) and G95 (feed per revolution), is a modal G-code.
- Since they are not processed as modal codes in the G93 mode, F-codes 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

<table>
<thead>
<tr>
<th>No.</th>
<th>Message</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>940</td>
<td>NO INVERSE TIME OPTION</td>
<td>The Inverse Time Feed option is not present.</td>
</tr>
<tr>
<td>941</td>
<td>G93 MODE</td>
<td>An illegal G-code* has been set in the G93 mode.</td>
</tr>
</tbody>
</table>

* Illegal G-codes are:

<table>
<thead>
<tr>
<th>G-code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G31</td>
<td>Skip</td>
</tr>
<tr>
<td>G32, G33</td>
<td>Threading</td>
</tr>
<tr>
<td>G7, G8, G2</td>
<td>Fixed cycle</td>
</tr>
</tbody>
</table>

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
```

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:

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

   1. The setting range for dwell time is as follows:

<table>
<thead>
<tr>
<th>Unit of data setting</th>
<th>Range for address X</th>
<th>Range for address P</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001 mm, 0.0001 inches</td>
<td>0.001 to 999999.999 (sec)</td>
<td>1 to 99999999 (× 0.001 sec)</td>
</tr>
</tbody>
</table>

   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. 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 sec
  G04 X 5000 ; ........................................Dwell time = 5.0 sec
  G04 X 5. ; ...........................................Dwell time = 5.0 sec
  G04 P 5000 ; ........................................Dwell time = 5.0 sec
  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 sec (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:

<table>
<thead>
<tr>
<th>Unit of data setting</th>
<th>Range for address X</th>
<th>Range for address P</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001 mm, 0.0001 inches</td>
<td>0.001 to 99999.999 (rev)</td>
<td>1 to 99999999 (× 0.001 rev)</td>
</tr>
</tbody>
</table>

   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.

   ![Diagram](TEP053)
   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.

6. This function cannot be used unless the position detecting encoder is provided to the spindle.


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. Program 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 M99 and M99 are internally processed by the NC M-code signals ans 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 (form 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.

<table>
<thead>
<tr>
<th>No. 2 miscellaneous functions</th>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>×</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>B</td>
<td>O</td>
<td>×</td>
<td>O</td>
</tr>
<tr>
<td>C</td>
<td>O</td>
<td>O</td>
<td>×</td>
</tr>
</tbody>
</table>

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 10 V or –8 to +8 V
- Resolution................................................................ 1/4096 (2 to the power of –12)
- Load conditions ....................................................... 10 kiloohms
- 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 Constant Surface Speed Control ON/OFF: G96/G97

1. Function and purpose

This function controls automatically the spindle speed as the coordinates are changed during cutting in diametral direction so as to execute cutting by keeping constant the relative speed between tool tip and workpiece.

2. Programming format

G96 Ss Pp Rr; ..... Constant surface speed control ON
  s: Surface speed
  p: Axis for constant surface speed control
  r: Spindle for constant surface speed control

G97; ..................... Constant surface speed control OFF

3. Detailed description

1. Axis for constant surface speed control is to be set by address P.

   P1: First axis
   P2: Second axis

   X-axis (the first axis) is automatically selected if argument P is omitted.
2. Spindle for constant surface speed control is to be set by address R.
   R1: Turning spindle (see the figure below)
   R2: Turning spindle (see the figure below)

![Spindle Diagram]

The default value is “R1” (automatically set if argument R is omitted).

3. Control change program and actual movement

\[
\begin{align*}
G90 & \quad G96 & \quad G01 & \quad X50. & \quad Z100. & \quad S200; & \text{Spindle speed is controlled for a surface speed of 200 m/min.} \\
G97 & \quad G01 & \quad X50. & \quad Z100. & \quad F300 & \quad S500; & \text{Spindle speed is controlled for 500 rpm.} \\
M02;
\end{align*}
\]

The initial modal state will be resumed.

4. Remarks

1. The initial modal state (G96 or G97) can be selected by parameter F93 bit 0.
   \[
   \begin{align*}
   F93 & \text{ bit } 0 = 0: & \text{G97 (Constant surface speed control OFF)} \\
   & \text{ bit } 1: & \text{G96 (Constant surface speed control ON)}
   \end{align*}
   \]

2. The function is not effective for blocks of rapid motion (G00).
   The spindle speed calculated for the surface speed at the ending point is applied to the entire motion of a block of G00.

3. The last value of S in the control mode of G96 is stored during cancellation of the control (G97) and automatically made valid upon resumption of the control mode (G96).
   **Example:**
   \[
   \begin{align*}
   G96 \quad S50; & \quad 50 \text{ m/min or } 50 \text{ ft/min} \\
   G97 \quad S1000; & \quad 1000 \text{ rpm} \\
   G96 \quad X3000; & \quad 50 \text{ m/min or } 50 \text{ ft/min}
   \end{align*}
   \]

4. The constant surface speed control is effective even during machine lock.

5. Cancellation of the control mode (G96) by a command of G97 without specification of S (revs/min) retains the spindle speed which has resulted at the end of the last spindle control in the G96 mode.
   **Example:**
   \[
   \begin{align*}
   G97 \quad S800; & \quad 800 \text{ rpm} \\
   G96 \quad S100; & \quad 100 \text{ m/min or } 100 \text{ ft/min} \\
   G97; & \quad x \text{ rpm}
   \end{align*}
   \]
   The speed \(x\) denotes the spindle speed of G96 mode at the end of the preceding block.

6. The constant surface speed control does not apply to the milling spindle.
10-3 Spindle Clamp Speed 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 clamping

3. Detailed description

1. 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 (rpm). 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.

2. Spindle for speed clamping is to be set by address R.

   - R1: Turning spindle (see the figure below)
   - R2: Turning spindle (see the figure below)
   - R3: Milling spindle

![Diagram of spindle and turret configurations](D740PB006)

**Note:** The default value is “R1” (automatically set if argument R is omitted). In this case the speed of turning spindle 2 can be raised up to the highest value in accordance with the machine specification concerned.
- NOTE -
## 11 TOOL FUNCTIONS

### 11-1 Tool Function [for ATC Systems]

A next tool and tool offset number can be designated for the machine provided with ATC function by commanding T-code in the format shown below. The next tool refers to a tool used for the next machining, which can be assigned when it is currently accommodated in the magazine. The next tool in the magazine can be indexed at ATC position beforehand by commanding the next tool, thus permitting reduced ATC time.

\[ T \text{◇◇◇} \text{◇◇◇} \text{M6} ; \]

- ◇◇◇: Number of the tool to be changed for
- ◇◇◇: Tool ID code
- ◇◇◇: Number of the tool to be used next

Use two digits after the decimal point as follows to designate the tool ID code with reference to the settings on the TOOL DATA display:

#### <Normal tools>

<table>
<thead>
<tr>
<th>ID code</th>
<th>w/o</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
<th>J</th>
<th>K</th>
<th>L</th>
<th>M</th>
</tr>
</thead>
<tbody>
<tr>
<td>◇◇◇</td>
<td>00</td>
<td>01</td>
<td>02</td>
<td>03</td>
<td>04</td>
<td>05</td>
<td>06</td>
<td>07</td>
<td>08</td>
<td>09</td>
<td>11</td>
<td>12</td>
<td>13</td>
</tr>
<tr>
<td>ID code</td>
<td>N</td>
<td>P</td>
<td>Q</td>
<td>R</td>
<td>S</td>
<td>T</td>
<td>U</td>
<td>V</td>
<td>W</td>
<td>X</td>
<td>Y</td>
<td>Z</td>
<td></td>
</tr>
<tr>
<td>◇◇◇</td>
<td>14</td>
<td>15</td>
<td>16</td>
<td>17</td>
<td>18</td>
<td>19</td>
<td>21</td>
<td>22</td>
<td>23</td>
<td>24</td>
<td>25</td>
<td>26</td>
<td></td>
</tr>
</tbody>
</table>

#### <Heavy tools>

<table>
<thead>
<tr>
<th>ID code</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
<th>J</th>
<th>K</th>
<th>L</th>
<th>M</th>
</tr>
</thead>
<tbody>
<tr>
<td>◇◇◇</td>
<td>61</td>
<td>62</td>
<td>63</td>
<td>64</td>
<td>65</td>
<td>66</td>
<td>67</td>
<td>68</td>
<td>69</td>
<td>71</td>
<td>72</td>
<td>73</td>
</tr>
<tr>
<td>ID code</td>
<td>N</td>
<td>P</td>
<td>Q</td>
<td>R</td>
<td>S</td>
<td>T</td>
<td>U</td>
<td>V</td>
<td>W</td>
<td>X</td>
<td>Y</td>
<td>Z</td>
</tr>
<tr>
<td>◇◇◇</td>
<td>74</td>
<td>75</td>
<td>76</td>
<td>77</td>
<td>78</td>
<td>79</td>
<td>81</td>
<td>82</td>
<td>83</td>
<td>84</td>
<td>85</td>
<td>86</td>
</tr>
</tbody>
</table>

### 11-2 Tool Function [4-Digit T-Code for Turret-Indexing Systems]

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.

Use bit 4 of parameter F162 to select the number of digits for the tool function (0 or 1 for 4- or 6-digit T-code).

\[ T \text{◇◇◇} \text{◇◇◇} ; \]

- Tool ID code
- Tool offset number
- Tool 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.
11-3 Tool Function [6-Digit T-Code for Turret-Indexing Systems]

This function is also used to designate the tool number and offset number. Of a six-digit integer at address T, upper and lower three digits are respectively used to specify the tool number and offset number. See the above description of the 4-digit T-code for the meaning of the decimal fractions.

The available range of T-codes depends on the machine specifications. For further details, refer to the operating manual of the relevant machine.

Only one T-code can be included in a block.

Use bit 4 of parameter F162 to select the number of digits for the tool function (0 or 1 for 4- or 6-digit T-code).

\[ T \quad \text{Tool ID code} \quad \text{Tool offset number} \quad \text{Tool number} \]

11-4 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.
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).

Data items used for turning tools are as follows:
- X, Y, Z, and Nose-R of Geometric Offset,
- X, Y, Z, and Nose-R of Wear Compensation, and
- Direction.

Data items used for milling tools are as follows:
- Z and Nose-R of Geometric Offset, and
- Z and Nose-R of Wear Compensation.

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.

\[
(H_n) = c_n + f_n \\
(D_n) = g_n + k_n \\
(H_n) (x, y, z) = (a_n + d_n, b_n + e_n, c_n + f_n) \\
(D_n) = g_n + k_n
\]
3. Setting TOOL OFFSET data

As tabulated below, various types of offset data can be set for one offset number: Geometric offset and Wear compensation data (X, Y, Z) for tool length, Geometric offset and Wear compensation data for tool diameter, and Direction.

Data items used for milling tools are: Geometric offset Z and Wear compensation Z (Length) and Geometric offset and Wear compensation (Diameter).
Data items used for turning tools are: Geometric offset X, Y, Z and Wear compensation X, Y, Z (Length), Geometric offset and Wear compensation (Diameter), and Direction.

<table>
<thead>
<tr>
<th>Tool offset No.</th>
<th>Length (H)</th>
<th>Diameter (D)/(Position offset) Nose-R (D)</th>
<th>Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X Y Z</td>
<td>Geometric offset Wear comp. X Y Z</td>
<td>Nose-R</td>
</tr>
<tr>
<td>1</td>
<td>a₁ b₁ c₁ d₁ e₁ f₁</td>
<td>g₁ k₁ l₁</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>a₂ b₂ c₂ d₂ e₂ f₂</td>
<td>g₂ k₂ l₂</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>a₃ b₃ c₃ d₃ e₃ f₃</td>
<td>g₃ k₃ l₃</td>
<td></td>
</tr>
<tr>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>n</td>
<td>aₙ bₙ cₙ dₙ eₙ fₙ</td>
<td>gₙ kₙ lₙ</td>
<td></td>
</tr>
</tbody>
</table>

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 diameter.
- 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 alarm 839 ILLEGAL OFFSET No. will result if an offset number is set which is greater than the maximum available number.
- 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.

<table>
<thead>
<tr>
<th>Metric Offset</th>
<th>Inch Offset</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric offset XYZ</td>
<td>±1999.9999 mm ±84.5000 in.</td>
</tr>
<tr>
<td>Geometric offset Nose-R</td>
<td>±999.9999 mm ±84.5000 in.</td>
</tr>
<tr>
<td>Wear compensation XYZ</td>
<td>±99.9999 mm ±9.9999 in.</td>
</tr>
<tr>
<td>Wear compensation Nose-R</td>
<td>±9.9999 mm ±0.9999 in.</td>
</tr>
</tbody>
</table>

Note: The tool offset number (H- or D-code) is not 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 execution of move commands to be shifted through the previously set offset amount for each axis. Any deviations between programmed tool lengths/diameters and actual lengths/diameters can be set as offset data using these commands to make the program more flexible.

2. Programming format

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

There are two types of tool length offset: for milling tools and for turning tools.

For milling tools: Length offsetting is executed on the axis specified in the G43 or G44 block (unless the length offset axis is fixed to “Z” by a parameter setting [F92 bit 3 = 1]).

For turning tools: Length offsetting is executed on all axes for which offset amounts are registered (and G49 cancels all offset amounts concerned).

Add an argument P as follows to designate the tool type. Note that the offset type for turning tools is to be selected in a measurement program using a touch sensor.

<table>
<thead>
<tr>
<th>Tool type</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Milling tool</td>
<td>The value of P is 0 (P0), or P is omitted.</td>
</tr>
<tr>
<td>Turning tool</td>
<td>The value of P is 1 (P1).</td>
</tr>
</tbody>
</table>

3. Detailed description

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

A. Tool length offsetting for milling tools

1. Z-axis motion distance

   G43Z±zHh₁ ±z + {±(ℓ₁) – (±ℓ₀)} Positive-direction offset by length offset amount
   G44Z±zHh₁ ±z + {±(ℓ₁) – (±ℓ₀)} Negative-direction offset by length offset amount
   G49Z±z ±z – (±ℓ₀) Cancellation of the offset amount

   ℓ₁: BA62 + Value of offset No. h₁
   ℓ₀: 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

For absolute data input
(H01: Z = 95.)

N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0 X0 B0
N003 T01 T00 M06
N004 G90 G54 G00 B45.
N005 G68 X0 Y0 Z0 I0 J1 K0 R45.
N006 G00 X0 Y0
N007 G43 Z5. H01
N008 G01 Z-50. F100
3. Supplement

1) Tool length offset data can be set for the X-axis, the Y-axis, and additional axes, as well as the Z-axis. Whether the offset data is to be used for the Z-axis only or for the axis specified in the G43 or G44 block can be selected using bit 3 of parameter F92.

2) Even if multiple axis addresses are programmed in one block, offsetting will be performed on only one of the axes and the priority in this case is as follows:

\[ C > Z > B > Y > X > A \]

**Example:**

```
G43 Xx1 Hh1
    ...  \{ Positive-direction offset on the X-axis, and cancellation \}
G49 Xx2
G44 Yy3 Hh3
    ...  \{ Negative-direction offset on the Y-axis, and cancellation \}
G49 Yy4
G43 αα5 Hh5
    ...  \{ Pos.-direct. offset on the additional axis, and cancellation \}
G49 αα6
G43 Xx7 Yy7 Zz7 Hh7 ............... Positive-direction offset on the Z-axis
```

3) Offsetting is always performed on the Z-axis if no axis addresses are programmed in the G43 or G44 block.

**Example:**

```
G43 Hh1
    ...  \{ Offsetting on the Z-axis, and cancellation \}
G49
```

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

**Example:**

```
G43 Hh1
    ...  \{ Upon completion of return to the reference point (zero point), the offset stroke is cleared. \}
G28 Zz2
G43 Hh3
G49 G28 Zz2 \{ Reference point return after a Z-axis motion at the current position for clearing the offset amount \}
```

5) 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 parameter \( F114 \) bit 3 = 0, the execution of G49 may result in interference with the workpiece since automatic cancellation moves the tool on the Z-axis in minus direction through the distance equivalent to the tool length plus parameter \( BA62 \) (offset amount from the B-axis to the spindle nose). When parameter \( F114 \) bit 3 = 1, axis motion commands in the block of G49 will all be ignored.

Use an H00 command, rather than a G49 command, if G43/G44 mode is to be cancelled temporarily.

6) The alarm \( 839 \) ILLEGAL OFFSET No. will occur if an offset number exceeding the machine specifications is set.

7) When tool offset data and MAZATROL tool data are both validated, offsetting is executed by the sum of the two data items concerned.
8) In order to apply length offset for a milling tool in its axial direction, give the corresponding G68 command (Coordinate Rotation) following a B-axis angular motion command.

B. Tool length offsetting for turning tools

1. Z- and X-axis motion distance

\[
\begin{align*}
G43X\pm x & \pm zHh_1P1 \quad \{ \pm x + [\pm h_{i1x} - (\pm h_{b0x})] \} \quad \text{Positive-direction length offset} \\
G44X\pm x & \pm zHh_1P1 \quad \{ \pm x + [\pm h_{i1x} - (\pm h_{b0x})] \} \quad \text{Negative-direction length offset} \\
G49X\pm x & \pm zHh_1P1 \quad \{ \pm z - (\pm h_{i1z}) \} \quad \text{Cancellation of the offset amount} \\
& \quad \{ \pm x - (\pm h_{i1x}) \} \quad \text{Cancellation of the offset amount}
\end{align*}
\]

\( h_{i1x} \): BA62 + X-axis value of offset No. \( h_1 \)
\( h_{i1z} \): Z-axis value of offset No. \( h_1 \)
\( h_{b0x} \): X-axis offset amount existing before the G43 or G44 block
\( h_{b0z} \): Z-axis offset amount existing before the G43 or G44 block

P1: Selection of the length offsetting type for turning tools

Irrespective of whether absolute or incremental programming method is used, the actual ending point coordinates are calculated by offsetting the programmed end point coordinates through the offset amount. Offsetting for a turning tool is executed on all axes for which offset amounts are registered.

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

As for an angular application of the tool, the X- and Z-axis component vectors for length offsetting are automatically computed for the particular application angle, as shown below:

X-axis offset amount = (“Geometric” Z + BA62) sin \( \theta \) + (“Geometric” X) cos \( \theta \)
Z-axis offset amount = (“Geometric” Z + BA62) cos \( \theta \) – (“Geometric” X) sin \( \theta \)

Example 1: B-axis position = 90°
Example 2: \( B\)-axis = 45°

<table>
<thead>
<tr>
<th>Geometric offset X</th>
<th>X-axis length offset amount</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric offset Z</td>
<td>Z-axis length offset amount</td>
</tr>
</tbody>
</table>

2. Sample programs

For absolute data input

\( H01: Z = 95, X = -5 \)

N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0
N003 T01 T00 M06
N004 G90 G54 X100. Y0
N005 G43 X10. Z5. H01 P1
N006 G01 Z-50. F100

For incremental data input

\( H01: Z = 95, X = -5 \)

N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0
N003 T01 T00 M06
N004 G90 G54 X100. Y0
N005 G91 G43 X-90. Z-195. H01 P1
N006 G01 Z-55. F100
Workpiece zero point

For absolute data input

(H01: Z = 95. X = –5.)
N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0 B0
N003 T01 T00 M06
N004 G90 G0 B45.
N005 G54
N006 G43 X10. Z5. H01 P1
N007 G01 Z–50. F100

For incremental data input

(H01: Z = 95. X = –5.)
N001 G90 G94 G00 G40 G80
N002 G91 G28 X0 Z0 B0
N003 T01 T00 M06
N004 G90 G0 B45.
N005 G54
N007 G01 Z–55. F100

3. MAZATROL “Wear Compensation” data for turning tools

Of MAZATROL tool data items, “Length A” and “Length B” correspond to the length and width of the tool, respectively, and “Wear Comp” values are used for tool compensation on the relevant controlled axes.

Set the following parameter to “1” to use the MAZATROL wear compensation data.

<table>
<thead>
<tr>
<th>F111 bit 5</th>
<th>MAZATROL wear comp. valid/invalid</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Invalid (Not used for EIA/ISO programs)</td>
</tr>
<tr>
<td>1</td>
<td>Valid (Used also for EIA/ISO programs)</td>
</tr>
</tbody>
</table>
4. Supplement

1) For turning tools, length offsetting is executed on all axes for which offset amounts are registered (and G49 cancels all offset amounts concerned).

Set “P1” in the block of G43 or G44 to select the length offsetting type for turning tools.

Example:

G43 Xx₁ Zz₁ Hh₁ P1 Positive-direction offset on X and Z (and Y)

G49 Xx₂ Cancellation of offsetting on X and Z (and Y)

2) Offsetting is always performed on all the axes concerned even if no axis addresses are programmed in the G43 or G44 block.

Example:

G43 Hh₁ P1

M

G49

3) If reference point (zero point) return is performed in the offsetting mode, the mode is cancelled after completion of the returning operation (if F94 bit 2 = 0).

Example:

G43 Hh₁

M

G28 Zz₂

G43 Hh₁

G49 G28 Zz₂

4) If command G49 or H00 is executed, offsetting will be immediately cancelled (the corresponding axis will move to clear the offset amount to zero).

When parameter F114 bit 3 = 0, the execution of G49 may result in interference with the workpiece since automatic cancellation moves the tool on the Z-axis in minus direction through the distance equivalent to the tool length plus parameter BA62 (offset amount from the B-axis to the spindle nose). When parameter F114 bit 3 = 1, axis motion commands in the block of G49 will all be ignored.

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) As for offsetting by T-codes, offset amount is not actually made valid until a movement command is executed.

Example:

G28 Xx₃

G28 Zz₃

T01 M6 ............Offset amount of T01 validated, but no axis movement.

G00 Xx₃ ............Motion on the X-axis only with offsetting.

Zz₃ ............Motion on the Z-axis with offsetting.

7) Length offset is automatically executed in the axial direction of a turning tool for any angle of the B-axis. There is no need to give a command of G68 (Coordinate Rotation), which is required in the case of milling tools.
C. Tool length offsetting for angular/cranked milling tools

1. Z- and X-axis motion distance

\[
\begin{align*}
\text{G43} & \quad X_{\pm x} Z_{\pm z} H_{h1} P2 \quad \{ z + [\pm (l_{h1z} - (\pm l_{h0z}))] \quad \text{Positive-direction length offset} \\
\text{G44} & \quad X_{\pm x} Z_{\pm z} H_{h1} P2 \quad \{ x + [\pm (l_{h1x} - (\pm l_{h0x}))] \quad \text{Positive-direction length offset} \\
\text{G49} & \quad X_{\pm x} Z_{\pm z} \quad [ z - (\pm l_{h1z})] \quad \text{Cancellation of the offset amount} \\
& \quad [ x - (\pm l_{h1x})] \quad \text{Cancellation of the offset amount}
\end{align*}
\]

\( l_{h1x} \): BA62 + X-axis value of offset No. \( h_1 \)

\( l_{h1z} \): Z-axis value of offset No. \( h_1 \)

\( l_{h0x} \): X-axis offset amount existing before the G43 or G44 block

\( l_{h0z} \): Z-axis offset amount existing before the G43 or G44 block

P2: Selection of the length offsetting type for angular/cranked milling tools

Irrespective of whether absolute or incremental programming method is used, the actual ending point coordinates are calculated byoffsetting the programmed end point coordinates through the offset amount. Offsetting for an angular/cranked milling tool is executed on all axes for which offset amounts are registered.

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

2. Sample programs

```
D740PB0077

For absolute data input
(H01: Z = 100, X = –30.)
N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0 X0 Y0
N003 T01 T00 M06
N004 G90 G54 X100.
N005 G43 Z30. X60. H01 P2
N006 G01 Z-50. F100
N007 G19
N008 G82R1.Z0.F10.
N009 G80G17
```

D740PB0077
3. Supplement

1) Tool length offsetting can be executed on the Y-axis as well as on the Z- and X-axis. When parameter F92 bit 3 = 1, however, length offsetting (by a block of G43/G44 P2) only occurs on the Z-axis.

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

   **Example:**
   
   ```
   G43 P2 Hh1
   G28 Xx2 Zz2
   G43 Hh1
   G49 G28 Xx2 Zz2
   ```

   Upon completion of return to the reference point (zero point), the offset amount is cleared.

   Reference point return after an X- and Z-axis motion at the current position for clearing the offset amount

3) If command G49 or H00 is executed, offsetting will be immediately cancelled (the corresponding axis will move to clear the offset amount to zero).

   When parameter F114 bit 3 = 0, the execution of G49 may result in interference with the workpiece since automatic cancellation moves the tool on the Z-axis in minus direction through the distance equivalent to the tool length plus parameter BA62 (offset amount from the B-axis to the spindle nose). When parameter F114 bit 3 = 1, axis motion commands in the block of G49 will all be ignored.

   Use an H00 command, rather than a G49 command, if G43/G44 mode is to be cancelled temporarily.

4) The alarm **839 ILLEGAL OFFSET No.** will occur if an offset number exceeding the machine specifications is set.

5) When tool offset data and MAZATROL tool data are both validated, offsetting is executed by the sum of the two data items concerned.

6) In order to apply length offset for a milling tool in its axial direction, give the corresponding G68 command (Coordinate Rotation) following a B-axis angular motion command.

7) Parameter BA62 is out of all relation to tool length offsetting in the mode of ram spindle offset (G52.1).
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).

<table>
<thead>
<tr>
<th>Pattern</th>
<th>Data items used (Display and Data item names)</th>
<th>Parameter</th>
<th>Programming method</th>
</tr>
</thead>
<tbody>
<tr>
<td>[1] TOOL OFFSET</td>
<td>Offset data items</td>
<td>F94 bit 7</td>
<td>F93 bit 3</td>
</tr>
<tr>
<td>[2] TOOL DATA</td>
<td>LENGTH</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td>LENGTH + LENG. No. LENGTH + LENG. CO.</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>[3] TOOL DATA</td>
<td>LENG. No. LENG. CO.</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>[4] TOOL OFFSET + TOOL DATA</td>
<td>Offset data items LENGTH</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>
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.

<table>
<thead>
<tr>
<th>Type</th>
<th>Inclusion of offset amount for the axis of rotation</th>
<th>Exclusion of offset amount for the axis of rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>F168 bit 2 = 0</td>
<td>F168 bit 2 = 1</td>
</tr>
<tr>
<td>Description</td>
<td>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.</td>
<td>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.)</td>
</tr>
</tbody>
</table>

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:

<table>
<thead>
<tr>
<th>Operation</th>
<th>Motion commands for orthogonal axes</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43.1Hh;</td>
<td>G90; G43.1XxYyZzHh;</td>
</tr>
<tr>
<td>not given</td>
<td>given</td>
</tr>
</tbody>
</table>

- No motions occur at all. (A motion by the offset amount does not occur.)
- A motion occurs for a point of the specified coordinates, inclusive of the amount for length offset in tool-axis direction.

- “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.

Machining program

<table>
<thead>
<tr>
<th>Block</th>
<th>Command Description</th>
<th>X Position</th>
<th>Z Position</th>
</tr>
</thead>
<tbody>
<tr>
<td>N01</td>
<td>G29XY2BC</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N02</td>
<td>G90G54</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N03</td>
<td>G0 B90.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N04</td>
<td>G43.1H1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N05</td>
<td>X85.000 Y0.000 Z0.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N06</td>
<td>G1 F10000.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N07</td>
<td>X85.000 Z0.000 B90.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N08</td>
<td>X55.000 Z0.000 B89.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N09</td>
<td>X54.992 Z2.878 B87.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N10</td>
<td>X54.866 Z23.837 B86.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N12</td>
<td>X54.699 Z25.749 B84.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N13</td>
<td>X54.590 Z26.703 B83.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N14</td>
<td>X54.465 Z27.655 B82.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N15</td>
<td>X54.323 Z28.604 B81.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N16</td>
<td>X54.164 Z29.551 B80.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N17</td>
<td>X54.000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N18</td>
<td>G49</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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.

<table>
<thead>
<tr>
<th>Block</th>
<th>Initial offset amount</th>
<th>Initial offset amount + 5</th>
<th>Initial offset amount – 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>N08</td>
<td>X55.000, Z0.000</td>
<td>X60.000, Z0.000</td>
<td>X50.000, Z0.000</td>
</tr>
<tr>
<td>N09</td>
<td>X54.992, Z0.960</td>
<td>X59.992, Z1.047</td>
<td>X49.992, Z0.873</td>
</tr>
<tr>
<td>N10</td>
<td>X54.966, Z1.919</td>
<td>X59.963, Z2.094</td>
<td>X49.970, Z1.745</td>
</tr>
<tr>
<td>N11</td>
<td>X54.925, Z2.878</td>
<td>X59.918, Z3.140</td>
<td>X49.931, Z2.617</td>
</tr>
<tr>
<td>N12</td>
<td>X54.866, Z3.837</td>
<td>X59.854, Z4.185</td>
<td>X49.878, Z3.488</td>
</tr>
<tr>
<td>N13</td>
<td>X54.791, Z4.794</td>
<td>X59.772, Z5.229</td>
<td>X49.810, Z4.358</td>
</tr>
<tr>
<td>N14</td>
<td>X54.699, Z5.749</td>
<td>X59.671, Z6.272</td>
<td>X49.726, Z5.226</td>
</tr>
<tr>
<td>N15</td>
<td>X54.590, Z6.703</td>
<td>X59.553, Z7.312</td>
<td>X49.627, Z6.093</td>
</tr>
</tbody>
</table>
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

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

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

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

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.

<table>
<thead>
<tr>
<th>G45 command</th>
<th>G46 command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extended thru offset stroke only</td>
<td>Contracted thru offset stroke only</td>
</tr>
<tr>
<td>Internal calculation</td>
<td>Internal calculation</td>
</tr>
<tr>
<td>Moving stroke</td>
<td>Moving stroke</td>
</tr>
<tr>
<td>Starting point</td>
<td>Ending point</td>
</tr>
<tr>
<td>Starting point</td>
<td>Ending point</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>G47 command</th>
<th>G48 command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extended thru twice the offset stroke</td>
<td>Contracted thru twice the offset stroke</td>
</tr>
<tr>
<td>Internal calculation</td>
<td>Internal calculation</td>
</tr>
<tr>
<td>Moving stroke</td>
<td>Moving stroke</td>
</tr>
<tr>
<td>Starting point</td>
<td>Ending point</td>
</tr>
<tr>
<td>Starting point</td>
<td>Ending point</td>
</tr>
</tbody>
</table>

\[
\text{(Program command value)} \pm \text{(Offset stroke)} = \text{(Moving stroke after offset)}
\]

2. Programming format

<table>
<thead>
<tr>
<th>Programming format</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G45 Xx Dd</td>
<td>To extend a moving stroke by the offset stroke which has been set in the offset memory.</td>
</tr>
<tr>
<td>G46 Xx Dd</td>
<td>To contract a moving stroke by the offset stroke which has been set in the offset memory.</td>
</tr>
<tr>
<td>G47 Xx Dd</td>
<td>To extend a moving stroke by twice the offset stroke which has been set in the offset memory.</td>
</tr>
<tr>
<td>G48 Xx Dd</td>
<td>To contract a moving stroke by twice the offset stroke which has been set in the offset memory.</td>
</tr>
</tbody>
</table>
3. Detailed description

- Programming based on incremental data is shown below.

<table>
<thead>
<tr>
<th>Tape command</th>
<th>Stroke of movement by equivalent tape command (selected offset stroke = ( l ))</th>
<th>Example (with ( x = 1000 ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>G45 Xx Dd</td>
<td>( X { x + l } )</td>
<td>( \ell = 10 ) ( X = 1010 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \ell = -10 ) ( X = 990 )</td>
</tr>
<tr>
<td>G45 X–x Dd</td>
<td>( X { x - l } )</td>
<td>( \ell = 10 ) ( X = 990 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \ell = -10 ) ( X = 1010 )</td>
</tr>
<tr>
<td>G46 Xx Dd</td>
<td>( X { x + 2l } )</td>
<td>( \ell = 10 ) ( X = 1020 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \ell = -10 ) ( X = 980 )</td>
</tr>
<tr>
<td>G46 X–x Dd</td>
<td>( X { x - 2l } )</td>
<td>( \ell = 10 ) ( X = 980 )</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( \ell = -10 ) ( X = 1020 )</td>
</tr>
</tbody>
</table>

- 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.

Program command: G48 X20.000
Offset stroke: + 15.000
Real move: X–10.000

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

<table>
<thead>
<tr>
<th>NC command</th>
<th>G45 X0 D01</th>
<th>G45 X–0 D01</th>
<th>G46 X0 D01</th>
<th>G46 X–0 D01</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent command</td>
<td>X1234</td>
<td>X–1234</td>
<td>X–1234</td>
<td>X1234</td>
</tr>
</tbody>
</table>

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 arc 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

Program path

Tool center path

N1 G46 G00 Xx1 Yy1 Dd1
N2 G45 G01 Yy2 Ff2
N3 G45 G03 Xx3 Yy3 Ii3
N4 G01 Xx4

X

Y
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.

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

```
N100 G91 G46 G00 X40. Y40. D01
N101 G45 G01 X100. F200
N103 G45 G01 Y40.
N104 G46 X0
N106 G45 G01 Y0
N107 G47 X-30.
N110 Y30.
N111 G45 X-30.
N113 G45 G01 Y-20.
N114 X10.
N115 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, moreover, nose-R compensation can be performed according to the designated direction.

2. Programming format

<table>
<thead>
<tr>
<th>Programming format</th>
<th>Function</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40X_ Y_</td>
<td>To cancel a tool radius compensation</td>
<td></td>
</tr>
<tr>
<td>G41X_ Y_</td>
<td>To compensate a tool radius (to the left)</td>
<td></td>
</tr>
<tr>
<td>G42X_ Y_</td>
<td>To compensate a tool radius (to the right)</td>
<td></td>
</tr>
<tr>
<td>G38 I_ J_</td>
<td>To change and hold an offset vector</td>
<td>These commands can be given in the mode of tool radius compensation.</td>
</tr>
<tr>
<td>G39</td>
<td>To interpolate a corner arc</td>
<td></td>
</tr>
</tbody>
</table>

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 offsetting is performed for axes other than those corresponding or parallel to the selected plane. See the section on “Plane Selection” 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 compensation 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 five blocks in succession is completed, irrespective of whether the single-block operation mode is used. During compensation, five 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

\[ \theta_s \ G42 \ r (Offset \ stroke) \]
Starting point

\[ Linear \rightarrow Linear \]
Programmed path
Tool center path

\[ Linear \rightarrow Arc \]
Programmed path
Tool center path

B. For the corner exterior (obtuse angle) \([90^\circ \leq \theta < 180^\circ]\)

(Type A/B selection is possible with a predetermined parameter.)

\[ \theta_s \ G41 \ r \ (Offset \ stroke) \]
Starting point

\[ Linear \rightarrow Linear \ (Type \ A) \]
Programmed path
Tool center path

\[ Linear \rightarrow Arc \ (Type \ A) \]
Programmed path
Tool center path

\[ Linear \rightarrow Linear \ (Type \ B) \]
Programmed path
Point of intersection

\[ Linear \rightarrow Arc \ (Type \ B) \]
Programmed path
Point of intersection

MEP068
MEP069
MEP070
C. **For the corner exterior (sharp angle) \([\theta < 90^\circ]\)**

(Type A/B selection is possible with a predetermined parameter.)

<table>
<thead>
<tr>
<th>Linear (\rightarrow) Linear (Type A)</th>
<th>Linear (\rightarrow) Arc (Type A)</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Linear (\rightarrow) Linear (Type A) diagram" /></td>
<td><img src="image2" alt="Linear (\rightarrow) Arc (Type A) diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Linear (\rightarrow) Linear (Type B)</th>
<th>Linear (\rightarrow) Arc (Type B)</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image3" alt="Linear (\rightarrow) Linear (Type B) diagram" /></td>
<td><img src="image4" alt="Linear (\rightarrow) Arc (Type B) diagram" /></td>
</tr>
</tbody>
</table>

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 **four or more blocks** that do not involve movement of axes during the compensation mode may result in excessive or insufficient cutting.
A. For the corner exterior

**Linear → Linear (90° ≤ θ < 180°)**

**Linear → Linear (0° < θ < 90°)**

**Linear → Arc (90° ≤ θ < 180°)**

**Linear → Arc (0° < θ < 90°)**

**Arc → Linear (90° ≤ θ < 180°)**

**Arc → Linear (0° < θ < 90°)**

**Arc → Arc (90° ≤ θ < 180°)**

**Arc → Arc (0° < θ < 90°)**
B. For the corner interior

Linear → Linear (Obtuse angle)

Linear → Arc (Obtuse angle)

Arc → Linear (Obtuse angle)

Arc → Arc (Obtuse angle)

Arc → Arc (Sharp angle)
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 li 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.

<table>
<thead>
<tr>
<th>G-code</th>
<th>Offset stroke sign</th>
<th>+</th>
<th>–</th>
</tr>
</thead>
<tbody>
<tr>
<td>G41</td>
<td>Left side offset</td>
<td></td>
<td>Right side offset</td>
</tr>
<tr>
<td>G42</td>
<td>Right side offset</td>
<td></td>
<td>Left side offset</td>
</tr>
</tbody>
</table>

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 12-5-7, 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

![Diagram of reference-point return command](image)

(G41)
N7 X–30. Y+60.
N8 X–70. Y–40.

Intermediate point

5. Blocks that do not include movement

The blocks listed below are referred to as those which do not include movement:

- M03................................................. M command
- 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 block that does not include movement is set at the start of compensation

Vertical offsetting will be performed on the next move block.

![Diagram of vertical offsetting](image)
Offset vectors, however, will not be generated if four or more blocks that do not include move commands appear in succession.

```
N1  X30. Y60.
N2  G41 D10
N3  G4 X1000
N4  F100
N5  S500
N6  M3
```

(Point of intersection)

**MEP087**

B. **When a block that does not include movement is set in the compensation mode**

Usual crossing-point vectors will be generated unless four or more blocks that do not include movement appear in succession.

```
N1  G41 X30. Y60. D10
N2  G4 X1000
N3  F100
N4  S500
N5  M3
```

(Point of intersection)

**MEP088**

Vertical offset vectors will be generated at the end point of preceding block if four or more blocks that do not include movement appear in succession.

```
N6  G91 X100. Y200.
N7  G04 F1000
N8  X200.
```

**MEP089**
C. When a block without move commands 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.
12-5-4 Corner movement

If multiple offset vectors are generated at connections between move command blocks, the tool will move linearly between those vectors. This action is referred to as corner movement. If the multiple vectors do not agree, the tool will move around the corresponding corners (but this movement belongs to the next block). During single-block operation, the section of (Preceding block + Corner movement) is executed as one block and the remaining section of (Connections movement + Next block) is executed during next movement as another block.

![Diagram of corner movement]( MEP096)

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

![Diagram of interruption without movement]( MEP097)
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 Nose-R compensation

1. Tool nose point (Direction)

   To apply the tool radius compensation function to turning tools for nose-R compensation, register
   the data sets of nose radius and hypothetical nose point (“Nose-R” and “Direction”) for the
   required tools on the TOOL OFFSET display (type C). “Hypothetical nose point” refers here to
   the reference position for preparing program data of machining with the particular tool (see the
   figure below).

   ![Diagram showing Nose-R compensation]

2. Detailed description

   1. Register the compensation amount (nose radius) together with the nose point No.
      (Direction) under a tool offset number.

   2. If four or more blocks without move commands exist in five continuous blocks, overcutting or
      undercutting may result. However, blocks for which optional block skip is valid are ignored.

   3. Nose radius compensation function is also valid for turning fixed cycles. A roughing cycle,
      however, is carried out with respect to the finishing contour compensated for nose-R with
      the compensation being temporarily canceled, and upon completion of the roughing, the
      compensation mode is retrieved.

   4. For threading commands, compensation is temporarily cancelled in one block before.

   5. The compensation plane, movement axes and next advance direction vectors depend upon
      the plane selection with G17, G18 or G19.

      G17  XY plane; X, Y; I, J
      G18  ZX plane; Z, X; K, I
      G19  YZ plane; Y, Z; J, K
12-5-7 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 diameter offsetting cancellation mode. If such updating is done during the offset 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).

   ![Diagram](MEP101)

   G41 offset stroke positive or G42 offset stroke negative
   (a)

   G41 offset stroke negative or G42 offset stroke positive
   (b)

4. Offset data item “Direction”
   As for data item “Direction” on the TOOL OFFSET display, specify the nose point direction for turning tools. Always set “Direction = 0” for offset numbers to be used for radius compensation of milling tools.
5. M-codes that prohibit pre-reading

When the presence of an M-code that prohibits pre-reading (M0, M1, or M2) interrupts the normal pre-reading of 5 blocks, the calculation occurs on the basis of the less information and the obtained path of radius compensation may differ from that which would result in normal cases.

12-5-8 Offset number updating during compensation mode

In principle, offset numbers should not be updated during the compensation mode. If updating is done, the tool center will move as shown below.

If an offset number (offset data) is updated

\[
\begin{align*}
G41 & \quad G01 & \quad Dr_1 \\
\vdots & \quad \vdots & \quad \alpha = 0, 1, 2, 3 \\
N101 & \quad G0\alpha & \quad Xx_1 \quad Yy_1 \\
N102 & \quad G0\alpha & \quad Xx_2 \quad Yy_2 \quad Dr_2 \quad \text{To change an offset number} \\
N103 & \quad Xx_3 \quad Yy_3
\end{align*}
\]

1. Line-to-line movement
2. Line-to-arc movement

3. Arc-to-arc movement
12-5-9 Excessive cutting due to tool radius compensation

If an interference check function (as described in 12-5-10) is not provided, excessive cutting may result in the following 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.

   ![Diagram]( MEP105)

   **Programmed path**

   **Programmed arc**

   **Tool center**

   **Excessive cutting**

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.

   ![Diagram]( MEP106)

   **Programmed path**

   **Opposite direction**

   **Tool center path**

3. **Machining of a stepped section smaller than the tool radius**

   ![Diagram]( MEP107)

   **Tool center path**

   **Programmed path**

   **Excessive cutting**
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:

```gcode
N1 G91 G00 G41 X500. Y500. D1
N2 S1000
N3 M3
N4 G01 Z-300. F1
N6 Y100. F2
```

With this program, all blocks up to N6 can be read during the start of offsetting based on N1. Thus, the NC unit will judge the relationship between N1 and N6 and correctly perform the offset operation as shown in the diagram above.

A sample program in which the N4 block in the program shown above has been split into two parts is shown below.

```gcode
N1 G91 G00 G41 X500. Y500. D1
N2 S1000
N3 M3
N4 Z-250.
N5 G01 Z-50. F1
N6 Y100. F2
```

In this case, the N2 through N5 blocks do not have any command corresponding to the X-Y plane and the relevant block N6 cannot be read during the start of offsetting based on N1. As a result, offsetting will be based only on the information contained in the N1 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.
For the sample program shown above, correct offsetting is ensured since the moving direction of the tool center at N2 is the same as at N6.

12-5-10 Interference check

1. Overview

Even a tool which is radially offset by the usual tool radius compensation based on two-block pre-reading may move into the workpiece to cut it. This status is referred to as interference, and a function for the prevention of such interference is referred to as interference check.

The following two types of interference check are provided and their selection is to be made using bit 5 of parameter F92.

<table>
<thead>
<tr>
<th>Function</th>
<th>Parameter</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interference check and alarm</td>
<td>Interference check and prevention off</td>
<td>The system will stop, with a program error resulting before executing the cutting block.</td>
</tr>
<tr>
<td>Interference check and prevention</td>
<td>Interference check and prevention on</td>
<td>The path is changed to prevent cutting from taking place.</td>
</tr>
</tbody>
</table>
Example:

```
(G41)
N1  G90  G1 X-50. Y-100.
N2  X-70. Y-100.
N3  X-120. Y0
```

Interference prevention path
Tool outside diameter
Cutting by N2

- For the alarm function
  An alarm occurs before N1 is executed. Machining can therefore be proceeded with by updating the program into, for example,
  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
↓
```

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 five 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 of the five to be pre-read is not possible (since the three 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:
Tool center path with interference prevented
Tool center path without interference check
Programmed path

Linear move
Tool center path with interference prevented
Tool center path without interference check
Programmed path
Arc center

Tool center path with interference prevented
Tool center path without interference check
Programmed path

Once all interference prevention linear vectors have been erased, a new prevention vector is made as shown at right. Thus, interference is prevented.
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, a program error 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, a program error will occur at the ending point of N1.

- For the diagram shown below, the direction of movement becomes opposite at N2. At this time, a program error 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. A program error 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:

\[
\begin{align*}
H_x &= \frac{I}{\sqrt{I^2 + J^2 + K^2}} \cdot r \\
H_y &= \frac{J}{\sqrt{I^2 + J^2 + K^2}} \cdot r \\
H_z &= \frac{K}{\sqrt{I^2 + J^2 + K^2}} \cdot r
\end{align*}
\]

Hence, the tool center coordinates \((x', y', z')\) are expressed as

\[
\begin{align*}
x' &= x + H_x \\
y' &= y + H_y \\
z' &= z + H_z
\end{align*}
\]

where \((x, y, z)\) denote the program 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

<table>
<thead>
<tr>
<th>G-code</th>
<th>Offset stroke positive</th>
<th>Offset stroke negative</th>
<th>Offset No. D00</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40</td>
<td>To cancel the 3-dimensional tool radius compensation</td>
<td>To cancel</td>
<td>To cancel</td>
</tr>
<tr>
<td>G41</td>
<td>To compensate in (I, J, K) direction</td>
<td>To compensate in the direction opposite to (I, J, K)</td>
<td>To cancel</td>
</tr>
<tr>
<td>G42</td>
<td>To compensate in the direction opposite to (I, J, K)</td>
<td>To compensate in (I, J, K) direction</td>
<td>To cancel</td>
</tr>
</tbody>
</table>

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 compensation is to be performed

The space in which compensation 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 $X_1 \ Y_1 \ Z_1 \ I_1 \ J_1 \ K_1$ XYZ space
- G41 $Y_2 \ I_2 \ J_2 \ K_2$ XYZ space
- G41 $X_3 \ V_3 \ Z_3 \ I_3 \ K_3$ XVZ space
- G41 $W_4 \ I_4 \ J_4 \ K_4$ 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.

Example:

- G41 (G42) $X_1 \ Y_1 \ Z_1 \ I_1 \ J_1 \ K_1 \ D_1$
  
  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:

$$G41 \ X_{x1} \ Y_{y1} \ Z_{z1} \ I_{i1} \ J_{j1} \ K_{k1} \ D_{d1}$$

![Diagram showing tool center path and 3-dimensional offset vector]

Example 2: If move commands are not present:

$$G41 \ I_{i2} \ J_{j2} \ K_{k2} \ D_{d2}$$

![Diagram showing tool center path and 3-dimensional offset vector]

5. During three-dimensional tool radius compensation

Set move commands and new plane-normal vector commands as follows:

$$X_{x3} \ Y_{y3} \ Z_{z3} \ I_{i3} \ J_{j3} \ K_{k3}$$

Example 1: If move commands and plane-normal vector commands are present:

$$X_{x3} \ Y_{y3} \ Z_{z3} \ I_{i3} \ J_{j3} \ K_{k3}$$

![Diagram showing new vector, old vector, tool center path, and programmed path]
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.

Note: 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 835 G41, G42 FORMAT ERROR.
Example 5: For changing the offset direction:

```
G41  Xx0  Yy0  Zz0  Ii0  Jj0  Kk0  Dd1
G42  Xx0  Yy0  Zz0  Ii0  Jj0  Kk0
```

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:

```
G40  Xx7  Yy7  Zz7
```

Example 2: If move commands are not present:

```
G40  (or D00)
```
12-6-3 Correlations 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 X, Y and Z space
   
   
   G41 U_ Y_ Z_ I_ J_ K_  To offset in X, Y and Z space while the U axis moves by the command value

4. Selection of an offset number falling outside the range from 1 to 128 (for standard machine specifications) or from 1 to 512 (for optional machine specifications) 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. A program error 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, work 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_ X_ Y_ Z_ α_ (α: Additional axis)
  
P: 0.....Coordinate shift (Added feature)
  1....G54
  2....G55
  3....G56
  4....G57
  5....G58
  6....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 (option)
  
  G10 L20 P_ X_ Y_ Z_ 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:

<table>
<thead>
<tr>
<th></th>
<th>Metric</th>
<th>Inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear axis</td>
<td>±99999.9999 mm</td>
<td>±99999.9999 in.</td>
</tr>
<tr>
<td>Rotational axis</td>
<td>±99999.9999°</td>
<td>±99999.9999°</td>
</tr>
</tbody>
</table>
B. Programming tool offsets

G10 L10 P_R_ Length offset; Geometric Z
G10 L11 P_R_ Length offset; Wear comp. Z
G10 L12 P_R_ Diameter/Nose-R offset (Geometric)
G10 L13 P_R_ Diameter/Nose-R offset (Wear comp.)
G10 L14 P_R_ Length offset; Geometric X
G10 L15 P_R_ Length offset; Wear comp. X
G10 L16 P_R_ Length offset; Geometric Y
G10 L17 P_R_ Length offset; Wear comp. Y
G10 L18 P_R_ Nose-R offset; Direction

The setting ranges for programming tool offset amount (R) are as follows:

<table>
<thead>
<tr>
<th>TOOL OFFSET</th>
<th>Metric</th>
<th>Inch</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geom. XYZ</td>
<td>±1999.9999 mm</td>
<td>±84.50000 in.</td>
</tr>
<tr>
<td>Geom. Nose R</td>
<td>±999.9999 mm</td>
<td>±9.99999 in.</td>
</tr>
<tr>
<td>Wear XYZ</td>
<td>±99.9999 mm</td>
<td>±0.99999 in.</td>
</tr>
<tr>
<td>Wear Nose R</td>
<td>±9.9999 mm</td>
<td>±0.99999 in.</td>
</tr>
<tr>
<td>Direction</td>
<td>0 - 9</td>
<td>0 - 9</td>
</tr>
</tbody>
</table>
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:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>N: Number</th>
<th>P: Axis No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1 to 200</td>
<td>1001 to 1200</td>
</tr>
<tr>
<td>B</td>
<td>1 to 200</td>
<td>2001 to 2200</td>
</tr>
<tr>
<td>C</td>
<td>1 to 200</td>
<td>3001 to 3200</td>
</tr>
<tr>
<td>D</td>
<td>1 to 144</td>
<td>4001 to 4144</td>
</tr>
<tr>
<td>E</td>
<td>1 to 144</td>
<td>5001 to 5144</td>
</tr>
<tr>
<td>F</td>
<td>1 to 168 (47 to 66 excluded)</td>
<td>6001 to 6168</td>
</tr>
<tr>
<td>I</td>
<td>1 to 24</td>
<td>9001 to 9024</td>
</tr>
<tr>
<td>J</td>
<td>1 to 144</td>
<td>10001 to 10144</td>
</tr>
<tr>
<td>K</td>
<td>1 to 144</td>
<td>11001 to 11144</td>
</tr>
<tr>
<td>L</td>
<td>1 to 144</td>
<td>12001 to 12144</td>
</tr>
<tr>
<td>M</td>
<td>1 to 48</td>
<td>13001 to 13048</td>
</tr>
<tr>
<td>N</td>
<td>1 to 48</td>
<td>14001 to 14048</td>
</tr>
<tr>
<td>P</td>
<td>1 to 5</td>
<td>150001 to 150005</td>
</tr>
<tr>
<td>S</td>
<td>1 to 48</td>
<td>16001 to 16048</td>
</tr>
<tr>
<td>SV</td>
<td>1 to 384</td>
<td>17001 to 17384</td>
</tr>
<tr>
<td>SP</td>
<td>1 to 256</td>
<td>18001 to 18256</td>
</tr>
<tr>
<td>SA</td>
<td>1 to 144</td>
<td>19001 to 19144</td>
</tr>
<tr>
<td>BA</td>
<td>1 to 132</td>
<td>20001 to 20132</td>
</tr>
<tr>
<td>TC</td>
<td>1 to 154</td>
<td>21001 to 21154</td>
</tr>
<tr>
<td>SU</td>
<td>1 to 168</td>
<td>22001 to 22168</td>
</tr>
<tr>
<td>SD</td>
<td>1 to 168</td>
<td>23001 to 23168</td>
</tr>
</tbody>
</table>

Note: As for the setting ranges of parameter data, refer to the separate Parameter List/Alarm List/M-code List.
### 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 a program error.
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:
   - Omit both L-code and P-code commands only when
     The axial data should refer to the coordinate system that was last selected.
   - 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.
   - 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–10000 Z–1000 B–10000
   
   The above command sets the following data:
   - Inch system: X –100. Y –0.1 Z –0.01 B –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 a program error.
3. Depending upon the data input mode — absolute (G90) or incremental (G91) — the designated data will overwrite, or will be added to, the existing data.
4. Offset data (R) without a decimal point can be entered in the range from –999999 to +999999 for geometric offset, or in the range from –99999 to +99999 for wear compensation. The data settings at that time depend upon the data input unit.

**Example:**

G10 L10 P1 R10000

The above command sets the following data:

- Metric system: 1.
- Inch system: 0.1

5. The offset data updated by a G10 command are not indicated as they are on the **TOOL OFFSET** display until that display has been selected anew.

6. Setting an illegal L-code value causes an alarm.

7. A command of “G10 P_ R_” without an L-code is also available for tool offset data input.

8. Setting an illegal P-code value causes an alarm.

9. Setting an illegal offset value (R) causes an alarm.

10. The G10 command is invalid (or skipped) during tool path check.

### C. Parameter 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 a program error.

3. Other NC statements must not be given in the parameter input mode.

4. No sequence number must be designated with address N in the parameter input mode.

5. Irrespective of the data input mode — absolute (G90) or incremental (G91) — the designated data will overwrite the existing parameter. Moreover, describe all the data in decimal numbers (hexadecimal and bit type data, therefore, must be converted).

**Example:** For changing a bit type data of 00110110 to 00110111:

Since \((00110111)_2 = (55)_{10}\) [a binary number of 00110111 corresponds to “55” in decimal notation], set 55 with address R.

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

<table>
<thead>
<tr>
<th>bit</th>
<th>7</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>2</th>
<th>1</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>S-shaped speed filter</td>
<td>7.1 ms</td>
<td></td>
<td>S-shaped speed filter</td>
<td>14.2 ms</td>
<td></td>
</tr>
</tbody>
</table>

Setting “1” for bits 2 and 3, for example, could not make valid a speed filter of 21.3 msec (= 7.1 + 14.2).

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
```

Note: Final offset stroke: H10 = –5000

MEP134
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:

\[ X = -10.000 \quad Y = -10.000 \]

: \nN100 G00 G90 G54 X0 Y0
N101 G10 L2 P1 X–15.000 Y–15.000
N102 X0 Y0 :
: M02

![Diagram showing the change in coordinate systems before and after offset application.]( MEP135)
**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.

\[
\begin{align*}
X &= 0 \quad \Rightarrow \quad X = +5.000 \\
Y &= 0 \quad \Rightarrow \quad Y = +5.000
\end{align*}
\]

**Note 2:** Prepare the following program to set workpiece coordinate system offset data in G54 to G59:

\[
\begin{align*}
G10L2P1X &-10.000 & Y &-10.000 \\
G10L2P2X &-20.000 & Y &-20.000 \\
G10L2P3X &-30.000 & Y &-30.000 \\
G10L2P4X &-40.000 & Y &-40.000 \\
G10L2P5X &-50.000 & Y &-50.000 \\
G10L2P6X &-60.000 & Y &-60.000
\end{align*}
\]

**D. Programming for using one workpiece coordinate system as multiple workpiece coordinate systems**

**Main program**

\[
\begin{align*}
&\text{M}\#1=-50. \text{ M}\#2=10. \\
&\text{M98} \text{ P200} \text{ L5} \\
&\text{M02}
\end{align*}
\]

**Subprogram** (O200)

\[
\begin{align*}
N1 & \text{ G90 G54 G10 L2 P1 X#1 Y#1} \\
N2 & \text{ G00 X0 Y0} \\
N3 & \text{ X-5. F100} \\
N4 & \text{ X0 Y-5.} \\
N5 & \text{ Y0} \\
N6 & \text{ #1=#1+#2} \\
N7 & \text{ M99}
\end{align*}
\]

\[
\begin{align*}
-\text{X} & -\text{Y} & M
\end{align*}
\]

Fundamental machine coordinate system zero point

 MEP136
E. Programming for parameter data input

<table>
<thead>
<tr>
<th>Parameter input mode ON</th>
</tr>
</thead>
<tbody>
<tr>
<td>G10L50</td>
</tr>
<tr>
<td>N4017R10</td>
</tr>
<tr>
<td>N6088R96</td>
</tr>
<tr>
<td>N12067R-1000</td>
</tr>
<tr>
<td>N12072R67</td>
</tr>
<tr>
<td>N150004P1R50</td>
</tr>
<tr>
<td>G11</td>
</tr>
</tbody>
</table>

Parameter input mode OFF

5. Related alarms

<table>
<thead>
<tr>
<th>Alarm No.</th>
<th>Alarm message</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
<tbody>
<tr>
<td>807</td>
<td>ILLEGAL FORMAT</td>
<td>Work offset input:</td>
<td>Review the program data.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>P-command is omitted in a block of G10 L20 (or L2) although the last</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>selected coordinate system is one of the systems from G54 to G59 (or</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>of the G54.1 systems).</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Parameter input:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>An illegal parameter number is set.</td>
<td></td>
</tr>
<tr>
<td>809</td>
<td>ILLEGAL NUMBER INPUT</td>
<td>Work offset input:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The setting range of the coordinate system number or the offset data</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>is overstepped.</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Tool offset input:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The setting range of the offset data is overstepped.</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Parameter input:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The axis number is not specified for an axis type parameter.</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The setting range of the axis number or the parameter data is overstepped.</td>
<td></td>
</tr>
<tr>
<td>839</td>
<td>ILLEGAL OFFSET No.</td>
<td>Tool offset input:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>The specified offset number is greater than the number of available</td>
<td>Correct the offset number</td>
</tr>
<tr>
<td></td>
<td></td>
<td>data sets.</td>
<td>according to the number of</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>available data sets.</td>
</tr>
<tr>
<td>903</td>
<td>ILLEGAL G10 L NUMBER</td>
<td>Work offset input:</td>
<td>Give an available L-code</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A command of G10 L20 is set although the corresponding function for</td>
<td>command.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>the G54.1 coordinate systems is not provided.</td>
<td></td>
</tr>
</tbody>
</table>
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/Nose-R compensation uses the MAZATROL tool data ACT-φ/NOSE-R.

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

<table>
<thead>
<tr>
<th>Data items used</th>
<th>Parameter</th>
<th>Data items used</th>
<th>Programming format</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOOL OFFSET</td>
<td>Tool offset No.</td>
<td>F93 bit 3</td>
<td>F94 bit 7</td>
<td>G43/G44 H_. (P_.)</td>
</tr>
<tr>
<td>TOOL DATA (MAZATROL)</td>
<td>LENGTH [1]</td>
<td>0</td>
<td>0</td>
<td>T_.</td>
</tr>
<tr>
<td></td>
<td>OFFSET No. or LENG CO. [2]</td>
<td>0</td>
<td>1</td>
<td>G43/G44 H_.</td>
</tr>
<tr>
<td>TOOL OFFSET + TOOL DATA</td>
<td>Tool offset No. + LENGTH [1]</td>
<td>1</td>
<td>0</td>
<td>(G43/G44 H_) + (T_) (P_.)</td>
</tr>
</tbody>
</table>

[2] LENG CO. data are only used for milling tools.
- Set G28/G30 before tool change command (when F94 bit 2 = 0).

2. Tool radius compensation

<table>
<thead>
<tr>
<th>Data items used</th>
<th>Parameter</th>
<th>Programming format</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOOL OFFSET</td>
<td>Tool offset No.</td>
<td>G41/G42 D_.</td>
</tr>
<tr>
<td>TOOL DATA (MAZATROL)</td>
<td>ACT-φ + ACT-φ CO. or ACT-φ + OFFSET No.</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td>ACT-φ CO. or OFFSET No.</td>
<td>0</td>
</tr>
<tr>
<td>TOOL OFFSET + TOOL DATA</td>
<td>Tool offset No. + ACT-φ</td>
<td>1</td>
</tr>
</tbody>
</table>
3. Nose-R compensation

<table>
<thead>
<tr>
<th>Data items used</th>
<th>Parameter</th>
<th>Programming format</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOOL OFFSET</td>
<td>F92 bit 7</td>
<td>G41/G42 D_</td>
</tr>
<tr>
<td>TOOL DATA (MAZATROL) (*)</td>
<td>F94 bit 7</td>
<td></td>
</tr>
<tr>
<td>NOSE-R + OFFSET No.</td>
<td>1 1</td>
<td>G41/G42 T_</td>
</tr>
<tr>
<td>OFFSET No.</td>
<td>0 1</td>
<td>G41/G42 T_</td>
</tr>
<tr>
<td>TOOL OFFSET + TOOL DATA (*)</td>
<td>1 0</td>
<td>G41/G42 D_ + T_</td>
</tr>
</tbody>
</table>

(*1) Since it is not calculated automatically from the CUT DIR. setting on the TOOL DATA display, the direction of the virtual tool nose must be set manually (under DIRCTN) on the TOOL OFFSET display.

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. The table below shows how and when the tool length offsetting actually takes place.

<table>
<thead>
<tr>
<th>F94 bit 7</th>
<th>How and when the tool length offsetting actually takes place</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>For milling tools: Length offsetting in the first movement on the Z-axis. For turning tools: Simultaneous offsetting by LENGTH A and B in the first axis movement, be it on the X-, Y-, Z-, or B-axis.</td>
</tr>
<tr>
<td>1</td>
<td>For milling tools: Length offsetting in the first movement on the Z-axis. For turning tools: Offset by LENGTH A in the first movement on the Z-axis, and by LENGTH B in the first movement on the X-axis.</td>
</tr>
</tbody>
</table>

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

For milling tools

Offsetting values: 
(LENGTH = 95.)

N001 G90 G94 G00 G40 G80
N002 G91 G28 Z0
N003 T01 T00 M06
N004 G90 G54 X-100. Y0
N005 G0 Z5.
N006 G01 Z-50. F100
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 compensation.

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: Compensation 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 12-8-2).

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.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>TOOL NOM-φ</th>
<th>ACT-φ</th>
<th>LENGTH</th>
<th>LENG COMP.</th>
<th>THRUST F/ HORSE PW</th>
<th>LIFE TIME</th>
<th>CUT TIME</th>
<th>MAT.</th>
<th>MAX-ROT.</th>
</tr>
</thead>
<tbody>
<tr>
<td>L57 = 0</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>L57 = 1</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>

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 (C-axis) 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.

- 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 \(x\ y\ f\)
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.

![Diagram of tool nose path](MEP306)

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 (+).

![Diagram of tool orientation and C-axial angle](MEP307)
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 (≤ 180°).

- Selection in a single-command block

```
G41.1 execution
N1 G01 Xx1 Yy1 Ff1
N2 G41.1
N3 Xx2 Yy2
```

- Selection in a block containing motion command

```
G41.1 execution
N1 G01 Xx1 Yy1 Ff1
N2 G41.1 Xx2 Yy2
```

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

```
G40.1 execution
N1 Xx1 Yy1
N2 G40.1
N3 Xx2 Yy2
```

- Cancellation in a block containing motion command

```
G40.1 execution
N1 Xx1 Yy1
N2 G40.1 Xx2 Yy2
```
- Cancellation in a block containing motion command

![Diagram showing tool nose path and G40.1 execution](image1)

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.

![Diagram showing arc center and G02 interpolation](image2)
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 $K2$ ($\varepsilon$: 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
  $\theta$: Rotational angle required
  $\varepsilon$: 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: $\alpha$

<table>
<thead>
<tr>
<th>$-\epsilon &lt; \alpha &lt; +\epsilon$</th>
<th>G41.1</th>
<th>G42.1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Diagram" /></td>
<td><img src="image2" alt="Diagram" /></td>
<td><img src="image3" alt="Diagram" /></td>
</tr>
</tbody>
</table>

C-axis rotation suppressed

<table>
<thead>
<tr>
<th>$+\epsilon &lt; \alpha &lt; (180^\circ - \epsilon)$</th>
<th>G41.1</th>
<th>G42.1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image4" alt="Diagram" /></td>
<td><img src="image5" alt="Diagram" /></td>
<td><img src="image6" alt="Diagram" /></td>
</tr>
</tbody>
</table>

Alarm 147 C AXIS TURNING ANGLE OVER

<table>
<thead>
<tr>
<th>$(180^\circ - \epsilon) \leq \alpha \leq (180^\circ + \epsilon)$</th>
<th>G41.1</th>
<th>G42.1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image7" alt="Diagram" /></td>
<td><img src="image8" alt="Diagram" /></td>
<td><img src="image9" alt="Diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$(180^\circ + \epsilon) \leq \alpha \leq (360^\circ - \epsilon)$</th>
<th>G41.1</th>
<th>G42.1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image10" alt="Diagram" /></td>
<td><img src="image11" alt="Diagram" /></td>
<td><img src="image12" alt="Diagram" /></td>
</tr>
</tbody>
</table>

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:

  \[
  F_c = \frac{F}{R} \times \frac{180}{\pi} \text{ (deg/min)}
  \]

  If parameter $K1$ (radius of C-axis rotation) ≠ 0

  \[
  F_c = F \times \frac{180}{\pi} \text{ (deg/min)}
  \]

  $F$: 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.

  Similar formulae apply to the rapid traverse.

- 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.
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 servo-axis (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.

8. The setting in bit 4 of parameter F85 (rotational axis feed rate × 1/10) is ignored in the mode of the shaping for inch system.
## 12-9-5 Compatibility with the other functions

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>One-way positioning</td>
<td>Shaping control is suppressed. (Note 1)</td>
</tr>
<tr>
<td>Helical interpolation</td>
<td>Shaping is realized adequately.</td>
</tr>
<tr>
<td>Spiral interpolation</td>
<td>Shaping cannot be realized correctly since the starting and ending point do not lie on one and the same circumference. (Note 2)</td>
</tr>
<tr>
<td>Synchronous feed</td>
<td>The designated feed rate cannot be obtained since the work spindle is controlled as the C-axis.</td>
</tr>
<tr>
<td>Shape correction</td>
<td>Shaping cannot be realized correctly since the control for constant acceleration and deceleration is not applicable to the rotation on the C-axis.</td>
</tr>
<tr>
<td>High-speed machining</td>
<td>Alarm 807 ILLEGAL FORMAT will be caused.</td>
</tr>
<tr>
<td>Exact-stop check</td>
<td>Deceleration and stop do not occur for the rotation on the C-axis.</td>
</tr>
<tr>
<td>Error detection</td>
<td>Deceleration and stop do not occur for the rotation on the C-axis.</td>
</tr>
<tr>
<td>Overriding</td>
<td>Overriding is applied adequately to the rotation on the C-axis.</td>
</tr>
<tr>
<td>Figure rotation</td>
<td>Shaping control is performed for the rotated figure.</td>
</tr>
<tr>
<td>Coordinates system rotation</td>
<td>Shaping control is performed for the rotated figure.</td>
</tr>
<tr>
<td>Scaling</td>
<td>Shaping control is performed for the scaled figure.</td>
</tr>
<tr>
<td>Mirror image</td>
<td>Shaping control is performed for the mirrored figure.</td>
</tr>
<tr>
<td>Linear angle command</td>
<td>Shaping control is performed for the calculated connection between linear segments.</td>
</tr>
<tr>
<td>Return to reference point</td>
<td>Shaping control is suppressed. (Note 3)</td>
</tr>
<tr>
<td>Return to starting point</td>
<td>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)</td>
</tr>
<tr>
<td>Workpiece coordinate system setting</td>
<td>The rotation on the C-axis is performed with reference to the coordinate system established in the shaping mode.</td>
</tr>
<tr>
<td>Local coordinate system setting</td>
<td>The rotation on the C-axis is performed with reference to the coordinate system established in the shaping mode.</td>
</tr>
<tr>
<td>Dry run</td>
<td>The speed of C-axis rotation is also modified by the external signal.</td>
</tr>
<tr>
<td>Modal restart</td>
<td>Restart from a block in the shaping mode can be performed with adequate control of the C-axis.</td>
</tr>
<tr>
<td>Non-modal restart</td>
<td>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.</td>
</tr>
<tr>
<td>Tool path check (plane)</td>
<td>The rotation on the C-axis cannot be displayed.</td>
</tr>
<tr>
<td>Virtual Machining/ Safety Shield</td>
<td>The rotation on the C-axis cannot be displayed.</td>
</tr>
<tr>
<td>Three-dimensional coordinate conversion</td>
<td>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.</td>
</tr>
</tbody>
</table>

(Note 1) Without rotation

(Note 2) Ideal normal

(Note 3) Without rotation

(Note 4) F91 bit 6=0 (Interpolation-type G00)

F91 bit 6=1 (Non-interpolation G00)
12-9-6 Sample program

Main program
WNo. 1000

G1000
G91 G28 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

G1001
G17 G91 G01 Y20. , R10. Z0. 01
X-70. , R10.
Y-40. , R10.
X70. , R10.
Y20.
M99
%

WNo. 1002

G1002
G17 G91 G01 Y20. , R10.
X-70. , R10.
Y-40. , R10.
X70. , R10.
Y20.
M99
%

MEP321

W: Workpiece origin of G54
- NOTE -
13 PROGRAM SUPPORT FUNCTIONS

13-1 Fixed Cycles

13-1-1 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

<table>
<thead>
<tr>
<th>G-Code</th>
<th>Description</th>
<th>Arguments</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>G71.1</td>
<td>Chamfering cutter (CW)</td>
<td>[X, Y] Z, Q, R, F [P, D]</td>
<td></td>
</tr>
<tr>
<td>G72.1</td>
<td>Chamfering cutter (CCW)</td>
<td>[X, Y] Z, Q, R, F [P, D]</td>
<td></td>
</tr>
<tr>
<td>G73</td>
<td>High-speed deep-hole drilling</td>
<td>[X, Y] Z, Q, R, F [P, D, K, I, J(B)]</td>
<td></td>
</tr>
<tr>
<td>G74</td>
<td>Reverse tapping</td>
<td>[X, Y] Z, R, F [P, D, J(B), H]</td>
<td>Dwell in seconds</td>
</tr>
<tr>
<td>G75</td>
<td>Boring</td>
<td>[X, Y] Z, R, F [Q, P, D, K, I, J(B)]</td>
<td></td>
</tr>
<tr>
<td>G77</td>
<td>Back spot facing</td>
<td>[X, Y] Z, R, F [Q, P, E, J(B)]</td>
<td>Return to initial point only.</td>
</tr>
<tr>
<td>G78</td>
<td>Boring</td>
<td>[X, Y] Z, R, F [Q, P, D, K]</td>
<td></td>
</tr>
<tr>
<td>G81</td>
<td>Spot drilling</td>
<td>[X, Y] Z, R, F</td>
<td></td>
</tr>
<tr>
<td>G82</td>
<td>Drilling</td>
<td>[X, Y] Z, R, F [P, D, I, J(B)]</td>
<td></td>
</tr>
<tr>
<td>G82.2</td>
<td>Pecking</td>
<td>[X, Y] Z, Q, R, F [P, D, K, I, H]</td>
<td></td>
</tr>
<tr>
<td>G83</td>
<td>Deep-hole drilling</td>
<td>[X, Y] Z, Q, R, F [P, D, K, I, J(B)]</td>
<td></td>
</tr>
<tr>
<td>G84</td>
<td>Tapping</td>
<td>[X, Y] Z, R, F [P, D, J(B), H]</td>
<td>Dwell in seconds</td>
</tr>
<tr>
<td>G84.2</td>
<td>Synchronous tapping</td>
<td>[X, Y] Z, R, F [P]</td>
<td></td>
</tr>
<tr>
<td>G84.3</td>
<td>Synchronous reverse tapping</td>
<td>[X, Y] R, Z [P] F</td>
<td></td>
</tr>
<tr>
<td>G86</td>
<td>Boring</td>
<td>[X, Y] Z, R, F [P]</td>
<td></td>
</tr>
<tr>
<td>G87</td>
<td>Back boring</td>
<td>[X, Y] Z, R, F [Q, P, D, J(B)]</td>
<td>Return to initial point only.</td>
</tr>
<tr>
<td>G89</td>
<td>Boring</td>
<td>[X, Y] Z, R, F [P]</td>
<td></td>
</tr>
</tbody>
</table>

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
13-1-2 Fixed-cycle machining data format

1. Setting fixed-cycle machining data

Set fixed-cycle machining data as follows:

```
G□□X_Y_Z_Q_R_P_D_K_I_J(B)_E_H_F_L
```

- 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.

<table>
<thead>
<tr>
<th>G90</th>
<th>G91</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Diagram of G90 mode" /></td>
<td><img src="image2" alt="Diagram of G91 mode" /></td>
</tr>
</tbody>
</table>

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.

```
G□□X_Y_Z_Q_R_P_D_K_I_J(B)_E_H_F_L_
```

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.
13-1-3  G71.1 [Chamfering cutter CW]

G71.1 [Xx Yy] Rr Zz Qq0 [Pp0 Dd0] Ff0

Initial point

G98

R-point

G99

Point D

Point Z

 MEP140

\[ q_0 \] : 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 a program error.
13-1-4 G72.1 [Chamfering cutter CCW]

G72.1 [Xx Yy] Rr Zz Qq₀ [Pp₀ Dd₀] Ff₀

- X, Y, P, and/or D can be omitted.
- Omission of Q or setting “Q0” results in a program error.
13-1-5  G73 [High-speed deep-hole drilling]

G73 [Xx Yy] Rr Zz Qtz [Ptc] Ff0 [Dd0 Kk0 Ii0 Jj0(Bb0)]

- 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 $f_1$.  

\[ t_z : \text{Depth of cut per pass} \quad j_0 : \text{Feed override ratio (\%)} \]
\[ t_c : \text{Dwell (in time or No. of revolutions)} \quad (b_0) \]
\[ d_0 : \text{Return distance} \quad f_0 : \text{Feed rate} \]
\[ k_0 : \text{Distance from R-point to the starting point of cutting feed} \quad f_1 : \text{Feed overridden} \quad f_1 = f_0 \times j_0(b_0)/100 \]
\[ i_0 : \text{Feed override distance} \quad f_2 : \text{Return speed (fixed)} \]

Max. speed: 9999 mm/min (for mm-spec.)
999.9 in./min (for in.-spec.)
13-1-6  G74 [Reverse tapping]

G74 [Xx Yy] Rr Zz [Pt_l] Ff_0 [J_(B) Dd_0 Hh_0 Kk_0]

- 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, the selection between synchronous/asynchronous tapping is performed by the bit 6 of parameter F94.
- For synchronous tapping, see Subsection 13-1-21.
- 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.
13-1-7  G75 [Boring]

G75 [Xx Yy] Rr Zz [Ptq Qq0] [Ff0] [Dd0 Jj0(Bb0) Kk0 Ii0]

- X, Y, P, Q, D, J(B), K, and/or I 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.
13-1-8  G76 [Boring]

G76 [Xx Yy] Rr Zz [Pt Qq0] Ff1 [Dd0 Jj0(Bb0)]

- Initial point
- R-point
- Point D
- Point Z
- M19: Dwell

\[ t_c : \text{Dwell (in time or No. of revolutions)} \]
\[ q_0 : \text{Amount of relief on the XY-plane} \]
\[ (\text{Direction determined by bits 3 & 4 of I14}) \]
\[ f_1 : \text{Feed rate} \]
\[ j_0 : 0 \text{ or omitted} \]
\[ (b_0) \text{ Value except 0} \]
\[ \cdots \cdots \text{M03 after machining} \]
\[ \cdots \cdots \text{M04 after machining} \]

- 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.
13-1-9  **G77 [Back spot facing]**

G77 [Xx Yy] Rr Zz [Pt, Qt] Ff0 [Ef, Jj0(Bb0) Dd0]

- 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).
13-1-10 G78 [Boring]

G78 [Xx Yy] Rr Zz [Pt] Ff0 [Dd0 Kk0 Qi0]

- X, Y, P, D, K, and/or Q can be omitted.
13-1-11 G79 [Boring]

G79 [Xx Yy] Rr Zz [Pt] Ff0 [Dd0 Kk0 Qi0 Ef1]

- Asynchronous feed is used for f1.
  - If, however, f1 is set equal to 0 or is not set, then the tool is fed at the setting of f0.
- X, Y, P, D, K, Q, and/or E can be omitted.

13-1-12 G81 [Spot drilling]

G81 [Xx Yy] Rr Zz

- X and/or Y can be omitted.
13-1-13 G82 [Drilling]

G82 [Xx Yy] Rr Zz [Ptₜ] Ff₀ [Dd₀ l₀ J₀(B₀)]

- 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.
13-1-14 G83 [Deep-hole drilling]

G83 [Xx Yy] Rr Zz Qtz Ff0 [Dd0 Kk0 Ii0 Jj0(Bb0)]

- 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.
- 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 \( f_1 \).
13-1-15 G84 [Tapping]

G84 [Xx Yy] Rr Zz [Pt] Ff [Jj0(Bb0) Dd0 Hh0 Kk0]

- 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, the selection between synchronous/asynchronous tapping is performed by the bit 6 of parameter F94.
- For synchronous tapping, see Subsection 13-1-21.
- 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.
13-1-16 **G85 [Reaming]**

G85 [Xx Yy] Rr Zz [Ptz] Fl [Ef1 Dd0]

- 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.

13-1-17 **G86 [Boring]**

G86 [Xx Yy] Rr Zz [Ptz]

- X, Y, and/or P can be omitted.
13-1-18 G87 [Back boring]

G87 [Xx Yy] Rr Zz [Ptq Qq0] Ff0 [Dd0 Jj0(Bb0)]

- 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).
13-1-19 G88 [Boring]

G88 [Xx Yy] Rr Zz [Ptₜ]

- Initial point
- G98
- R-point
- Point Z
- Dwell, M05, M00
- tc : Dwell (in time or No. of revolutions)
- X, Y, and/or P can be omitted.
- At the hole bottom, M05 and M00 are outputted.

13-1-20 G89 [Boring]

G89 [Xx Yy] Rr Zz [Ptₜ]

- Initial point
- G98
- R-point
- Point Z
- Dwell
- tc : Dwell (in time or No. of revolutions)
- X, Y, and/or P can be omitted.
13-1-21 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.] Ff0 [Jj0(Bb0) Dd0 Hh0 Kk0]
```

- **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, the selection between synchronous/asynchronous tapping is performed by the bit 6 of parameter **F94**.
- **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. **H** becomes invalid for a machine not capable of synchronous tapping, or if your machine has synchronous tapping function but bit 6 of parameter **F94** is not set to 1.
- 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.] Ff0 [Jj0(Bb0) Dd0 Hh0 Kk0]

- **t₀**: 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
- **k₀**: Distance from R-point
- **d₀**: Distance from R-point (Tap lifting distance)
- **h₀**: Return speed override (%)
  - h₀ = 0 .... Asynchronous tapping
  - h₀ ≥ 1 .... Synchronous tapping

- 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, the selection between synchronous/asynchronous tapping is performed by the bit 6 of parameter F94.

- 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. H becomes invalid for a machine not capable of synchronous tapping, or if your machine has synchronous tapping function but bit 6 of parameter F94 is not set to 1.

- 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] Ff₀

- t₀ : 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.
- G84.2 and G84.3 always performs a synchronous tapping, irrespective of the setting in bit 6 of parameter **F94**.
- 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 [Ptc] Ff₀

- \( t₀ \) : 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.
- G84.2 and G84.3 always performs a synchronous tapping, irrespective of the setting in bit 6 of parameter \( F94 \).
- 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 (%).
13-2 Hole Machining Pattern Cycles: G34.1/G35/G36/G37.1

13-2-1 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

<table>
<thead>
<tr>
<th>G-code</th>
<th>Description</th>
<th>Argument addresses</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>G34.1</td>
<td>Holes on a circle</td>
<td>X, Y, I, J, K</td>
<td></td>
</tr>
<tr>
<td>G35</td>
<td>Holes on a line</td>
<td>X, Y, I, J, K</td>
<td></td>
</tr>
<tr>
<td>G36</td>
<td>Holes on an arc</td>
<td>X, Y, I, J, P, K</td>
<td></td>
</tr>
<tr>
<td>G37.1</td>
<td>Holes on a grid</td>
<td>X, Y, I, J, K</td>
<td></td>
</tr>
</tbody>
</table>
13-2-2 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 (G281 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 Jθ 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 (Settin range: from –9999 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 programs

Given below is an example of G81 hole machining with a figure representing the hole positions.

N001 G91;
N002 G81 Z-10. R5. K0. F200;
N004 G80;
N005 G90 G0 X500. Y100.;

3. Notes

- 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.)
13-2-3 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

\[
\text{G35 } Xx \ Yy \ Id \ J \ \theta \ Kn; \\
\text{X, Y : Coordinates of the starting point.} \\
\text{I : Interval (d) between holes. Change of sign for argument I causes a centrically symmetric hole arrangement with the starting point as the center.} \\
\text{J : Angle (θ) of the line. Positive angles refer to counterclockwise measurement.} \\
\text{K : Number (n) of holes to be machined (from 1 to 9999), inclusive of the starting point.} \\
\]

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. K0. F100;
N004 G80;

3. Notes

- Omission of argument K or setting “K0” will result in a programming error. 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.
13-2-4 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 “Δθ”, 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 J0 PΔθ 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 (Δθ) 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 programes

   Given below is an example of G81 hole machining with a figure representing the hole positions.

   N001 G91;
   N002 G81 Z-10. R5. F100;
   N004 G80;
13-2-5 Holes on a grid: G37.1

As shown in the format below, a command of G37.1 determines a grid pattern of \( [\Delta x] \times [nx] \) by \( [\Delta y] \times [ny] \) 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 “\( \Delta x \)”, and “ny” in number along the Y-axis at intervals of “\( \Delta y \)”. 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

   \[
   \text{G37.1 } \text{Xx } \text{Yy } \text{I}\Delta\text{x Pnx } \text{J}\Delta\text{y Kny;}
   \]

   X, Y : Coordinates of the starting point.

   I : Hole interval (\( \Delta x \)) 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 (\( \Delta y \)) 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;
   N003 G37.1 X300. Y–100. I50. P10 J100. K8;
   N004 G80;

3. Notes

   - Omission of argument P or K, or setting “P0” or “K0” will result in a programming error. 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.
13-3 Hole Machining Fixed Cycles: G80, G283 to G289

13-3-1 Outline

1. Function and purpose

When performing predetermined sequences of machining operations such as positioning, hole machining, boring and tapping, these functions permit to command in a single block the machining program which is normally commanded in several blocks. In other words, they simplify the machining program.

The following types of fixed cycles for hole machining are available.

<table>
<thead>
<tr>
<th>G-code</th>
<th>Hole machining axis</th>
<th>Hole machining operation</th>
<th>Operation at hole bottom</th>
<th>Return movement</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>G80</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>Cancel</td>
</tr>
<tr>
<td>G283</td>
<td>Z</td>
<td>Cutting feed, intermittent feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>Deep hole drilling cycle</td>
</tr>
<tr>
<td>G284</td>
<td>Z</td>
<td>Cutting feed</td>
<td>Dwell, spindle reverse rotation</td>
<td>Cutting feed</td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G285</td>
<td>Z</td>
<td>Cutting feed</td>
<td>Dwell</td>
<td>Cutting feed</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G284.2</td>
<td>Z</td>
<td>Cutting feed</td>
<td>Spindle reverse rotation</td>
<td>Cutting feed</td>
<td>Synchronous tapping cycle</td>
</tr>
<tr>
<td>G287</td>
<td>X</td>
<td>Cutting feed, intermittent feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>Deep hole drilling cycle</td>
</tr>
<tr>
<td>G288</td>
<td>X</td>
<td>Cutting feed</td>
<td>Dwell, spindle reverse rotation</td>
<td>Cutting feed</td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G289</td>
<td>X</td>
<td>Cutting feed</td>
<td>Dwell</td>
<td>Cutting feed</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G288.2</td>
<td>X</td>
<td>Cutting feed</td>
<td>Spindle reverse rotation</td>
<td>Cutting feed</td>
<td>Synchronous tapping cycle</td>
</tr>
</tbody>
</table>

A fixed cycle mode is cancelled when the G80 or any G-code in the 01 group is set. The various data will also be cleared simultaneously to zero.

2. Programming format

A. Face hole machining

G28ΔX_ C_ Z_ R_ Q_ P_ F_ K_ M_ D_;  

- Spindle (for G284.2 only)
- M-code
- Number of repetitions
- Hole machining data
- Hole positioning data
- Hole machining mode (G283, G284, G284.2, G285)

B. Outside hole machining

G28D Z_ C_ X_ R_ Q_ P_ F_ K_ M_;  

- M-code
- Number of repetitions
- Hole machining data
- Hole positioning data
- Hole machining mode (G287, G288, G288.2, G289)

C. Cancel

G80;
D. Data outline and corresponding address

- Hole machining modes:
  These are the fixed cycle modes for drilling (G283, G287), tapping (G284, G284.2, G288, G288.2) and boring (G285, G289).
  These are modal commands and once they have been set, they will remain valid until another hole machining mode command, the cancel command for the hole machining fixed cycle or a G command in the 01 group is set.

- Hole positioning data:
  These are for the positioning of the X(Z)- and C-axes.
  These are unmodal data, and they are commanded block by block when the same hole machining mode is to be executed continuously.

- Hole machining data:
  These are the actual machining data.
  Except for Q, they are modal. Q in the G283 or G287 command is unmodal and is commanded block by block as required.

- Number of repetitions:
  This number is designated for machining holes at equal intervals when the same cycle is to be repeated.
  The setting range is from 0 through 9999 and the decimal point is not valid.
  The number is unmodal and is valid only in the block in which it has been set. When this number is not designated, it is treated as K1. When K0 is designated, the hole machining data are stored in the memory but no holes will be machined.

- M-code:
  Commanding M210 causes M-code for C-axis clamping to be outputted at the start of operation 2 (described later), and M-code for C-axis unclamping to be outputted at the end of operation 5.
  For G284 (G288) and G284.2 (G288.2), M-code for the direction of spindle revolution is specified. If not specified, the preset data of the respective parameter will be used.

- Spindle (for G284.2 only):
  Use the address D as follows to specify the spindle used for hole machining. The default value is “0” (Milling spindle selection).

<table>
<thead>
<tr>
<th>Argument D</th>
<th>Spindle used for hole machining</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>For Upper turret</td>
</tr>
<tr>
<td>0</td>
<td>Milling spindle</td>
</tr>
<tr>
<td>1</td>
<td>Turning spindle 1</td>
</tr>
<tr>
<td>2</td>
<td>Turning spindle 2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Address</th>
<th>Signification</th>
</tr>
</thead>
<tbody>
<tr>
<td>G</td>
<td>Selection of hole machining cycle sequence (G80, G283, G284, G284.2, G285, G287, G288, G288.2, G289)</td>
</tr>
<tr>
<td>X(Z)*, C</td>
<td>Designation of hole position initial point (absolute/incremental value)</td>
</tr>
<tr>
<td>Z(X)*</td>
<td>Designation of hole bottom position (absolute/incremental value from reference point)</td>
</tr>
<tr>
<td>R</td>
<td>Designation of (rapid feed)-point position (incremental value from initial point) (sign ignored.)</td>
</tr>
<tr>
<td>Q</td>
<td>Designation of cut amount for each cutting pass in G283 (G287); always incremental value, radial value (Decimal point cannot be used.)</td>
</tr>
<tr>
<td>P</td>
<td>Designation of dwell time at hole bottom point; relationship between time and designated value is same as for G04.</td>
</tr>
<tr>
<td>F</td>
<td>Designation of feed rate for cutting feed</td>
</tr>
<tr>
<td>K</td>
<td>Designation of number of repetitions, 0 to 9999 (Default = 1)</td>
</tr>
<tr>
<td>M</td>
<td>Designation of M-code</td>
</tr>
<tr>
<td>D</td>
<td>Selection of the spindle used for hole machining (for G284.2 only)</td>
</tr>
</tbody>
</table>

* Addresses in parentheses apply for commands G287, G288 and G289.
E. Use of the hole machining fixed cycles on the 2nd spindle side

The hole machining fixed cycles can also be used for the lower turret on the 2nd spindle side with the aid of the related G-code (G109 L2).

3. Outline drawing

The hole machining axes for the hole machining fixed cycle and the positioning are shown in the outline drawing below.

During the hole machining cycle, the C-axis (spindle) is clamped so that it does not move.

4. Operations

There are 7 actual operations which are each described in turn below.

Operation 1 : Positionning by rapid feed to the X(Z) and C-axis initial point
Operation 2 : Output of the M-code for C-axis clamping if it is set
Operation 3 : Positionning to the R-point by rapid feed
Operation 4 : Hole machining by cutting feed
Operation 5 : Operation at the hole bottom position which differs according to the fixed cycle mode. Possible actions include rotary tools reverse rotation (M4), rotary tools forward rotation (M3) and dwell.
Operation 6 : Return to the R-point
Operation 7 : Return to the initial point at rapid feed
(Operation 6 and 7 may be a single operation depending on the fixed cycle mode.)
Whether the fixed cycle is to be completed at operation 6 or 7 can be selected by the user parameter **F162** bit 3.

Parameter **F162** bit 3 = 0: Initial level return  
= 1: R-point level return

### 13-3-2 Face/Outside deep hole drilling cycle: G283/G287

1. **When the Q command is present** (deep hole drilling)

G283(G287)X(Z)_ C_ Z(X)_ Rr Qq Pp Ff Kk Mm ;

<table>
<thead>
<tr>
<th>Parameter <strong>F162</strong> bit 3 = 0</th>
<th>Parameter <strong>F162</strong> bit 3 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="D740PB0029" alt="Diagram" /></td>
<td><img src="TEP158" alt="Diagram" /></td>
</tr>
</tbody>
</table>

- Return distance “d” is set by the parameter (**F12**: Pecking return distance in drilling process).  
The tool returns at rapid feed.

- **(Mα)**: The C-axis clamping M-code (Mm) is outputted here if specified.

- **(Mβ)**: The C-axis unclamping M-code (C-axis clamp M-code + 1 = Mm+1) is output when there is a C-axis clamping M-code command (Mm).

- **(P)**: Dwell is performed for the duration equivalent to the time designated by P.

2. **When the Q command is not present** (drilling)

G283 (G287) X(Z)_ C_ Z(X)_ R_ P_ F_ K_ M_ ;

<table>
<thead>
<tr>
<th>Parameter <strong>F162</strong> bit 3 = 0</th>
<th>Parameter <strong>F162</strong> bit 3 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="D740PB0029" alt="Diagram" /></td>
<td><img src="TEP158" alt="Diagram" /></td>
</tr>
</tbody>
</table>

See 1 for details on (Mα), (Mβ) and (P).
13-3-3  Face/Outside tapping cycle: G284/G288

G284 (G288) X(Z)_ C_ Z(X)_ R_ P_ F_ K_ M_ ;

<table>
<thead>
<tr>
<th>Parameter</th>
<th>F162 bit 3 = 0</th>
<th>Parameter</th>
<th>F162 bit 3 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotary tool</td>
<td>Initial point</td>
<td>Rotary tool</td>
<td>Initial point</td>
</tr>
<tr>
<td>(Mα)</td>
<td>R-point (Mβ)</td>
<td>(Mα)</td>
<td>R-point (Mβ)</td>
</tr>
<tr>
<td>Initial point</td>
<td>Z point (P)</td>
<td>Initial point</td>
<td>Z point (P)</td>
</tr>
<tr>
<td>Reverse rotation of rotary tool</td>
<td></td>
<td>Forward rotation of rotary tool</td>
<td></td>
</tr>
</tbody>
</table>

- (Mα), (Mβ) and (P) are as with G283.
- During the execution G284 (G288), the override cancel status is established and 100% override is automatically applied. Dry run is also ignored.
- When feed hold is applied during the execution of G284 (G288), block stop results after return movement.
- The in-tapping signal is output in a G284 (G288) modal operation.

13-3-4  Face/Outside boring cycle: G285/G289

G285 (G289) X(Z)_ C_ Z(X)_ R_ P_ F_ K_ M_ ;

<table>
<thead>
<tr>
<th>Parameter</th>
<th>F162 bit 3 = 0</th>
<th>Parameter</th>
<th>F162 bit 3 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial point</td>
<td>R-point (Mβ)</td>
<td>Initial point</td>
<td>R-point (Mβ)</td>
</tr>
<tr>
<td>(Mα)</td>
<td>2f</td>
<td>(Mα)</td>
<td>2f</td>
</tr>
<tr>
<td>Z point (P)</td>
<td></td>
<td>Z point (P)</td>
<td></td>
</tr>
</tbody>
</table>

- (Mα), (Mβ) and (P) are as with G283.
- The tool returns to the R-point at a cutting feed rate which is double the designated feed rate command. However, it does not exceed the maximum cutting feed rate.
13-3-5  Face/Outside synchronous tapping cycle: G284.2/G288.2

G284.2 (G288.2) X(Z)_ C_ Z(X)_ R_ P_ F_ K_ M_: 

<table>
<thead>
<tr>
<th>Parameter F162 bit 3 = 0</th>
<th>Parameter F162 bit 3 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotary tool</td>
<td>Rotary tool</td>
</tr>
<tr>
<td>(Mα)</td>
<td>(Mα)</td>
</tr>
<tr>
<td>Initial point</td>
<td>Initial point</td>
</tr>
<tr>
<td>R-point (Mβ)</td>
<td>R-point (Mβ)</td>
</tr>
<tr>
<td>Forward rotation of rotary tool</td>
<td>Forward rotation of rotary tool</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>Z point</th>
<th>R point</th>
</tr>
</thead>
<tbody>
<tr>
<td>M03</td>
<td>M04</td>
<td>M03</td>
</tr>
<tr>
<td>M04</td>
<td>M03</td>
<td>M04</td>
</tr>
</tbody>
</table>

- As for synchronous tapping on the face (G284.2), the combination of the direction of the Z-axis movement (in the workpiece coordinate system) and that of the spindle rotation determines the type of tapping: normal or reverse.

<table>
<thead>
<tr>
<th>Type of tapping</th>
<th>Z-axis movement direction (in the workpiece coordinate system)</th>
<th>Command for the direction of spindle rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal tapping</td>
<td>Negative</td>
<td>M03</td>
</tr>
<tr>
<td></td>
<td>Positive</td>
<td>M04</td>
</tr>
<tr>
<td>Reverse tapping</td>
<td>Negative</td>
<td>M04</td>
</tr>
<tr>
<td></td>
<td>Positive</td>
<td>M03</td>
</tr>
</tbody>
</table>

Programming example:
1) G00 Z0.
   G284.2 Z10. F0.1 M4 ...... Normal tapping
2) G00 Z0.
   G284.2 Z-10. F0.1 M4 ...... Reverse tapping

- When G284.2 is commanded by feed per revolution (G95), where the unit of cutting feed rate F is set to mm/rev or inch/rev, tap thread pitch can be commanded directly.
- Give a G288.2 command, in place of G284.2, to use the X-axis as a hole machining axis.
- In tapping cycle (G284), the feed rate of Z-axis per spindle rotation must be equal to the thread pitch of a tap. This means that the most desirable tapping always fills the following conditions.

\[ P = \frac{F}{S} \]

- Tap thread pitch (mm)
- Z-axis feed rate (mm/min)
- Spindle speed (rpm)

Spindle rotation and Z-axis feed are independently controlled in tapping cycle (G284). Therefore, the above condition are not always filled. Spindle rotation and Z-axis feed are both decelerated and stopped particularly at the hole bottom, and then the spindle and Z-axis move in the reverse direction, giving acceleration.

Since each acceleration and deceleration are independently performed, the above conditions are not filled usually. As a result, for improving the accuracy of tapping, it is customary to compensate the feed by mounting a spring in the tap holder.

On the other hand, for synchronous tapping cycle (G284.2), spindle rotation and Z-axis feed are controlled so that they are always synchronized. In other words, for normal rotation, the spindle is controlled only in relation to speed. However, for synchronous tapping, position control is given also to spindle rotation. And spindle rotation and Z-axis feed are controlled as the linear interpolation of two axes. This fills the condition of \( P = \frac{F}{S} \) even in deceleration and acceleration at the hole bottom, permitting tapping of high accuracy.

2. Remarks

1. Synchronous tapping cycle (G284.2) and tapping cycle (G284) differ only in the control method of spindle when Z-axis moves from point R to point Z and when it does from point Z to point R. In synchronous tapping, the movement of spindle is detected by the position coder as shown below, and position control is given. Spindle motor is controlled like a servo motor to give the linear interpolation of two axes of Z-axis and spindle.

The movement distance of linear interpolation of Z-axis and spindle as well as the feed rate are as given below.

<table>
<thead>
<tr>
<th>Movement distance</th>
<th>Feed rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z-axis: ( z ) = Distance between point R and point Z (mm, inch)</td>
<td>( F_Z = \frac{F}{F_{\text{command value}}} \times 360 ) (deg)</td>
</tr>
<tr>
<td>Spindle: ( s = z \times \left( \frac{S}{F_{\text{command value}}} \right) \times 360 ) (deg)</td>
<td>( F_S = S ) command value (rpm)</td>
</tr>
</tbody>
</table>

Synchronous tapping cycle is as with G284 except that it differs from tapping cycle in the control method of spindle when Z-axis moves from point R to point Z and when it does from point Z to point R. Refer to the section of fixed cycle G284 for the notes including programming.

2. Z-axis is used as a hole machining axis in the above description. Give a G288.2 command to use the X-axis as a hole machining axis.

**Example:**

\[ \text{G288.2 Z_ C_ X_ R_ F_;} \]  X-axis is used as a hole machining axis.

3. For synchronous tapping cycle (G284.2), feed override is invalid, and it is fixed to 100%.

13-3-6 Hole machining fixed cycle cancel: G80

This command cancels the hole machining fixed cycles (G283, G284, G284.2, G285, G287, G288, G288.2, G289). The hole machining mode as well as the hole machining data are cancelled.
13-3-7 General notes on the hole machining fixed cycles

1. When the G284 and G288 fixed cycle commands are set, the rotary tool must be rotated in the designated direction beforehand using a miscellaneous function (M3, M4).

2. If the basic axis, additional axis and R data are present in a block, hole machining is performed in a fixed cycle mode; it will not be performed if these data are not present. Even if the X-axis data are present, hole machining will not be executed if a dwell (G04) command is present in the block.

3. The hole machining data (Q, P) should be commanded in the block (block including the basic axis, additional axis and R data) in which the holes are machined. The modal data will not be updated even if these data are commanded in a non-hole machining block.

4. When resetting is applied during the execution of the G285 (G289) command, the hole machining data will be erased.

5. The hole machining fixed cycles are also cancelled by any G code in the 01 group besides G80. If it is commanded in the same block as the fixed cycle, the fixed cycle will be ignored. m = 01 group code, n = hole machining fixed cycle code

   - Gm Gn X(Z) C_ Z(X)_ R_ Q_ P_ K_ F_;
   - Gn Gm X(Z) C_ Z(X)_ R_ Q_ P_ K_ F_;

   executed ignored executed ignored memorized

   In both cases, G01 X100.C30.Z50.F100. is executed.

6. When a miscellaneous command is set in the same block as the fixed cycle command, it is outputted after the initial positioning. However, the C-axis unclamping M-code (clamp M + 1) is output after the holes have been machined and the tool returns to the return point. When the number of repetitions has been designated, the M command execution in above condition is exercised only for the initial operation except for the C-axis clamping M-code. In the case of the C-axis clamping/unclamping M commands, as they are modal, the codes are outputted with each repetitions until the operation is cancelled by the fixed cycle cancel command.

7. When a tool position offset command (T function) is set in a hole machining fixed cycle mode, execution will follow the tool position offset function.

8. When a hole machining fixed cycle command is set during tool nose radius compensation, program error occurs.

9. Cutting feed rate by F will be kept after cancelling the drilling cycle.

10. Waiting commands (corresponding M- and P-codes) are all ignored in a hole machining fixed cycle mode (G283 to G289). Cancel the hole machining fixed cycle mode beforehand with G80 to give a waiting command.
### 13-3-8 Sample programs with fixed cycles for hole machining

<table>
<thead>
<tr>
<th>1. Face deep hole drilling cycle (G283)</th>
<th>5. Outside tapping cycle (G288)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 G97 G94;</td>
<td>G00 G97 G94;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M200;</td>
<td>M200;</td>
</tr>
<tr>
<td>M03 S800;</td>
<td>M03 S600;</td>
</tr>
<tr>
<td>G90 X100.Z2.C0;</td>
<td>G90 X102.Z–50.C0;</td>
</tr>
<tr>
<td>G80;</td>
<td>G80;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M30;</td>
<td>M30;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>2. Face tapping cycle (G284)</th>
<th>6. Outside boring cycle (G289)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 G97 G94;</td>
<td>G00 G97 G94;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M200;</td>
<td>M200;</td>
</tr>
<tr>
<td>M03 S600;</td>
<td>M03 S800;</td>
</tr>
<tr>
<td>G90 X100.Z2.C0;</td>
<td>G90 X102.Z–50.C0;</td>
</tr>
<tr>
<td>G80;</td>
<td>G80;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M30;</td>
<td>M30;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>3. Face boring cycle (G285)</th>
<th>7. Face synchronous tapping cycle (G284.2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 G97 G94;</td>
<td>G00 G97 G94;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M200;</td>
<td>M200;</td>
</tr>
<tr>
<td>M03 S600;</td>
<td>M03 S600;</td>
</tr>
<tr>
<td>G90 X100.Z2.C0;</td>
<td>G90 X100.Z2.C0;</td>
</tr>
<tr>
<td>G80;</td>
<td>G80;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M30;</td>
<td>M30;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>4. Outside deep hole drilling cycle (G287)</th>
<th>8. Outside synchronous tapping cycle (G288.2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 G97 G94;</td>
<td>G00 G97 G94;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M200;</td>
<td>M200;</td>
</tr>
<tr>
<td>M03 S800;</td>
<td>M03 S600;</td>
</tr>
<tr>
<td>G80;</td>
<td>G80;</td>
</tr>
<tr>
<td>G91 G28 XZ;</td>
<td>G91 G28 XZ;</td>
</tr>
<tr>
<td>M30;</td>
<td>M30;</td>
</tr>
</tbody>
</table>
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:

<table>
<thead>
<tr>
<th>Number of holes</th>
<th>Sample program</th>
<th>G98 (At power-on or after cancellation using M02, M30, or RESET key)</th>
<th>G99</th>
</tr>
</thead>
<tbody>
<tr>
<td>Only one</td>
<td>G81 X100. Y100. Z–50. R25. F1000</td>
<td><img src="image1" alt="Initial point" /> <img src="image2" alt="R-point" /> Return to initial point level.</td>
<td><img src="image1" alt="Initial point" /> <img src="image2" alt="R-point" /> Return to R-point level.</td>
</tr>
<tr>
<td>Two or more</td>
<td>G81 X100. Y100. Z–50. R25. L5 F1000</td>
<td><img src="image3" alt="1st hole" /> <img src="image4" alt="2nd hole" /> <img src="image5" alt="Last hole" /> Always return to initial point.</td>
<td><img src="image3" alt="1st hole" /> <img src="image4" alt="2nd hole" /> <img src="image5" alt="Last hole" /> MEP158</td>
</tr>
</tbody>
</table>
13-5 Fixed Cycles for Turning

When performing roughing and other such operations during turning, these functions permit to command in a single block the machining program which is normally commanded in several blocks. In other words, they simplify the machining program. The following types of fixed cycles for turning are available.

<table>
<thead>
<tr>
<th>G-code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G290</td>
<td>Longitudinal turning cycle</td>
</tr>
<tr>
<td>G292</td>
<td>Threading cycle</td>
</tr>
<tr>
<td>G294</td>
<td>Transverse turning cycle</td>
</tr>
</tbody>
</table>

1. The programming format is as follows:
   
   G290 X_ Z_ R_ F_ ;
   (Same for G292, G294)

   The taper values of fixed cycles G290, G292 and G294 are to be specified by argument R.

2. Fixed cycle commands are modal G-codes and so they are valid until another command in the same modal group or a cancel command is set. The following G-code cancels fixed cycle commands.

   G00, G01, G02, G03,
   G07,
   G09,
   G10,
   G27, G28, G29, G30,
   G31,
   G32, G33, G34,
   G52, G53,
   G92

3. There are two types of fixed cycle call, move command block call and block-by-block call. These are selected by a parameter setting.

   A move command block call calls the fixed cycle macro subprogram only when there is an axial move command in the fixed cycle mode. The block-by-block call calls the fixed cycle macro subprogram in each block in the fixed cycle mode. Both types are executed until the fixed cycle is cancelled.

4. A manual interruption can be applied while a fixed cycle for turning (G290, G292 and G294) is being executed. Upon completion of the interruption, however, the tool must be returned to the position where the manual interruption was applied and then the fixed cycle for turning should be restarted. If it is restarted without the tool having been returned, all subsequent operation movements will deviate by an amount equivalent to the manual interruption value.
13-5-1 Longitudinal turning cycle: G290

1. Straight turning

Continuous straight turning operations can be initiated by the following instruction:

G290 X_ Z_ F_

![Diagram of straight turning cycle](image1)

2. Taper turning

Continuous taper turning operations can be initiated by the following instruction:

G290 X_ Z_ R_ F_

![Diagram of taper turning cycle](image2)
3. Remarks

- In the single-block operation mode, the tool stops either at the ending points of operations 1, 2, 3 and 4, or only on completion of one cycle (depending on bit 7 of parameter F111).
- Depending on the signs of X, Z and R, the following shapes are created.
  * The explanation below refers to incremental data input (under G91).

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>[1] X &lt; 0, Z &lt; 0, R &lt; 0</td>
<td>[2] X &lt; 0, Z &lt; 0, R &gt; 0</td>
<td></td>
</tr>
<tr>
<td>![Diagram 1]</td>
<td>![Diagram 2]</td>
<td></td>
</tr>
</tbody>
</table>

<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>[3] X &gt; 0, Z &lt; 0, R &lt; 0</td>
<td>[4] X &gt; 0, Z &lt; 0, R &gt; 0</td>
<td></td>
</tr>
<tr>
<td>![Diagram 3]</td>
<td>![Diagram 4]</td>
<td></td>
</tr>
</tbody>
</table>

A programming error (**899 ILLEGAL TAPER LENGTH**) occurs for shapes [2] and [3] unless the following condition is satisfied.

| X | ≥ | R |
13-5-2 Threading cycle: G292

1. Straight threading

Straight or cylindrical threading operations can be initiated by the following instruction:

\[
\text{G292 } X_\_ Z_\_ F/E_\_
\]

![Diagram of straight threading](TEP121)

2. Taper threading

Taper threading operations can be initiated by the following instruction:

\[
\text{G292 } X_\_ Z_\_ R_\_ F/E_\_
\]

![Diagram of taper threading](TEP122)
3. Remarks

1. Details of thread run-out

- **α**: Length of thread run-out
  - Can be set in a parameter (TC82) to a value from 0.0L to 4.0L (L=Thread lead) in 0.1 steps.
- **θ**: Run-out angle of threading
  - Can be set in a parameter (F28) to 45 or 60 degrees.

2. In the single-block operation mode, the tool stops either at the ending points of operations 1, 2, 3 and 4, or only on completion of one cycle (depending on bit 7 of parameter F111).

3. The feed hold function causes a non-threading operation (1, 3, or 4) to be stopped immediately, but the threading operation (2) is either not stopped till completion of the next step (3) or immediately changed into run-out and then stopped (according to the setting of bit 2 of parameter F111).

4. During threading, use or disuse of dry run cannot be changed.

5. For machines with the optional function for automatic correction of threading start position, the thread cutting conditions can be changed by “overriding” the spindle speed. See Subsection 6-15-6 for more information.

6. Depending on the signs of X, Z and R, the following shapes are created.

   - The explanation below refers to incremental data input (under G91).

   | [1] X < 0, Z < 0, R < 0 | [2] X < 0, Z < 0, R > 0 |
   | [3] X > 0, Z < 0, R < 0 | [4] X > 0, Z < 0, R > 0 |

   A programming error (899 ILLEGAL TAPER LENGTH) occurs for shapes [2] and [3] unless the following condition is satisfied.

   \[ |X| \geq |R| \]
13-5-3 Transverse turning cycle: G294

1. Straight facing

Continuous straight facing operations can be initiated by the following instruction:
G294 X_ Z_ F_

2. Bevel facing

Continuous bevel facing operations can be initiated by the following instruction:
G294 X_ Z_ R_ F_
3. Remarks

1. In the single-block operation mode, the tool stops either at the ending points of operations 1, 2, 3 and 4, or only on completion of one cycle (depending on bit 7 of parameter F111).

2. Depending on the signs of X, Z and R, the following shapes are created.

   * The explanation below refers to incremental data input (under G91).

   - **[1] X < 0, Z < 0, R < 0**
     - X < 0, Z < 0, R > 0
     - X/2
     - 1(R)
     - 2(F)
     - 4(R)
     - 3(F)

   - **[2] X < 0, Z < 0, R > 0**
     - X < 0, Z < 0, R > 0
     - X/2
     - 1(R)
     - 2(F)
     - 4(R)
     - 3(F)

   - **[3] X > 0, Z < 0, R < 0**
     - X > 0, Z < 0, R > 0
     - X/2
     - 1(R)
     - 2(F)
     - 4(R)
     - 3(F)

   - **[4] X > 0, Z < 0, R > 0**
     - X > 0, Z < 0, R > 0
     - X/2
     - 1(R)
     - 2(F)
     - 4(R)
     - 3(F)

   A programming error (899 ILLEGAL TAPER LENGTH) occurs for shapes [2] and [3] unless the following condition is satisfied.

   \[ |Z| \geq |R| \]
13-6 Compound Fixed Cycles for Turning

These functions permit various fixed cycles to be initiated by a single instruction with the corresponding preparatory functions.

The available types of compound fixed cycle for turning are as follows:

<table>
<thead>
<tr>
<th>G-code</th>
<th>Function</th>
<th>Compound fixed cycles I</th>
<th>Compound fixed cycles II</th>
</tr>
</thead>
<tbody>
<tr>
<td>G270</td>
<td>Finishing cycle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G271</td>
<td>Longitudinal roughing cycle (leaving finishing allowance)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G272</td>
<td>Transverse roughing cycle (leaving finishing allowance)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G273</td>
<td>Contour-parallel roughing cycle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G274</td>
<td>Longitudinal cut-off cycle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G275</td>
<td>Transverse cut-off cycle</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G276</td>
<td>Compound threading cycle</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- If the finishing contour program has not been entered in the memory, any of the above functions for the compound fixed cycles I (G270 to G273) cannot be used.

- The programming formats are as follows.

<table>
<thead>
<tr>
<th>G-code</th>
<th>Programming format</th>
</tr>
</thead>
<tbody>
<tr>
<td>G270</td>
<td>G270 A_P_Q_ ;</td>
</tr>
<tr>
<td>G271</td>
<td>G271 U_R_ ;</td>
</tr>
<tr>
<td>G272</td>
<td>G272 U_W_R_ ;</td>
</tr>
<tr>
<td>G273</td>
<td>G273 U_W_R_ ;</td>
</tr>
<tr>
<td>G274</td>
<td>G274 R_ ;</td>
</tr>
<tr>
<td>G275</td>
<td>G275 R_ ;</td>
</tr>
<tr>
<td>G276</td>
<td>G276 P_Q_R_ ;</td>
</tr>
</tbody>
</table>
13-6-1 Longitudinal roughing cycle: G271

1. Overview

With commands as shown below for the finishing contour between (A) to (H), roughing by cutting depth \( \Delta d \) will be executed by leaving finishing allowances \( U \) and \( W \).

![Diagram](D740PB0058)

2. Programming format

\[
\begin{align*}
\text{G271 } U \Delta d & \hspace{0.5em} R_
\text{G271 } A_ \hspace{0.5em} P_ \hspace{0.5em} Q_ \hspace{0.5em} U \Delta u \hspace{0.5em} W_ \hspace{0.5em} F_ \hspace{0.5em} S_ \hspace{0.5em} T_
\end{align*}
\]

- \( U \Delta d \): Depth of cut
  - Set an absolute value (in radius value).
  - The value is modal and remains valid until it is overwritten with a new value.
- \( R \): Escape distance
  - The value is modal and remains valid until it is overwritten with a new value.
  - (The escape motion in each pass of the roughing cycle is detailed later under "4. Escape patterns".)
- \( A \): Finishing contour program No.
- \( P \): Head sequence No. for finishing contour
- \( Q \): End sequence No. for finishing contour
- \( U \Delta u \): Finishing allowance and direction along the X-axis (in diameter or radius value)
- \( W \): Finishing allowance and direction along the Z-axis
- \( F \_ S \_ T \_ \): Feed, Spindle and Tool functions

The roughing cycle is executed using the F-, S- and T-functions specified in or before the G271 block, in stead of those existing in the program section designated by \( P \) and \( Q \).

**Note 1:** Even if F- and S-codes exist in the program section designated by \( P \) and \( Q \), they are considered as for the finishing cycle only and, therefore, ignored in the roughing cycle.

**Note 2:** \( \Delta d \) and \( \Delta u \) are both specified with address \( U \). The differentiation depends on whether \( P \) and \( Q \) are specified in the same block.

**Note 3:** The block of G271 \( U \Delta d \hspace{0.5em} R \_ \) can be omitted when the external settings in parameters \textbf{SU103} and \textbf{SU102} are to be used respectively as arguments \( U \ (\Delta d) \) and \( R \).
3. Detailed description

The contour of machining by G271 may be one of the four combinations below. Machining is basically executed by feed motions along the Z-axis. Finishing allowances U and W may have different signs.

![Diagram of contour combinations](TEP129)

The section from A to B is to be described in the block designated with P. See the description under “4. Escape patterns” for more information. For the section between B and C, a maximum of 49 recesses can be specified. Depending upon whether the modal value for the block of the motion from A to B is G00 or G01, the infeed on the X-axis is repeated by rapid motion or cutting feed during the roughing cycle.

![Diagram of tool path and programmed contour](TEP130)

4. Escape patterns

There are two patterns provided for escape motion during cycle operation. The selection between the two patterns occurs automatically according to the finishing contour program as follows:

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Pattern 2</th>
<th>Pattern 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>With convex sections</td>
<td>The first block (designated with P) is of single-axis movement (on the X-axis).</td>
<td>Without convex sections</td>
</tr>
<tr>
<td>Without convex sections</td>
<td>The first block (designated with P) is of two-axis movement (in the ZX-plane).</td>
<td>With convex sections</td>
</tr>
<tr>
<td>Pattern 2</td>
<td>Pattern 1</td>
<td>Pattern 2</td>
</tr>
</tbody>
</table>
Note 1: “Convex” section here denotes a prominence in the X-axis direction, as illustrated below.

Examples of finishing contour with convex sections

- Pattern 1 (as illustrated below)
  Escape motion is performed with an escape angle of 45° at the rate of cutting feed (G01).

- Pattern 2 (as illustrated below)
  Final escape motion occurs along the X-axis at the rate of rapid traverse (as for G00) after an ending step of contour-parallel turning according to the G-code used for the element concerned of the finishing contour.
**Note 2:** As shown below, the roughing cycle using the escape pattern 1 ends with a complementary pass of contour-parallel turning, which does not apply to the roughing cycle using the escape pattern 2.

As illustrated here, the last repetitive pass of the roughing cycle using the escape pattern 1 is followed by a complementary pass of contour-parallel turning (according to the G-codes to be used for the finish turning), which is not the case with the roughing cycle using the escape pattern 2.

---

**5. Remarks**

1. Subprograms can be called from the program section designated by P and Q.
2. The last element of the contour described in the program section designated with P and Q may be circular (G02 or G03). Do not fail to give the desired G-code explicitly for the section next to the roughing cycle.
3. The maximum permissible number of blocks for the finishing contour is 100, including blocks automatically inserted by the NC (e.g. for nose radius compensation).

The Z-axis position, however, must be monotonic increasing or decreasing. For shapes as shown below, the alarm **898 LAP CYCLE ILLEGAL SHAPE DESIGN.** will occur.
4. Monotonousness of the Z-axis motion is the only requirement for the finishing contour. Even the initial cut-in may be of simultaneous control of the X- and Z-axis, as shown below. See the description above under "4. Escape patterns" for the escape motion.

5. If a roughing cycle command is given in the mode of nose radius compensation, the roughing cycle operations are executed for the finishing contour accordingly compensated.

6. Set “W0” in general for a contour if it is not monotonic changing on the X-axis. Otherwise excessive cutting is caused on either side.

7. If the machining contour from A through B to C should contain an intermediate point whose X-position is not "below" A (as is the case with B’ in the figure below), the alarm 898 LAP CYCLE ILLEGAL SHAPE DESIGN. will occur. Case (I) is correct, while case (II) will cause the alarm. This also applies when the case in question results from the nose radius compensation, the tool length offset, and the tool wear compensation.

8. M-codes in the program section designated by P and Q are only effective in the mode of G270 (finishing). They are ignored in the cycle of G271 (also G272 and G273).

9. Cutting path is drawn as follows:
6. Sample programs

![Workpiece diagram with dimensions and notes](image)

Cutting depth: 5.
Escape distance: 1.
Finishing allowance X: 2.
Z: 2.

N001 G00 G96 G94
N002 G91 G28 X0 Z0
N003 G10.9X1
N010 G271 U5. R1.
N012 G00 X60.S200
N013 G01 Z-30.F100
N014 G03 X120.Z-60.R30.
N015 G01 Z-100.
N017 G270 P012 Q016
N018 G91 G28 X0 Z0 M205
N019 M30
13-6-2 Transverse roughing cycle: G272

1. Programming format

\[ \text{G272 } W\Delta d \text{ R}_n \]
\[ \text{G272 A}_n \text{ P}_n \text{ Q}_n \text{ U}_n \text{ W}_n \text{ F}_n \text{ S}_n \text{ T}_n \]

\( W\Delta d \): Depth of cut

For the other addresses and remarks see the description given for G271.

The contour of machining by G272 may be one of the four combinations below.
Machining is basically executed by feed motions along the X-axis. Finishing allowances \( U \) and \( W \) may have different signs.

- The section from A to B is to be described in the block designated with P. See the description under “2. Escape patterns” for more information.
- For the section between B and C, a maximum of 49 recesses can be specified.
- Depending upon whether the modal value for the block of the motion from A to B is G00 or G01, the infeed on the Z-axis is repeated by rapid motion or cutting feed during the roughing cycle.
- Nose radius compensation amount is added to the finishing allowances \( U \) and \( W \).
2. Escape patterns

There are two patterns provided for escape motion during cycle operation. The selection between the two patterns occurs automatically according to the finishing contour program as follows:

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Pattern 2</th>
<th>Pattern 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>With convex sections</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Without convex sections</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note 1: In contradistinction to G271, the automatic selection of escape pattern for G272 does not depend upon whether the first block (designated with P) of the finishing contour program is of single-axis or two-axis movement.

Note 2: “Convex” section here denotes a prominence in the Z-axis direction, as illustrated below.

Examples of finishing contour with convex sections

- Pattern 1 (as illustrated below)
  Escape motion is performed with an escape angle of 45° at the rate of cutting feed (G01).
- Pattern 2 (as illustrated below)

Final escape motion occurs along the Z-axis at the rate of rapid traverse (as for G00) after an ending step of contour-parallel turning according to the G-code used for the element concerned of the finishing contour.

Pattern 2: Final escape motion occurs on the Z-axis through the escape distance (designated with R) after an ending step of contour-parallel turning for a Z-axis range of the depth of cut (\( \Delta d \) designated with W).

Note 3: As is the case with G271, the roughing cycle using the escape pattern 1 ends with a complementary pass of contour-parallel turning, which does apply to the roughing cycle using the escape pattern 2.

3. Sample programs

```
N001 G00 G96 G94
N002 G91 G28 X0 Z0
N003 T01 T00 M06
N004 G10.9X1
N005 G18 G90 X176. Z2.
N011 G272 P012 Q018 U4.W2.F100 S100 M203
N012 G00 Z=80.S150
N013 G01 X120.Z=72.F100
N014 Z=62.
N015 X82.Z=51.
N016 Z=31.
N017 X35.Z=10.
N018 Z2.
N019 G270 P012 Q018
N020 G91 G28 X0 Z0 M205
N021 M30
```
13-6-3 Contour-parallel roughing cycle: G273

1. Overview

This function allows efficient execution in roughing when cast or forged parts are to be cut along the finishing contour.

2. Programming format

G273 U\(\Delta i\) W\(\Delta k\) Rd;
G273 P_ Q_ U\(\Delta u\) W\(\Delta w\) F_ S_ T_; 

\(\Delta i\) : Escape distance and direction along the X-axis (in radius value)
The value is modal and remains valid until it is overwritten with a new value.

\(\Delta k\) : Escape distance and direction along the Z-axis
The value is modal and remains valid until it is overwritten with a new value.

d : Number of divisions
This denotes the number of roughing operations to be repeated. The value is modal and remains valid until it is overwritten with a new value.

For other addresses see the description given for G271.

Note: Even if F- and S-codes exist in the program section designated by P and Q, they are considered as for the finishing cycle only and, therefore, ignored in the roughing cycle.
3. Detailed description

- Finishing contour
  The finishing program has to describe the contour A→B→C as shown in the figure below.
  The section from B to C must be monotonic increasing or decreasing on both the X- and Z-axis.

- One cycle configuration
  A cycle is composed as shown below.

- Nose radius compensation
  If a roughing cycle command is given in the mode of nose-R compensation, the roughing cycle operations are executed for the finishing contour accordingly offset with the compensation being temporarily canceled, and upon completion of the roughing, the compensation mode is retrieved (for the first block of the finishing cycle).

- Infeed direction
  The shift direction for the infeed is determined by the shape in the finishing program, as shown in the table below.

<table>
<thead>
<tr>
<th>Trace</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial X-axis</td>
<td>“−” direction</td>
<td>−</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td>Overall Z-axis</td>
<td>“−” direction</td>
<td>+</td>
<td>+</td>
<td>−</td>
</tr>
<tr>
<td>X-axis cutting</td>
<td>“+” direction</td>
<td>+</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Z-axis cutting</td>
<td>“+” direction</td>
<td>−</td>
<td>−</td>
<td>+</td>
</tr>
</tbody>
</table>
4. Related parameters

- The X-axis escape distance and direction can be set by parameter TC117, whose value will be overridden with the programmed value.
- The Z-axis escape distance and direction can be set by parameter TC118, whose value will be overridden with the programmed value.
- The number of divisions can be set by parameter TC72, whose value will be overridden with the programmed value.

5. Remarks

- The contour-parallel roughing cycle is executed using the F-, M-, S- and T-functions specified in or before the G273 block, in stead of those existing in the program section designated by P and Q.
- There are four patterns of machining as shown below. Take care to assign correct signs to the $\Delta u$, $\Delta w$, $\Delta i$ and $\Delta k$ values.

$\Delta i$ and $\Delta k$ and $\Delta u$ and $\Delta w$ are both specified with addresses U and W, respectively. The differentiation is given by whether P and Q are specified in the same block. That is, addresses U and W without P and Q in a G273 block refer to $\Delta i$ and $\Delta k$, while those with P and Q denote $\Delta u$ and $\Delta w$, respectively.

- When the cycle terminates, the tool is returned to point A.
- If the contour-parallel roughing cycle is executed in the nose-R compensation mode where the tool nose center is aligned with the starting point, the amount of nose radius compensation is added to $\Delta u$ and $\Delta w$.
- Others are as with G271.
6. Sample programs

N001 G00 G96 G94
N002 G91 G28 X0 Z0
N003 T01T00M06
N004 G10.9X1
N005 G18 G90 X150. Z5.
N016 G00 X50.
N017 G01 Z-30
N019 Z-75.
N020 X120.Z-90.
N021 G270 P016 Q020
N022 G91 G28 X0 Z0 M205
N023 M30
13-6-4 Finishing cycle: G270

After roughing by the commands G271 to G273, finishing can be initiated by the following
programming format.

G270 A_ P_ Q_;

- A : No. of the finishing contour program (program being executed in default)
- P : Head sequence No. for finishing contour (program head in default)
- Q : End sequence No. for finishing contour (end of program in default)
  Up to an M99 block if it is read before the block designated with Q.

- The finishing cycle is executed using the F-, S- and T-functions specified in the program
  section that is designated with A, P and Q.
- When a G270 cycle is completed, the tool returns to the starting point in rapid motion and the
  next block is read.

Example 1: Designation of sequence numbers only

| N100 G270 P200 Q300; |
| N110 |
| N120 |
| N200 |
| N300 |
| N310 |

Example 2: Designation of a program number

| N100 G270 A100; |
| N110 |
| N120 |
| O100 |
| G01 X100 250 F0.5; |
| M99; |

In both examples 1 and 2, the N110 block is executed after the execution of the N100 cycle.
13-6-5 Longitudinal cut-off cycle: G274

1. Overview

This function is used for smooth disposal of machining chips in longitudinal cut-off machining. For SS materials which produce hard-to-cut machining chips this function can be managed for easy machining chip disposal.

2. Programming format

G274 Re;
G274 Xx Zz Pa Qk RΔd Ff Ss;

- **e**: Return distance
  - The value is modal and remains valid until it is overwritten with a new value.
- **x**: Final X-axis position in absolute/incremental data
- **z**: Final Z-axis position in absolute/incremental data
- **Δi**: X-axis movement step (in an absolute value)
- **Δk**: Z-axis depth of cut (in an absolute value)
- **Δd**: Tool escape distance at the bottom of cut
  - Normally set in an absolute value. When omitting the arguments X and P, however, set the value with a sign as required for the direction of escape.
- **f**: Feed function (rate of feed)
- **s**: Spindle function

The distance “e” is set by parameter TC74 (pecking return distance in grooving process).
3. **Detailed description**

1. X, P and RΔd are not required for drilling.

   ![Diagram of drilling operation]
   
   ```
   G00 X0 Z5.0;
   G274 Z-50.0 Q15.0 F0.2;
   ```

2. No escape motion is performed in default of RΔd.
   Normally set the RΔd data in an absolute value. When omitting the arguments X and P for outside turning or inside boring, however, set the data with a sign as required for the direction of escape.

   ![Diagram showing four combinations of G274]
   
   Four combinations of G274

   - Δd > 0 for (A), (B)
   - Δd < 0 for (C), (D)

3. In the single-block operation mode, all the blocks are executed step by step.
4. Remarks

1. In the single-block operation mode, all the blocks are executed step by step.
2. Omission of address X, P and R\(d\) provides the operation of Z-axis alone, resulting in a deep-hole drilling cycle.
3. “e” and “\(\Delta d\)” are both specified with address R. The differentiation is given by whether Z is commanded together or not. That is, the R value given with Z is processed as “\(\Delta d\)”.
4. The cycle operation is initiated by a G274 block containing the Z data.

5. Sample programs

```
G00 G96 G94
G91 G28 X0 Z0
T01T00 M06
G10.9X1
G18 G90 X100.Z2.
G274 R2.
G91 G28 X0 Z0
M30
```
13-6-6 Transverse cut-off cycle: G275

1. Overview

This function is used for smooth disposal of machining chips in transverse cut-off machining. This allows easy disposal of machining chips in face turning as well.

2. Programming format

\[
\begin{align*}
\text{G275} & \quad \text{Re} ; \\
\text{G275} & \quad X_\_ Z_\_ P_\_ Q_\_ R_\Delta d \quad F_\_ S_\_ T_\_; \\
\end{align*}
\]

G275 executes cycle operations as shown below.
3. Detailed description

1. \( Z, Q, \) and \( R\Delta d \) are not required for grooving on the outside or inside cylinder surface.

\[
\begin{align*}
G00 & \; X105.0 \; Z-60.0; \\
G275 & \; X90.0 \; P2.0 \; F0.05;
\end{align*}
\]

2. No escape motion is performed in default of \( R\Delta d \).
   Normally set the \( R\Delta d \) data in an absolute value. When omitting the arguments \( Z \) and \( Q \) for facing, however, set the data with a sign as required for the direction of escape.

\[
\begin{align*}
\Delta d > 0 \; \text{for (A), (B)} \\
\Delta d < 0 \; \text{for (C), (D)}
\end{align*}
\]

3. In the single-block operation mode, all the blocks are executed step by step.

4. Remarks

1. Both G274 and G275, which are used for cutting off, grooving or drilling, are a cycle to give the escape of a tool automatically. Four patterns which are symmetrical with each other are available.

2. The return distance “e” can be set by parameter TC74, whose value will be overridden with the programmed value.

3. In the single-block operation mode, all the blocks are executed step by step.
5. Sample programs

G00 G96 G94
G91 G28 X0 Z0
T01 T00 M06
G10.9 X1
G18 G90 X102 Z-20.
G275 R2.
G275 Z-35 X70 P6 Q5 F150 S100 M203
G91 G28 X0 Z0
M30
13-6-7 Compound threading cycle: G276

1. Cycle configuration

Feed (by F- or E-code)  
Rapid traverse  

TC82: Length of thread run-out (Parameter)  

SU105: Thread finishing allowance  
(Parameter) (Diameter value)
2. Programming format

G276 Pmra Rd; (omission allowed)
G276 Xx Zz Ri Pk Qd Fz S_ T_;

m : Repeat times of final finishing (1 to 99)
The value is modal and remains valid until it is overwritten with a new value.
r : Length of thread run-out
Express in a two-digit number (00 to 99) the tenfold of that multiplier to the thread lead (r) which produces the desired length of run-out.
The value is modal and remains valid until it is overwritten with a new value.
a : Tool tip angle (thread angle)
Choose an angle from among 80°, 60°, 55°, 30°, 29° and 0°, and set the numeric value in two digits. The value is modal and remains valid until it is overwritten with a new value.
d : Finishing allowance
The value is modal and remains valid until it is overwritten with a new value.
i : Radial difference of threading portion
Set i = 0 for parallel thread cutting.
k : Thread height (along the X-axis, in radius value)
Δd : First cutting depth (in radius value)
ℓ : Lead of thread (as with the threading by G32/G33)
S, T: As with G271.

Note: Set m, r and a together with address P.
When m = 2 (times), r = 1.2 (× Lead) and a = 60 (deg.), set the argument P as follows:
P 02 12 60
m r a
3. Detailed description

1. Length of thread run-out can be set by parameter TC82 by $0.1 \times L$ units in a range from $0.1 \times L$ to $4.0 \times L$ (L as lead).

2. Cut depth is determined with $\Delta d$ for initial cut, and $\Delta d\sqrt{n}$ for n-th cut to have a constant depth for each cut.

---

**Four combinations of G276**

- (A)
- (B)
- (C)
- (D)

Feed (by F- or E-code)

Rapid traverse

$i < 0$ for (A), (D)

$i > 0$ for (B), (C)
3. One cycle configuration

The tool moves at rapid traverse for operations [1], [2], [5] and [6] in the cycle and at the cutting feed based on the value designated to F for operations [3] and [4].

When R is negative

When R is positive

\[ \Delta d \text{ for first cutting pass} \]

Second cutting pass \( \Delta d \times \sqrt{2} \)

n-th cutting pass \( \Delta d \times \sqrt{n} \)

"Finishing allowance d"/2
(cutting results for "m" number of passes)
4. Remarks

1. When the Feed Hold button is pressed during execution of a G276 cycle, undergoing threading is either not stopped till completion of the next step or immediately changed into run-out and then stopped (according to the setting of bit 2 of parameter F111), as is the case with G292. (The Feed Hold lamp lights up immediately and then goes off when automatic operation is stopped.) If threading is not being carried out, the feed hold state is immediately established and the Feed Hold lamp lights up.

2. During execution of a G276 cycle the machining does not stop till completion of operations [1], [4] and [5] when the mode is switched to another automatic mode or to a manual operation mode, or in the single-block operation mode.

3. During a G276 cycle, validity or invalidity of dry run cannot be changed while threading is under way.

4. In the single-block operation mode, all the blocks are executed step by step. For blocks of threading, however, the subsequent block is continuously executed.

5. For machines with the optional function for automatic correction of threading start position, the thread cutting conditions can be changed by “overriding” the spindle speed. See Subsection 6-15-6 for more information.

5. Related parameters

1. The number of repeatitions of final finishing can be set by parameter TC81, whose value will be overridden with the programmed value.

2. Length of thread run-out can be set by parameter TC82, whose value will be overridden with the programmed value.

3. The tool tip angle can be set by parameter TC80, whose value will be overridden with the programmed value.

4. The finishing allowance can be set by parameter SU105, whose value will be overridden with the programmed value.
6. Detailed description

1. Setting the tool tip angle provides the machining of a single tip, permitting the decrease in a load applied to the tool tip.

2. Cut amount is held constant by setting the first cut depth as $\Delta d$ and $n$-th cut depth as $\Delta d\sqrt{n}$.

3. According to the sign of each address, four patterns are available, and inside thread can also be cut.

4. Threading cycle provides the cutting feed designated by the F- or E-code only from C to D, and rapid traverse for other movements.

5. For the cycle shown above, the signs of increment are as follows:
   - $u, w$ ............According to the direction of paths $A \rightarrow C$ and $C \rightarrow D$.
   - $i$ .................According to the direction of path $A \rightarrow C$.
   - $k$ .................Plus (always plus)
   - $\Delta d$ ...........Plus (always plus)

6. Finishing allowance ($d$; diameter value) can be set by parameter (SU105) within the range as follows:
   - 0 to 65.535 mm (6.5535 inches)
7. Sample programs

G00 G97 G95
G10.9X1
G91 G28 X0 Z0
T01T00M06
S500 M203
G18 G90 X100.Z20.
G276 P011060 R0.2
G276 X60.64 Z-80.P3.68 Q1.8 F6.0
G91 G28 X0 Z0 M205
M30
8. Notes

1. For G276 cycle, the notes on threading are as with G32/G33 and G292 threading. The feed hold function causes the threading operation to be changed immediately into run-out (when F111 bit 2 = 1) and to be stopped thereafter, as described in item 3 below. Refer to G292 threading cycle for details.

2. The angle of run-out can be set in parameter F28 within a range from 0° to 89°, but only 45° or 60° can actually be applied.
   Setting of 90° or more validates 45°.
   Setting of 0° to 45° and of 46° to 89° is taken respectively as that of 45° and 60°.

3. According to a parameter (F111 bit 2), the Feed Hold function brings the current threading operation in one of the following ways to a stop:
   - After continued execution of the next block, or
   - After run-out operation started immediately at an angle of 60°.
   The feed is stopped immediately except during threading.
   Pressing the Cycle Start button again causes the tool to be returned in rapid motion by simultaneous control of the X- and Z-axis to the starting point, and the cycle operation is continued.

4. An alarm occurs in the cases below.
   - Either X or Z is not specified.
   - The displacement distance on the X- or Z-axis is 0.
   - The thread angle exceeds the range from 0° to 120°.

5. In the single-block operation mode, all the blocks are executed step by step. For blocks of threading, however, the next block is also executed in sequence.

6. The data items designated with P, Q and R are differentiated by whether addresses X and Z are specified in the same block.

7. The cycle operation is initiated by a G276 block containing the X and Z data.

8. For machines with the optional function for automatic correction of threading start position, the thread cutting conditions can be changed by “overriding” the spindle speed. See Subsection 6-15-6 for more information.

13-6-8 General notes on the compound fixed cycles G270 to G276

1. Except for the preset parameters, set all the required parameters or arguments in the blocks of the compound fixed cycle G-codes.

2. Provided that the finishing contour program has been registered in the memory, compound fixed cycle I commands can be executed in the memory, MDI or tape operation mode.

3. Take care not to use twice in the program concerned the sequence numbers that are to be designated with P and Q in the blocks of G270 to G273 for fetching the description of the finishing contour.

4. The maximum permissible number of blocks for the finishing contour (in the section from P to Q specified in the blocks G271 to G273) is 100, including blocks automatically inserted by the NC (for corner chamfering or rounding, nose radius compensation, etc.). If this number is exceeded, a program error occurs.

5. The finishing contour to be used for the roughing cycles of G271 to G273 must be monotonic increasing or decreasing on either of, or both, the X- and Z-axis as appropriate.

6. Blocks without motion commands in the finishing contour section are ignored.
7. N, F, S, M and T commands in the finishing contour section are ignored during roughing.

8. A program error occurs if any of the following commands are present in the finishing contour section.
   - Commands related to reference point return (G27, G28, G29, G30)
   - Threading (G32/G33)
   - Fixed cycles
   - Skip functions (G31, G31.1, G31.2, G31.3, G37)

9. Subprograms can be called from the blocks in the finishing contour section.

10. Except for threading cycles, operation stops at the ending (starting) point of each block in the single-block operation mode.

11. Note that the next block upon completion of the G271, G272 or G273 cycle differs depending on whether a program number or the sequence numbers only are designated.
   - Designation of sequence numbers only:
     The next block is that which follows the block designated by Q.
     The N600 block is executed in the example upon completion of the cycle.

   - Designation of a program number:
     The next block is that which follows the cycle command block.
     The N200 block is executed in the example upon completion of the cycle.

12. The next block upon completion of the G270 command is that which follows the command block.
    The N1100 block is executed in the example upon completion of the G270 command.

13. Manual interruption can be applied during a compound fixed cycle command (G270 to G276). However, upon completion of the interruption, the tool must first be returned to the position where the interruption was applied and then the compound fixed cycle must be restarted.
    If it is restarted without the tool having been returned, all subsequent movements will deviate by an amount equivalent to the manual interruption amount.
14. Compound fixed cycle commands are unmodal commands and so they must be set every time they are required.

15. The alarm No. **898 LAP CYCLE ILLEGAL SHAPE DESIGN**. is caused for G271 and G272 if, because of nose-R compensation, there is no further displacement of the Z-axis in the second block or a reverse motion on the Z-axis should be made.

16. Prohibited commands in the finishing contour program section (designated with P and Q in a block of G270 to G273) are as follows:
   
   M98/M99,  
   T code,  
   G20, G21, G94, G95, G52, G53, G68, G69,  
   G32/G33, G290, G292, G294,  
   G10, G27, G28, G29, G30.

17. Take care not to use twice in the program concerned the sequence numbers that are to be designated with P and Q in the blocks of G270 to G273.

18. If the last motion command in the finishing contour program section (designated with P and Q in a block of G270 to G273) is for corner chamfering (G01X_, Z_, C_) or corner rounding (G01X_, Z_, R_), the alarm **NO DIRECTIVE FOR NEXT MOVE R/C** occurs.

19. The block designated with P in a G271, G272 or G273 command must be in the G00 or G01 mode.

20. For a manual “handle” interruption (to shift the tool position) after feed hold during execution of the G270 to G276 cycle, the tool must be returned to the original position before pressing the Cycle Start button to continue the operation.

   If the operation is restarted without the tool having been returned, all subsequent movements will be shifted by the amount of the manual interruption.

   Handle interruption values are cancelled by resetting the NC.

21. The example below is given to show the execution timing of an M- or T-function code designated in the command block of G270 to G276.

   ```
   N041 G90 G00 X100.Z0;
   N042 G91 G271 P101 Q103 X0.5 Z0.5
   D4000 F0.5 S150 M08;
   ;
   N101 G90 G01 X90.F0.5;
   N102 Z-20.;
   N103 X100.;
   ```

   ![Execution Point](tep155.png)
13-7 Workpiece Coordinate Setting in a Fixed-Cycle Mode

The specified axis moves in the workpiece coordinate system currently valid. The new coordinate system does not become valid for Z-axis till the R-point positioning or another Z-axis movement after completion of positioning on X-Y plane.

Note: For axis addresses and R, reprogramming must be done during workpiece coordinate updating even if their settings are the same as respective previous ones.

Example:

```
G54 Xx1 Yy1 ZZ1
G81 Xx2 Yy2 ZZ2 Rr2
   ...
   ...
G55 Xx3 Yy3 ZZ2 Rr2
   xx4 Yy4
   xx5 Yy5
   ...
   ...
```

Reprogramming is required even if Z and R are of the same values as for the last time.
13-8 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 \textbf{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.

Program errors occur in the following cases:

- If scaling is specified for a machine not capable of scaling (Alarm \textbf{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 \textbf{809 ILLEGAL NUMBER INPUT}) (All scaling factors less than 0.000001 are processed as 1.)

\textbf{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.

\textbf{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.
   \begin{verbatim}
   G51 P0.5  0.5 time
   P0.5 G51  1 time (regarded as P = 0)
   P500000 G51 0.5 time
   G51 P500000 0.5 time
   \end{verbatim}
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
N05 G01Z-250.F1000
N06 Y-150.F200
N07 X-150.
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

[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
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.
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
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.
5. 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
```
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 $l_1$ 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 G90 G54 G00 X0 Y0
N02 G51 X–100. Y–100. P0.5
N03 G65 P100
N04 G90 G55 G00 X0 Y0
N05 G65 P100

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

```gcode
N01 G92X0Y0
N02 G90G51X0Y0P0.5
N03 G00X-100.Y-100.
N04 M98P200I-50.L8 %
```

**Machining program**

**Scaling center**

**After scaling**

**Machining program**
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
G90G00X100.
M98P300I-100.L4
G90G00X0Y0
M02
```

```
O300
G91G51X0Y0P0.5
G00X-40.
G01Y-40.F1000
X40.
G03Y80.J40.
G01X-40.
Y-40.
G00G50X40.
X-100.Y100.
M99
%
```

Machining program

After scaling
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  
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  

(Coordinate rotation data setting)
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
```
13-9 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 arc 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)
   G00 G90 G40 G49 G80
   M98 P100
   G51.1 X0
   M98 P100
   G51.1 Y0
   M98 P100
   G50.1 X0
   M98 P100
   G50.1 Y0
   M30

   (Subprogram O100)
   G91 G28 X0 Y0
   G90 G00 X20 Y20.
   G42 G01 X40 D01 F120
   Y40.
   X20.
   Y20.
   G40 X0 Y0
   M99
13-10 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.

<table>
<thead>
<tr>
<th>Function</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Memory operation</td>
<td>O</td>
<td>O</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>2. Tape editing (main memory)</td>
<td>O</td>
<td>O</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>3. Subprogram call</td>
<td>x</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td>4. Subprogram nesting level call</td>
<td>x</td>
<td>O</td>
<td>O</td>
<td>x</td>
</tr>
<tr>
<td>5. Fixed cycles</td>
<td>x</td>
<td>x</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>6. Fixed cycle subprogram editing</td>
<td>x</td>
<td>x</td>
<td>O</td>
<td>O</td>
</tr>
</tbody>
</table>

Notes:
1. "O" denotes a function which can be used and "x" a function which cannot be used.
2. The nesting depth can include as many as 8 levels.
2. Programming format

Subprogram call

M98 < > Q L;

- Number of subprogram repetitions (L1 if omitted)
- Sequence number in subprogram to be called (head block if omitted)
- Program name of subprogram to be called (own program if omitted)
  (Use the address H for G-code series M.)

Alternatively,

M98 P Q L;

- Number of subprogram repetitions (L1 if omitted)
- Sequence number in subprogram to be called (head block if omitted)
- Program number [composed of numerals only] of subprogram to be called (own program if omitted). P can only be omitted during memory operation.

(Use the address H for G-code series M.)

Program name of subprogram to be called (own program if omitted).
Can only be omitted during memory operation.

Return to main program from subprogram

M99 P L;

- Number of times after repetition number has been changed
- Sequence number of return destination (returned to block following block of call if omitted)

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.

Program number as subprogram

...............;
...............;
...............;
...............;
M99;
%(EOR)

Prepare programs of the above format and store them in the NC memory. Refer to Chapter 31 for details on program preparation.

Only those subprograms numbers ranging from 1 to 99999999 designated by the optional specifications can be used. When a program loaded by the tape input function has no program number specified, the setting number for “program input” is used.

Up to 8 nesting levels can be used for calling programs from subprograms, and program error 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.)
Example:

; 
O○○○○;
............;
;
M99;
%

; 
O△△△△;
............;
;
M99;
%

; 
O□□□□;
............;
;
M99;
%

Subprogram A
Subprogram B
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 \langle \rangle Q_\_ L_; or M98 P_\_ Q_\_ L_; 

Where \langle \rangle : Name of the subprogram to be called (up to 32 characters)
P : Number of the subprogram to be called (up to 8 digits)
Q : Any sequence number within the subprogram to be called (up to 5 digits)
(Use the address H for G-code series M.)
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 ;

\[
\begin{align*}
\text{O1; } & \quad \text{Subprogram 1} \\
\vdots & \\
M99; & \\
\% & \\
\text{O2; } & \\
\vdots & \\
\text{N200 } & \quad \text{Subprogram 2} \\
\vdots & \\
M99; & \\
\% & \\
\text{O3; } & \\
\vdots & \\
\text{N200 } & \quad \text{Subprogram 3} \\
\vdots & \\
M99; & \\
\%
\end{align*}
\]

- 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,) is to be repeatedly used, it will be repeatedly executed for I, times provided that M98 Pp I Ll ; is programmed.

5. Other precautions

- Programming error 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 _;
In the execution of a subprogram composed of sections each proper to either of the upper and lower turrets with the aid of G109L_ blocks, only the program sections for that system (headstock or turret) which is currently active in the main program at the call of the subprogram will selectively be executed with the other sections being appropriately skipped, as shown below:

**Pattern 1:**

**Main program (EIA)**

```
G109L1  : (for machining with Upper turret)
M98<1000>
```

**Subprogram (WNo. 1000)**

```
G109L1  (for machining with Upper turret)
:  M99
G109L2  (for machining with Lower turret)
:  Skipped
```

**Pattern 2:**

**Main program (EIA)**

```
:  G109L2  (for machining with Lower turret)
M98<2000>
```

**Subprogram (WNo. 2000)**

```
G109L1  (for machining with Upper turret)
:  Skipped
G109L2  (for machining with Lower turret)
:  M99  Executed
```
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)

Note 1: 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.

Note 2: MAZATROL programs (with commands for both the upper and lower turret) can successfully be called up as a subprogram from two positions of an EIA program of similar structure on condition that one and the same program is called up on completion of blocks of the same waiting command.

Example:
B. Programming format

M98 <_> L_; or M98 P_ L_;  
< > or P: Name, or number, of the MAZATROL machining program to be called. 
When omitted, the alarm NO DESIGNATED PROGRAM will be displayed. Also, when the specified program is not stored, the alarm NO DESIGNATED PROGRAM will be displayed. 
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.

   ![Diagram](image)

   Also, REPEAT and SHIFT are ignored even if they are specified.

2. MAZATROL program execution
   When MAZATROL program is called from 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 nose 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.
13-11 Figure Rotation: M98 (Option)

1. Function and purpose
The figures commanded by subprograms can be executed after rotation by using subprogram call-out, center commands I, J, K, and word L.

2. Programming format
M98 P/\_< > H_I_J_ (K_) L_   (for G17:  I, J  for G18: K, I  for G19: J, K)
M98:  M-code for subprogram call
P/<_>: Number or name of the subprogram to be called
H:  Sequence number in the subprogram to be called
I, J, K: Incremental values of figure rotation-center (incremental from the starting point)
L:  Number of repetitions of subprogram execution (when L < 1, it is not regarded as the figure rotation)

3. Detailed description
1. Subprograms are executed by the above format commands and they are completed one time by M99 subprogram return, then the subprogram commands are rotated on the rotation information which consists of the start point, the center, and the end point. The rotation angle can be added up every one-time completion by specifying repeat times as two and over, so the figures commanded by subprograms can be arranged by the times specified after rotation with the center coordinates as their standard.
2. The first time of subprograms by subprogram called-out is executed on the basis of 0 rotation angle to trace the locus specified. All blocks in the subprogram are rotated.
3. When the start point and the end point of the subprogram are not on the same circle with the figure rotation center as the center, the interpolation based on the command, in which the subprogram end point is its start point and the end point of the first move command block of the subprogram rotated is its end point, is carried out.
4. Cooperative use of absolute value and incremental value commands
In figure rotation subprograms, the cooperative use of absolute values and incremental values is possible. For the absolute value mode the rotation of two times and over can be executed by the same command if the standard figure is programmed by absolute values.
5. Subprogram controlling
The nesting of subprograms is possible even during figure rotation. However, one-time completion of figure rotation is no other than M99 of the nesting level called-out by figure rotation subprogram.

Note 1: Figure rotation is carried out on the workpiece coordinate system, so it can be shifted by the commands G92, G52, G54 to G59 (workpiece coordinate system shift).

Note 2: Figure rotation is carried out on the workpiece coordinate system, so the functions on the machine coordinate system (return to the zero point, one-direction positioning, etc.) are not rotated.

Note 3: The figure rotation command during diagram rotation will cause an alarm (849 FIGURE ROTATE NESTING EXCEEDED).

Note 4: Figure rotation and program coordinate rotation cannot be commanded simultaneously. If so commanded, it will cause an alarm (850 G68 AND M98 COMMANDS SAME BLOCK).
4. Sample programs

Example 1: Cutting into a gear shape
Prepare the program for machining of one tooth using a subprogram first and then designate the teeth quantity during program call:

```
G92X0Y0
G90G00X50.
M98P7L36I-50.
G00X0Y0
M02
O7
G03X54.358Y.190J50.F100
X54.135Y4.927I-54.354J-.190
X49.810Y4.358J-50.
X49.240Y8.682I-49.810J-4.358
M99
%
```

Main program
Startpoint positioning and call-out of diagram rotation of gear cut

Subprogram (O7)
Data of gear basic form
Example 2: Cooperative use with workpiece offset function
Diagram rotation can be carried out on the workpiece coordinate system.

G54 (G55, G56) G90XY
G90X0Y0
M98P10H1I-10.J10.L4F100
M98P10H2I-10.J10.L4
M98P10H3I-10.J10.L4
M02
%
O10
N1G01X-5.Y10.
X0Y20.
M99
N2G01X5.Y10.
X0Y20.
M99
X0Y20.
M99
N4G01X15.Y10.
X0Y20.
M99
%

Subprogram (O10)

Main program

Workpiece coordinate offset data

<table>
<thead>
<tr>
<th>G54</th>
<th>G55</th>
<th>G56</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>0</td>
<td>60.</td>
</tr>
<tr>
<td>Y</td>
<td>0</td>
<td>20.</td>
</tr>
</tbody>
</table>
Example 3: Using the fixed-cycle machining function
Make a subprogram which contains only the positioning data for fixed-cycle operation first and then store fixed-cycle hole-machining data into the memory during subprogram call (executing such a program allows rotation of the setting position in the subprogram, and hence, bolt-hole circle machining).

G92X0Y0Z0
G91X50.
M98P101I–50.L8
G00X0Y0
M02
%
O101
X35.355Y35.355
M99%

Main program
Hole-machining data storage and subprogram call

Subprogram (O101)
Positioning data (absolute data)
Example 4: Application to fixed-cycle operation (Application to arcs)

M98P102I-iJ-jL5
M02
%
O102
XxYy
M99
%

Main program
Start point positioning, hole-machining, figure rotation call

Subprogram (O102)
Positioning data
13-12 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<OOOOO>**
  - MAZATROL or EIA/ISO program is called from EIA/ISO program and executed as the next program.

- **M999<OOOOO>**
  - MAZATROL or EIA/ISO program is only called from EIA/ISO program and the operation is terminated.
**Note 1:** The programs to be called up at the end of both the upper and lower turrets' program sections must be of the same work number; otherwise an alarm will be caused. Moreover, use either M998 or M999 for both turrets' sections in their respective ending blocks; otherwise an alarm will likewise be caused.

**Example 1:** Correct use

**Main program (EIA)**

```
G109L1
:  
M950
M999<1000>
```

**Subprogram (WNo. 1000)**

```
G109L1
```

**Main program (EIA)**

```
G109L2
:  
M950
M999<1000>
```

**Subprogram (WNo. 1000)**

```
G109L2
```

**Example 2:** Wrong use

**Main program (EIA)**

```
G109L1
:  
M950
M998<1000>
```

**Subprogram (WNo. 1000)**

```
G109L1
```

**Main program (EIA)**

```
G109L2
:  
M950
M999<1000>
```

**Subprogram (WNo. 1000)**


**Note 2:** 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 3:** 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.
13-13 Chamfering and Corner Rounding at Arbitrary Angle Corner

Chamfering or corner rounding at any angle corner is performed automatically by adding ",C_" or ",R_" to the end of the block to be commanded first among those command blocks which form the corner with lines only.

13-13-1 Chamfering at arbitrary angle corner: , C_

1. Function

   The arbitrary corner is chamfered between two points on the two lines which form this corner and displaced by the lengths commanded by ", C_" from their intersection point.

2. Programming format

   N100 G01 X_ Z_ ,C_;  \{ Chamfering is performed at the point where N100 and N200 intersect. \}
   N200 G01 X_ Z_;  \{ Length up to chamfering starting point or ending point from virtual corner intersection point \}

3. Example of program

   (a) G91 G01 Z100.,C10.F100;
   (b) X280.Z100.;

4. Detailed description

   1. The starting point of the block following the corner chamfering is the virtual corner intersection point.
   2. When the comma in ", C_" is not present, it is considered as a C command.
   3. When both, C_ and , R_ are commanded in the same block, the latter command is valid.
   4. Tool offset is calculated for the shape which has already been subjected to corner chamfering.
   5. Program error occurs when the block following the block with corner chamfering does not contain a linear interpolation command.
   6. Program error occurs when the movement amount in the block commanding corner chamfering is less than the chamfering amount.
7. Program error occurs when the movement amount in the block following the block commanding corner chamfering is less than the chamfering amount.

13-13-2 Rounding at arbitrary angle corner: , R_

1. Function

The arbitrary corner is rounded with the arc whose radius is commanded by ,R_ and whose center is on the bisector of this corner angle.

2. Programming format

\[
\begin{align*}
N100 & \ G01 \ X_\ _ \ Z_\ , \ R_\ ; \\
N200 & \ G01 \ X_\ _ \ Z_\ ;
\end{align*}
\]

Rounding is performed at the point where N100 and N200 intersect.

3. Example of program

(a) G91 G01 Z100.,R10.F100;
(b) X280.Z100.;

4. Detailed description

1. The starting point of the block following the corner rounding is the virtual corner intersection point.
2. When the comma in , R_ is not present, it is considered as an R command.
3. When both , C_ and , R_ are commanded in the same block the latter command is valid.
4. Tool offset is calculated for the shape which has already been subjected to corner rounding.
5. Program error occurs when the block following the block with corner rounding does not contain a linear command.
6. Program error occurs when the movement amount in the block commanding corner rounding is less than the R value.
7. Program error occurs when the movement amount in the block following the block commanding corner rounding is less than the R value.
13-14 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)  
   N2 G01 A–a₂ Zz₂ Xx₂  
   (Setting Aa₃ means the same as setting A–a₂.)

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.
13-15 Macro Call Function: G65, G66, G66.1, G67

13-15-1 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.
13-15-2 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.

Format:
G65 <__> L__ <argument>

Repeat times
Program Name (When omitted, own program will be repeated.)

Alternatively,
G65 P__ L__ <argument>

Repeat times
Program No. (When P is omitted, own program will be repeated.)

<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.
  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:

<table>
<thead>
<tr>
<th>Address specified using method I</th>
<th>Variable in macro-program</th>
<th>Call commands and usable addresses</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>G65, G66</td>
<td>G66.1</td>
</tr>
<tr>
<td>A</td>
<td>#1</td>
<td>O</td>
</tr>
<tr>
<td>B</td>
<td>#2</td>
<td>O</td>
</tr>
<tr>
<td>C</td>
<td>#3</td>
<td>O</td>
</tr>
<tr>
<td>D</td>
<td>#7</td>
<td>O</td>
</tr>
<tr>
<td>E</td>
<td>#8</td>
<td>O</td>
</tr>
<tr>
<td>F</td>
<td>#9</td>
<td>O</td>
</tr>
<tr>
<td>G</td>
<td>#10</td>
<td>×</td>
</tr>
<tr>
<td>H</td>
<td>#11</td>
<td>O</td>
</tr>
<tr>
<td>I</td>
<td>#4</td>
<td>O</td>
</tr>
<tr>
<td>J</td>
<td>#5</td>
<td>O</td>
</tr>
<tr>
<td>K</td>
<td>#6</td>
<td>O</td>
</tr>
<tr>
<td>L</td>
<td>#12</td>
<td>×</td>
</tr>
<tr>
<td>M</td>
<td>#13</td>
<td>O</td>
</tr>
<tr>
<td>N</td>
<td>#14</td>
<td>×</td>
</tr>
<tr>
<td>O</td>
<td>#15</td>
<td>×</td>
</tr>
<tr>
<td>P</td>
<td>#16</td>
<td>×</td>
</tr>
<tr>
<td>Q</td>
<td>#17</td>
<td>O</td>
</tr>
<tr>
<td>R</td>
<td>#18</td>
<td>O</td>
</tr>
<tr>
<td>S</td>
<td>#19</td>
<td>O</td>
</tr>
<tr>
<td>T</td>
<td>#20</td>
<td>O</td>
</tr>
<tr>
<td>U</td>
<td>#21</td>
<td>O</td>
</tr>
<tr>
<td>V</td>
<td>#22</td>
<td>O</td>
</tr>
<tr>
<td>W</td>
<td>#23</td>
<td>O</td>
</tr>
<tr>
<td>X</td>
<td>#24</td>
<td>O</td>
</tr>
<tr>
<td>Y</td>
<td>#25</td>
<td>O</td>
</tr>
<tr>
<td>Z</td>
<td>#26</td>
<td>O</td>
</tr>
</tbody>
</table>

O: 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:

<table>
<thead>
<tr>
<th>Argument specification II addresses</th>
<th>Variables in macro-programs</th>
<th>Argument specification II addresses</th>
<th>Variables in macro-programs</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>#1</td>
<td>K5</td>
<td>#18</td>
</tr>
<tr>
<td>B</td>
<td>#2</td>
<td>I6</td>
<td>#19</td>
</tr>
<tr>
<td>C</td>
<td>#3</td>
<td>J6</td>
<td>#20</td>
</tr>
<tr>
<td>I1</td>
<td>#4</td>
<td>K6</td>
<td>#21</td>
</tr>
<tr>
<td>J1</td>
<td>#5</td>
<td>I7</td>
<td>#22</td>
</tr>
<tr>
<td>K1</td>
<td>#6</td>
<td>J7</td>
<td>#23</td>
</tr>
<tr>
<td>I2</td>
<td>#7</td>
<td>K7</td>
<td>#24</td>
</tr>
<tr>
<td>J2</td>
<td>#8</td>
<td>I8</td>
<td>#25</td>
</tr>
<tr>
<td>K2</td>
<td>#9</td>
<td>J8</td>
<td>#26</td>
</tr>
<tr>
<td>I3</td>
<td>#10</td>
<td>K8</td>
<td>#27</td>
</tr>
<tr>
<td>J3</td>
<td>#11</td>
<td>I9</td>
<td>#28</td>
</tr>
<tr>
<td>K3</td>
<td>#12</td>
<td>J9</td>
<td>#29</td>
</tr>
<tr>
<td>I4</td>
<td>#13</td>
<td>K9</td>
<td>#30</td>
</tr>
<tr>
<td>J4</td>
<td>#14</td>
<td>I10</td>
<td>#31</td>
</tr>
<tr>
<td>K4</td>
<td>#15</td>
<td>J10</td>
<td>#32</td>
</tr>
<tr>
<td>I5</td>
<td>#16</td>
<td>K10</td>
<td>#33</td>
</tr>
<tr>
<td>J5</td>
<td>#17</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

Variables
#1: 1.1
#2: -2.2
#3:
#4: 4.4
#5:
#6:
#7: 7.7

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 \( l \) number of times for the first call, or once for subsequent calls.

For modal call of type A, the methods of specifying \(<\text{argument}>\) are the same as used for single call.

```
Format:
G66 \_<\_>_ L\_<\_>_ <\text{argument}>

Alternatively,
G66 P\_<\_>_ L\_<\_>_ <\text{argument}>
```

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 \textbf{857 INCORRECT USER MACRO G67 PROG}.  

Drilling cycle

Main program

N1G90G54G0X0Y0Z0
N5X–50.
N6G67

Subprogram

O9010
N10G00Z#18M03
N20G09G01Z#26F#9
N30G00Z–[#18+#26]
N99

To subprogram after execution of axis command
To subprogram after execution of axis command
To main program

To subprogram

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.

Format:

G66.1 <__> L__ <argument>
Repeat times
Program name

Alternatively,

G66.1 P__ L__ <argument>
Repeat times
Program No.
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

\[\text{N100G01G90X100. Y200. F400R1000}\]

in the G66.1P1000 mode is handled as equivalent to

\[\text{N100G65P1000G01G90X100. Y200. F400R1000.}\]

**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: G×× <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.

\[\text{M98PΔΔΔΔ}\]

\[\text{G65PΔΔΔΔ <argument>}\]

\[\text{G66PΔΔΔΔ <argument>}\]

\[\text{G66.1PΔΔΔΔ <argument>}\]

- Use parameters to set the relationship between G×× (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 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:

\[ \text{Mm (or Ss, Tt and Bb)} \]

\[ \text{M (or S, T and B) code which calls macro-subprogram} \]

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.

\[ \text{M98P} \]

\[ \text{G65P} \]

\[ \text{M66P} \]

\[ \text{G66.1P} \]

- Use parameter to set the relationship between Mm (macro call M-code) and P (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 eight 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

\[
\begin{align*}
& \vdots \\
& \text{M96}< > _L \text{ (or M96P } _L \text{) } \\
& \vdots \\
& \text{M97 (Branching mode off)} \\
& \vdots \\
\end{align*}
\]

(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.
13-15-3 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
#5=#[{#1}]
```

From #1 = 10, #[{#1}] = #[{#10}] will result. Therefore #5 = #20, i.e. #5 = 30 will result.

```
#1=10  #10=20  #20=30
#5=1000
#[{#1}]=#5
```

From #10 = 20, #[{#10}] = #20 will result. Therefore #20 = #5, i.e. #20 = 1000 will result.

Example 2: Replacing variables identifiers with <expression>

```
#10=5
#[{#10+1}]=1000
#[{#10-1}]=-1000
#[{#10+3}]=100
#[{#10/2}]=100
```

#6 = 1000 will result.

#4 = -1000 will result.

#15 = 100 will result.

#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.

<table>
<thead>
<tr>
<th></th>
<th>If #101 = &lt;empty&gt;</th>
<th>If #101 = 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>#101EQ#0</td>
<td>&lt;empty&gt; = &lt;empty&gt; holds.</td>
<td>#101EQ#0</td>
</tr>
<tr>
<td>#101NE#0</td>
<td>&lt;empty&gt; \neq 0 holds.</td>
<td>#101NE#0</td>
</tr>
<tr>
<td>#101GE#0</td>
<td>&lt;empty&gt; \geq &lt;empty&gt; holds.</td>
<td>#101GE#0</td>
</tr>
<tr>
<td>#101GT#0</td>
<td>&lt;empty&gt; &gt; 0 does not hold.</td>
<td>#101GT#0</td>
</tr>
</tbody>
</table>

Hold-conditions and not-hold-conditions list
(For conditional expressions including undefined variables)

<table>
<thead>
<tr>
<th>Left side</th>
<th>Right side</th>
<th>EQ</th>
<th>NE</th>
<th>GT</th>
<th>LT</th>
<th>GE</th>
<th>LE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Empty</td>
<td>Empty</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
</tr>
<tr>
<td>Constant</td>
<td>Constant</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
<td>H</td>
</tr>
</tbody>
</table>

H: Holds (The conditional expression holds.)
Blank: The conditional expression does not hold.
13-15-4 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.

\[
\text{G65P} p_1 \text{L} _1 <\text{argument}>
\]

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:

<table>
<thead>
<tr>
<th>Call commands</th>
<th>Argument address</th>
<th>Local variable</th>
<th>Call commands</th>
<th>Argument address</th>
<th>Local variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>G65 G66</td>
<td></td>
<td></td>
<td>G65 G66</td>
<td></td>
<td></td>
</tr>
<tr>
<td>O O</td>
<td>A</td>
<td>#1</td>
<td>O O</td>
<td>R</td>
<td>#18</td>
</tr>
<tr>
<td>O O</td>
<td>B</td>
<td>#2</td>
<td>O O</td>
<td>S</td>
<td>#19</td>
</tr>
<tr>
<td>O O</td>
<td>C</td>
<td>#3</td>
<td>O O</td>
<td>T</td>
<td>#20</td>
</tr>
<tr>
<td>O O</td>
<td>D</td>
<td>#7</td>
<td>O O</td>
<td>U</td>
<td>#21</td>
</tr>
<tr>
<td>O O</td>
<td>E</td>
<td>#8</td>
<td>O O</td>
<td>V</td>
<td>#22</td>
</tr>
<tr>
<td>O O</td>
<td>F</td>
<td>#9</td>
<td>O O</td>
<td>W</td>
<td>#23</td>
</tr>
<tr>
<td>× ×*</td>
<td>G</td>
<td>#10</td>
<td>O O</td>
<td>X</td>
<td>#24</td>
</tr>
<tr>
<td>O O</td>
<td>H</td>
<td>#11</td>
<td>O O</td>
<td>Y</td>
<td>#25</td>
</tr>
<tr>
<td>O O</td>
<td>I</td>
<td>#4</td>
<td>O O</td>
<td>Z</td>
<td>#26</td>
</tr>
<tr>
<td>O O</td>
<td>J</td>
<td>#5</td>
<td></td>
<td>–</td>
<td>#27</td>
</tr>
<tr>
<td>O O</td>
<td>K</td>
<td>#6</td>
<td></td>
<td>–</td>
<td>#28</td>
</tr>
<tr>
<td>× ×*</td>
<td>L</td>
<td>#12</td>
<td></td>
<td>–</td>
<td>#29</td>
</tr>
<tr>
<td>O O</td>
<td>M</td>
<td>#13</td>
<td></td>
<td>–</td>
<td>#30</td>
</tr>
<tr>
<td>× ×*</td>
<td>N</td>
<td>#14</td>
<td></td>
<td>–</td>
<td>#31</td>
</tr>
<tr>
<td>× ×*</td>
<td>O</td>
<td>#15</td>
<td></td>
<td>–</td>
<td>#32</td>
</tr>
<tr>
<td>× ×*</td>
<td>P</td>
<td>#16</td>
<td></td>
<td>–</td>
<td>#33</td>
</tr>
<tr>
<td>O O</td>
<td>Q</td>
<td>#17</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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.
1. Local variables for a subprogram can be defined by specifying <argument> when calling a macro.

```
G65P9900A60.S100.F800
M02
```

```
#5=#4010
G91G01 X[#19*COS[#1]]
    Y[#19*SIN[#1]]F#9
M99
```

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.

2. Within a subprogram, local variables can be freely used.

```
G65P1A100.B50.J10.F500
M02
```

```
#30=FUP[#2/#5/2]
#5=#2/#30/2
M98H100L#30
X#1
M99
N100G1X#1F#9
M99
N100G1X#5F#9
X#5
M99
```

```
A (#1)=100.000
B (#2)=50.000
F (#9)=500
J (#5)=10.000 → 8.333
   (8.333)
   (#30)
```

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.

<table>
<thead>
<tr>
<th>System variable</th>
<th>Points</th>
<th>Interface input signal</th>
<th>System variable</th>
<th>Points</th>
<th>Interface input signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1000</td>
<td>1</td>
<td>Register R72, bit 0</td>
<td>#1016</td>
<td>1</td>
<td>Register R73, bit 0</td>
</tr>
<tr>
<td>#1001</td>
<td>1</td>
<td>Register R72, bit 1</td>
<td>#1017</td>
<td>1</td>
<td>Register R73, bit 1</td>
</tr>
<tr>
<td>#1002</td>
<td>1</td>
<td>Register R72, bit 2</td>
<td>#1018</td>
<td>1</td>
<td>Register R73, bit 2</td>
</tr>
<tr>
<td>#1003</td>
<td>1</td>
<td>Register R72, bit 3</td>
<td>#1019</td>
<td>1</td>
<td>Register R73, bit 3</td>
</tr>
<tr>
<td>#1004</td>
<td>1</td>
<td>Register R72, bit 4</td>
<td>#1020</td>
<td>1</td>
<td>Register R73, bit 4</td>
</tr>
<tr>
<td>#1005</td>
<td>1</td>
<td>Register R72, bit 5</td>
<td>#1021</td>
<td>1</td>
<td>Register R73, bit 5</td>
</tr>
<tr>
<td>#1006</td>
<td>1</td>
<td>Register R72, bit 6</td>
<td>#1022</td>
<td>1</td>
<td>Register R73, bit 6</td>
</tr>
<tr>
<td>#1007</td>
<td>1</td>
<td>Register R72, bit 7</td>
<td>#1023</td>
<td>1</td>
<td>Register R73, bit 7</td>
</tr>
<tr>
<td>#1008</td>
<td>1</td>
<td>Register R72, bit 8</td>
<td>#1024</td>
<td>1</td>
<td>Register R73, bit 8</td>
</tr>
<tr>
<td>#1009</td>
<td>1</td>
<td>Register R72, bit 9</td>
<td>#1025</td>
<td>1</td>
<td>Register R73, bit 9</td>
</tr>
<tr>
<td>#1010</td>
<td>1</td>
<td>Register R72, bit 10</td>
<td>#1026</td>
<td>1</td>
<td>Register R73, bit 10</td>
</tr>
<tr>
<td>#1011</td>
<td>1</td>
<td>Register R72, bit 11</td>
<td>#1027</td>
<td>1</td>
<td>Register R73, bit 11</td>
</tr>
<tr>
<td>#1012</td>
<td>1</td>
<td>Register R72, bit 12</td>
<td>#1028</td>
<td>1</td>
<td>Register R73, bit 12</td>
</tr>
<tr>
<td>#1013</td>
<td>1</td>
<td>Register R72, bit 13</td>
<td>#1029</td>
<td>1</td>
<td>Register R73, bit 13</td>
</tr>
<tr>
<td>#1014</td>
<td>1</td>
<td>Register R72, bit 14</td>
<td>#1030</td>
<td>1</td>
<td>Register R73, bit 14</td>
</tr>
<tr>
<td>#1015</td>
<td>1</td>
<td>Register R72, bit 15</td>
<td>#1031</td>
<td>1</td>
<td>Register R73, bit 15</td>
</tr>
<tr>
<td>#1032</td>
<td>32</td>
<td>Register R72 and R73</td>
<td>#1033</td>
<td>32</td>
<td>Register R74 and R75</td>
</tr>
<tr>
<td>#1034</td>
<td>32</td>
<td>Register R76 and R77</td>
<td>#1035</td>
<td>32</td>
<td>Register R78 and R79</td>
</tr>
</tbody>
</table>

**Note:** The following interface input signals are used exclusively in the NC system operation (cannot be used for other purposes).

<table>
<thead>
<tr>
<th>Interface input signal</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Register R72, bit 0</td>
<td>Touch sensor mounted in the spindle</td>
</tr>
<tr>
<td>Register R72, bit 4</td>
<td>X- and Y-axis machine lock ON</td>
</tr>
<tr>
<td>Register R72, bit 5</td>
<td>M-, S-, T-code lock ON</td>
</tr>
<tr>
<td>Register R72, bit 6</td>
<td>Z-axis machine lock ON</td>
</tr>
</tbody>
</table>
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.

<table>
<thead>
<tr>
<th>System variable</th>
<th>Points</th>
<th>Interface output signal</th>
<th>System variable</th>
<th>Points</th>
<th>Interface output signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1100</td>
<td>1</td>
<td>Register R172, bit 0</td>
<td>#1116</td>
<td>1</td>
<td>Register R173, bit 0</td>
</tr>
<tr>
<td>#1101</td>
<td>1</td>
<td>Register R172, bit 1</td>
<td>#1117</td>
<td>1</td>
<td>Register R173, bit 1</td>
</tr>
<tr>
<td>#1102</td>
<td>1</td>
<td>Register R172, bit 2</td>
<td>#1118</td>
<td>1</td>
<td>Register R173, bit 2</td>
</tr>
<tr>
<td>#1103</td>
<td>1</td>
<td>Register R172, bit 3</td>
<td>#1119</td>
<td>1</td>
<td>Register R173, bit 3</td>
</tr>
<tr>
<td>#1104</td>
<td>1</td>
<td>Register R172, bit 4</td>
<td>#1120</td>
<td>1</td>
<td>Register R173, bit 4</td>
</tr>
<tr>
<td>#1105</td>
<td>1</td>
<td>Register R172, bit 5</td>
<td>#1121</td>
<td>1</td>
<td>Register R173, bit 5</td>
</tr>
<tr>
<td>#1106</td>
<td>1</td>
<td>Register R172, bit 6</td>
<td>#1122</td>
<td>1</td>
<td>Register R173, bit 6</td>
</tr>
<tr>
<td>#1107</td>
<td>1</td>
<td>Register R172, bit 7</td>
<td>#1123</td>
<td>1</td>
<td>Register R173, bit 7</td>
</tr>
<tr>
<td>#1108</td>
<td>1</td>
<td>Register R172, bit 8</td>
<td>#1124</td>
<td>1</td>
<td>Register R173, bit 8</td>
</tr>
<tr>
<td>#1109</td>
<td>1</td>
<td>Register R172, bit 9</td>
<td>#1125</td>
<td>1</td>
<td>Register R173, bit 9</td>
</tr>
<tr>
<td>#1110</td>
<td>1</td>
<td>Register R172, bit 10</td>
<td>#1126</td>
<td>1</td>
<td>Register R173, bit 10</td>
</tr>
<tr>
<td>#1111</td>
<td>1</td>
<td>Register R172, bit 11</td>
<td>#1127</td>
<td>1</td>
<td>Register R173, bit 11</td>
</tr>
<tr>
<td>#1112</td>
<td>1</td>
<td>Register R172, bit 12</td>
<td>#1128</td>
<td>1</td>
<td>Register R173, bit 12</td>
</tr>
<tr>
<td>#1113</td>
<td>1</td>
<td>Register R172, bit 13</td>
<td>#1129</td>
<td>1</td>
<td>Register R173, bit 13</td>
</tr>
<tr>
<td>#1114</td>
<td>1</td>
<td>Register R172, bit 14</td>
<td>#1130</td>
<td>1</td>
<td>Register R173, bit 14</td>
</tr>
<tr>
<td>#1115</td>
<td>1</td>
<td>Register R172, bit 15</td>
<td>#1131</td>
<td>1</td>
<td>Register R173, bit 15</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>System variable</th>
<th>Points</th>
<th>Interface output signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1132</td>
<td>32</td>
<td>Register R172 and R173</td>
</tr>
<tr>
<td>#1133</td>
<td>32</td>
<td>Register R174 and R175</td>
</tr>
<tr>
<td>#1134</td>
<td>32</td>
<td>Register R176 and R177</td>
</tr>
<tr>
<td>#1135</td>
<td>32</td>
<td>Register R178 and R179</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Range of variable Nos.</th>
<th>Item</th>
</tr>
</thead>
<tbody>
<tr>
<td>#100001 - #100000+n</td>
<td>#10001 - #10000+n</td>
</tr>
<tr>
<td>#2001 - #2000+n</td>
<td>Geometric offset Z</td>
</tr>
<tr>
<td>#110001 - #110000+n</td>
<td>#11001 - #11000+n</td>
</tr>
<tr>
<td>#2201 - #2200+n</td>
<td>Wear compensation Z</td>
</tr>
<tr>
<td>#160001 - #160000+n</td>
<td>#16001 - #16000+n</td>
</tr>
<tr>
<td>#2401 - #2400+n</td>
<td>Geometric offset Nose-R</td>
</tr>
<tr>
<td>#170001 - #170000+n</td>
<td>#17001 - #17000+n</td>
</tr>
<tr>
<td>#2601 - #2600+n</td>
<td>Wear compensation Nose-R</td>
</tr>
<tr>
<td>#120001 - #120000+n</td>
<td>#12001 - #12000+n</td>
</tr>
<tr>
<td>#2101 - #2100+n</td>
<td>Geometric offset X</td>
</tr>
<tr>
<td>#130001 - #130000+n</td>
<td>#13001 - #13000+n</td>
</tr>
<tr>
<td>#2301 - #2300+n</td>
<td>Wear compensation X</td>
</tr>
<tr>
<td>#140001 - #140000+n</td>
<td>#14001 - #14000+n</td>
</tr>
<tr>
<td>#2401 - #2400+n</td>
<td>Geometric offset Y</td>
</tr>
<tr>
<td>#150001 - #150000+n</td>
<td>#15001 - #15000+n</td>
</tr>
<tr>
<td>#2501 - #2500+n</td>
<td>Wear compensation Y</td>
</tr>
<tr>
<td>#180001 - #180000+n</td>
<td>#18001 - #18000+n</td>
</tr>
<tr>
<td></td>
<td>Offsetting direction</td>
</tr>
</tbody>
</table>

n: Number of tools available (according to the specification of the machine)

<table>
<thead>
<tr>
<th>Variable numbers</th>
<th>Maximum of n</th>
</tr>
</thead>
<tbody>
<tr>
<td>#100001 - #184000</td>
<td>4000</td>
</tr>
<tr>
<td>#10001 - #18999</td>
<td>999</td>
</tr>
<tr>
<td>#2001 - #2800</td>
<td>200</td>
</tr>
</tbody>
</table>

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

```
G28 X0 T01
M06
#1=#5003
G00 Z-500.
G31 Z-100 F100
#10001=#5063-#1
```

**Program example**

<table>
<thead>
<tr>
<th>Program example</th>
<th>After execution</th>
<th>Common variables</th>
<th>Tool offset data</th>
</tr>
</thead>
<tbody>
<tr>
<td>#101=1000</td>
<td>#10001=#101</td>
<td>#101=1000.0</td>
<td>H1=1000.000</td>
</tr>
<tr>
<td>#102=#10001</td>
<td></td>
<td>#102=1000.0</td>
<td></td>
</tr>
</tbody>
</table>

**After execution**

<table>
<thead>
<tr>
<th>H1=1000.000</th>
</tr>
</thead>
</table>

**Example:** Tool offset data measuring

<table>
<thead>
<tr>
<th>Return to zero point</th>
<th>Tool change (Spindle T01)</th>
<th>Starting point memory</th>
<th>Rapid feed to safe position</th>
<th>Skip measuring</th>
<th>Measuring distance calculation and tool offset data setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>G00</td>
<td>G31</td>
<td>#5063</td>
<td>Sensor</td>
<td>#5003</td>
</tr>
</tbody>
</table>

**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.

<table>
<thead>
<tr>
<th>Axis No.</th>
<th>1st axis</th>
<th>2nd axis</th>
<th>3rd axis</th>
<th>16th axis</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>SHIFT</td>
<td>#5201</td>
<td>#5202</td>
<td>#5203</td>
<td>#5216</td>
<td>An external data input/output optional spec. is required.</td>
</tr>
<tr>
<td>G54</td>
<td>#5221</td>
<td>#5222</td>
<td>#5223</td>
<td>#5236</td>
<td>A workpiece coordinate system offset feature is required.</td>
</tr>
<tr>
<td>G55</td>
<td>#5241</td>
<td>#5242</td>
<td>#5243</td>
<td>#5256</td>
<td></td>
</tr>
<tr>
<td>G56</td>
<td>#5261</td>
<td>#5262</td>
<td>#5263</td>
<td>#5276</td>
<td></td>
</tr>
<tr>
<td>G57</td>
<td>#5281</td>
<td>#5282</td>
<td>#5283</td>
<td>#5296</td>
<td></td>
</tr>
<tr>
<td>G58</td>
<td>#5301</td>
<td>#5302</td>
<td>#5303</td>
<td>#5316</td>
<td></td>
</tr>
<tr>
<td>G59</td>
<td>#5321</td>
<td>#5322</td>
<td>#5323</td>
<td>#5336</td>
<td></td>
</tr>
</tbody>
</table>

(Example 1)

N1 G28X0Y0Z0
N2 #5221=-20.#5222=-20.
N3 G90G00G54X0Y0
N10 #5221=-90.#5222=-10.
N11 G90G00G54X0Y0
M02

(Example 2)

N100 #5221=$5221+$5201
#5222=$5222+$5202
#5241=$5241+$5201
#5242=$5242+$5202
#5201=0 #5202=0

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.

<table>
<thead>
<tr>
<th>Data name</th>
<th>Axis No.</th>
<th>1st axis</th>
<th>2nd axis</th>
<th>3rd axis</th>
<th>4th axis</th>
<th>16th axis</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>G54.1P1</td>
<td>#70001</td>
<td>#70002</td>
<td>#70003</td>
<td>#70004</td>
<td>#70016</td>
<td></td>
<td>Only available with the optional function for additional coordinate system offset.</td>
</tr>
<tr>
<td>G54.1P2</td>
<td>#70021</td>
<td>#70022</td>
<td>#70023</td>
<td>#70024</td>
<td>#70036</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G54.1P299</td>
<td>#75961</td>
<td>#75962</td>
<td>#75963</td>
<td>#75964</td>
<td>#75976</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G54.1P300</td>
<td>#75981</td>
<td>#75982</td>
<td>#75983</td>
<td>#75984</td>
<td>#75996</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Alternatively, variables numbered 7001 to 7956 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:

\[7000 + (n – 1) \times 20 + k\]

**Note:** The total number of controllable axes depends on the machine specifications.

<table>
<thead>
<tr>
<th>Data name</th>
<th>Axis No.</th>
<th>1st axis</th>
<th>2nd axis</th>
<th>3rd axis</th>
<th>4th axis</th>
<th>16th axis</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>G54.1P1</td>
<td>#7001</td>
<td>#7002</td>
<td>#7003</td>
<td>#7004</td>
<td>#7016</td>
<td></td>
<td>Only available with the optional function for additional coordinate system offset.</td>
</tr>
<tr>
<td>G54.1P2</td>
<td>#7021</td>
<td>#7022</td>
<td>#7023</td>
<td>#7024</td>
<td>#7036</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G54.1P3</td>
<td>#7041</td>
<td>#7042</td>
<td>#7043</td>
<td>#7044</td>
<td>#7056</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G54.1P48</td>
<td>#7941</td>
<td>#7942</td>
<td>#7943</td>
<td>#7944</td>
<td>#7956</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

7. **Local coordinate system offset (#5381 to #5386)**

As shown below, there are variable numbers provided for reading the local offsetting values on the current workpiece coordinate system.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#5381</td>
<td>Local offsetting value for the 1st axis.</td>
</tr>
<tr>
<td>#5382</td>
<td>Local offsetting value for the 2nd axis.</td>
</tr>
<tr>
<td>#5383</td>
<td>Local offsetting value for the 3rd axis.</td>
</tr>
<tr>
<td>#5384</td>
<td>Local offsetting value for the 4th axis.</td>
</tr>
<tr>
<td>#5385</td>
<td>Local offsetting value for the 5th axis.</td>
</tr>
<tr>
<td>#5386</td>
<td>Local offsetting value for the 6th axis.</td>
</tr>
</tbody>
</table>

8. **Angle of rotation of the workpiece coordinate system (#5387 and #5397)**

As shown below, there are variable numbers provided for reading the angle of rotation of the workpiece coordinate system.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#5387</td>
<td>Rotation angle (as specified with argument R in a G92.5 command)</td>
</tr>
<tr>
<td>#5397</td>
<td>Rotation angle (by G92.5) + Angle of rotation of the local coordinate system (by G68)</td>
</tr>
</tbody>
</table>
9. Positions of zero points

As shown below, there are variable numbers provided for reading the axis positions of the 1st to 4th zero point (as set by parameters M4 to M7).

<table>
<thead>
<tr>
<th></th>
<th>1st axis</th>
<th>2nd axis</th>
<th>......</th>
<th>16th axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>M4</td>
<td>#5701</td>
<td>#5702</td>
<td>......</td>
<td>#5716</td>
</tr>
<tr>
<td>M5</td>
<td>#5721</td>
<td>#5722</td>
<td>......</td>
<td>#5736</td>
</tr>
<tr>
<td>M6</td>
<td>#5741</td>
<td>#5742</td>
<td>......</td>
<td>#5756</td>
</tr>
<tr>
<td>M7</td>
<td>#5761</td>
<td>#5762</td>
<td>......</td>
<td>#5776</td>
</tr>
</tbody>
</table>

10. NC alarm (#3000)

The NC unit can be forced into an alarm status using variables number 3000.

#3000 = 70 (CALL#PROGRAMMER#TEL#530)

Alarm No.  Alarm message

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.

<table>
<thead>
<tr>
<th>Designated alarm No.</th>
<th>Displayed alarm No.</th>
<th>Displayed alarm message</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 to 20</td>
<td>[Designated alarm No.] + 979</td>
<td>Message preset for the displayed alarm No.  *1</td>
</tr>
<tr>
<td>21 to 6999</td>
<td>[Designated alarm No.] + 3000</td>
<td>Designated alarm message as it is  *2</td>
</tr>
</tbody>
</table>

*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.

Program ex. 1 (Command for display of "980 MACRO USER ALARM 1" on condition of #1=0)

```plaintext
: IF[#1NE0]GOTO100
#3000=1
N100**********
:
```

Program ex. 2 (Command for display of "3021#ORIGINAL#ALARM#1" on condition of #2=0)

```plaintext
: IF[#2NE0]GOTO200
#3000=21(#ORIGINAL#ALARM#1)
N200**********
:
```

Operation stop by NC alarm
11. Integrated time (#3001, #3002)

Using variables #3001 and #3002, you can read the integrated time existing during automatic operation or assign data.

<table>
<thead>
<tr>
<th>Type</th>
<th>Variable No.</th>
<th>Unit</th>
<th>Data at power-on</th>
<th>Initialization</th>
<th>Counting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Integrated time 1</td>
<td>3001</td>
<td>msec</td>
<td>Same as at power-off</td>
<td>Data is assigned in variables.</td>
<td>Always during power-on</td>
</tr>
<tr>
<td>Integrated time 2</td>
<td>3002</td>
<td></td>
<td></td>
<td></td>
<td>During auto-starting</td>
</tr>
</tbody>
</table>

The integrated time is cleared to 0 after having reached about $2.44 \times 10^{11}$ msec (about 7.7 years).

12. 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).

<table>
<thead>
<tr>
<th>#3003</th>
<th>Single block stop</th>
<th>Auxiliary-function completion signal</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Effective</td>
<td>Wait</td>
</tr>
<tr>
<td>1</td>
<td>Ineffective</td>
<td>Wait</td>
</tr>
<tr>
<td>2</td>
<td>Effective</td>
<td>No wait</td>
</tr>
<tr>
<td>3</td>
<td>Ineffective</td>
<td>No wait</td>
</tr>
</tbody>
</table>

Note: Variable #3003 is cleared to 0 by resetting.
13. 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.

<table>
<thead>
<tr>
<th>#3004</th>
<th>Bit 0</th>
<th>Bit 1</th>
<th>Bit 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Feed hold</td>
<td>Feed rate override</td>
<td>G09 check</td>
</tr>
<tr>
<td>1</td>
<td>Ineffective</td>
<td>Effective</td>
<td>Effective</td>
</tr>
<tr>
<td>2</td>
<td>Effective</td>
<td>Ineffective</td>
<td>Effective</td>
</tr>
<tr>
<td>3</td>
<td>Ineffective</td>
<td>Ineffective</td>
<td>Effective</td>
</tr>
<tr>
<td>4</td>
<td>Effective</td>
<td>Effective</td>
<td>Ineffective</td>
</tr>
<tr>
<td>5</td>
<td>Ineffective</td>
<td>Effective</td>
<td>Ineffective</td>
</tr>
<tr>
<td>6</td>
<td>Effective</td>
<td>Ineffective</td>
<td>Ineffective</td>
</tr>
<tr>
<td>7</td>
<td>Ineffective</td>
<td>Ineffective</td>
<td>Ineffective</td>
</tr>
</tbody>
</table>

**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.

14. 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.

15. 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.

```
<table>
<thead>
<tr>
<th>Bit</th>
<th>15</th>
<th>14</th>
<th>13</th>
<th>12</th>
<th>11</th>
<th>10</th>
<th>9</th>
<th>8</th>
<th>7</th>
<th>6</th>
<th>5</th>
<th>4</th>
<th>3</th>
<th>2</th>
<th>1</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axis No.</td>
<td>6</td>
<td>5</td>
<td>4</td>
<td>3</td>
<td>2</td>
<td>1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
```
16. G-command modal status

The G-command modal status in the previous 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.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Previous block</td>
<td>Block executed</td>
</tr>
<tr>
<td>#4001</td>
<td>#4201</td>
</tr>
<tr>
<td>#4002</td>
<td>#4202</td>
</tr>
<tr>
<td>#4003</td>
<td>#4203</td>
</tr>
<tr>
<td>#4004</td>
<td>#4204</td>
</tr>
<tr>
<td>#4005</td>
<td>#4205</td>
</tr>
<tr>
<td>#4006</td>
<td>#4206</td>
</tr>
<tr>
<td>#4007</td>
<td>#4207</td>
</tr>
<tr>
<td>#4008</td>
<td>#4208</td>
</tr>
<tr>
<td>#4010</td>
<td>#4210</td>
</tr>
<tr>
<td>#4011</td>
<td>#4211</td>
</tr>
<tr>
<td>#4012</td>
<td>#4212</td>
</tr>
<tr>
<td>#4013</td>
<td>#4213</td>
</tr>
<tr>
<td>#4014</td>
<td>#4214</td>
</tr>
<tr>
<td>#4015</td>
<td>#4215</td>
</tr>
<tr>
<td>#4016</td>
<td>#4216</td>
</tr>
<tr>
<td>#4017</td>
<td>#4217</td>
</tr>
<tr>
<td>#4018</td>
<td>#4218</td>
</tr>
<tr>
<td>#4019</td>
<td>#4219</td>
</tr>
<tr>
<td>#4020</td>
<td>#4220</td>
</tr>
<tr>
<td>#4021</td>
<td>#4221</td>
</tr>
<tr>
<td>#4022</td>
<td>#4222</td>
</tr>
<tr>
<td>#4023</td>
<td>#4223</td>
</tr>
<tr>
<td>#4024</td>
<td>#4224</td>
</tr>
<tr>
<td>#4025</td>
<td>#4225</td>
</tr>
<tr>
<td>#4026</td>
<td>#4226</td>
</tr>
<tr>
<td>#4027</td>
<td>#4227</td>
</tr>
</tbody>
</table>
17. Other modal information

Modal information about factors other than the G-command modal status in the previous 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.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Modal information</th>
<th>Variable Nos.</th>
<th>Modal information</th>
</tr>
</thead>
<tbody>
<tr>
<td>Previous block</td>
<td>Block executed</td>
<td>Previous block</td>
<td>Block executed</td>
</tr>
<tr>
<td>#4101 #4301</td>
<td>#4113 #4313</td>
<td>Miscellaneous function---M</td>
<td></td>
</tr>
<tr>
<td>#4102 #4302</td>
<td>No. 2 miscellaneous function</td>
<td>#4114 #4314</td>
<td>Sequence No.---N</td>
</tr>
<tr>
<td>#4103 #4303</td>
<td>#4115 #4315</td>
<td>Program No.---O</td>
<td></td>
</tr>
<tr>
<td>#4104 #4304</td>
<td>#4116 #4316</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4105 #4305</td>
<td>#4117 #4317</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4106 #4306</td>
<td>#4118 #4318</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4107 #4307</td>
<td>Tool radius comp. No.---D</td>
<td>#4119 #4319</td>
<td>Spindle function---S</td>
</tr>
<tr>
<td>#4108 #4308</td>
<td>#4120 #4320</td>
<td>Tool function---T</td>
<td></td>
</tr>
<tr>
<td>#4109 #4309</td>
<td>Rate of feed---F</td>
<td>#4130 #4330</td>
<td>Addt. workpiece coordinate system G54-G59: 0, G54.1F1-P300: 1-300</td>
</tr>
<tr>
<td>#4110 #4310</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4111 #4311</td>
<td>Tool length offset No.---H</td>
<td>#4131</td>
<td></td>
</tr>
<tr>
<td>#4112 #4312</td>
<td>#4132</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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).

Example: Given below is an example to show the difference between variables #4113 and #4313 (for checking modal information about M-codes) in the information that is obtained in the parallel flow of actual execution and internal pre-reading (analysis) for automatic operation.

Previous block (#4113) Block being executed (#4313)

\[
\begin{align*}
\text{N001} & \text{ M3} \\
\text{N002} & \quad \text{In execution} \\
\text{N003} & \\
\text{N006 M200} & \quad \text{In analysis} \\
\text{N007 \#100=\#4113} & \quad \text{In analysis} \\
\text{N008} & \quad \text{In analysis} \\
\end{align*}
\]

Reading the information about M-codes (M200) at the block previous to the currently analyzed one. Reading the information about M-codes (M3) at the block being executed.
18. Position information

Using variables numbers from #5001 to #5176, you can check the ending-point coordinates of the previous block, machine coordinates, workpiece coordinates, skip coordinates, skip coordinate during 3-D coordinate conversion, tool position offset coordinates, and servo deviation amounts.

<table>
<thead>
<tr>
<th>Axis No.</th>
<th>Position information</th>
<th>End point coordinates of previous block</th>
<th>Machine coordinate</th>
<th>Workpiece coordinate</th>
<th>Skip coordinate</th>
<th>Skip coordinate during 3-D coordinate conversion</th>
<th>Tool position offset coordinates</th>
<th>Servo deviation amount</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>#5001</td>
<td>#5021</td>
<td>#5041</td>
<td>#5061</td>
<td>#5161</td>
<td>#5081</td>
<td>#5101</td>
<td>#5116</td>
</tr>
<tr>
<td>2</td>
<td>#5002</td>
<td>#5022</td>
<td>#5042</td>
<td>#5062</td>
<td>#5162</td>
<td>#5082</td>
<td>#5102</td>
<td>#5116</td>
</tr>
<tr>
<td>3</td>
<td>#5003</td>
<td>#5023</td>
<td>#5043</td>
<td>#5063</td>
<td>#5163</td>
<td>#5083</td>
<td>#5103</td>
<td>#5116</td>
</tr>
<tr>
<td>16</td>
<td>#5016</td>
<td>#5036</td>
<td>#5056</td>
<td>#5076</td>
<td>#5176</td>
<td>#5096</td>
<td>#5116</td>
<td></td>
</tr>
</tbody>
</table>

Remarks (Reading during move) Possible Impossible Impossible Possible Possible Impossible Possible Possible

Note 1: The number of controllable axes depends on the machine specifications.

Note 2: Axis number refers to the currently active system.

1. The ending-point coordinates and skip coordinates read refer to the workpiece coordinate system. Skip coordinates during 3-D coordinate conversion are read with respect to the workpiece coordinate system converted by G68 (Coordinate system rotation).

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.

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 the section (Chapter 15) on skip functions 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:

```
G65P9031X100.Y100.Z-10.F200
```

### Example 1

**O9031**

```
N1 #180=#4003
N2 #30=#5001#31=#5002
N3 G91G0Z#26F#9
N4 G31X#24Y#25F#9
N5 G90G00X#301#31
N6 #101=#30-#5061#102=#31-#5062
N7 #103=SQR[#101?#101+#102?#102]
N8 G91G0Z-#26
N9 IF[#180EQ91]GOTO11
N10 G90
N11 M99
```

**Argument (Local variable)**

<table>
<thead>
<tr>
<th>F (#9)</th>
<th>200</th>
</tr>
</thead>
<tbody>
<tr>
<td>X (#24)</td>
<td>100.000</td>
</tr>
<tr>
<td>Y (#25)</td>
<td>100.000</td>
</tr>
<tr>
<td>Z (#26)</td>
<td>-10.000</td>
</tr>
</tbody>
</table>

**To subprogram**

| #101 | 87.245 |
| #102 | 87.245 |
| #103 | 123.383 |

**Common variable**

| #111 = -75. + ε |
| #112 = -75. + ε |
| #121 = -25. + ε |
| #122 = -75. + ε |

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.

Example 2:  Skip signal input coordinates reading:

```
N1 G91G28X0Y0
N2 G90G00X0Y0
N3 X0Y-100.
N4 G31X-150.Y-50.F80
N5 #111=#5061#112=#5062
N6 G00Y0
N7 G31X0
N8 #121=#5061#122=#5062
N9 M02
```

**Example 2**

```
#111 = -75. + ε #112 = -75. + ε
#121 = -25. + ε #122 = -75. + ε
```

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.
19. Measurement parameters

Variable numbers provided for reading measurement parameters are as follows:

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
<th>Parameter</th>
<th>Register R</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3071</td>
<td>Specification of tolerance (Lower limit) for workpiece measurement</td>
<td>K17</td>
<td></td>
</tr>
<tr>
<td>#3072</td>
<td>Specification of tolerance (Upper limit) for workpiece measurement</td>
<td>K18</td>
<td></td>
</tr>
<tr>
<td>#3086</td>
<td>Rate of skip feed for measurement</td>
<td>K13</td>
<td></td>
</tr>
<tr>
<td>#3087</td>
<td>Rate of approach feed for measurement</td>
<td>K14</td>
<td></td>
</tr>
<tr>
<td>#3088</td>
<td>Rate of skip feed for measurement (for the C-axis)</td>
<td>K15</td>
<td></td>
</tr>
<tr>
<td>#3089</td>
<td>Rate of approach feed for measurement (for the C-axis)</td>
<td>K16</td>
<td></td>
</tr>
<tr>
<td>#5501</td>
<td>Stylus eccentricity of touch sensor (X-component)</td>
<td>L1</td>
<td></td>
</tr>
<tr>
<td>#5502</td>
<td>Stylus eccentricity of touch sensor (Y-component)</td>
<td>L2</td>
<td></td>
</tr>
<tr>
<td>#5503</td>
<td>Radius of stylus ball of touch sensor (X-component)</td>
<td>L3</td>
<td></td>
</tr>
<tr>
<td>#5504</td>
<td>Radius of stylus ball of touch sensor (Y-component)</td>
<td>L4</td>
<td></td>
</tr>
<tr>
<td>#58165</td>
<td>Retry frequency for workpiece measurement</td>
<td>K23</td>
<td>R8164, 8165</td>
</tr>
<tr>
<td>#58167</td>
<td>Rate of skip feed for measurement</td>
<td>K13</td>
<td>R8166, 8167</td>
</tr>
<tr>
<td>#58169</td>
<td>Rate of approach feed for measurement</td>
<td>K14</td>
<td>R8168, 8169</td>
</tr>
<tr>
<td>#58171</td>
<td>Measurement stroke for workpiece measurement</td>
<td>K19</td>
<td>R8170, 8171</td>
</tr>
<tr>
<td>#58172</td>
<td>Safety contour clearance – Outside diameter (Radius value)</td>
<td>TC37</td>
<td>R8172</td>
</tr>
<tr>
<td>#58173</td>
<td>Safety contour clearance – Inside diameter (Radius value)</td>
<td>TC38</td>
<td>R8173</td>
</tr>
<tr>
<td>#58174</td>
<td>Safety contour clearance – Front face</td>
<td>TC39</td>
<td>R8174</td>
</tr>
<tr>
<td>#58175</td>
<td>Safety contour clearance – Back face</td>
<td>TC40</td>
<td>R8175</td>
</tr>
<tr>
<td>#58205</td>
<td>Measurement stroke for tool nose measurement</td>
<td>K20</td>
<td></td>
</tr>
<tr>
<td>#58221</td>
<td>Width of the tool-nose measurement sensor along the X-axis</td>
<td>L22</td>
<td></td>
</tr>
<tr>
<td>#58223</td>
<td>Width of the tool-nose measurement sensor along the Z-axis</td>
<td>L23</td>
<td></td>
</tr>
<tr>
<td>#58225</td>
<td>Reference position of the tool-nose measurement sensor in X</td>
<td>L24</td>
<td></td>
</tr>
<tr>
<td>#58227</td>
<td>Reference position of the tool-nose measurement sensor in Z</td>
<td>L25</td>
<td></td>
</tr>
<tr>
<td>#58229</td>
<td>Reference position of the tool-nose measurement sensor in Y</td>
<td>L26</td>
<td></td>
</tr>
<tr>
<td>#58259</td>
<td>Z-axis escape distance from the approach point after TOOL EYE measurement</td>
<td>L28</td>
<td></td>
</tr>
</tbody>
</table>

20. 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.

<table>
<thead>
<tr>
<th>System variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#51999</td>
<td>Tool number of the spindle tool</td>
</tr>
<tr>
<td>#3020</td>
<td>Number of tool data index of the spindle tool</td>
</tr>
</tbody>
</table>

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.
21. **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.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
</table>
| #3022     | Designation of the required tool (for writing only). As is the case with a T-code, use the integral and decimal parts respectively for specifying the required tool with its number and suffix. #3022 = OOO. ΔΔ  
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:**

<table>
<thead>
<tr>
<th>TNo.</th>
<th>#3022 setting</th>
<th>Reading in #3023</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A</td>
<td>1.01</td>
</tr>
<tr>
<td>1</td>
<td>B</td>
<td>1.02</td>
</tr>
<tr>
<td>1</td>
<td>C</td>
<td>1.03</td>
</tr>
<tr>
<td>2</td>
<td>A</td>
<td>2.61</td>
</tr>
<tr>
<td>2</td>
<td>B</td>
<td>2.62</td>
</tr>
<tr>
<td>2</td>
<td>C</td>
<td>2.63</td>
</tr>
<tr>
<td>3</td>
<td>H</td>
<td>3.08</td>
</tr>
<tr>
<td>3</td>
<td>V</td>
<td>3.22</td>
</tr>
<tr>
<td>3</td>
<td>Z</td>
<td>3.26</td>
</tr>
<tr>
<td>:</td>
<td>:</td>
<td>:</td>
</tr>
<tr>
<td>:</td>
<td>:</td>
<td>:</td>
</tr>
<tr>
<td>Failure</td>
<td>–</td>
<td>:</td>
</tr>
</tbody>
</table>
22. MAZATROL tool data

Using variables tabulated below, MAZATROL tool data can be read or written, as required.

Variables from #60001 ............ Tool quantity: 400 (maximum)
Variables from #600001 .......... Tool quantity: 4000 (maximum)

The maximum applicable tool quantity depends on the machine specifications.

<table>
<thead>
<tr>
<th>System variables</th>
<th>MAZATROL tool data</th>
</tr>
</thead>
<tbody>
<tr>
<td>#60001 to #6000+n</td>
<td>#600001 to #60000+n</td>
</tr>
<tr>
<td>#61001 to #6100+n</td>
<td>#610001 to #61000+n</td>
</tr>
<tr>
<td>#62001 to #6200+n</td>
<td>#620001 to #62000+n</td>
</tr>
<tr>
<td>#63001 to #6300+n</td>
<td>#630001 to #63000+n</td>
</tr>
<tr>
<td>#64001 to #6400+n</td>
<td>#640001 to #64000+n</td>
</tr>
<tr>
<td>#65001 to #6500+n</td>
<td>#650001 to #65000+n</td>
</tr>
<tr>
<td>#66001 to #6600+n</td>
<td>#660001 to #66000+n</td>
</tr>
<tr>
<td>#67001 to #6700+n</td>
<td>#670001 to #67000+n</td>
</tr>
<tr>
<td>#68001 to #6800+n</td>
<td>#680001 to #68000+n</td>
</tr>
</tbody>
</table>

Note 1: During tool path check, tool data can be read but cannot be written.

Note 2: Tool life flags (#62001 to #62400, #620001 to #624000) and tool damage flags (#63001 to #634000, #630001 to #634000) can take either 1 or 0 as their logical states (1 for ON, 0 for OFF).

Note 3: Tool life flags (#62001 to #62400, or #620001 to #624000) only refer to tool life management according to the tool’s application time. The flags for life management according to the tool’s application quantity and wear amount are stored in variables #49001 to #49960 or #490001 to #494000.

Example: Shown below is an example of programming for reading MAZATROL tool data with the aid of variables #3022 and #3023.

```plaintext
: #3022 = 1.01;  # For reading the number of index of the tool with TNo. 1 and suffix A.
#510 = #3023;  # Storing the read number of index (#3023) into variable #510.
#500 = #600000+#510;  # Storing Length A (turning) or Length (milling) into #500.
#501 = #610000+#510;  # Storing Nose R (turning) or Diameter (milling) into #501.
#502 = #620000+#510;  # Storing Tool life flag into #502.
#503 = #630000+#510;  # Storing Tool damage flag into #503.
#504 = #640000+#510;  # Storing Wear compensation X into #504.
#505 = #650000+#510;  # Storing Wear compensation Y into #505.
#506 = #660000+#510;  # Storing Wear compensation Z into #506.
#507 = #670000+#510;  # Storing Group number into #507.
#508 = #680000+#510;  # Storing Length B (turning) into #508.
:
```

Execution of the programmed data above stores particular data items on the TOOL DATA display into the relevant variables on the MACRO VARIABLE display, as illustrated below.
23. EIA/ISO tool data

Using variables tabulated below, EIA/ISO tool data (tool life management data) can be read or written, as required.

Variables from #40001 .............. Tool quantity: 960 (maximum)
Variables from #400001 .......... Tool quantity: 4000 (maximum)

The maximum applicable tool quantity depends on the machine specifications.

<table>
<thead>
<tr>
<th>System variables</th>
<th>Corresponding data</th>
</tr>
</thead>
<tbody>
<tr>
<td>#40001 to #40000+n</td>
<td>#400001 to #400000+n</td>
</tr>
<tr>
<td>#41001 to #41000+n</td>
<td>#410001 to #410000+n</td>
</tr>
<tr>
<td>#42001 to #42000+n</td>
<td>#420001 to #420000+n</td>
</tr>
<tr>
<td>#43001 to #43000+n</td>
<td>#430001 to #430000+n</td>
</tr>
<tr>
<td>#44001 to #44000+n</td>
<td>#440001 to #440000+n</td>
</tr>
<tr>
<td>#45001 to #45000+n</td>
<td>#450001 to #450000+n</td>
</tr>
<tr>
<td>#46001 to #46000+n</td>
<td>#460001 to #460000+n</td>
</tr>
<tr>
<td>#47001 to #47000+n</td>
<td>#470001 to #470000+n</td>
</tr>
<tr>
<td>#48001 to #48000+n</td>
<td>#480001 to #480000+n</td>
</tr>
<tr>
<td>#49001 to #49000+n</td>
<td>#490001 to #490000+n</td>
</tr>
<tr>
<td>0: No expiration of serviceable life</td>
<td></td>
</tr>
<tr>
<td>1: Expiration in terms of application quantity</td>
<td></td>
</tr>
<tr>
<td>2: Expiration in terms of application time</td>
<td></td>
</tr>
<tr>
<td>4: Expiration in terms of X-axis wear amount</td>
<td></td>
</tr>
<tr>
<td>8: Expiration in terms of Y-axis wear amount</td>
<td></td>
</tr>
<tr>
<td>16: Expiration in terms of Z-axis wear amount</td>
<td></td>
</tr>
</tbody>
</table>

Note 1: During tool path check, tool data can be read but cannot be written.

Note 2: Tool life flags (#42001 to #42960, #420001 to #424000) and tool damage flags (#43001 to #43960, #430001 to #434000) can take either 1 or 0 as their logical states (1 for ON, 0 for OFF).
Note 3: Tool life flags (#42001 to #42960, or #420001 to #424000) only refer to tool life management according to the tool’s application time. The flags for life management according to the tool’s application quantity and wear amount are stored in variables #49001 to #49960 or #490001 to #494000.

Note 4: Tool life flags (#49001 to #49960, or #490001 to #494000) contain the sum of the values resulting from all the available methods of tool life management.

Example: When the life of a tool is judged to have expired in terms of application quantity (1) and Z-axis wear amount (16), then the life flag of that tool amounts to “1 + 16 = 17”.

Note 5: The writing data of tool life flag always overwrites the current value.

Example: A writing value of 4 (for expiration of tool life only in terms of X-axis wear amount) will clear all the other flag components. It will not result, for example, in 5 (= 4 + 1) being set even when the current value would have to contain 1 (for expiration in terms of application quantity).

Note 6: The writing of tool application time (or quantity), tool life time (or quantity), or wear compensation amount cannot cause the respective life flag component to be turned on, which would correspond to the resulting conditions.

Example: When the current values of life quantity setting and application quantity are respectively 100 and 85 (no expiration of life in terms of application quantity), overwriting the application quantity with 105 will not cause the corresponding tool life flag to be changed accordingly.

Note 7: The identification between number and amount of tool length offset or radius compensation is made by referring to the tool data flag.

<table>
<thead>
<tr>
<th>Tool data flag</th>
<th>bit 0</th>
<th>bit 1</th>
<th>bit 2</th>
<th>bit 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length offset data No.</td>
<td>0</td>
<td>0</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Length offset amount</td>
<td>0</td>
<td>1</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>Radius comp. data No.</td>
<td>–</td>
<td>–</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Radius comp. amount</td>
<td>–</td>
<td>–</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

Remark: For information as to how to use the above system variables in programming, refer to the example given for the similar variables that are provided for MAZATROL tool data.

24. Date and time (Year-month-day and hour-minute-second)

Variables numbered 3011 and 3012 can be used to read date and time data.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3011</td>
<td>Date (Year-month-day)</td>
</tr>
<tr>
<td>#3012</td>
<td>Time (Hour-minute-second)</td>
</tr>
</tbody>
</table>

The date stored in #3011 consists of 8 digits (4, 2, and 2 digits denote respectively year, month, and day), and the time in #3012 is composed of three sets of two-digit data (for hour, minute, and second in that order).

Example: If the date is December 15, 2006 and the time is 16:45:10, data is set as follows in the corresponding system variables:

```
#3011 = 20061215
#3012 = 164510.
```
25. 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.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3901</td>
<td>Total number of machined parts</td>
</tr>
<tr>
<td>#3902</td>
<td>Number of parts required</td>
</tr>
</tbody>
</table>

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.

26. 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 alphanumerics or less that begin with a letter of the alphabet.

Format: SETVNNo [NAME1, NAME2, ......]

- Starting number of the variable to be named
- Name of #n (Variables name)
- Name of #n + 1 (Variable 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 ← Variables name assigned to #502
F50

27. System unit and Input unit: Inch or Metric (#3094)

Variable numbered 3094 can be used to secure information about the type and combination of system unit and input unit (inch or metric).

<table>
<thead>
<tr>
<th>#3094 value</th>
<th>System unit</th>
<th>Input unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Metric</td>
<td>G21 (Metric)</td>
</tr>
<tr>
<td>0</td>
<td>Inch</td>
<td>G20 (Inch)</td>
</tr>
<tr>
<td>1</td>
<td>Metric</td>
<td>G20 (Inch)</td>
</tr>
<tr>
<td>2</td>
<td>Inch</td>
<td>G21 (Metric)</td>
</tr>
</tbody>
</table>
28. **Inch data input as initial mode ON/OFF (#51000)**

Variable numbered 51000 can be used to secure information as to whether or not inch data input is set (by F91 bit 4) as the initial mode.

<table>
<thead>
<tr>
<th>#51000 value</th>
<th>F91 bit 4</th>
<th>Initial mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>0</td>
<td>Metric data input</td>
</tr>
<tr>
<td>10000</td>
<td>1</td>
<td>Inch data input</td>
</tr>
</tbody>
</table>

29. **C-axis clamping M-code (#3095)**

Variable numbered 3095 can be used to secure information on the number of the C-axis clamping M-code. Refer to the M-code list in the operating manual of the machine for details.

30. **MAZATROL coordinate system valid/invalid (#3098)**

Variable numbered 51000 can be used to secure information as to whether bit 1 of parameter F91 is set to “1” or “0” (to make MAZATROL coordinate system valid or invalid).

31. **Basic data for tool life management (#3102)**

Variable numbered 3102 can be used to obtain information about the data on the basis of which the tool life management is performed.

The bases for tool life management are to be set using bits 4, 5, 6, and 7 of parameter F113. See the separate Parameter List/Alarm List/M-code List for more information.

<table>
<thead>
<tr>
<th>#3102 value</th>
<th>Basic data for tool life management</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Application quantity of the tool (Number of workpieces machined with the tool)</td>
</tr>
<tr>
<td>2</td>
<td>Application time of the tool</td>
</tr>
<tr>
<td>4</td>
<td>X-axis wear compensation amount</td>
</tr>
<tr>
<td>8</td>
<td>Y-axis wear compensation amount</td>
</tr>
<tr>
<td>16</td>
<td>Z-axis wear compensation amount</td>
</tr>
</tbody>
</table>

Variable #3102 can take a value of the sum of multiple values enumerated above when the tool life management is currently done on the basis of the respective data types.

**Example:** When the tool life management is currently done with respect to the X-axis wear compensation amount (4) and the Z-axis wear compensation amount (16), then the variable in question amounts to “4 + 16 = 20”.

32. **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.

<table>
<thead>
<tr>
<th>#3200 setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 to 16</td>
<td>Serial numbers of significant axis settings</td>
</tr>
<tr>
<td></td>
<td>within the current system</td>
</tr>
</tbody>
</table>

**Example:** #3200 = 1; Designation of the first significant axis setting (normally: X) within the system.

<table>
<thead>
<tr>
<th>Variable Nos.</th>
<th>S parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3212</td>
<td>S12</td>
</tr>
<tr>
<td>#3223</td>
<td>S23</td>
</tr>
</tbody>
</table>

**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.

33. Dynamic offsetting II

Using variables tabulated below, it is possible to read dynamic offsetting values (X, Y, Z), coordinates of the center of table rotation (X, Y, Z), and the dynamic offset number.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#5121</td>
<td>Dynamic offsetting value X</td>
</tr>
<tr>
<td>#5122</td>
<td>Dynamic offsetting value Y</td>
</tr>
<tr>
<td>#5123</td>
<td>Dynamic offsetting value Z</td>
</tr>
<tr>
<td>#50700</td>
<td>X-coordinate of the center of table rotation</td>
</tr>
<tr>
<td>#50705</td>
<td>Y-coordinate of the center of table rotation</td>
</tr>
<tr>
<td>#50701</td>
<td>Z-coordinate of the center of table rotation</td>
</tr>
<tr>
<td>#5510</td>
<td>Dynamic offset number (0 to 8)</td>
</tr>
</tbody>
</table>

Moreover, use the following variables to read and write the values of the reference dynamic offset:

<table>
<thead>
<tr>
<th>Variables</th>
<th>1st axis</th>
<th>2nd axis</th>
<th>. . .</th>
<th>16th axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>G54.2P1</td>
<td>#5521</td>
<td>#5522</td>
<td>. . .</td>
<td>#5536</td>
</tr>
<tr>
<td>G54.2P2</td>
<td>#5541</td>
<td>#5542</td>
<td>. . .</td>
<td>#5556</td>
</tr>
<tr>
<td>G54.2P3</td>
<td>#5561</td>
<td>#5562</td>
<td>. . .</td>
<td>#5576</td>
</tr>
<tr>
<td>G54.2P4</td>
<td>#5581</td>
<td>#5582</td>
<td>. . .</td>
<td>#5596</td>
</tr>
<tr>
<td>G54.2P5</td>
<td>#5601</td>
<td>#5602</td>
<td>. . .</td>
<td>#5616</td>
</tr>
<tr>
<td>G54.2P6</td>
<td>#5621</td>
<td>#5622</td>
<td>. . .</td>
<td>#5636</td>
</tr>
<tr>
<td>G54.2P7</td>
<td>#5641</td>
<td>#5642</td>
<td>. . .</td>
<td>#5656</td>
</tr>
<tr>
<td>G54.2P8</td>
<td>#5661</td>
<td>#5662</td>
<td>. . .</td>
<td>#5676</td>
</tr>
</tbody>
</table>

34. Basic coordinates system of a MAZATROL program

Using variables tabulated below, it is possible to read and write the data of the currently active basic coordinates system of a MAZATROL program.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#5341</td>
<td>WPC-X</td>
<td>#5344</td>
<td>WPC-4</td>
</tr>
<tr>
<td>#5342</td>
<td>WPC-Y</td>
<td>#5345</td>
<td>WPC-C</td>
</tr>
<tr>
<td>#5343</td>
<td>WPC-Z</td>
<td>#5347</td>
<td>WPC-th</td>
</tr>
</tbody>
</table>

Note: When writing a rotational axis value in the mode of inch data input (to be checked with #4206), set the system variable concerned to one tenth (1/10) of the desired angle.

Example: To set WPC-C to 100 (degrees), enter

\[
#5345 = 10
\]

or with a decimal point

\[
#5345 = 10.0
\]
35. Current tool number and Current position in machine coordinates

Using variables tabulated below, it is possible to read the number of the system currently active, the number of the tool currently used, the machine coordinates of the current position, and the current Z- and C-offset values.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
<th>Register R</th>
</tr>
</thead>
<tbody>
<tr>
<td>#8000</td>
<td>Number of the system currently active</td>
<td></td>
</tr>
<tr>
<td>#8001</td>
<td>No. of the current tool (System 1)</td>
<td>R1999</td>
</tr>
<tr>
<td>#8002</td>
<td>No. of the current tool (System 2)</td>
<td>R3299</td>
</tr>
<tr>
<td>#8003</td>
<td>No. of the current tool (System 3)</td>
<td>R3399</td>
</tr>
<tr>
<td>#8004</td>
<td>No. of the current tool (System 4)</td>
<td>R3499</td>
</tr>
<tr>
<td>#8010</td>
<td>Machine coordinate X (System 1)</td>
<td></td>
</tr>
<tr>
<td>#8011</td>
<td>Machine coordinate Y (System 1)</td>
<td></td>
</tr>
<tr>
<td>#8012</td>
<td>Machine coordinate Z (System 1)</td>
<td></td>
</tr>
<tr>
<td>#8020</td>
<td>Machine coordinate X (System 2)</td>
<td></td>
</tr>
<tr>
<td>#8021</td>
<td>Machine coordinate Y (System 2)</td>
<td></td>
</tr>
<tr>
<td>#8022</td>
<td>Machine coordinate Z (System 2)</td>
<td></td>
</tr>
<tr>
<td>#8030</td>
<td>Machine coordinate X (System 3)</td>
<td></td>
</tr>
<tr>
<td>#8031</td>
<td>Machine coordinate Y (System 3)</td>
<td></td>
</tr>
<tr>
<td>#8032</td>
<td>Machine coordinate Z (System 3)</td>
<td></td>
</tr>
<tr>
<td>#8040</td>
<td>Machine coordinate X (System 4)</td>
<td></td>
</tr>
<tr>
<td>#8041</td>
<td>Machine coordinate Y (System 4)</td>
<td></td>
</tr>
<tr>
<td>#8042</td>
<td>Machine coordinate Z (System 4)</td>
<td></td>
</tr>
</tbody>
</table>

36. Positional information for the powered tailstock

Variables numbered 56154 and 56156 can be used to assign the particular positions as required for moving a powered tailstock.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#56154</td>
<td>Tailstock position 1</td>
</tr>
<tr>
<td>#56156</td>
<td>Tailstock position 2</td>
</tr>
</tbody>
</table>

Note 1: The setting range is from –9999.999 to 0 for metric data input, or from –999.9999 to 0 for inch data input.

Note 2: Use the M-codes concerned to move the tailstock to the above positions. See the M-code list in the operating manual of the machine for details.
37. Amount of workpiece transfer (#3024)

Variable numbered 3024 can be used to set the amount of a workpiece transfer operation which is performed within an EIA/ISO program. The block of setting the variable #3024 does not cause any axis motion on the machine, but informs the NC unit of the workpiece being transferred so that a tool path avoiding collision with the shifted workpiece can be drawn timely for the succeeding process by a MAZATROL program on the side of the turning spindle No. 2.

<Example of programming>

```
MAZATROL program

HEAD 1 selection
Machining on the turning spindle No. 1
SUB PRO unit
HEAD 2 selection
Machining on the turning spindle No. 2
END unit
```

```
EIA/ISO program

M540 (Transfer mode ON)
G90 G1 W-1200.4 (Transfer)
#3024=1200.4 (Setting transfer amount)
M99
```

Note: For a restart operation, based on a MAZATROL program of the above structure, from a block of machining on the side of the turning spindle No. 2, enter a block of "#3024 = 1200.4" in the MDI mode before starting the operation.

38. Serial number of the B- and C-axis (#50505 and #50506)

Variables numbered 50505 and 50506 can be used to read the serial number of setting of the B- and C-axis.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#50505</td>
<td>B-axis</td>
</tr>
<tr>
<td>#50506</td>
<td>C-axis</td>
</tr>
</tbody>
</table>

39. Data on the AUTO MEASURE display

Use the variables tabulated below to read and write the data on the AUTO MEASURE display.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
<th>Reading</th>
<th>Writing</th>
</tr>
</thead>
<tbody>
<tr>
<td>#58066</td>
<td>SNS-TOOL (Sensor tool)</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58067</td>
<td>OFS-TOOL (Tool to be offset)</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58069</td>
<td>TARGET DATA X</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58071</td>
<td>TARGET DATA Y</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58073</td>
<td>TARGET DATA Z</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58075</td>
<td>MEASURED DATA X</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58077</td>
<td>MEASURED DATA Y</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58079</td>
<td>MEASURED DATA Z</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58080</td>
<td>MEASURE POINT #1 (Axis name in ASCII code)</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>#58081</td>
<td>MEASURE POINT #2 (Axis name in ASCII code)</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>#58083</td>
<td>MEASURE POINT #1 (Position)</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>#58085</td>
<td>MEASURE POINT #2 (Position)</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>#58087</td>
<td>OFFSET AMOUNT X</td>
<td>O</td>
<td></td>
</tr>
<tr>
<td>#58089</td>
<td>OFFSET AMOUNT Y</td>
<td>O</td>
<td></td>
</tr>
</tbody>
</table>
### 40. Data in register R (#500000 to #516383, #550000 to #566383)

Use the following variables to read 2-byte data of register R:

\[ \#500000 + \text{No. of register R} \quad (#500000 \text{ to } #516383) \]

**Example 1:** Use the variable numbered 500100 for the R100 register.

Use the following variables to read 4-byte data of register R:

\[ \#550000 + \text{No. of register R} \quad (#550000 \text{ to } #566383) \]

**Example 2:** Use the variable numbered 560100 for the R10100/R10101 register.

### 41. Information about threading position storage (#3062 and #3063)

Variables numbered 3062 and 3063 can be used respectively to check whether or not the threading position data (MACHINE Z and SPINDLE) are currently stored for turning spindle No. 1 and No. 2.

<table>
<thead>
<tr>
<th>Variables</th>
<th>Description</th>
</tr>
</thead>
</table>
| #3062     | 0: No threading position storage for turning spindle No. 1  
            | 1: Threading position storage for turning spindle No. 1 |
| #3063     | 0: No threading position storage for turning spindle No. 2  
            | 1: Threading position storage for turning spindle No. 2 |

**Note:** Threading position storage is an optional function.
13-15-5 Arithmetic operation commands

Various operations can be carried out between variables using the following format.

\[ #i = \text{<expression>} \]

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 \(#k\).

<table>
<thead>
<tr>
<th>[1] Definition/replacement of variables</th>
<th>#i=#j</th>
<th>Definition/replacement</th>
</tr>
</thead>
<tbody>
<tr>
<td>[2] Additional-type operations</td>
<td>#i=#j+k</td>
<td>Addition</td>
</tr>
<tr>
<td></td>
<td>#i=#j–#k</td>
<td>Subtraction</td>
</tr>
<tr>
<td></td>
<td>#i=#jOR#k</td>
<td>Logical addition (For each of 32 bits)</td>
</tr>
<tr>
<td></td>
<td>#i=#jXOR#k</td>
<td>Exclusive OR (For each of 32 bits)</td>
</tr>
<tr>
<td>[3] Multiplicative-type operations</td>
<td>#i=#j∗#k</td>
<td>Multiplication</td>
</tr>
<tr>
<td></td>
<td>#i=#j/#k</td>
<td>Division</td>
</tr>
<tr>
<td></td>
<td>#i=#jMOD#k</td>
<td>Surplus</td>
</tr>
<tr>
<td></td>
<td>#i=#jAND#k</td>
<td>Logical product (For each of 32 bits)</td>
</tr>
<tr>
<td>[4] Functions</td>
<td>#i=SIN[#j]</td>
<td>Sine</td>
</tr>
<tr>
<td></td>
<td>#i=COS[#j]</td>
<td>Cosine</td>
</tr>
<tr>
<td></td>
<td>#i=TAN[#j]</td>
<td>Tangent (tanq is used as sinq/cosq.)</td>
</tr>
<tr>
<td></td>
<td>#i=ATAN[#j]</td>
<td>Arc-tangent (Either ATAN or ATN can be used.)</td>
</tr>
<tr>
<td></td>
<td>#i=ACOS[#j]</td>
<td>Arc-cosine</td>
</tr>
<tr>
<td></td>
<td>#i=SQRT[#j]</td>
<td>Square root (Either SQRT or SQR is available.)</td>
</tr>
<tr>
<td></td>
<td>#i=ABS[#j]</td>
<td>Absolute value</td>
</tr>
<tr>
<td></td>
<td>#i=BIN[#j]</td>
<td>BINARY conversion from BCD</td>
</tr>
<tr>
<td></td>
<td>#i=BCD[#j]</td>
<td>BCD conversion from BINARY</td>
</tr>
<tr>
<td></td>
<td>#i=ROUND[#j]</td>
<td>Rounding to the nearest whole number (Either ROUND or RND is available.)</td>
</tr>
<tr>
<td></td>
<td>#i=FIX[#j]</td>
<td>Cutting away any decimal digits</td>
</tr>
<tr>
<td></td>
<td>#i=FUP[#j]</td>
<td>Counting any decimal digits as 1s</td>
</tr>
<tr>
<td></td>
<td>#i=LN[#j]</td>
<td>Natural logarithm</td>
</tr>
<tr>
<td></td>
<td>#i=EXP[#j]</td>
<td>Exponent with the base of e (= 2.718 ...)</td>
</tr>
</tbody>
</table>

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:

```
#101=1000
#10001=#101
#102=#10001
```

Note 3: The \(<expression>\) after a function must be enclosed in brackets (\([\text{\ }]\)).
1. **Operation priority**

Higher priority is given to functions, multiplicative operations, and additive operations, in that order.

\[ \#101 = \#111 + \#112 \times \sin[\#113] \]

|--------------|---------------------|---------------|

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.

\[ \#101 = \sqrt{[[\#111 - \#112] \times \sin[\#113] + \#114] \times \#15]} \]

3. **Examples of operation instructions**

<table>
<thead>
<tr>
<th>[1] Main program and argument specification</th>
<th>#1</th>
<th>#2</th>
<th>#3</th>
<th>#4</th>
<th>#5</th>
</tr>
</thead>
<tbody>
<tr>
<td>G65 P100 A10 B20.</td>
<td>10.000</td>
<td>20.000</td>
<td>100.000</td>
<td>200.000</td>
<td>-10.000 Offset amount</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[2] Definition, replacement =</th>
<th>#1</th>
<th>#2</th>
<th>#3</th>
<th>#4</th>
<th>#5</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1 = 1000</td>
<td>1000.000</td>
<td>1000.000</td>
<td>100.000</td>
<td>200.000</td>
<td>-10.000 Offset amount</td>
</tr>
<tr>
<td>#2 = 1000.000</td>
<td>1000.000</td>
<td>1000.000</td>
<td>1000.000</td>
<td>1000.000</td>
<td>-10.000 Offset amount</td>
</tr>
<tr>
<td>#3 = #101</td>
<td>Data of common variables</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
</tr>
<tr>
<td>#4 = #102</td>
<td></td>
<td>200.000</td>
<td>200.000</td>
<td>200.000</td>
<td>200.000</td>
</tr>
<tr>
<td>#5 = #5081</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[3] Addition, subtraction + –</th>
<th>#11</th>
<th>#12</th>
<th>#13</th>
<th>#14</th>
<th>#15</th>
</tr>
</thead>
<tbody>
<tr>
<td>#11 = #1 + 1000</td>
<td>2000.000</td>
<td>950.000</td>
<td>1100.000</td>
<td>-13.000</td>
<td>190.000</td>
</tr>
<tr>
<td>#12 = #2 – 50</td>
<td>1000.000</td>
<td>900.000</td>
<td>900.000</td>
<td>900.000</td>
<td>900.000</td>
</tr>
<tr>
<td>#13 = #101 + #1</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
</tr>
<tr>
<td>#14 = #5081 + #101</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
</tr>
<tr>
<td>#15 = #5081 + #102</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
<td>100.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[4] Logical addition OR</th>
<th>#3 = 100</th>
<th>#4 = #3OR14</th>
<th>#11</th>
<th>#12</th>
<th>#13</th>
<th>#14</th>
<th>#15</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3 = 100</td>
<td>110010100</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4 = #3OR14</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[5] Exclusive OR XOR</th>
<th>#3 = 100</th>
<th>#4 = #3XOR14</th>
<th>#11</th>
<th>#12</th>
<th>#13</th>
<th>#14</th>
<th>#15</th>
</tr>
</thead>
<tbody>
<tr>
<td>#3 = 100</td>
<td>110010100</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#4 = #3XOR14</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td>110011110</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[6] Multiplication, Division * /</th>
<th>#21 = 100 * 100</th>
<th>#22 = 100.000 * 100.000</th>
<th>#23 = 100 * 100.000</th>
<th>#24 = 100.000 * 100.000</th>
<th>#25 = 100 * 1000</th>
<th>#26 = 1000 * 1000</th>
<th>#27 = 1000.000 * 1000.000</th>
<th>#28 = 1000.000 * 1000.000</th>
<th>#29 = 1000.000 / 100</th>
<th>#30 = 1000.000 / 100</th>
<th>#31 = 1000.000 / 100</th>
</tr>
</thead>
<tbody>
<tr>
<td>#21 = 10000.000</td>
<td>#22 = 10000.000</td>
<td>#23 = 10000.000</td>
<td>#24 = 10000.000</td>
<td>#25 = 1000</td>
<td>#26 = 1000</td>
<td>#27 = 1000</td>
<td>#28 = 1000</td>
<td>#29 = 1000</td>
<td>#30 = 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#21 = 10000.000</td>
<td>#22 = 10000.000</td>
<td>#23 = 10000.000</td>
<td>#24 = 10000.000</td>
<td>#25 = 1000</td>
<td>#26 = 1000</td>
<td>#27 = 1000</td>
<td>#28 = 1000</td>
<td>#29 = 1000</td>
<td>#30 = 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#21 = 10000.000</td>
<td>#22 = 10000.000</td>
<td>#23 = 10000.000</td>
<td>#24 = 10000.000</td>
<td>#25 = 1000</td>
<td>#26 = 1000</td>
<td>#27 = 1000</td>
<td>#28 = 1000</td>
<td>#29 = 1000</td>
<td>#30 = 100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>#21 = 10000.000</td>
<td>#22 = 10000.000</td>
<td>#23 = 10000.000</td>
<td>#24 = 10000.000</td>
<td>#25 = 1000</td>
<td>#26 = 1000</td>
<td>#27 = 1000</td>
<td>#28 = 1000</td>
<td>#29 = 1000</td>
<td>#30 = 100</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[7] Surplus MOD</th>
<th>#31 = #19MOD#20</th>
<th>#19 = 48</th>
<th>#20 = 9</th>
<th>#30 = 5 surplus 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>#31 = #19MOD#20</td>
<td>5 surplus 3</td>
<td>10000</td>
<td>10000</td>
<td>10000</td>
</tr>
<tr>
<td>#19 = 48</td>
<td>5 surplus 3</td>
<td>10000</td>
<td>10000</td>
<td>10000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>[8] Logical product AND</th>
<th>#9 = 100</th>
<th>#10 = #9AND#15</th>
<th>#9 = 01100100</th>
<th>#10 = 00001111</th>
</tr>
</thead>
<tbody>
<tr>
<td>#9 = 01100100</td>
<td>#10 = 00001111</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#9 = 01100100</td>
<td>#10 = 00001111</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>#9 = 01100100</td>
<td>#10 = 00001111</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### [9] Sine

#### SIN

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#501 = SIN[60]</td>
<td>0.866</td>
</tr>
<tr>
<td>#502 = SIN[60.]</td>
<td>0.866</td>
</tr>
<tr>
<td>#503 = 1000 * SIN[60]</td>
<td>866.025</td>
</tr>
<tr>
<td>#504 = 1000 * SIN[60.]</td>
<td>866.025</td>
</tr>
<tr>
<td>#505 = 1000. * SIN[60]</td>
<td>866.025</td>
</tr>
<tr>
<td>#506 = 1000. * SIN[60.]</td>
<td>866.025</td>
</tr>
</tbody>
</table>

**Note:** SIN[60] is equal to SIN[60.].

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#501</td>
<td>0.866</td>
</tr>
<tr>
<td>#502</td>
<td>0.866</td>
</tr>
<tr>
<td>#503</td>
<td>866.025</td>
</tr>
<tr>
<td>#504</td>
<td>866.025</td>
</tr>
<tr>
<td>#505</td>
<td>866.025</td>
</tr>
<tr>
<td>#506</td>
<td>866.025</td>
</tr>
</tbody>
</table>

### [10] Cosine

#### COS

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#541 = COS[45]</td>
<td>0.707</td>
</tr>
<tr>
<td>#542 = COS[45.]</td>
<td>0.707</td>
</tr>
<tr>
<td>#543 = 1000 * COS[45]</td>
<td>707.107</td>
</tr>
<tr>
<td>#544 = 1000 * COS[45.]</td>
<td>707.107</td>
</tr>
<tr>
<td>#545 = 1000. * COS[45]</td>
<td>707.107</td>
</tr>
<tr>
<td>#546 = 1000. * COS[45.]</td>
<td>707.107</td>
</tr>
</tbody>
</table>

**Note:** COS[45] is equal to COS[45.].

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#541</td>
<td>0.707</td>
</tr>
<tr>
<td>#542</td>
<td>0.707</td>
</tr>
<tr>
<td>#543</td>
<td>707.107</td>
</tr>
<tr>
<td>#544</td>
<td>707.107</td>
</tr>
<tr>
<td>#545</td>
<td>707.107</td>
</tr>
<tr>
<td>#546</td>
<td>707.107</td>
</tr>
</tbody>
</table>


#### TAN

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#551 = TAN[60]</td>
<td>1.732</td>
</tr>
<tr>
<td>#552 = TAN[60.]</td>
<td>1.732</td>
</tr>
<tr>
<td>#553 = 1000 * TAN[60]</td>
<td>1732.051</td>
</tr>
<tr>
<td>#554 = 1000 * TAN[60.]</td>
<td>1732.051</td>
</tr>
<tr>
<td>#555 = 1000. * TAN[60]</td>
<td>1732.051</td>
</tr>
<tr>
<td>#556 = 1000. * TAN[60.]</td>
<td>1732.051</td>
</tr>
</tbody>
</table>

**Note:** TAN[60] is equal to TAN[60.].

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#551</td>
<td>1.732</td>
</tr>
<tr>
<td>#552</td>
<td>1.732</td>
</tr>
<tr>
<td>#553</td>
<td>1732.051</td>
</tr>
<tr>
<td>#554</td>
<td>1732.051</td>
</tr>
<tr>
<td>#555</td>
<td>1732.051</td>
</tr>
<tr>
<td>#556</td>
<td>1732.051</td>
</tr>
</tbody>
</table>

### [12] Arc-tangent

#### ATAN

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#561 = ATAN[173205/1000000]</td>
<td>60.000</td>
</tr>
<tr>
<td>#562 = ATAN[173.205/100.]</td>
<td>60.000</td>
</tr>
<tr>
<td>#563 = ATAN[1.732]</td>
<td>59.999</td>
</tr>
</tbody>
</table>

### [13] Arc-cosine

#### ACOS

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#521 = ACOS[100000/141421]</td>
<td>45.000</td>
</tr>
<tr>
<td>#522 = ACOS[100./141.421]</td>
<td>45.000</td>
</tr>
<tr>
<td>#523 = ACOS[1000/1414.213]</td>
<td>45.000</td>
</tr>
<tr>
<td>#524 = ACOS[10./14.142]</td>
<td>44.999</td>
</tr>
<tr>
<td>#525 = ACOS[0.707]</td>
<td>45.009</td>
</tr>
</tbody>
</table>

### [14] Square root

#### SQRT

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#571 = SQRT[1000]</td>
<td>31.623</td>
</tr>
<tr>
<td>#572 = SQRT[1000.]</td>
<td>31.623</td>
</tr>
<tr>
<td>#573 = SQRT[10.*10.+20.*20.]</td>
<td>22.361</td>
</tr>
<tr>
<td>#574 = SQRT[#14*#14+#15*#15]</td>
<td>190.444</td>
</tr>
</tbody>
</table>

**Note:** For enhanced accuracy, perform operations within [ ] as far as possible.

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#571</td>
<td>31.623</td>
</tr>
<tr>
<td>#572</td>
<td>31.623</td>
</tr>
<tr>
<td>#573</td>
<td>22.361</td>
</tr>
<tr>
<td>#574</td>
<td>190.444</td>
</tr>
</tbody>
</table>

### [15] Absolute value

#### ABS

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#576 = –1000</td>
<td>-1000.000</td>
</tr>
<tr>
<td>#577 = ABS[#576]</td>
<td>1000.000</td>
</tr>
<tr>
<td>#3 = 70.</td>
<td>70.</td>
</tr>
<tr>
<td>#4 = –50.</td>
<td>–50.</td>
</tr>
<tr>
<td>#580 = ABS[#4–#3]</td>
<td>120.000</td>
</tr>
</tbody>
</table>

### [16] BIN, BCD

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1 = 100</td>
<td>64</td>
</tr>
<tr>
<td>#11 = BIN[#1]</td>
<td>11</td>
</tr>
<tr>
<td>#12 = BCD[#1]</td>
<td>256</td>
</tr>
</tbody>
</table>

### [17] Rounding into the nearest whole number

#### ROUND

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#21 = ROUND[14/3]</td>
<td>5</td>
</tr>
<tr>
<td>#22 = ROUND[14./3]</td>
<td>5</td>
</tr>
<tr>
<td>#23 = ROUND[14/3.]</td>
<td>5</td>
</tr>
<tr>
<td>#24 = ROUND[14./3.]</td>
<td>5</td>
</tr>
<tr>
<td>#25 = ROUND[–14/3]</td>
<td>–5</td>
</tr>
<tr>
<td>#26 = ROUND[–14./3]</td>
<td>–5</td>
</tr>
<tr>
<td>#27 = ROUND[–14/3.]</td>
<td>–5</td>
</tr>
<tr>
<td>#28 = ROUND[–14./3.]</td>
<td>–5</td>
</tr>
</tbody>
</table>

### [18] Cutting away any decimal digits

#### FIX

<table>
<thead>
<tr>
<th>Expression</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>#21 = FIX[14/3]</td>
<td>4.000</td>
</tr>
<tr>
<td>#22 = FIX[14./3]</td>
<td>4.000</td>
</tr>
<tr>
<td>#23 = FIX[14/3.]</td>
<td>4.000</td>
</tr>
<tr>
<td>#24 = FIX[14./3.]</td>
<td>4.000</td>
</tr>
<tr>
<td>#25 = FIX[–14/3]</td>
<td>–4.000</td>
</tr>
<tr>
<td>#26 = FIX[–14./3]</td>
<td>–4.000</td>
</tr>
<tr>
<td>#27 = FIX[–14/3.]</td>
<td>–4.000</td>
</tr>
<tr>
<td>#28 = FIX[–14./3.]</td>
<td>–4.000</td>
</tr>
</tbody>
</table>
### 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.

<table>
<thead>
<tr>
<th>Operation format</th>
<th>Mean error</th>
<th>Max. error</th>
<th>Kind of error</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a = b + c)</td>
<td>(2.33 \times 10^{-10})</td>
<td>(5.32 \times 10^{-10})</td>
<td>Min. (</td>
</tr>
<tr>
<td>(a = b - c)</td>
<td>(1.55 \times 10^{-10})</td>
<td>(4.66 \times 10^{-10})</td>
<td>Relative error (\frac{c}{a})</td>
</tr>
<tr>
<td>(a = b \cdot c)</td>
<td>(4.66 \times 10^{-10})</td>
<td>(1.86 \times 10^{-9})</td>
<td></td>
</tr>
<tr>
<td>(a = b/c)</td>
<td>(1.24 \times 10^{-9})</td>
<td>(3.73 \times 10^{-9})</td>
<td></td>
</tr>
<tr>
<td>(a = \sin b)</td>
<td>(5.0 \times 10^{-9})</td>
<td>(1.0 \times 10^{-8})</td>
<td>Absolute error (</td>
</tr>
<tr>
<td>(a = \tan^{-1} b/c)</td>
<td>(1.8 \times 10^{-6})</td>
<td>(3.6 \times 10^{-6})</td>
<td></td>
</tr>
</tbody>
</table>

**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 substraction, so be careful to errors. For example, to judge whether #10 is equal to #20 of the above example, the conditional expression

\[
\text{IF } [#10 \text{EQ} #20]
\]

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.

\[
\text{IF } [\text{ABS} (#10 - #20) \text{LT} 200000]
\]

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.

13-15-6 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:

\[
\begin{array}{|c|}
\hline
\#i \text{ EQ } \#j \quad = \quad (\#i \text{ is equal to } \#j) \\
\#i \text{ NE } \#j \quad \neq \quad (\#i \text{ is not equal to } \#j) \\
\#i \text{ GT } \#j \quad > \quad (\#i \text{ is larger than } \#j) \\
\#i \text{ LT } \#j \quad < \quad (\#i \text{ is smaller than } \#j) \\
\#i \text{ GE } \#j \quad \geq \quad (\#i \text{ is equal to } \#j, \text{ or larger than } \#j) \\
\#i \text{ LE } \#j \quad \leq \quad (\#i \text{ is equal to } \#j, \text{ or smaller than } \#j) \\
\hline
\end{array}
\]

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.
PROGRAM SUPPORT FUNCTIONS

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.
13 PROGRAM SUPPORT FUNCTIONS

[1] Same identifying No. can be used repeatedly.

Usable

WHILE ~ DO1

END1

Usable

WHILE ~ DO1

END1

[2] The identifying No. of WHILE ~ DOm is arbitrary.

WHILE ~ DO1

{ END1

WHILE ~ DO3

{ END3

WHILE ~ DO2

{ END2

WHILE ~ DO1

{ END1

[3] Up to 27 levels of WHILE ~ DOm can be used.

m can be 1 to 127, independent of the depth of nesting.

WHILE ~ DO1

WHILE ~ DO2

WHILE ~ DO27

WHILE ~ DO28

END28

END27

END2

END1

Note: For nesting, m once used cannot be used again.

[4] The total number of levels of WHILE ~ DOm must not exceed 27.

WHILE ~ DO1

WHILE ~ DO2

WHILE ~ DO27

WHILE ~ DO28

END28

END27

END2

END1

[5] WHILE ~ DOm must precede ENDm.

END1

WHILE ~ DO1

Unusable

[6] WHILE ~ DOm must correspond to ENDm one-to-one in the same program.

WHILE ~ DO1

WHILE ~ DO1

Unusable

[7] WHILE ~ DOm must not overlap.

WHILE ~ DO1

WHILE ~ DO2

END1

END2

Unusable

[8] Outward branching from the range of WHILE ~ DOm is possible.

WHILE ~ DO1

IF ~ GOTOOn

END1

Nn-......
### [9] Branching into WHILE - DOm is not allowed.

Unusable

<table>
<thead>
<tr>
<th>IF ~ GOTOon</th>
</tr>
</thead>
<tbody>
<tr>
<td>WHILE ~ DO1</td>
</tr>
<tr>
<td>Nn........</td>
</tr>
<tr>
<td>END1</td>
</tr>
</tbody>
</table>

### [10] Subprogram can be called using M98, G65, G66, etc. from the midway of WHILE - DOm.

Main program

- WHILE ~ DO1
- END1
- WHILE ~ DO1
- Nn........
- END1

Subprogram

- WHILE ~ DO1
- G65 P100
- END1
- M99

### [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.

Main program

- WHILE ~ DO1
- G65 P100
- END1

Subprogram

- WHILE ~ DO1
- END1
- M99

### [12] If WHILE and END are not included in pairs in subprogram (including macro subprogram), a program error will result at M99.

Main program

- WHILE ~ DO1
- M65 P100
- END1
- M99

Subprogram

- Alarm 868 DO-END MIS-MATCH
13-15-7 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

```
<table>
<thead>
<tr>
<th>POPEN</th>
<th>POPEN</th>
<th>BPRNT</th>
<th>DPRNT</th>
<th>PCLOS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Open command</td>
<td>Data output commands</td>
<td>Close command</td>
<td></td>
</tr>
</tbody>
</table>
```

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[#1#v1[c1]/2#v2[c2]......]
```

- **Effective digits after decimal point**
- **Variable number**
- **Character string**

**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[#1#y1[d1 c1]/2#v2[c2]......]
```

- **Effective digits below decimal point**
- **Effective digits above decimal point**
- **Variable number**
- **Character string**

\[c + d \leq 8\]

**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.
13-15-8 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: DPR8 × 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\n
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.

<table>
<thead>
<tr>
<th>Program</th>
<th>Output example</th>
</tr>
</thead>
<tbody>
<tr>
<td>G28XYZ</td>
<td>print.txt</td>
</tr>
<tr>
<td>POPEN</td>
<td></td>
</tr>
<tr>
<td>DPRNT [OOOOOOOOOOOOO]</td>
<td>OOOOOOOOOOOOO</td>
</tr>
<tr>
<td>DPRNT [XXXXXXXXXXXXX]</td>
<td>XXXXXXXXXXXXX</td>
</tr>
<tr>
<td>DPRNT [IIIIIIIIIIII]</td>
<td>IIIIIIIIIIIII</td>
</tr>
<tr>
<td>PCLOS</td>
<td></td>
</tr>
<tr>
<td>G0X100.Y100.Z100.</td>
<td></td>
</tr>
<tr>
<td>M30</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>DPR14: 4</td>
<td></td>
</tr>
<tr>
<td>DPR15: No setting</td>
<td></td>
</tr>
</tbody>
</table>

4. Related alarms

The alarm given for text file output is described below.

<table>
<thead>
<tr>
<th>No.</th>
<th>Message</th>
<th>Argument 1</th>
<th>Argument 2</th>
<th>Argument 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>887</td>
<td>TAPE I/O ERROR</td>
<td>–100 File open error</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>–111 File write error</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>–112 File size too great</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
13-15-9 Precautions

Use of user macro commands allows a machining program to be created by combining arithmetic operation, judgment, branchining, 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
```

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.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Off</th>
<th>On</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program analysis</td>
<td>N1 N2 N3 N4 N5 N6 N7</td>
<td>N1 N2 N3 N4 N5 N6 N7</td>
</tr>
<tr>
<td>Macroprogram statement treatment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Next command</td>
<td>N2 N3</td>
<td>N2 N3</td>
</tr>
<tr>
<td>NC statement treatment</td>
<td></td>
<td></td>
</tr>
<tr>
<td>In execution</td>
<td>N1 N2 N3 N4 N5 N6</td>
<td>N1 N2 N3 N4 N5 N6</td>
</tr>
</tbody>
</table>

Machining program data is displayed as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Off</th>
<th>On</th>
</tr>
</thead>
<tbody>
<tr>
<td>(In execution)</td>
<td>N3 G00X–100,Y–100.</td>
<td>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.</td>
</tr>
<tr>
<td>(Next command)</td>
<td>N6 G01X#101Y#102F800</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(In execution)</td>
<td>N3 G00X–100,Y–100.</td>
<td>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.</td>
</tr>
<tr>
<td>(Next command)</td>
<td>N4 #101=100.*COS[210.]</td>
<td></td>
</tr>
</tbody>
</table>

13-163
13-15-10 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

G65 Pp1 Aa1 Bb1 Cc1 Ff1

A: Initial value 0°
B: Final value 360°
C: R of R=\sin(\theta)
F: Feed rate

\[(\sin \theta)
\]

X

\[\begin{array}{c}
0 \\
90. \\
180. \\
270. \\
360.
\end{array}\]

Y

\[\begin{array}{c}
-100. \\
0 \\
100. \\
200. \\
300. \\
400.
\end{array}\]

Note

O9910 (Subprogram)

\[
\text{WHILE}[\#1<\#2]DO1
\]
\[
\#10=\#3\times\sin[\#1]
\]
\[
G90G1X\#1Y\#10F\#9
\]
\[
\#1=\#1+10.
\]
END1
M99

Note:

G90G01X\#1Y[\#3\times\sin[\#1]]F\#9

makes one block command available.
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.

Main program

```
G812-100.R50.F300
G65P9920Aa1Bb1CRr1XX1YY1
```

O9920 (Subprogram)

```
#101=0
#102=#4003
#103=#5001
#104=#5002
#111=#1

WHILE[#101LT#2]DO1
    #120=#24+#18*COS[#111]
    #121=#25+#18*SIN[#111]
    #122=#120 #123=#121
    IF[#102EQ90]GOTO100

    #122=#120-#103
    #123=#121-#104
    #103=#120
    #104=#121

N100X#122Y#123
#101=#101+1
#111=#1+360.*#101/#2
END1
M99
```

Note: The processing time can be reduced by one-block reduced programming.

Note:
- #101 = Hole count
- #102 = G90 or G91
- #103 = X-axis present position
- #104 = Y-axis present position
- #111 = Start angle

Note:
- #120 = Hole position X coordinate
- #121 = Hole position Y coordinate
- #122 = X-axis absolute value
- #123 = Y-axis absolute value

Drilling command

```
#101+1 -#101
360°*#101/#111 + #1 -#111
```

Note:
- Mode judge G90, G91
- #122 = X-axis incremental value
- #123 = Y-axis incremental value
- X-axis present position renewal
- Y-axis present position renewal

Note:
- #111 = Hole position angle

Note:
- #120 = Hole position X coordinate
- #121 = Hole position Y coordinate
- #122 = X-axis absolute value
- #123 = Y-axis absolute value

Drilling command

```
#101+1 -> #101
360°*#101/#111 + #1 -> #111
```

Note:
- #111 = Hole position angle

Note:
- The processing time can be reduced by one-block reduced programming.
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 Zz Rr Ff
G65 Pp Xx Yy Ii Jj Aa Bb

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.
O9930 (Subprogram)

N2 #113=0
#103=#9

WHILE[#105GT0]DO1
#105=#105-1
X#101 Y#102
IF[#112EQ1]GOTO10
IF[#111NE1]GOTO10
#103=#103-#103
#112=1
N10 #113=#103
END1
N100 #106=#106-1
#112=0
#110=#110+1
IF[#106LT0]GOTO200
#105=#1
#102=#102+#104
#111=#111
#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 → #106
0 → #110
0 → #111
0 → #112

#105 > 0
X direction hole drilling completion check
Yes
No

#101 + #113 → #101
#105 – 1 → #105
X#101 Y#102
Drilling command

#112 = 1
X-axis drilling direction reverse switch check
Yes
No

#111 = 1
For even times (#111 = 0), X distance is same as command. For odd times (#111 = 1), X-axis drilling direction reverse switch on
Yes
No

0 – #103 → #103
1 → #112

#103 → #113
X-axis drilling distance reset

#106 – 1 → #106
0 → #112
#110 + 1 → #110

#106 < 0
Y direction hole drilling completion check
Yes
No

#1 → #105
#102 + #104 → #102
#110 → #111
#111AND 1 → #111

X-axis hole quantity reset
Y coordinates renewal

Even times = 0
Odd times = 1

END

Note:
The processing time can be reduced by one-block reduced programming.
13-16 Geometric Commands

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 Aa1 Ff1 Specify the angle and speed for the first block.
N2 Xxe Zze Aa2 (a’2) Ff2 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 a program error will result.
- Any speed can be specified for each block.
- A program error will result if the angle of the crossing point of the two lines is 1 degree or less.
- A program error 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.
- A program error will result if the first block or the second block is not linear.
4. Correlations to other functions

Geometric command can be set following a linear angle command.

\[ \text{N1} \times x_1, a_1 \]
\[ \text{N2} a_2 \]
\[ \text{N3} x_3, z_3, a_3 \]
14 COORDINATE SYSTEM SETTING FUNCTIONS

14-1 Coordinate System Setting Function: G92

1. Function and purpose

Use a G92 command to modify the current workpiece coordinate system, without any actual motion, in order that the current tool position be indicated by the specified coordinates. The coordinate system can be placed anywhere, but normally its X- and Y-axis zero points are to be set on the workpiece center, and the Z-axis zero point on the workpiece end face.

2. Programming format

G92 Xx Yy Zz αα  (α: Additional axis)

3. Detailed description

For moving the tool by absolute command, the coordinate system needs to be determined in advance. The coordinate system can be set by a command as below.

G92 X_ Z_ C_;

This command allows to set a coordinate system where a point on a tool, for example the tool tip position, can be represented with coordinates (X, Z). This coordinate system is called the workpiece coordinate system.

Once a coordinate system is set, coordinates by absolute command will represent the positions on this coordinate system.

The command has not to be used for all axes at the same time.

For changing coordinate systems in the midway on a program, command only the axis for which change is required.

Example 1:

G92 X350. Z300.;

For setting a coordinate system with point A as zero point, command as follows:

G92 X350. Z350.;
Example 2:

For setting at a reference point, command as follows:

G92 X700. Z300.;

This coordinate system uses the center of turret rotation as a reference point. Any point can be used as a reference. Use tool offsetting functions for $\delta X$ and $\delta Z$. For details, refer to Section 12-1.

Remarks

- The base machine coordinate system is shifted by G92 command to set a virtual machine coordinate system.
- Spindle clamp revolution speed is set by G92 with S or Q command. (Refer to the section for spindle clamp speed setting.)
- If the MAZATROL coordinate system is selected, validity or invalidity of G92 coordinate system can be selected by parameter setting.
- If a coordinate system is set by G92 during compensation, the coordinate system will be set on which the position specified by G92 is the position without compensation.
- Nose radius compensation is temporarily cancelled by G92.
- S data in a block with G92 will be regarded as spindle clamp revolution speed setting, but not as ordinary S data.
### 4. G92 command and indications on POSITION and MACHINE counters

**Example:**

<table>
<thead>
<tr>
<th>Program</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>POSITION (0, 0)</td>
<td>MACHINE (0, 0)</td>
</tr>
<tr>
<td>N01 G28 X0.20.;</td>
<td>(0, 0)</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>N02 G92 X0.20.;</td>
<td>(0, 0)</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>N03 T001T000M06;</td>
<td>(0, 0)</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>N04 G43H1;</td>
<td>(0, 0)</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>N05 G00 X50.250.;</td>
<td>(50, 50)</td>
<td>(40, 40)</td>
</tr>
<tr>
<td>N06 G92 X0.20.;</td>
<td>(0, 0)</td>
<td>(40, 40)</td>
</tr>
<tr>
<td>N07 G00 X50.250.;</td>
<td>(50, 50)</td>
<td>(90, 90)</td>
</tr>
<tr>
<td>N08 G49;</td>
<td>(50, 50)</td>
<td>(100, 100)</td>
</tr>
<tr>
<td>N09 G00 X0.20.;</td>
<td>(0, 0)</td>
<td>(50, 50)</td>
</tr>
<tr>
<td>N10 G28 X0.20.;</td>
<td>(–50, –50)</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>N11 M02;</td>
<td>(–50, –50)</td>
<td>(0, 0)</td>
</tr>
</tbody>
</table>
14-2 Selection of Workpiece Coordinate System: G54 to G59

1. Function and purpose

G54; Workpiece coordinate system 1
G55; Workpiece coordinate system 2
G56; Workpiece coordinate system 3
G57; Workpiece coordinate system 4
G58; Workpiece coordinate system 5
G59; Workpiece coordinate system 6

Commanding the above permits selection/change of one of six coordinate systems specified beforehand that are agreed with the machine.

By this command, subsequent axis commands are used as positioning at selected workpiece coordinate system until the reset key is pressed.

For the six workpiece coordinate systems, set the distance of each axis from the machine zero point to the zero point of each coordinate system on the WORK OFFSET display.

2. Remarks

1. When G54 to G59 and move command are given in the same block, the coordinate system is changed to the specified one to move to the specified position on a new coordinate system.

2. When G54 to G59 are changed independently, the counter display of current position changes to a value on the specified coordinate system. (Machine does not move.)

3. Workpiece coordinate systems 1 to 6 are correctly established after reference point return after the power is turned on.

4. When the power is turned on or when the reset key is pressed, G54 is selected.

5. The distance when coordinate system is moved by G92 is added subsequently to all workpiece zero point offset values. For example, when coordinate system is moved by “G92 X_ Z_” command in the selection of G54, G55 to G59 also move by the same distance. Therefore, take care when having changed to G55.

6. The coordinate system cannot be established exactly for the C-axis by a command of G54 to G59 if it is given with the C-axis not being connected. Do not fail, therefore, to select the milling mode (for C-axis connection) before entering G54 to G59 as required for the C-axis.

7. The values for setting coordinate origin must be specified separately as required for the systems of spindle 1 and spindle 2 (even when the same value is to be set).
14-3 Additional Workpiece Coordinate System Setting and Selection: G54.1 (Option)

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:** G54.1P300  Selection of P300 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**
   
   G90 Xx Yy Zz
   
   **Example:** G54.1P1  Selection of P1 system
   
   X0Y0Z0  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:** G10L20P30X–255.Z–50.
   
   Replacement of the X and Z origin data of the P30-system.
   
   
   Incremental processing on the X and Z origin data of the P30-system.

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  or 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:
  
<table>
<thead>
<tr>
<th>G-code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G04 Pp</td>
<td>Dwell</td>
</tr>
<tr>
<td>G30 Pp</td>
<td>Reference-point return</td>
</tr>
<tr>
<td>G72 to G89</td>
<td>Fixed cycle</td>
</tr>
<tr>
<td>G65 Pp, M98 Pp</td>
<td>Subprogram call</td>
</tr>
</tbody>
</table>

- 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 can be read and written by using the system variables concerned. See Article 6 in Subsection 13-16-4 for more information.
4. Sample programs

1. Consecutive setting of origin data for all the 48 sets of additional workpiece coordinate system

Setting in format “G10L20PpXxYyZz”

O100

\[
\begin{align*}
\#100 &= 1 \\
\#101 &= 10. \\
\text{WHILE} [\#100 LT 49] DO1 \\
G10L20F\#100X\#101Y\#101 \\
\#100 &= \#100 + 1 \\
\#101 &= \#101 + 10. \\
\text{END1}
\end{align*}
\]

M30 %

Setting in assignment of variables

O200

\[
\begin{align*}
\#100 &= 7001 \\
\#101 &= 10. \\
\#102 &= 1 \\
\text{WHILE} [\#102 LT 49] DO1 \\
\#103 &= 0 \\
\text{WHILE} [\#103 LT 2] DO2 \\
\#100 &= \#101 \\
\#100 &= \#100 + 1 \\
\#103 &= \#103 + 1 \\
\text{END2} \\
\#100 &= \#100 + 18 \\
\#101 &= \#101 + 10. \\
\#102 &= \#102 + 1 \\
\text{END1}
\end{align*}
\]

M30 %

P-No. initial.  Origin setting  P-No. count-up  Sys.-var.-No. 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 program)
G91 G28XYZ
#100=1
G90
WHILE[#100LT49]DO1
G54.1P#100
M98P1001
#100=#100+1
END1
G91 G28Z
G28XY
M02
%

Reference-pt. return
P-No. initial.
Repeat while P-No. < 49
Wpc. coordn. sys. Setting
Subprogram call
P-No. count-up
Reset to reference-pt.

O1001 (Subprogram)
G98G81X0.Z0.R5.F40 Drilling
G80
M99
%
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:

![Diagram of workpiece fixed on rotary table]

O2000 (Main program)

```
G91 G28 X Y Z C
G90
G00 C0
G65 P2001 A1
M98 P2002
G00 C90.
G65 P2001 A7
M98 P2002
G00 C180.
G65 P2001 A13
M98 P2002
G00 C270.
G65 P2001 A19
M98 P2002
G91 G28 X Y Z C
M02
```

O2001 (Origin-data transference)

```
O2002 (Drilling subprg.)

```

O2002 (Drilling subprg.)

```
G54 M98 H100
G55 M98 H100
G56 M98 H100
G57 M98 H100
G58 M98 H100
G59 M98 H100
G28 Z0
M99
```

Fixed cycle for drilling

```
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:

![Diagram of workpiece and rotary table](image_url)

**Program Code: O3000**

- **G91G28XYZC**: Reference-point return
- **G90**: Selection of absolut data input
- **G00 C0**: Indexing of table for 1st surface
- **G65 P3001A1**: Indexing of table for 2nd surface
- **G00 C90.**: Indexing of table for 3rd surface
- **G65 P3001A7**: Indexing of table for 4th surface
- **G00 C180.**: Reset to reference-point
- **G65 P3001A1**: Initialization of P-No.
- **G65 P3001A7**: Initialization of counter
- **G65 P3001A13**: Setting of additional workpiece coordinate system
- **G00 C270.**: Call for drilling subroutine
- **G65 P3001A19**: P-No. count-up
- **G91G28XYC**: Checking counter count-up
- **M30**: Fixed cycle for drilling

**Initialize P-No.**

```plaintext
O3000
G91G28XYZC
G90
G00 C0
G65 P3001A1
G00 C90.
G65 P3001A7
G00 C180.
G65 P3001A13
G00 C270.
G65 P3001A19
G91G28XYC
M30
%
```

**Reference-point return**

```plaintext
O3001
#100=#1
#101=0
WHILE[#101LT6]DO1
G54.1P#100
M98H100
#100=#100+1
#101=#101+1
END1
G28Z0
M99
%
```

**Selection of absolut data input**

- **Fix the cycle for drilling**

```plaintext
G80
G28Z
M99
%
```
14-4 Workpiece Coordinate System Shift

Difference may be caused between the workpiece coordinate system considered in programming and the coordinate system specified actually by G92 command or G54 to G59 command. The coordinate system being specified can be shifted then. The amount to be shifted is specified at the SHIFT item on the WORK OFFSET display.

All the six workpiece coordinate systems are shifted by the coordinate shift amount.
- When the shift amount is specified, the workpiece coordinate system is shifted immediately. (The shift amount is reflected in the current position counter.)
- When “G92 X_ Y_ Z_” is specified after the shift amount has been specified, the shift amount is ignored.

14-5 Change of Workpiece Coordinate System by Program Command

G10 L2 P_ X_ Z_ C_ ;

P = 0: Coordinate shift amount is specified.
P = 1 to 6: Specified to workpiece coordinate system 1 to 6
X, Z, C: Workpiece zero point offset value of each axis

The above command permits rewriting a workpiece offset value to change the position of workpiece coordinate system. To move the workpiece coordinate system for each program, it is commanded at the head of a program.

Note: The timing at which the rewritten value becomes effective is after G54 to G59 command is subsequently executed.
14-6 Selection of Machine Coordinate System: G53

1. Function and purpose

   G53 X_ Y_ Z_ ;
   The above command permits moving the tool to the commanded position in the machine coordinate system at rapid feed. G53 is valid only for the commanded block.
   To move a tool to the position specifically set for the machine including tool change position, command G53 using the machine coordinate system.
   A base point on the machine is referred to as the machine zero point. Machine zero point depends on machine specifications.
   A coordinate system using machine zero point as the zero point of coordinate system is referred to as machine coordinate system.
   The tool cannot always move to the machine zero point. In some cases, machine zero point is set at a position to which the tool cannot move.
   Machine coordinate system is established when the reference point return is executed after the power is turned on.
   Once the machine coordinate system is established, it is not changed by reset, workpiece coordinate system setting (G92), local coordinate system setting (G52) and other operation unless the power is turned off.
   Stored stroke limit (G22, G23), which specifies the stroke of the machine, must be set using the coordinate value of the machine coordinate system.

2. Remarks

   1. When G53 is commanded, tool offset and tool nose radius compensation must be cancelled. (Because they are not incorporated when G53 is commanded.)
   2. Since the machine coordinate system must be set before G53 is commanded, at least one manual reference point return or automatic reference point return by G28 should be executed after the power is turned on.
   3. G53 with incremental command can be commanded, but it is meaningless.
   4. Virtual axes such as Y-axis cannot be commanded. (The execution gives an alarm.)
14-7 Selection of Local Coordinate System: G52

G52 X_ Y_ Z_ ;

The above command allows a coordinate system (local coordinate system) to be established on the basis of the current workpiece coordinate system (G54 to G59).

With the local coordinate system established, axis motion commands are all executed with respect to that system of coordinates.

To change local coordinate system, specify anew with G52 the zero point position of a new local coordinate system using workpiece coordinates.

To cancel the local coordinate system, align the zero point of coordinate system with that of workpiece coordinate system. That is, “G52 X0Y0Z0 ;” must be commanded.

Note: G52 can be used in place of G92 command. However, the distance through which coordinate system is shifted by G52 is not added to other workpiece zero point offset values.
14-8 Ram Spindle Offset ON/OFF: G52.1/G52.2

1. Overview

The G52.1 function allows a new (local) coordinate system to be established which corresponds to a translation of the current workpiece coordinate system (of G54 to G59, or G54.1 P1 to P300) according to the setting of parameter S27 (Offset amounts from the main spindle to the ram spindle). With the local coordinate system established, axis motion commands are all executed with respect to that system of coordinates. The ram spindle can be used for machining in two programming types which differ from each other only in that, as for the direction of workpiece axis, the preparatory positioning and the movement in the offset mode are to be programmed with axis names Z and W, respectively, or vice versa.

Use the G52.2 function to cancel the local coordinate system established by G52.1.

Whether or not the G52.1 coordinate system is cancelled by resetting, depends upon the setting of the following parameter:

F34 bit 0: Cancellation of the G52.1 coordinate system by resetting
0: Invalid (G52.1 is not cancelled),
1: Valid.

2. Programming format

G52.1XYZ (or XYW)......The shifting amounts concerned on the three designated axes are included in the axis movement to be executed next.

- G52.1XYZ: W for preparatory positioning and Z for movement in the mode
- G52.1XYW: Z for preparatory positioning and W for movement in the mode

G52.2.............................The shifting amounts concerned on the three designated axes are cancelled in the axis movement to be executed next.

Note 1: Giving a G52.1 command without the required axis addresses will result in an alarm (807 ILLEGAL FORMAT).

Note 2: Do not give a G52.1 command in the mode of three-dimensional coordinate conversion; otherwise an alarm will be caused (943 CONVERTING IN 3-D COORDINATES).

Note 3: The ram spindle cannot be applied to synchronous tapping. Giving a synchronous tapping command in the G52.1 mode will result in an alarm.
3. Sample programs with explanatory illustrations

A. W for preparatory positioning and Z for movement in the mode

N101 G91G28XYZ  ..................Return to zero point
N102 G28BCW  ..................Return to zero point (1)
N103 M700  ..................Selection of ram spindle mode
N104 M600T801  ..................Tool change on the ram spindle
N105 G54  ..................Selection of workpiece coordinate system
N106 G53W-300.  ..................Positioning on the W-axis
N107 M303  ..................Normal rotation of the ram spindle
N108 G52.1XYZ  ..................Offsetting the coordinate system for the ram spindle
N109 G90X0.Y0.Z100.  ..................Movement to the initial point (2)
N110 G43P2H1Z-150.  ..................Tool length offset ON (3)
N111 G0X-150.  ..................Approach to the machining position (4)
N112 G19  ..................Plane selection (for the YZ-plane)
N113 G82R-5.Z-10.F100.  ..................Fixed cycle (Drilling)
N114 G80  ..................Fixed cycle OFF
N115 G17  ..................Plane selection (for the XY-plane)
N116 G49  ..................Tool length offset OFF
N117 G52.2  ..................Offsetting for the ram spindle OFF
N118 M702  ..................Cancellation of ram spindle mode

1. N102: Completion of zero point return for all axes

![Diagram of main and ram spindles with zero point return](image)
2. **N109: Initial positioning after offsetting the coordinate system for the ram spindle**

![Diagram](D740PB0084)

3. **N110: Tool length offset (G43P2)**

![Diagram](D740PB0085)
4. N111: Approach to the machining position

B. Z for preparatory positioning and W for movement in the mode

- N101 G91 G28 XYZ: Return to zero point
- N102 G28 BCW: Return to zero point (1)
- N103 M700: Selection of ram spindle mode
- N104 M600 T801: Tool change on the ram spindle
- N105 G54: Selection of workpiece coordinate system
- N106 G53 Z-300.: Positioning on the Z-axis
- N107 M303: Normal rotation of the ram spindle
- N108 G52.1 XY W: Offsetting the coordinate system for the ram spindle
- N109 G110 Z: Axis name change (Z → W)
- N110 G90 X0 Y0 Z100.: Movement to the initial point (2)
- N111 G43 P2 H1 Z-150.: Tool length offset ON (3)
- N112 G82 R-5 Z-10 F100.: Approach to the machining position (4)
- N113 G19: Plane selection (for the YZ-plane)
- N114 G82 R-5 Z-10 F100.: Fixed cycle (Drilling)
- N115 G80: Fixed cycle OFF
- N116 G17: Plane selection (for the XY-plane)
- N117 G49: Tool length offset OFF
- N118 G111: Axis name change OFF
- N119 G52.2: Offsetting for the ram spindle OFF
- N120 M702: Cancellation of ram spindle mode
1. N102: Completion of zero point return for all axes

2. N110: Initial positioning after offsetting the coordinate system for the ram spindle
3. N111: Tool length offset (G43P2)

4. N112: Approach to the machining position

4. Supplementary notes

1. Commands for movement on the W- and Z-axis cannot be given in the program sections under G52.1XYZ and G52.1XYW, respectively. The mode in question must be cancelled temporarily by G52.2 to give such a command.

2. The setting of parameter BA62 is not used at all for the tool length offset in the mode of G52.1.
3. In the program section after the axis name change command (G110Z) all Z-axis motion commands are used for the corresponding motions on the W-axis until the cancellation code (G111) is designated.

4. Give a command for axis name change (G110Z) beforehand to use the tool length offsetting function (G43) in the mode of G52.1XYW. Otherwise the tool length offset will be started up with a movement on the Z-axis.

5. Give a command of G43P1 to use the tool length offsetting function for turning tools.

6. Set to one and the same value the Z- and W-axis components of the origin of the workpiece coordinate system to be used for ram spindle offset.

7. The tool on the ram spindle cannot be selected for an ordinary machining unit of the MAZATROL program. A subprogram is to be prepared as required, and to be called up with a subprogram unit.

8. The axis name change command (G110Z) must be followed by a command (be it given anew) for activating the function of tool length offset.

   With the MAZATROL tool data being valid for EIA/ISO programs
   Give a motion command for the Z-axis (W-axis) after G110Z.
   
   Data for machining:
   :
   G110Z
   G0Z150.

   With the MAZATROL tool data being invalid for EIA/ISO programs
   Give a command for tool length offset (with G43 or G44) after G110Z.
   
   Data for machining:
   :
   G110Z
   G43P2H1G0Z150.

9. The tool length offsetting mode (G43) selected in a section under G110Z (axis name change) must be cancelled (by G49) before the cancellation (by G111) of axis name change.

   Sample program for the use of the ram spindle
   
   Data for machining:
   :
   G110Z
   G43P2H1Z150.
   G49
   G111

   Tool length offset OFF
   Axis name change OFF.
10. In the mode of offsetting for the ram spindle (G52.1) do not attempt to change the controlled axes between Z and W. The command of axis name change must be preceded by the cancellation (with G52.2) of the current G52.1 mode and the reselection (with G52.1) of the desired offsetting mode.

**Example:** Z for preparatory positioning and W for movement in the mode

```
N101 G91 G28 XYZ  
N102 G28 BCW  
N103 M700  
N104 M600 T801  
N105 G54  
N106 G52.1 XYZ  
N107 G90 G0 X0 Y0 Z100.  
N108 M303  
N109 G52.2  
N110 G52.1 XY W  
N111 G110 Z  
N112 G0 Z-150 X-150.  
N113 G19  
N114 G82 R-5 Z-10 F100.  
N115 G80  
N116 G17  
N117 G49  
N118 G111  
N119 G52.2  
N120 M702  
```

11. The digital indications of the tool tip position are given as X, Y, and Z values and X, Y, and W values, respectively, for the program sections under G52.1 XYZ and G52.1 XYW. The digital indications concerned with axis position are as illustrated below.
14-9 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.

For rewriting the basic coordinates, select the [MEASURE MACRO] menu function to set data in the subprogram unit for the macroprogram concerned.

14-9-1 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.

14-9-2 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)

<table>
<thead>
<tr>
<th>Variables number</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>#5341</td>
<td>WPC-X</td>
</tr>
<tr>
<td>#5342</td>
<td>WPC-Y</td>
</tr>
<tr>
<td>#5343</td>
<td>WPC-Z</td>
</tr>
<tr>
<td>#5344</td>
<td>WPC-4</td>
</tr>
<tr>
<td>#5345</td>
<td>WPC-5</td>
</tr>
<tr>
<td>#5347</td>
<td>WPC-th</td>
</tr>
</tbody>
</table>
14-9-3 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 following format:

1. Macroprogramming format

```
G65P9998X_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:
  
  D: WPC-th  B: WPC-4  C: WPC-5
  
- 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
#5342=#25
#50443=#2
#50447=#25
#50467=#50467OR256
N45
N20
IF[#3EQ#0]GOTO50
IF[#26EQ#0]GOTO30
#5343=#26
#50445=#26
#50467=#50467OR128
N50
```

Note: An alarm will occur when executing the macroprogram if no basic coordinates of MAZATROL program are currently validated.
14-10 Automatic Return to Reference Point (Zero Point): 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 traverse 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 traverse rate independently along each specified axis, and then a G00 (rapid) positioning to the specified position.

2. Programming format

G28 Xx1 Yy1 Zz1 αα1;
G29 Xx2 Yy2 Zz2 αα2;

(α: Additional axis) Automatic return to reference point
(α: Additional axis) Return to start point

3. Detailed description

1. Command G28 is equivalent to the following commands:
   G00 Xx1 Yy1 Zz1 αα1;
   G00 Xx3 Yy3 Zz3 αα3;
   where x3, y3, z3 and α3 denote the coordinates of the appropriate reference point, determined by the parameter as the distance from the zero point of the base machine coordinate system.

2. Axes that have not been returned to the reference point (zero 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 designated direction. 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 return using the watchdog method can also be parameter-set for the second time onward.)

3. When return to reference point (zero point) is completed, a return complete output signal will be outputted and the monitor will display “#1” in the display field of the axis name.

4. Command G29 is equivalent to the following commands:
   G00 Xx1 Yy1 Zz1 αα1;
   G00 Xx2 Yy2 Zz2 αα2;
   \{ Independent rapid traverse on each axis \\
   where x1, y1, z1 and α1 are the coordinates of the intermediate point specified by the last G28 or G30 command. \}
5. A program error will result if G29 is executed without any preceding G28 (automatic reference point return command) after turning-on.

6. The coordinates of the intermediate point \((x_1, y_1, z_1, \alpha_1)\) must be given according to the type of dimensional data input (G90 or G91).

7. 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.

8. The tool offset will be temporarily cancelled during return to reference point unless it is already cancelled but the intermediate point will be the offset position.

9. During return to reference point under a machine lock status, movement from the intermediate point to the reference point is omitted. The next block is executed after the designated axis has arrived at the intermediate point.

10. During return to reference point 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 corresponding point. For movement from that point to the reference point, however, the mirror image become invalid and thus the axis moves to the reference point.

11. In the single-block operation mode, stop will be made at the intermediate point.

**4. Sample programs**

**Example 1:**

```
G28 Xx1 Zz1;
```

**Example 2:**

```
G29 Xx2 Zz2;
```

---

**Diagram**

- First movement after power-on
- Second and subsequent movements
- G00 Xx1 Zz1;
- Intermediate point \((x_1, z_1)\)
- G0 Xx1 Zz1;
- Reference point (#1)
- Rapid feed rate
- Watchdog
- Reference point (#1)
- Second and subsequent movements

---

**Diagram**

- \((G0)\) Xx1 Zz1;
- Intermediate point by G28, G30 \((x_1, z_1)\)
- G0 Xx2 Zz2;
- \((x_2, z_2)\)
Example 3:  

G28   \(x_1, z_1;\)  
\[\text{From point A to reference point}\]

G30   \(x_2, z_2;\)  
\[\text{From point B to second reference point}\]

G29   \(x_3, z_3;\)  
\[\text{From point C to point D}\]

14-11 Return to Second Reference Point (Zero Point): 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) \(x_1, y_1, z_1, \alpha_1;\) \((\alpha: \text{Addition axis.})\)

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 position specific to the machine. The coordinates can be checked by machine parameter M5, M6, or M7.

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. If the plane containing the designated reference point is currently undergoing tool nose radius compensation, the designated axis will move to the intermediate point according to the tool nose radius compensation data. The movement of the axis from the intermediate point to the second reference point will become free from that compensation data. For the next G29 command, movement from the reference point to the intermediate point will be based without the tool nose radius compensation data which will apply to the movement from the intermediate point to the point specified with G29 command.

6. After return to second reference point, the tool nose radius compensation data for the next movement is cancelled automatically.

7. During return to second reference point under a machine lock status, movement from the intermediate point to the reference point is omitted. The next block is executed after the designated axis has arrived at the intermediate point.
8. During return to second reference point in the mirror image mode, the mirror image is valid for movement from the starting point to the intermediate point and the movement is effectuated in an opposite direction to the corresponding point. For movement from that point to the reference point, however, the mirror image becomes invalid and thus the axis moves to that reference point.

14-12 Return to Reference Point Check Command: G27

1. Function and purpose

As with command G28, execution of command G27 will output a return to reference point return complete signal to the machine if the point at which the designated axis has been positioned by the program is the first reference point. Thus, if the axis is programmed to start moving from the first reference point and then returns to that reference point, you can check whether the axis has returned to the reference point after execution of the program.

2. Programming format

G27 Xx1 Yy1 Zz1 Pp1

Check No.

P1: First reference point check
P2: Second reference point check
P3: Third reference point check
P4: Fourth reference point check

Return control axis
Check command

3. Detailed description

- The first reference point check will occur if the P command is omitted.
- The number of axes for which reference point checks can be done at the same time depends on the number of simultaneously controllable axes.
- An alarm will result if the axis has not arrived at the designated reference point on completion of this command.
14-13 Three-Dimensional Coordinate Conversion ON/OFF: G68/G69

1. Outline

The three-dimensional coordinate conversion is used to determine a new coordinate system through the translation of the origin of the currently active workpiece coordinate system and the rotation on an axis of coordinate. Use this command to specify freely a plane in space which is convenient for programming.

2. Programming format

G68  Xx0 Yy0 Zz0 Ii Jj Kk Rr ; ............. 3D coordinate conversion mode ON
G69 ; ............. 3D coordinate conversion mode OFF

x0 y0 z0 : Coordinates of the center of rotation
Specify in absolute dimensions the translation of the workpiece origin.
i, j, k : Designation of the axis of rotation (1: valid, 0: invalid)
  I : X-axis
  J : Y-axis
  K : Z-axis
r : Angle and direction of rotation on the rotational axis
A positive value of angle refers to the left turn when seen from the positive side of the rotational axis.

3. Detailed description

- It is impossible to change the coordinate system in the G68 mode.
- The coordinate system set by a command of G68 is as indicated below.

After the selection of the G17 (XY) plane, the workpiece origin is shifted to the point (X, Y, Z) = (10, 0, –5), and the plane is rotated by 30 degrees on the Y'-axis. The new coordinate system (X', Y', Z') has thus been established.
- The cancel command G69 will set again the coordinate system subjected to the translation and rotation by the preceding G68 command.
- In the G68 mode all the dimensions must be entered in radius values.
4. Sample program

<table>
<thead>
<tr>
<th>N01</th>
<th>G90G00G40G49G80</th>
<th>G68I1J0K0R-90.</th>
</tr>
</thead>
<tbody>
<tr>
<td>G54X0Y-100.</td>
<td>......[1]</td>
<td>G00X0Y0</td>
</tr>
<tr>
<td>G01Z-1.F1200</td>
<td>......[3]</td>
<td>G01Z-1.</td>
</tr>
<tr>
<td>G02X100.R50.</td>
<td>......[5]</td>
<td>G02X100.R50.</td>
</tr>
<tr>
<td>G01Y-100.</td>
<td>......[6]</td>
<td>G01Y0</td>
</tr>
<tr>
<td>G01Z50.</td>
<td>......[7]</td>
<td>G01Z50.</td>
</tr>
<tr>
<td>G91G28Z0</td>
<td>......[8]</td>
<td>G69</td>
</tr>
<tr>
<td>G28X0Y0</td>
<td>......[9]</td>
<td>G91G28Z0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N02</th>
<th>G90G00G40G49G80</th>
<th>G28X0Y0</th>
<th>......[18]</th>
</tr>
</thead>
<tbody>
<tr>
<td>G54G00A90.</td>
<td></td>
<td>M30</td>
<td></td>
</tr>
</tbody>
</table>

![Diagram](NM221-00050)
5. Restrictions

1. The G68 command cannot be given in the following modes:
   - Tool nose radius compensation (G40 mode not selected)
   - Fixed cycle (G80 not selected in the G-code group 09)

2. The G68 command cannot be given in the mode of cross machining.

3. No tool change commands by T-code can be given in the G68 mode. A T-code in this mode
   will be processed as a programming error.

4. A block in the G68 mode cannot be designated as restart position. The search for such a
   block as restart position will cause an alarm.

5. Certain G-codes cannot be given in the G68 mode without causing an alarm. Refer to
   Paragraph 6 below for the listing of illegal G-codes.

6. If the addresses X, Y and Z are all omitted, no translation of the origin will occur and the
   rotation will be performed on an existing axis of coordinate.

7. All the arguments I, J and K must be specified in general as required. If one of the
   arguments is omitted, such a block of G68 will be processed as a programming error.
   
   **Example 1:**
   
   G68 X10. Y0 Z0 I0 J1 R30.; ........ Format error
   
   If, in particular, all the arguments are omitted, then the axis perpendicular to the currently
   selected plane will be regarded as the axis of rotation.

   **Example 2:**
   
   G17;
   G68 X10. Y0 Z0 R30.; .... Equiv. to G68 X10. Y0 Z0 I0 J1 K0 R30.;

8. A block of G68 will be processed as a programming error if all the arguments I, J and K are
   specified with zero (0).
   
   **Example:**
   
   G68 X10. Y0 Z0 I0 J0 K0 R30.; .......... Format error

9. The codes G68 and G69 are not available for a system without the optional function of
   coordinate system rotation.

10. A MAZATROL program cannot be called up as subprogram in the G68 mode.

6. Relationship to other functions

A. Usable G-codes in the G68 mode of 3-D coordinate conversion

<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>00</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>01</td>
</tr>
<tr>
<td>Threading with C-axis interpolation</td>
<td>01.1</td>
</tr>
</tbody>
</table>
| Circular interpolation CW      | 02     | Note 1
| Circular interpolation CCW     | 03     | Note 1
| Dwell                           | 04     | Note 4
| Cylindrical interpolation      | 07.1   |
| Exact-stop check                | 09     |
| Programmed parameter input ON  | 10     | Note 4
| Diameter/Radius data input     | 10.9   |
| Programmed parameter input OFF | 11     | Note 4
<p>| Polar coordinate interpolation ON | 12.1  |
| Polar coordinate interpolation OFF | 13.1  |
| XY-plane selection             | 17     |</p>
<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Five-surface machining (Top surface)</td>
<td>17.1</td>
</tr>
<tr>
<td>Five-surface machining (0° surface)</td>
<td>17.2</td>
</tr>
<tr>
<td>Five-surface machining (90° surface)</td>
<td>17.3</td>
</tr>
<tr>
<td>Five-surface machining (180° surface)</td>
<td>17.4</td>
</tr>
<tr>
<td>Five-surface machining (270° surface)</td>
<td>17.5</td>
</tr>
<tr>
<td>Five-surface machining OFF</td>
<td>17.9</td>
</tr>
<tr>
<td>ZX-plane selection</td>
<td>18</td>
</tr>
<tr>
<td>YZ-plane selection</td>
<td>19</td>
</tr>
<tr>
<td>Return to reference point</td>
<td>28</td>
</tr>
<tr>
<td>Return to starting point</td>
<td>29</td>
</tr>
<tr>
<td>Return to 2nd, 3rd, 4th reference point</td>
<td>30</td>
</tr>
<tr>
<td>Tool radius compensation OFF</td>
<td>40</td>
</tr>
<tr>
<td>Shaping OFF</td>
<td>40.1</td>
</tr>
<tr>
<td>Tool radius compensation, to the left</td>
<td>41</td>
</tr>
<tr>
<td>Shaping, to the left</td>
<td>41.1</td>
</tr>
<tr>
<td>Tool radius compensation, to the right</td>
<td>42</td>
</tr>
<tr>
<td>Shaping, to the right</td>
<td>42.1</td>
</tr>
<tr>
<td>Tool-length offset (+)</td>
<td>43</td>
</tr>
<tr>
<td>Tool-length offset (–)</td>
<td>44</td>
</tr>
<tr>
<td>Tool-position offset, extension</td>
<td>45</td>
</tr>
<tr>
<td>Attachment compensation ON</td>
<td>45.1</td>
</tr>
<tr>
<td>Tool-position offset, reduction</td>
<td>46</td>
</tr>
<tr>
<td>Tool-position offset, double extension</td>
<td>47</td>
</tr>
<tr>
<td>Tool-position offset, double reduction</td>
<td>48</td>
</tr>
<tr>
<td>Tool-position offset OFF</td>
<td>49</td>
</tr>
<tr>
<td>Attachment compensation OFF</td>
<td>49.1</td>
</tr>
<tr>
<td>G-command mirror image OFF</td>
<td>50.1</td>
</tr>
<tr>
<td>Polygonal machining OFF</td>
<td>50.2</td>
</tr>
<tr>
<td>G-command mirror image ON</td>
<td>51.1</td>
</tr>
<tr>
<td>Polygonal machining ON</td>
<td>51.2</td>
</tr>
<tr>
<td>Local coordinate system setting</td>
<td>52</td>
</tr>
<tr>
<td>Machine coordinate system selection</td>
<td>53</td>
</tr>
<tr>
<td>One-way positioning</td>
<td>60</td>
</tr>
<tr>
<td>Exact-stop check mode</td>
<td>61</td>
</tr>
<tr>
<td>Geometry compensation</td>
<td>61.1</td>
</tr>
<tr>
<td>Inverse model control</td>
<td>61.4</td>
</tr>
<tr>
<td>Automatic corner override</td>
<td>62</td>
</tr>
<tr>
<td>Tapping mode</td>
<td>63</td>
</tr>
<tr>
<td>Cutting mode</td>
<td>64</td>
</tr>
<tr>
<td>User macro simple call</td>
<td>65</td>
</tr>
<tr>
<td>User macro modal call A</td>
<td>66</td>
</tr>
<tr>
<td>User macro modal call B</td>
<td>66.1</td>
</tr>
<tr>
<td>User macro modal call OFF</td>
<td>67</td>
</tr>
<tr>
<td>3-D coordinate conversion ON</td>
<td>68</td>
</tr>
<tr>
<td>Programmed coordinates rotation/3-D coordinate conversion OFF</td>
<td>69</td>
</tr>
<tr>
<td>Fixed cycle (chamfering cutter 1)</td>
<td>71.1</td>
</tr>
<tr>
<td>Fixed cycle (chamfering cutter 2)</td>
<td>72.1</td>
</tr>
</tbody>
</table>
### Function G-code

| Fixed cycle (high-speed deep-hole drilling) | 73   |
| Fixed cycle (reverse tapping)              | 74   |
| Fixed cycle (boring)                       | 75   |
| Fixed cycle (boring)                       | 76   |
| Fixed cycle (Back facing)                  | 77   |
| Fixed cycle (boring)                       | 78   |
| Fixed cycle (boring)                       | 79   |
| Fixed cycle OFF                            | 80   |
| Fixed cycle (drill/spot drill)             | 81   |
| Fixed cycle (drilling)                     | 82   |
| Fixed cycle (drilling)                     | 82.2 |
| Fixed cycle (deep hole drilling)           | 83   |
| Fixed cycle (tapping)                      | 84   |
| Fixed cycle (synchronous tapping)          | 84.2 |
| Fixed cycle (synchronous reverse tapping)  | 84.3 |
| Fixed cycle (reaming)                      | 85   |
| Fixed cycle (boring)                       | 86   |
| Fixed cycle (back boring)                  | 87   |
| Fixed cycle (boring)                       | 88   |
| Fixed cycle (boring)                       | 89   |
| Absolute data input                        | 90   |
| Incremental data input                     | 91   |
| Workpiece coordinate system rotation       | 92.5 |
| Inverse time feed                          | 93   | Note 3
| Asynchronous feed (feed per minute)        | 94   | Note 4
| Synchronous feed (feed per revolution)     | 95   | Note 4
| Initial level return in fixed cycle        | 98   |
| R-point level return in fixed cycle        | 99   |
| Axis name change ON                        | 110  |
| Axis name change OFF                       | 111  |
| Hob milling OFF                            | 113  |
| Hob milling ON                             | 114.3|

**Note 1:** Setting of helical interpolation results in an alarm.

**Note 2:** Only the intermediate point has its coordinates converted.

**Note 3:** To use G31 for measurement purposes, G68 (3D coordinate conversion mode) must be cancelled.

**Note 4:** The preparatory function is executed independently of the coordinate conversion.

**Note 5:** The operation is always performed in the original coordinate system (free from any conversion).
## B. Modes in which G68 is selectable

<table>
<thead>
<tr>
<th>Function</th>
<th>G-code</th>
<th>Function</th>
<th>G-code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>00</td>
<td>Exact-stop check mode</td>
<td>61</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>01</td>
<td>Geometry compensation</td>
<td>61.1</td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>13.1</td>
<td>Inverse model control</td>
<td>61.4</td>
</tr>
<tr>
<td>XY-plane selection</td>
<td>17</td>
<td>Automatic corner override</td>
<td>62</td>
</tr>
<tr>
<td>Five-surface machining (Top surface)</td>
<td>17.1</td>
<td>Tapping mode</td>
<td>63</td>
</tr>
<tr>
<td>Five-surface machining (0° surface)</td>
<td>17.2</td>
<td>Cutting mode</td>
<td>64</td>
</tr>
<tr>
<td>Five-surface machining (90° surface)</td>
<td>17.3</td>
<td>User macro modal call OFF</td>
<td>67</td>
</tr>
<tr>
<td>Five-surface machining (180° surface)</td>
<td>17.4</td>
<td>3-D coordinate conversion ON</td>
<td>68</td>
</tr>
<tr>
<td>Five-surface machining (270° surface)</td>
<td>17.5</td>
<td>Programmed coordinates rotation/3-D coordinate conversion OFF</td>
<td>69</td>
</tr>
<tr>
<td>Five-surface machining OFF</td>
<td>17.9</td>
<td>Fixed cycle OFF</td>
<td>80</td>
</tr>
<tr>
<td>ZX-plane selection</td>
<td>18</td>
<td>Absolute data input</td>
<td>90</td>
</tr>
<tr>
<td>YZ-plane selection</td>
<td>19</td>
<td>Incremental data input</td>
<td>91</td>
</tr>
<tr>
<td>Inch command</td>
<td>20</td>
<td>Inverse time feed</td>
<td>93</td>
</tr>
<tr>
<td>Metric command</td>
<td>21</td>
<td>Asynchronous feed (feed per minute)</td>
<td>94</td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>23</td>
<td>Synchronous feed (feed per revolution)</td>
<td>95</td>
</tr>
<tr>
<td>Attachment compensation ON</td>
<td>45.1</td>
<td>Constant surface speed control ON</td>
<td>96</td>
</tr>
<tr>
<td>Tool-position offset OFF</td>
<td>49</td>
<td>Constant surface speed control OFF</td>
<td>97</td>
</tr>
<tr>
<td>Attachment compensation OFF</td>
<td>49.1</td>
<td>Initial level return in fixed cycle</td>
<td>98</td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>50</td>
<td>R-point level return in fixed cycle</td>
<td>99</td>
</tr>
<tr>
<td>G-command mirror image OFF</td>
<td>50.1</td>
<td>Axis name change ON</td>
<td>110</td>
</tr>
<tr>
<td>Polygonal machining OFF</td>
<td>50.2</td>
<td>Axis name change OFF</td>
<td>111</td>
</tr>
<tr>
<td>Polygonal machining ON</td>
<td>51.2</td>
<td>Hob milling OFF</td>
<td>113</td>
</tr>
<tr>
<td>Local coordinate system setting</td>
<td>52</td>
<td>Hob milling ON</td>
<td>114.3</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 1</td>
<td>54</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of additional workpiece coordinate system</td>
<td>54.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 2</td>
<td>55</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 3</td>
<td>56</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 4</td>
<td>57</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 5</td>
<td>58</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 6</td>
<td>59</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
14-14 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 ..... ............XY-plane
(G18) G92.5 Zz Xx Rr ..... ............ZX-plane
(G19) G92.5 Yy Zz Rr ..... ............YZ-plane

or

(G17) G92.5 Xx Yy li Jj ............XY-plane
(G18) G92.5 Zz Xx Kk li ............ZX-plane
(G19) G92.5 Yy Zz Jj Kk ............YZ-plane

x, y, z : Coordinates of the rotational center.

The position along the two axes of the previously selected XY-, ZX-, or YZ-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

<table>
<thead>
<tr>
<th>Setting method</th>
<th>Setting range</th>
<th>Setting unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial component vectors (i, j, k)</td>
<td>Metric system</td>
<td>0 to ±99999.999</td>
</tr>
<tr>
<td></td>
<td>Inch system</td>
<td>0 to ±9999.9999</td>
</tr>
<tr>
<td>Angle of rotation (r)</td>
<td>Metric system</td>
<td>0 to ±99999.999°</td>
</tr>
<tr>
<td></td>
<td>Inch system</td>
<td>0 to ±9999.999°</td>
</tr>
</tbody>
</table>
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 XY-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.

```
```

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

```
```

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
%

- 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.............
N7 X100.
N8 X150.
N9 M30

- 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.

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).
3. Programmed coordinate rotation (G68) in the mode of G92.5

N1 G28X0Y0
N2 G17
N3 G55
N4 G90
N5 G92.5X0Y0R90. ............................ [1]
N7 G0X0Y0
N8 G1X100.F500
N9 Y100.
N10 X0
N11 Y0
N12 M30
%

G55 (Work Offset)
X100.
Y100.

Programmed contour without [2]
Programmed contour with both [1] and [2]
Programmed contour without both [1] and [2]

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
```

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
%

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

   G55 (Work Offset)
   X100.
   Y100.

   
   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

   Programmed contour without [1]
   Programmed contour without [2]
   Programmed contour with both [1] and [2]
b) **M-code mirror image**

N1  G28X0Y0  
N2  G17  
N3  G55  
N4  G90  
N5  G92.5X0Y0R90.  
N6  M91.  
N7  G0X0Y0  
N8  G1X100.F500  
N9  Y100.  
N10  X0Y0  
N11  M30  

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
```

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.

Programmed contour without [2]
Programmed contour with both [1] and [2]
Programmed contour without both [1] and [2]
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

![Circular interpolation diagram](image)

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.

<table>
<thead>
<tr>
<th>Function name</th>
<th>Workpiece coordinate system rotation</th>
<th>Programmed coordinate rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>System to be rotated</td>
<td>Workpiece coordinate system</td>
<td>Local coordinate system</td>
</tr>
<tr>
<td>Programming format</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(G17)</td>
<td>G92.5 Xx Yy Rr</td>
<td>(G17)</td>
</tr>
<tr>
<td>(G18)</td>
<td>G92.5 Yy Zz Rr</td>
<td>(G18)</td>
</tr>
<tr>
<td>(G19)</td>
<td>G92.5 Zz Xx Rr</td>
<td>(G19)</td>
</tr>
<tr>
<td>or</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(G17)</td>
<td>G92.5 Xx Yy li Jj</td>
<td></td>
</tr>
<tr>
<td>(G18)</td>
<td>G92.5 Yy Zz Jj Kk</td>
<td>(Vector comp.)</td>
</tr>
<tr>
<td>(G19)</td>
<td>G92.5 Zz Xx Kk li</td>
<td></td>
</tr>
<tr>
<td>Operation</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotational center coordinates</td>
<td>Designation at addresses X, Y, Z</td>
<td>Designation at addresses X, Y, Z</td>
</tr>
<tr>
<td>Angle of rotation</td>
<td>Designation at R (angle) or at I, J, K (vector components)</td>
<td>Designation at R (angle)</td>
</tr>
<tr>
<td>Information on center and angle of rotation cleared?</td>
<td>Power-off → on</td>
<td>Retained</td>
</tr>
<tr>
<td></td>
<td>M02/M30</td>
<td>Retained</td>
</tr>
<tr>
<td></td>
<td>Reset key</td>
<td>Retained</td>
</tr>
<tr>
<td></td>
<td>Resumption of readiness after emergency stop</td>
<td>Retained</td>
</tr>
</tbody>
</table>

**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.
15 MEASUREMENT SUPPORT FUNCTIONS

Measurement by EIA/ISO is basically the same as that by MAZATROL. Information given by MAZATROL may be executed by preparation function below.

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 Zz Yy Ff ;
   x, y, z : The coordinates of the respective axes. These coordinates are to be designated using absolute or incremental data.
   f : Rate of feed (mm/min)

3. Detailed description
   1. An asynchronous feed rate commanded previously will be used as feed rate. If an asynchronous feed command is not given previously and if Ff is not commanded, the value set by the parameter K41 will be used as the feed rate. 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 nose radius compensation results in a program error.
   7. Under a machine lock status the skip signals will be valid.
4. Execution of G31

**Example 1:** When the next block is an incremental value command

G31 Z10. F100;
G91 G01 X20. Z10.;

**Example 2:** When the next block is a one axis move command with absolute value

G31 Z10. F100;
G01 X10.;

**Example 3:** When the next block is a two axes move command with absolute value

G31 Z10. F100;
G01 X10. Z20.;
**15-1-2 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:

\[
\delta_0 = \frac{F}{60} \times T_p + \frac{F}{60} (t_1 \pm t_2) = \frac{F}{60} \times (T_p + t_1) \pm \frac{F}{60} \times t_2
\]

- \(\delta_0\): Amount of coasting (mm)
- \(F\): G31 skip rate (mm/min)
- \(T_p\): Position loop time constant (sec) = (Position loop gain)\(^{-1}\)
- \(t_1\): Response delay time (sec) = (The time from skip signal detection until arrival at NC through PC)
- \(t_2\): Response error time = 0.001 (sec)

When using command G31 for measurement purposes, measured data \(\delta_1\) can be corrected. Such corrections, however, cannot be performed for \(\delta_2\).

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 msec and, \(t_1\) to 5 msec.
15-1-3 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 Tp and cutting feed time constant Ts. 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.

\[
\varepsilon = \pm \frac{F}{60} \times t_2
\]

$\varepsilon$ : Reading error (mm)
$F$ : Feed rate (mm/min)
$t_2$ : Response delay time 0.001(sec)

The reading error at a feed rate of 60 mm/min

\[
\varepsilon = \pm \frac{60}{60} \times 0.001 = \pm 0.001 \text{ (mm)}
\]

and measured data stays within the reading error range of $\pm 1\mu$.

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.
15-2 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
1. 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
2. The program will skip when the skip signal input conditions appropriate for each of these G commands are satisfied.
3. Except for items other than 1 and 2 above, the description of command code G31 in Section 15-1 also applies.

4. Parameter setting
1. 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.
2. 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 00000111 makes the command concerned equivalent to G31.

<table>
<thead>
<tr>
<th>Parameter setting (K69 to K73)</th>
<th>Valid skip signals</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bit setting</td>
<td>A</td>
</tr>
<tr>
<td>7 6 5 4 3 2 1 0</td>
<td></td>
</tr>
<tr>
<td>0 0 0 0 0 0 0 1</td>
<td></td>
</tr>
<tr>
<td>0 0 0 0 0 0 1 0</td>
<td></td>
</tr>
<tr>
<td>0 0 0 0 0 1 1 1</td>
<td></td>
</tr>
<tr>
<td>0 0 0 0 1 0 0 0</td>
<td></td>
</tr>
<tr>
<td>0 0 0 0 1 0 1 0</td>
<td></td>
</tr>
<tr>
<td>0 0 0 1 1 1 0 0</td>
<td></td>
</tr>
<tr>
<td>0 0 1 1 1 1 0 0</td>
<td></td>
</tr>
</tbody>
</table>

---

15-5
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.

<table>
<thead>
<tr>
<th>Skip condition</th>
<th>Skip feed rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>K69 = 00000111 (for G31.1)</td>
<td>20.0 mm/min (f₁)</td>
</tr>
<tr>
<td>K70 = 00000011 (for G31.2)</td>
<td>5.0 mm/min (f₂)</td>
</tr>
<tr>
<td>K71 = 00000001 (for G31.3)</td>
<td>1.0 mm/min (f₃)</td>
</tr>
</tbody>
</table>

Sample program

```
N10  G31.1X200.
N20  G31.2X40.
N30  G31.3X1.
```

Note: During the machine action shown above, if input of skip signal C precedes that of skip signal B, 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 stored 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.

```
Stored stroke limit I, upper limit

Stored stroke limit II, upper limit

Before-movement stroke check, upper limit

(i, j, k) ~ Before-movement stroke check, lower limit

Stored stroke limit II, lower limit

Stored stroke limit I, lower limit

(x, y, z)
```

2. Programming format

```
G22 X_ Y_ Z_ I_ J_ K_ (Inside machining prohibit area specification)

Lower limit specification

Upper limit specification

G23 (Cancellation)
```

3. Detailed description

1. Both upper-limit and lower-limit values must be specified with machine coordinates.

2. 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.

\[
\]

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.
This chapter describes the measurement macros provided for implementing MAZATROL workpiece measuring and tool measuring functions on the EIA program. The measurement macroprograms are to be called up with G136 (Measurement Macro Call) and G137 (Correction Macro Call). The use of these G-codes requires the corresponding settings of the relevant parameters (J).

1. Detailed description

Give special macro call instructions (with G136 and G137) to use the measurement macros. The appropriate system of workpiece coordinates must be established beforehand for the macro to operate correctly. Moreover, the B-axis positioning in an angle and linear positioning to an intermediate point must be completed as appropriate before the macro call since such operations are not provided for in the macroprogram.

Note that modal information on G90/G91 (absolute/incremental programming) and the F-code (rate of feed) may be changed through the execution of the G136 measurement macro.

A. Related parameters for macro call (J29 to J36)

Set the related parameters (J) as instructed below in order that the relevant system macros may correctly be called by the special G-codes.

<table>
<thead>
<tr>
<th>No.1</th>
<th>No.2</th>
<th>No.3</th>
<th>No.4</th>
<th>No.5</th>
<th>No.6</th>
<th>No.7</th>
<th>No.8</th>
<th>No.9</th>
<th>No.10</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>J1</td>
<td>J5</td>
<td>J9</td>
<td>J13</td>
<td>J17</td>
<td>J21</td>
<td>J25</td>
<td>J29</td>
<td>J33</td>
<td>J37</td>
<td>Work No. of the called program</td>
</tr>
<tr>
<td>J2</td>
<td>J6</td>
<td>J10</td>
<td>J14</td>
<td>J18</td>
<td>J22</td>
<td>J26</td>
<td>J30</td>
<td>J34</td>
<td>J38</td>
<td>G-code for macro call</td>
</tr>
<tr>
<td>J3</td>
<td>J7</td>
<td>J11</td>
<td>J15</td>
<td>J19</td>
<td>J23</td>
<td>J27</td>
<td>J31</td>
<td>J35</td>
<td>J39</td>
<td>Type of macro call</td>
</tr>
</tbody>
</table>

Parameters for G136 (Measurement Macro Call)

J29 = 100009590
J30 = 136
J31 = 1 (G65)
J32 = 0

Parameters for G137 (Correction Macro Call)

J33 = 100009599
J34 = 137
J35 = 1 (G65)
J36 = 0

B. Setting the system of workpiece coordinates

The measurement of the outside or inside diameter of a workpiece requires the origins of the X- and Y-axis coordinates to be set on the axis of the turning spindle, as is the case with turning operations by a MAZATROL program.

There are no restrictions on the system of coordinates for other measurement patterns.

Set a code from G54 to G59, or use a G92 command, to designate the system of coordinates.

<table>
<thead>
<tr>
<th>Measurement pattern</th>
<th>Origin of workpiece coordinates</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X-axis</td>
</tr>
<tr>
<td>Workpiece measurement</td>
<td>Patterns except OD/ID</td>
</tr>
<tr>
<td></td>
<td>OD and ID</td>
</tr>
<tr>
<td>Tool measurement</td>
<td>Arbitrary (because the machine coordinate system is used)</td>
</tr>
</tbody>
</table>
C. Coordinates rotation (G68)

The programmed coordinates rotation mode (G68) must always be canceled before calling the measurement macro.

D. Tool length offset

The measurement macro performs length offsetting for the touch sensor in the same manner as for turning tools. That is, the appropriate compensation can be provided for any angular position on the B-axis without coordinate rotation, which is required in the case of milling tools.

E. Measurement patterns and required addresses

The arguments used in the measurement and correction macros are as tabulated below. Use address Q to specify the measurement pattern.

<table>
<thead>
<tr>
<th>Pattern</th>
<th>G-code</th>
<th>Address</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X Y Z A B C H I J K D M U V W R S T Q</td>
<td></td>
</tr>
<tr>
<td>X WIDTH</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 1</td>
</tr>
<tr>
<td>Y WIDTH</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 2</td>
</tr>
<tr>
<td>Z WIDTH</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 3</td>
</tr>
<tr>
<td>X GROOVE</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 4</td>
</tr>
<tr>
<td>Y GROOVE</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 5</td>
</tr>
<tr>
<td>Z GROOVE</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 6</td>
</tr>
<tr>
<td>+X STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 7</td>
</tr>
<tr>
<td>–X STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 8</td>
</tr>
<tr>
<td>+Y STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 9</td>
</tr>
<tr>
<td>–Y STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 10</td>
</tr>
<tr>
<td>+Z STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 11</td>
</tr>
<tr>
<td>–Z STEP</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 12</td>
</tr>
<tr>
<td>OUTER X</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 13</td>
</tr>
<tr>
<td>OUTER Y</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 14</td>
</tr>
<tr>
<td>INNER X</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 15</td>
</tr>
<tr>
<td>INNER Y</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 16</td>
</tr>
<tr>
<td>OUT. X (One-side)</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 17</td>
</tr>
<tr>
<td>INN. X (One-side)</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 18</td>
</tr>
<tr>
<td>IN WIDTH</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 19</td>
</tr>
<tr>
<td>IN GROOVE</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 20</td>
</tr>
<tr>
<td>EXT MILL</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 51</td>
</tr>
<tr>
<td>EXT TURN</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 51</td>
</tr>
<tr>
<td>Correction/Printout</td>
<td>G137</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
</tr>
<tr>
<td>MDI Laser meas.</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
</tr>
<tr>
<td>Auto Laser meas.</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
</tr>
<tr>
<td>TOOL EYE</td>
<td>G136</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
</tr>
<tr>
<td>Correction/Printout</td>
<td>G137</td>
<td>0 0 0 0 0 0 0 0 0 0 0 0 0 0 0</td>
</tr>
</tbody>
</table>

- A blank box indicates that specifying the argument is pointless.
- Tool change occurs in accordance with the argument S (sensor tool).
- (×) Do not enter the argument S (for sensor tool), in general, for external measurement.
- OD and ID measuring movements are performed on a YZ- or ZX-plane cutting the third axis (X or Y) in its origin.
- Set in general the origin of the X- and Y-axis coordinates on the axis of the turning spindle.
- Omit the argument S (sensor designation) as required to suppress tool change operation (e.g. for fear lest an interference should occur after tool change). In this case, no information on the sensor tool can be printed out by G137.

2. Sample instructions for macro call

A. Workpiece measurement (G136: Q1 - Q18)

: G54 .................................................. [1]


: G137 A1 B0 C0 V2.0 .................................................. [3]

[1] Coordinate system setting
[2] Tool change for “T4A” and the Protrusion WIDTH along the X-axis measured.
[3] Correction on the “T2” tool and the measurement results printed out.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Starting point X</td>
<td>X-axis coordinate of the starting measurement point. (Note 3)</td>
</tr>
<tr>
<td>Y</td>
<td>Starting point Y</td>
<td>Y-axis coordinate of the starting measurement point. (Note 3)</td>
</tr>
<tr>
<td>Z</td>
<td>Starting point Z</td>
<td>Z-axis coordinate of the starting measurement point.</td>
</tr>
<tr>
<td>A</td>
<td>Ending point X</td>
<td>X-axis coordinate of the ending measurement point. (Note 3)</td>
</tr>
<tr>
<td>B</td>
<td>Ending point Y</td>
<td>Y-axis coordinate of the ending measurement point. (Note 3)</td>
</tr>
<tr>
<td>C</td>
<td>Ending point Z</td>
<td>Z-axis coordinate of the ending measurement point.</td>
</tr>
<tr>
<td>I</td>
<td>Upper tolerance limit</td>
<td>Upper limit of tolerance. (Note 3)</td>
</tr>
<tr>
<td>J</td>
<td>Lower tolerance limit</td>
<td>Lower limit of tolerance. (Note 3)</td>
</tr>
<tr>
<td>K</td>
<td>Measurement stroke</td>
<td>Distance for measuring feed. (Notes 1, 3)</td>
</tr>
<tr>
<td>D</td>
<td>Measurement reference</td>
<td>0: Starting measurement point 1: Ending measurement point</td>
</tr>
<tr>
<td>M</td>
<td>Direction of approach</td>
<td>0: X (OD turning direction) 1: Z (Facing direction)</td>
</tr>
<tr>
<td>U</td>
<td>Approach point X</td>
<td>X-axis coordinate of the approach point. (Note 3)</td>
</tr>
<tr>
<td>V</td>
<td>Approach point Y</td>
<td>Y-axis coordinate of the approach point. (Note 3)</td>
</tr>
<tr>
<td>W</td>
<td>Approach point Z</td>
<td>Z-axis coordinate of the approach point.</td>
</tr>
<tr>
<td>S</td>
<td>Sensor tool</td>
<td>Tool No. and suffix (before and after decimal point) of touch sensor. (Notes 2, 4)</td>
</tr>
<tr>
<td>Q</td>
<td>Measurement pattern</td>
<td>Designation of measurement pattern.</td>
</tr>
</tbody>
</table>

- “Diameter” measuring movements must be performed on a YZ- or ZX-plane containing the axis of turning operations. For this reason, be sure to set the workpiece origins of the X- and Y-axis coordinates on the axis of the turning spindle.

- Omission of argument Q result in the alarm 994 MACRO MEASUREMENT ALARM 5.

Note 1: Argument K for EIA measurement has almost the same meaning as parameter K19 for MAZATROL programs. “K” and K19 denote respectively a distance from the center, and one from the contact point, of the probe, as shown below.

Note 2: See the chapter entitled TOOL FUNCTIONS for more information on the tool specifying method.
Example) T2.3 ・・・ Tool No. 2 with suffix C
**Note 3:** In the mode of “G10.9 X1” (Diameter data input for the X-axis), designate positions on the Y-axis (with addresses Y, B, or U) in diameter values as well as positions on the X-axis (with X, A, or U).

**Note 4:** Give commands for tool change and positioning the B-axis to 90° beforehand, as required for Z-axis groove/step width measurement, and omit the argument S in the G136 block.

### B. Workpiece measurement - External (G136: Q19, Q20)

```
G137 A1 B0 C0 V3.0  .................. [2]
```

[2] Correction on the tool “T3” and the measurement results printed out.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Data item to be corrected</td>
<td>Valid for turning tools only.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0: Wear compensation X</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1: Wear compensation Z (Note)</td>
</tr>
<tr>
<td>I</td>
<td>Upper tolerance limit</td>
<td>Upper limit of tolerance.</td>
</tr>
<tr>
<td>J</td>
<td>Lower tolerance limit</td>
<td>Lower limit of tolerance.</td>
</tr>
<tr>
<td>D</td>
<td>Change of sign of correction value</td>
<td>0: Not reversed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1: Reversed (Note)</td>
</tr>
<tr>
<td>M</td>
<td>Desired value</td>
<td>Desired value for the section to be measured externally.</td>
</tr>
<tr>
<td>Q</td>
<td>Measurement pattern</td>
<td>19: External measurement of a milling tool</td>
</tr>
<tr>
<td></td>
<td></td>
<td>20: External measurement of a turning tool</td>
</tr>
</tbody>
</table>

**Note:** Setting a value other than 0 and 1 functions the same as setting “1”.


C. Workpiece measurement - Correction/Printout (G137)


:  

G137 A1 B0 C0 V2.0  ······ [2]

[1] Protrusion WIDTH along the X-axis measured.
[2] Correction on the tool “T2” and output of the measurement results.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
</table>
| A       | Measurement results output pattern | 0: No output  
1: Output to a text file (C:\MC_sdg\print\print.txt) on the hard disk.  
The maximum permissible size of the text file is 100 MB. (The size limit can be changed by parameter DPR8.)  
2: Output to a printer (via RS232C) |
| B       | Data item to be corrected | Valid for milling tools only.  
0: Tool diameter  (Note 3)  
1: Tool length  (Note 5) |
| C       | Correction | 0: Executed  
1: Not executed  (Note 4) |
| D       | Type of the tool for correction | Used for an offset No. designated.  
0: Correction for a milling tool  
1: Correction for a turning tool  (Note 5) |
| V (Note 2) | Tool No. for correction | No. and suffix (before and after decimal point) of the tool concerned.  (Note 1)  
Used for correction on the MAZATROL tool data. |
| H (Note 2) | Offset No. for correction | No. of the offset data on which correction is to be conducted. |

Note 1: See the chapter entitled TOOL FUNCTIONS for more information on the tool specifying method.

Note 2: Setting both H (offset No.) and V (tool for correction) results in the alarm 993 MACRO MEASUREMENT ALARM 4.

Note 3: Setting a value other than 0 and 1 functions the same as setting “0” (for diameter).

Note 4: Setting a value other than 0 and 1 functions the same as setting “1”.

Note 5: “D0” for OD/ID measurement or “B1” for Width/Groove measurement leads to the alarm 192 EXECUTION IMPOSSIBLE.

<Data item for correction>

1. For an argument V (Tool for correction)
   Correction is conducted on the data of the TOOL DATA display.
   - For milling tools: LENGTH or ACT-φ
   - For turning tools: WEAR COMP. X or Z

2. For an argument H (Offset No. for correction)
   Correction is conducted on the data of the TOOL OFFSET display.
   - For milling tools: GEOMETRIC OFFSET Z (as length) or NOSE-R (as diameter)
   - For turning tools: WEAR COMP. Z or X
D. Tool measurement

1. MDI laser measurement (G136: Q51)

   M33 .................................. Laser measuring unit cover opened.
   M187 .................................. Status for laser measurement checked.
   G136 T1.2 M1 Q51 ................. Tool change for TNo. 1.B and length measured.
   G136 T1.2 M2 Q51 ................. Diameter of TNo. 1.B measured.
   G137 V1.2 .................................. Correction for length and diameter on the TNo. 1.B tool conducted.
   M34 .................................. Laser measuring unit cover closed.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
</table>
   | M       | Type of measurement   | 1: MDI tool length measurement [from the menu]  
           (for MDI only)  
           2: MDI tool diameter measurement [from the menu]  (Note 1) |
   | T       | Tool to be measured   | No. and suffix (before and after decimal point) of the tool to be measured. (Notes 2, 3) |
   | Q       | Measurement pattern   | 51: Laser measurement.          |

   Note 1: Do not set “M0”. Diameter measurement by “M2” is executed on the basis of the result from the preceding length measurement.

   Note 2: See the chapter entitled TOOL FUNCTIONS for a detailed description of tool designation.

   Example: Enter “T2.3” for the tool of TNo. 2 with suffix C.

   Note 3: The G136 macro of MDI laser measurement includes the change for the tool to be measured.

2. Automatic laser measurement [using MAZATROL tool data] (G136: Q51)

   T1.2T0M6 .................................. Tool change for TNo. 1.B.
   G020 .................................. Tool length offset conducted.
   M33 .................................. Laser measuring unit cover opened.
   M187 .................................. Status for laser measurement checked.
   G137 A0 C0 V1.2 .................................. Correction for length and diameter on the TNo. 1.B tool conducted.
   M34 .................................. Laser measuring unit cover closed.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
</table>
   | B       | Check for breakage    | 0: Not executed  
           1: Executed  (Note) |
   | I       | Tolerance for length  | Tolerance for tool length.  
           Omit the argument I if tool length measurement is not required. |
   | J       | Tolerance for diameter| Tolerance for tool diameter.  
           Omit the argument J if tool diameter measurement is not required. |
   | Q       | Measurement pattern   | 51: Laser measurement.          |

   Note: Setting a value other than 0 and 1 functions the same as setting “1”.

   - Be sure to give a G137 command, irrespective of whether checking for tool breakage is to be executed.

   - The G136 macro of automatic laser measurement does not include the change for the tool to be measured. Therefore, give commands for tool change and length offset beforehand.
3. Automatic laser measurement [using tool offset data] (G136: Q51)

- Length measurement, using tool offset data only

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43 H3</td>
<td>Tool length offset conducted using offset data No. 3.</td>
</tr>
<tr>
<td>M33</td>
<td>Laser measuring unit cover opened.</td>
</tr>
<tr>
<td>M187</td>
<td>Status for laser measurement checked.</td>
</tr>
<tr>
<td>G136 B0 I1. J0. Q51</td>
<td>Length measured on the basis of offset data No. 3.</td>
</tr>
<tr>
<td>G137 A0 C0 H3</td>
<td>Correction on offset data No. 3 conducted.</td>
</tr>
<tr>
<td>M34</td>
<td>Laser measuring unit cover closed.</td>
</tr>
</tbody>
</table>

- Diameter measurement, using tool offset data only

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G43 H3</td>
<td>Tool length offset conducted using offset data No. 3.</td>
</tr>
<tr>
<td>M33</td>
<td>Laser measuring unit cover opened.</td>
</tr>
<tr>
<td>M187</td>
<td>Status for laser measurement checked.</td>
</tr>
<tr>
<td>G136 B0 I0. J1. D2 Q51</td>
<td>Diameter measured on the basis of offset data No. 2.</td>
</tr>
<tr>
<td>G137 A0 C0 H2</td>
<td>Correction on offset data No. 2 conducted.</td>
</tr>
<tr>
<td>M34</td>
<td>Laser measuring unit cover closed.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Check for breakage</td>
<td>0: Not executed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1: Executed (Note 1)</td>
</tr>
<tr>
<td>I</td>
<td>Tolerance for length</td>
<td>Tolerance for tool length.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omit the argument I if tool length measurement is not required.</td>
</tr>
<tr>
<td>J</td>
<td>Tolerance for diameter</td>
<td>Tolerance for tool diameter.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omit the argument J if tool diameter measurement is not required.</td>
</tr>
<tr>
<td>D</td>
<td>Tool offset No.</td>
<td>Number of tool offset data to be used for diameter measurement. (Note 2)</td>
</tr>
<tr>
<td>Q</td>
<td>Measurement pattern</td>
<td>51: Laser measurement.</td>
</tr>
</tbody>
</table>

**Note 1:** Setting a value other than 0 and 1 functions the same as setting “1”.

**Note 2:** Use address D to specify the number of tool offset data to be used for diameter measurement in the EIA/ISO program execution.

The specified offset data is used for diameter measurement, irrespective of the setting in F92 bit 7 (Diameter offset by MAZATROL tool data). Omit argument D if the diameter value of the MAZATROL tool data concerned is to be used.

- Be sure to give a G137 command, irrespective of whether checking for tool breakage is to be executed.

- The G136 macro of automatic laser measurement does not include the change for the tool to be measured. Therefore, give commands for tool change and length offset beforehand.

- Tool length offset is indispensable for the measurement, be it of length or diameter, since the tool path for the execution of G136 macro is controlled using the previously specified value of length offset.
4. TOOL EYE measurement (G136: Q52)

M283
G136 B1 I-9.J-9.M1 R0.2 Q52
G137 A1 C0 V2.1
M284

TOOL EYE set in the measuring position.

TOOL EYE measurement #1 (with breakage check).

Correction and printout.

TOOL EYE returned to the waiting position.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Check for breakage</td>
<td>0: Not executed</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1: Executed (Note 1)</td>
</tr>
<tr>
<td>I</td>
<td>Tolerance for Wear X</td>
<td>Tolerance for Wear X data.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omit the argument I if tool measurement (for X) is not required.</td>
</tr>
<tr>
<td>J</td>
<td>Tolerance for Wear Z</td>
<td>Tolerance for Wear Z data.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omit the argument J if tool measurement (for Z) is not required.</td>
</tr>
<tr>
<td>M</td>
<td>Measurement section</td>
<td>Measurement section of the TOOL EYE.</td>
</tr>
<tr>
<td>R</td>
<td>Nose-R</td>
<td>Nose radius of the tool to be measured. (Note 2)</td>
</tr>
<tr>
<td>Q</td>
<td>Measurement pattern</td>
<td>52: TOOL EYE measurement.</td>
</tr>
</tbody>
</table>

Note 1: Setting a value other than 0 and 1 functions the same as setting “1”.

Note 2: Set the precise value of nose-R of the tool to be measured with address R; otherwise the tool cannot be brought properly into contact with the TOOL EYE sensor.
E. Tool measurement - Correction/Printout (G137)

G137 A1 C0 V3.0 ··········· Correction on the tool “T3” and output of the measurement results.

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Measurement results output pattern</td>
<td>0: No output 1: Output to a text file (C:\MC_sdg\print\print.txt) on the hard disk. The maximum permissible size of the text file is 100 MB. (The size limit can be changed by parameter DPR8.) 2: Output to a printer (via RS232C)</td>
</tr>
<tr>
<td>C</td>
<td>Correction</td>
<td>0: Executed 1: Not executed (Note 3)</td>
</tr>
<tr>
<td>V (Note 2)</td>
<td>Tool No. for correction</td>
<td>No. and suffix (before and after decimal point) of the tool concerned. (Note 1) Used for correction on the MAZATROL tool data.</td>
</tr>
<tr>
<td>H (Note 2)</td>
<td>Offset No. for correction</td>
<td>No. of the offset data on which correction is to be conducted.</td>
</tr>
</tbody>
</table>

Note 1: See the chapter entitled TOOL FUNCTIONS for more information on the tool specifying method.

Note 2: Setting both H (offset No.) and V (tool for correction) results in the alarm 993 MACRO MEASUREMENT ALARM 4.

Note 3: Setting a value other than 0 and 1 functions the same as setting “1”.

<Data item for correction>

1. For an argument V (Tool for correction)
   Correction is conducted on the data of the TOOL DATA display.
   - For milling tools: LENGTH or ACT-\( r \)
   - For turning tools: WEAR COMP. X or Z

2. For an argument H (Offset No. for correction)
   Correction is conducted on the data of the TOOL OFFSET display.
   - For milling tools: GEOMETRIC OFFSET Z (as length) or NOSE-R (as diameter)
   - For turning tools: WEAR COMP. Z or X

3. Illustration

A. Workpiece measurement

The example bellow refers to the measurement operation for Width X. The measuring tool path is determined by the positional arguments (for starting and ending points, approach point, and measurement stroke) designated in the macro-call block. The only difference from the corresponding MAZATROL measurement is that the approach point can be designated as desired.

1. Programming

   G54 ......Coordinate system setting
   T1 T0 M6 ......[1] Change for the touch sensor
   G0 Xx Yy Zz ......[2] Intermediate point setting
   G136 Xx Yy ZZ Aa BB CC II Jj XK DD Uu Vv WW Q1 ......[3] Measurement of Width X
   G137 A1 B0 C0 V2.0 ......Correction and printout
2. Tool path

- **Tool change position**
- **Intermediate pt.** \((x_1, y_1, z_1)\)
- **Approach pt.** \((u, v, w)\)
- **Starting meas. pt.** \((x, y, z)\)
- **Ending meas. pt.** \((a, b, c)\)

**Rapid motion**

Approaching feed (Parameter K14)

Skip feed (K13) for the designated argument K

---

[1] Tool change for the touch sensor.
[2] Intermediate point before the approach point; programmed beforehand as required.
[3] Rapid traverse (G0 mode) to the approach point \((u, v, w)\).

The measurement macro controls the operation starting from the point [2] through the measuring movements for the starting and ending points up to the escape on the Z-axis to the approach point level.

3. Approach point, escape position, and measurement stroke

- The measurement macro controls the movements starting with the motion to the approach point and ending with the escape to the approach point level for each pattern.

- The position of the approach point also determines the level to be reached by the final escape as well as the level for the transfer movements from the first for the second measurement point. This also applies to the Y-axis in the relevant motions on the XY- or YZ-plane. Take due care in designating the approach point, therefore, to allow for a safety clearance for transfer.
1) Protrusion/Groove Width measurement

2) Step Height measurement

3) Outside/Inside Diameter measurement

4) Inside Protrusion/Groove Width measurement

5) Outside/Inside X measurement (on one side)

○: Starting meas. pt. (x, y, z)
●: Ending meas. pt. (a, b, c)
B. Tool measurement

There are no differences in tool measurement between MAZATROL and EIA programming types.

4. Remarks on G10.9 (Diameter/Radius data input)

A. Workpiece measurement

The designated data of starting and ending points and tolerances in the program, and the related data on the AUTO MEASURE display, are processed and indicated respectively as follows:

<table>
<thead>
<tr>
<th></th>
<th>OD/ID measurements</th>
<th>Other measurement patterns</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Starting pt.</td>
<td>TARGET DATA</td>
</tr>
<tr>
<td></td>
<td>Ending pt.</td>
<td>MEASURED DATA</td>
</tr>
<tr>
<td></td>
<td>Tolerance</td>
<td>OFFSET AMOUNT</td>
</tr>
<tr>
<td>MAZATROL</td>
<td>TARGET DATA</td>
<td>START DATA</td>
</tr>
<tr>
<td>G10.9X0</td>
<td></td>
<td>Tolerance</td>
</tr>
<tr>
<td></td>
<td>Diameter data</td>
<td>OFFSET AMOUNT</td>
</tr>
<tr>
<td>G10.9X1</td>
<td></td>
<td>Radius data</td>
</tr>
<tr>
<td>EIA</td>
<td>G10.9X0 (Radius data input)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G10.9X1 (Diameter data input)</td>
<td></td>
</tr>
</tbody>
</table>

Note 1: The “G10.9X0” (Radius data input) mode is established upon completion of the work-piece measurement.

Note 2: The K19 parameter functions as a radius value even in the OD/ID measurements.

B. Tool measurement

All types of tolerance data are processed as radius values. Preparatory function G10.9 is not available.
18 TWO-SYSTEM CONTROL FUNCTION

18-1 Two-Process Control by One Program: G109

1. Outline

When machining of different processes are performed by respective systems on a machine with two systems of headstock (HD1 and HD2), or turret (TR1 and TR2), the two systems can be controlled by a single program.

The program section from “G109L;” to “%” or to “G109L*;” is used for controlling the O-system.

2. Programming format

G109 L_;
L = 1 : HD1 (or TR1)
2 : HD2 (or TR2)

The system number is to be specified by a value following the address L.

3. Notes

1. Even if a value following L includes a decimal point or negative sign (–), it is ignored.
2. In the mode of single-block operation, the stop can be performed after execution of G109 block. However, when the number specified by L belongs to the other system such as L2 in HD1 operation, the single-block stop does not occur.
3. G109 can be specified in the same block as G-codes other than of group 0. When specified in the same block as another G-code of group 0, the G-code specified later is effective.
4. The section from the head of a program to the place where G109 is commanded is common to HD1 and HD2, or TR1 and TR2.

Example:

```
G28 X Z;  // Common to HD1 and HD2 (or TR1 and TR2)
G109 L1; // HD1 (or TR1)
  :  M30;
G109 L2; // HD2 (or TR2)
  :  M30;
  %  // Common to HD1 and HD2 (or TR1 and TR2)
```

5. One block including more than 128 characters causes an alarm (ILLEGAL FORMAT).
6. In the remainder of this chapter, “HD1” and “HD2” generally refer to “TR1” and “TR2”, respectively, at once.
7. The surface speed (cutting speed) of the respective turning spindles is to be specified, with reference to the G109 condition, as follows:

```
G96S__R1 ..........to specify the surface speed for the 1st spindle under “G109L1”
G96S__R2 ..........to specify the surface speed for the 2nd spindle under “G109L1”
G96S__R2 ..........to specify the surface speed for the 1st spindle under “G109L2”
G96S__R1 ..........to specify the surface speed for the 2nd spindle under “G109L2”
```
18-2 Specifying/Cancelling Cross Machining Control Axis: G110/G111

1. Outline

Axis control of HD2 side by HD1 side or that of HD1 side by HD2 side is referred to as cross machining control. Cross machining control axis is specified by G110 and G111. Specify after G110 an axis address and the HD number controlling the axis.

2. Programming format

G110 X_ Y_ Z_; .................. Cross machining control axis and HD number are specified.

1: Axis controlled by HD1
2: Axis controlled by HD2

G111;............................. Cross machining control axis specified by G110 is returned to normal control (not cross machining).

Example:
Operation at HD1 side
G110 X2;  .......... Changed to X-axis of HD2
G00 X10. Z10.; .......... X of HD2 moves to 10, Z of HD1 moves to 10.
G110 Z2;  .......... Changed to Z-axis of HD2
G00 X20. Z20.; .......... X of HD2 moves to 20, Z of HD2 moves to 20.
G110 X1 Z1;  .......... Changed to X-axis and Z-axis of HD1
G00 X30. Z30.;  .......... X of HD1 moves to 30, Z of HD1 moves to 30.

Specify the Z-axis for the lower turret as follows:

Example:
G110 Z2;  .......... Selection of the lower turret’s Z-axis
G00 Z100.;  .......... All the Z-axial commands between G110 and G111 are processed as those for the lower turret.
G111;  .......... Cancellation of G110

Specify the C-axis on the 2nd headstock side as follows:

Example:
G110 C2;  .......... Selection of the 2nd headstock’s C-axis
G00 C45.123.;  .......... All the C-axial commands between G110 and G111 are processed as those for the 2nd headstock side.
G111;  .......... Cancellation of G110
Prepare a program as follows to use the C-axis settings on the WORK OFFSET display for the 2nd spindle:

**Example:**

```
M200;    ________ Milling mode selection for 1st spindle
G28XZC;
T001T000M6;
G54:    ________ Origin data of the G54 system: C = 30°
G00 C150.; ________ HD1 C-axis motion to 150° (POSITION) or 180° (MACHINE)
M202;    ________ Milling mode cancellation for 1st spindle
M902:    ________ 2nd spindle selection
M300:    ________ Milling mode selection for 2nd spindle
G110 C2;    ________ Selection of 2nd spindle C-axis
G00 C150.; ________ HD2 C-axis motion to 150° (POSITION) or 180° (MACHINE)
G55:    ________ Origin data of the G55 system: C = 50°
G00 C150.; ________ HD2 C-axis motion to 150° (POSITION) or 200° (MACHINE)
G56:    ________ Origin data of the G56 system: C = 100°
G00 C150.; ________ HD2 C-axis motion to 150° (POSITION) or 250° (MACHINE)
G111;    ________ Cancellation of G110
M302:    ________ Milling mode cancellation for 2nd spindle
```

Prepare a program as follows to use a fixed cycle for hole machining on the 2nd spindle side:

**Example:**

```
M902;    ________ 2nd spindle selection
M300;    ________ Milling mode selection for 2nd spindle
G110 C2;    ________ Selection of 2nd spindle C-axis
G00 C0.;    ________ HD2 C-axis positioning
G287Z–5.0X5.0P0.2M310;—— Clamping; Deep-hole drilling cycle
C45.;    ________ Unclamping, positioning, clamping; Deep-hole drilling cycle
C90.;    ________ Unclamping, positioning, clamping; Deep-hole drilling cycle
M312;    ________ Unclamping on the 2nd spindle side
G80;    ________ Fixed cycle cancellation
G111;    ________ Cancellation of G110
M30;    ________ Program end
```

### 3. Sample programs

Examples of programming for the machine specifications with the lower turret.

The major sections of a sample program for machines equipped with the lower turret are shown below.

```
O1234
G54    Coordinate system selection
#101=124.750 (SP1 COF) 1st spindle side C-axis offset
#102=10.664 (SP2 COF) 2nd spindle side C-axis offset
```
(MAIN SPINDLE SIDE) 1st spindle side machining program
G109L1
M901 1st spindle select mode (enter for machining at the 1st spindle side)
G92S3000 Spindle clamping speed setting
M202 1st spindle turning mode
G00W0. 2nd spindle side W-axis positioning
G91G00G28X0Y0Z0 1st spindle return to zero point (X, Y, Z)
T001T000M6 Tool selection
N101 (EDG-R)
G96S200 Surface speed setting
G00X0.0Y0.1 Positioning
G95G01X22.0F0.3 Cutting feed
G00Z0.8 Positioning
N102 (OUT-R)
G91G00G28X0Y0Z0 O.D. machining with 1st spindle
T001T000M6 (Machining program omitted for convenience's sake.)

(TRS CHK)
G91G00G28X0Y0Z0 Transfer program
M902 1st spindle return to zero point (X, Y, Z)
M302 2nd spindle selection
M200 (MAIN C-ON) 2nd spindle turning mode
G00C#101 1st spindle mill-point machining mode
G00C#102 1st spindle C-axis positioning (angle indexing)
M300 (SUB C-ON) 2nd spindle mill-point machining mode
G110C2 2nd spindle C-axis selection
G00C#102 2nd spindle C-axis positioning (angle indexing)
G111 2nd spindle C-axis selection revoking (G110 cancellation)
M306 2nd spindle chuck open
M540 TRS-CHK mode
G00W-686. 2nd spindle side W-axis positioning
G508 Start of pressing action on the 2nd spindle side
G91G31W-1.1F50 2nd spindle side W-axis positioning for pressing
M202 1st spindle turning mode
M509 2nd spindle M508 cancellation
M541 TRS-CHK mode cancellation
M307 2nd spindle chuck close
M206 1st spindle chuck open
M302 2nd spindle turning mode
G90G00W0. 2nd spindle side W-axis positioning

(SUB SPINDLE SIDE) 2nd spindle machining program
N301 (SP2 DRL) 2nd spindle selection (enter for machining at the 2nd spindle side)
M902 Tool selection
T003T000M6 Feed per minute and cancellation of constant surface speed control
G94G97 2nd spindle mill-point machining mode
M300 Milling speed selection and milling spindle normal rotation
M03S3184 G110C2 2nd spindle C-axis selection
G00C#102 2nd spindle C-axis positioning (angle indexing)
M310 2nd spindle C-axis clamping
G00X25.Z-5. Positioning
G287Z-5.X5.Q5000P0.2F200 Longitudinal deep-hole drilling cycle
M312 2nd spindle C-axis unclamping
G90 2nd spindle C-axis positioning (angle indexing)
M310 Cancellation of fixed hole-drilling cycle
G00C[#102+180.] 2nd spindle C-axis clamping
M310 Longitudinal deep-hole drilling cycle
G287Z-5.X5.Q5000P0.2F200 2nd spindle C-axis unclamping
M312 Cancellation of fixed hole-drilling cycle
G80 2nd spindle C-axis selection revoking
G111 Return to zero point (X, Y, Z)
G91G28X0Y0Z0 End of program
M30
4. Notes

1. G110 and G111 must always be given in a single-command block.

2. When a value following the axis address in the G110 block includes a decimal point or negative sign, it is ignored.

3. In the single-block operation mode, the stop is performed after execution of G110 and G111 blocks.

4. The tool information to be used in tool offsetting does not automatically change for the other system on the occasion of designating for cross machining control an axis which is in direct relation to tool movement. Use, therefore, a G53 command (for positioning in the machine coordinate system) as required.

5. As long as an axis in direct relation to tool movement is controlled for cross machining, do not change tools (by M6).

6. When the axis is changed by G110, the counterpart system must be in a state of automatic starting and standby.

   **State of standby**

   M-codes from M950 to M997 are used for waiting. When both HD1 and HD2 are operated and when machining is performed with HD1 and HD2 synchronized, M950 to M997 is used.

   A state of standby refers to the time before the same waiting M-code is outputted from the counterpart.

   For example, when M950 is outputted from HD1, HD1 is in a state of standby until M950 is outputted from HD2. (HD1 does not execute blocks subsequent to M950.) When M950 is outputted from HD2, HD1 executes the block following M950.

   **Program example**

   ```plaintext
   HD1                      HD2
   M950;                    M950;
   G110 X2;                 M951;
   X....                    X.....Z...
   X.....Z...
   ...
   M951;
   ```

   Indicates the waiting time for which HD2 is in a state of standby when X-axis of HD2 is controlled by HD1.

7. Give a command of G111 as required at the end of machining section in an EIA/ISO program which is to be called from a MAZATROL program as a subprogram for cross machining.

8. The axis being under cross machining control in automatic mode of operation cannot be controlled in manual mode. An attempt to do so will only result in the alarm **ILLEGAL COMMAND CROSS MACHINING**.

9. Barrier is effective also during axis change. In other words, barrier is checked in the region of HD1 side for the axis of HD1 side and in that of HD2 for the axis of HD2 independently of the axis change by G110.

10. Synchronous feed with, or control of feed per, revolution of the milling spindle is not available during cross machining control.
11. The alarm **CROSS MACHINING IMPOSSIBLE** will be caused when a command for cross machining control is given under one of the following incompatible modal conditions:
   - Nose R/Tool radius compensation
   - Polar coordinate interpolation
   - Cylindrical interpolation
   - Fixed cycle
   - 3-D coordinate conversion
   - Mirror image
   - Tool tip point control

12. C-axis commands in the cross machining mode can only be given for the preparatory functions (G-codes) enumerated below.

**Usable G-codes for C-axis commands in the cross machining mode**

<table>
<thead>
<tr>
<th>G-code</th>
<th>Group</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01</td>
<td>Rapid positioning</td>
</tr>
<tr>
<td>G01</td>
<td>01</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td>01</td>
<td>Circular interpolation CW</td>
</tr>
<tr>
<td>G03</td>
<td>01</td>
<td>Circular interpolation CCW</td>
</tr>
<tr>
<td>G10</td>
<td>00</td>
<td>Data setting/change</td>
</tr>
<tr>
<td>G27</td>
<td>00</td>
<td>Reference point return check</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>Reference point return</td>
</tr>
<tr>
<td>G29</td>
<td>00</td>
<td>Return from reference point</td>
</tr>
<tr>
<td>G30</td>
<td>00</td>
<td>Return to 2nd/3rd/4th reference point</td>
</tr>
<tr>
<td>G92</td>
<td>00</td>
<td>Coordinate system setting/Spindle limit speed setting</td>
</tr>
<tr>
<td>G53</td>
<td>00</td>
<td>Machine coordinate system selection</td>
</tr>
<tr>
<td>G65</td>
<td>00</td>
<td>Macro call</td>
</tr>
<tr>
<td>G66</td>
<td>14</td>
<td>Macro modal call</td>
</tr>
<tr>
<td>G283</td>
<td>09</td>
<td>Face drilling cycle</td>
</tr>
<tr>
<td>G284</td>
<td>09</td>
<td>Face tapping cycle</td>
</tr>
<tr>
<td>G284.2</td>
<td>09</td>
<td>Face synchronous tapping cycle</td>
</tr>
<tr>
<td>G285</td>
<td>09</td>
<td>Face boring cycle</td>
</tr>
<tr>
<td>G287</td>
<td>09</td>
<td>Outside drilling cycle</td>
</tr>
<tr>
<td>G288</td>
<td>09</td>
<td>Outside tapping cycle</td>
</tr>
<tr>
<td>G288.2</td>
<td>09</td>
<td>Outside synchronous tapping cycle</td>
</tr>
<tr>
<td>G289</td>
<td>09</td>
<td>Outside boring cycle</td>
</tr>
<tr>
<td>G110</td>
<td>00</td>
<td>Cross machining control axis selection</td>
</tr>
<tr>
<td>G111</td>
<td>00</td>
<td>Cross machining control axis cancellation</td>
</tr>
<tr>
<td>G112</td>
<td>00</td>
<td>M-, S-, T-, and B-code output to counterpart</td>
</tr>
</tbody>
</table>

13. When the axis (normally the X-axis) relevant to the constant surface speed control is designated for cross machining control, the speed of the turning spindle may change steeply in accordance with the change in positional information to be used in the calculation of spindle speed for a particular surface speed.

14. Do not fail to give the cancellation command G111 after the required cross machining.
18-3 M, S, T, B Output Function to Counterpart: G112

1. Outline
The function outputs M-, S-, T- and B-codes (second miscellaneous function) commanded after G112 to the counterpart system.

2. Programming format
G112 L_M_M_M_M_S_T_T_B;

Example 1: With an argument L specified.
<Given in a section for System 2>
G109L2;
;
G112 L1M3S1000; ..........Normal rotation of the upper turret's milling spindle.

Example 2: With an argument L omitted.
<Given in a section for System 1, with BA71 = 1 for that system>
G109L1;
;
G112 M3S1000; ................Normal rotation of the lower turret's milling spindle.

3. Notes
1. Do not give any other G-code in one block with a G112 command; otherwise the alarm ILLEGAL FORMAT will be caused.
2. Do not enter any codes concerned (M, S, T or second miscellaneous function) before G112L_ within a block; otherwise the alarm ILLEGAL FORMAT will be caused.
3. Entering values with any other address than N, M, S, T, and that for second miscellaneous function in one block with a G112 command will lead to the alarm ILLEGAL ADDRESS.
4. The alarm ILLEGAL NUMBER INPUT will be caused if any of the following commands is given in one block with a G112 command:
   M0, M1, M2, M30, M99, M-codes for waiting, and M-, S-, T- or second miscellaneous function code for macroprogram call.
5. Entering a number for the self-system or non-existent system with address L as well as in parameter BA71 will lead to the alarm ILLEGAL NUMBER INPUT.
6. The number of the codes concerned to be entered in a G112 block is limited as follows:
   4 for M, 1 for S, 2 for T, and 1 for the second miscellaneous function.
   Entering codes in excess will only result in the last ones within the limit being outputted.
7. The single-block stop can occur after the execution of a G112 block.
8. Use waiting M-codes so as to output the codes concerned (M, S, T or second miscellaneous function) to one and the same system at one time from multiple systems.
19 COMPOUND MACHINING FUNCTIONS

This chapter describes the functions proper to the machines equipped with two turrets (upper and lower) which can be operated independently from each other.

19-1 Programming for Compound Machining

1. Outline

The movement of the upper and lower turrets is to be controlled in a single program as follows:

G109 L1;....................................Selection of the upper turret

Commands for the upper turret

M30;

G109 L2;....................................Selection of the lower turret

Commands for the lower turret

M30;

2. Remarks

1. If an argument L includes a decimal point or negative sign (–), a programming error will result.

2. In the mode of single-block operation, the stop can be performed after execution of G109 block. However, when the number specified by L belongs to another system, the single-block stop does not occur.

3. G109 must be given in an independent block. If any other command is given in the same block, a programming error will result.

4. Note that the program section under no specification by the G109 command is used for all the systems without distinction.

5. The restart position for the [RESTART 2 NONMODAL] menu function must be set within a program section which is prepared commonly for all the systems.

6. The control for a constant surface speed (by G96) is always conducted with reference to that tool tip’s position of either turret which is nearer to the axis of turning.

7. The call command for a MAZATROL program must be given in program sections of both turrets for one and the same program. If it is given for either turret only, the flow of the called MAZATROL program will enter in a waiting state which cannot be cleared and, as a result, stop the machine operation.
19-2 Waiting Command: M950 to M997, P1 to P99999999

1. Outline

Waiting commands are used to time the operation of the upper and lower turrets as required. Two types of waiting command are provided: M-code and P-code, which can be used freely and even mixedly.

Note: Waiting commands (corresponding M- and P-codes) are all ignored in a hole machining fixed cycle mode (G283 to G289). Cancel the hole machining fixed cycle mode beforehand with G80 to give a waiting command.

2. Detailed description

A. M-codes for waiting

The execution of the commands for turret A will be stopped at the position of a waiting M-code with some number until the program flow for turret B reaches a waiting M-code with the same number.

Programming format

M***; (*** denotes a number from 950 to 997.)

Program structure

Commands for the upper turret

G109L1;

A

M950;

B

M951;

M997;

C

M30;

Commands for the lower turret

G109L2;

M950;

M951;

A

M997;

B

M997;

C

M30;

Operation

Upper turret

A

B

C

Lower turret

A

B

C

Note: A waiting M-code must be given in a single-command block. It may not function as waiting command if it is entered in the same block together with other instructions.
B. P-codes for waiting

The execution of the commands for turret A will be stopped at the position of a waiting P-code with some number until the program flow for turret B reaches a waiting P-code with the same or a larger number.

**Programming format**

P********; (******** denotes a number from 1 to 99999999.)

**Program structure**

<table>
<thead>
<tr>
<th>Commands for the upper turret</th>
<th>Commands for the lower turret</th>
</tr>
</thead>
<tbody>
<tr>
<td>G109L1;</td>
<td>G109L2;</td>
</tr>
<tr>
<td>A</td>
<td>P10;</td>
</tr>
<tr>
<td>P10;</td>
<td>P10;</td>
</tr>
<tr>
<td>B</td>
<td>P100;</td>
</tr>
<tr>
<td>P200;</td>
<td>P10;</td>
</tr>
<tr>
<td>P3000;</td>
<td>P100;</td>
</tr>
<tr>
<td>C</td>
<td>P3000;</td>
</tr>
<tr>
<td>M30;</td>
<td>M30;</td>
</tr>
</tbody>
</table>

**Operation**

A waiting P-code must be given in a single-command block. It may not function as waiting command if it is entered in the same block together with other instructions.

**Note 1:** Use the waiting P-codes in the ascending order of their number, since one turret cannot be released from the wait state until the program flow for the other turret reaches a waiting P-code with the same or a larger number.
19-3 Balanced Cutting

1. Outline

Balanced cutting is achieved through the symmetrical movement of the upper and lower turrets. It helps the reduction in the vibration of a long workpiece and permits the cutting speed to be doubled for the saving of the machining time. During the balanced cutting one turret acts as the main turret (master turret) and the other as the subordinate turret (servant turret). Enter the movement commands for the balanced cutting in a program section for the main turret.

2. Programming method

The balanced cutting can be achieved by combining the following three commands:
- Waiting command (M950 to M997 or P1 to P99999999)
- M562;........Coupling command for the two turrets
- M563;........Coupling cancellation command

The main points of programming the balanced cutting are the following:
1) Enter the waiting command just before the balanced cutting in order to synchronize the movement of both turrets.
2) Enter the command M562 for the main turret in order to couple both turrets. The subordinate turret must have been set in wait state.
3) Enter the movement commands for the main turret. The subordinate turret will be moved symmetrically during the balanced cutting.
4) Enter the command M563 after the movement commands for the master turret to cancel the coupling.
5) Enter the waiting command for the main turret to release the subordinate turret from the wait state.

3. Program structure

Given below is an example of program structure with the upper turret as the master.

Commands for the upper turret                  Commands for the lower turret
G109L1;                                      G109L2;

Waiting for the start of balanced cutting
P1000;                                    P1000;
M562;                                      P2000;

Start of coupling

Commands for balanced cutting
M563;                                      M30;
Cancellation of coupling
P2000;                                    M30;

Waiting for the end of balanced cutting
4. Sample program

N000 G109 L1;
M901;
N001 G00 X800.Z70.;
P10;
M03 S250;
T001T000M06D001;
N002 X132.Z60.M08;
M950;
M562;
N003 G01 X78.F0.35;
N004 G00 X156.Z63.;
N005 Z29.;
N006 G01 X150.;
N007 X148.Z30.;
N008 X128.;
N009 G00 X800.Z70.;
N100 G109 L2;
N101 G00 X800.Z200.;
P10;
M03 S250;
T001001;
N102 X92.Z65.M08;
M950;
P20;
N103 M09 M05;
N104 G00 X800.Z70.;
N105 Z29.;
N106 G00 X150.;
N107 X148.Z30.;
N108 X128.;
N109 G00 X800.Z70.;
M563;
P20;
N015 M09 M05;
N100 G109 L2;
M901;
N101 G00 X800.Z200.;
P10;
M03 S250;
T001001;
N102 X92.Z65.M08;
M950;
P20;
19-4 Milling with the Lower Turret

1. Programming format

The basic format of programming for milling with the lower turret is an application of the preparatory function G109 (Two processes in one program; see Section 18-1).

G109 L;
    L = 1: HD1 (TR1)
        2: HD2 (TR2)

Example:
G28 X Z; ................ Common to both spindles (both turrets)
G109 L1;
    .............. Commands for Upper turret
    ..............
G109 L2;
    .............. Selection of 2nd spindle (Lower turret)
    M200;
    M203;
    .............. Commands for Lower turret
    M210;
    M30
% ..................... Common to both spindles (both turrets)

<Usable machining patterns>

As shown in the table below, not only for turning can the lower turret be used, but also for milling.

Table 19-1 Machining patterns

<table>
<thead>
<tr>
<th>Lower turret</th>
<th>1st spindle</th>
<th>2nd spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turning</td>
<td>Milling</td>
<td>Turning</td>
</tr>
</tbody>
</table>

2. G-codes for milling

The G-codes of fixed cycle for hole machining are available for milling with the lower turret.
(See Section 13-3 for more information on the above G-codes.)
3. Sample program

N000 G00 G97 G98;
N001 G91 G28 X Z;
N100 G109 L1;  \(\text{Upper turret selection}\)
N101 T001T000M6D001;
N102 M901;  \(\text{1st spindle selection}\)
N103 M200;  \(\text{Point milling mode}\)
N104 M03 S800;  \(\text{Normal rotation of the milling spindle}\)
N105 G90 X102.Z-50.C0.;
N107 G80;
N108 M950;  \(\text{M950 for waiting}\)
N109 M30;
N200 G109 L2;
N201 T102022;
N202 M902;  \(\text{Lower turret selection}\)
N203 M300;
N204 M03 S800;
N207 G80;
N208 M950;
N209 M30;
## 19-5 Compound Machining Patterns

### 1. Overview


*1 M901 for the 1st spindle selection.
*2 M902 for the 2nd spindle selection.
*3 M200/M300 for milling mode selection for 1st/2nd spindle.
*4 M03 for milling spindle’s normal rotation.
*5 Machining data

### 2. Machining pattern list

**<Single workpiece>**

- **Separate machining (with either turret)**
  
<table>
<thead>
<tr>
<th>EIA</th>
<th>1st spindle</th>
<th>2nd spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Turning</td>
<td>Milling</td>
</tr>
<tr>
<td>Upper turret</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Lower turret</td>
<td>O</td>
<td>O</td>
</tr>
</tbody>
</table>

- **Parallel machining (on either spindle side with both turrets)**

<table>
<thead>
<tr>
<th>EIA</th>
<th>Upper turret</th>
<th>1st spindle</th>
<th>2nd spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Turning</td>
<td>Milling</td>
<td>Turning</td>
</tr>
<tr>
<td></td>
<td>O</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Milling</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lower turret</td>
<td>Turning</td>
<td>O (Note)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Miling</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Turning</td>
<td></td>
<td>O</td>
</tr>
<tr>
<td></td>
<td>Miling</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** Simultaneous milling is possible in the EIA programming format, indeed, but take care of a phase difference occurring for machine structural reasons.
<Dual workpiece>
Parallel machining (on both spindle sides with each turret)

<table>
<thead>
<tr>
<th>EIA</th>
<th>1st spindle</th>
<th>2nd spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Turning</td>
<td>Turning</td>
</tr>
<tr>
<td>Upper turre</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Turning</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>Milling</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>Lower turre</td>
<td>1st spindle</td>
<td>2nd spindle</td>
</tr>
<tr>
<td>Turning</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Milling</td>
<td>O</td>
<td>O</td>
</tr>
</tbody>
</table>

O: Possible  ---: Inapplicable
<table>
<thead>
<tr>
<th>No.</th>
<th>Machining pattern</th>
<th>Programming example</th>
</tr>
</thead>
</table>
| 1   | Upper turret — 1st spindle; Turning, Separate | G109L1  
M901  
M202  
M203 S000  
: Machining data  
M205  
M950  
M30  
G109L2  
M950  
M30 |
|     | ![Diagram 1](image1.png) | ![Diagram 2](image2.png) |
| 2   | Lower turret — 2nd spindle; Turning, Separate | G109L1  
M902  
M203 S000  
: Machining data  
M305  
M950  
M30  
G109L2  
M950  
M30 |
|     | ![Diagram 3](image3.png) | ![Diagram 4](image4.png) |
| 3   | Upper turret — 1st spindle; Milling, Separate | G109L1  
M901  
M200  
M03 S000  
: Machining data  
M05  
M202  
M950  
M30  
G109L2  
M950  
M30 |
|     | ![Diagram 5](image5.png) | ![Diagram 6](image6.png) |
| 4   | Upper turret — 2nd spindle; Milling, Separate | G109L1  
M902  
M300  
M03 S000  
: Machining data  
M05  
M302  
M950  
M30  
G109L2  
M950  
M30 |
<table>
<thead>
<tr>
<th>No.</th>
<th>Machining pattern</th>
<th>Programming example</th>
</tr>
</thead>
</table>
| 5   | Upper turret — 2nd spindle; Turning, Separate | G109L1  
M902  
M302  
M303  
M303  
M303  |
|     |                   | G109L2  
M950  
M30 |
| 6   | Lower turret — 2nd spindle; Milling, Separate | G109L1  
M950  
M30 |
|     |                   | G109L2  
M950  
M30 |
| 7   | Lower turret — 1st spindle; Milling, Separate | G109L1  
M950  
M30 |
|     |                   | G109L2  
M901  
M200  
M03  |
| 8   | Lower turret — 1st spindle; Turning, Separate | G109L1  
M950  
M30 |
|     |                   | G109L2  
M901  
M202  
M203  |

Machining data:
<table>
<thead>
<tr>
<th>No.</th>
<th>Machining pattern</th>
<th>Programming example</th>
</tr>
</thead>
</table>
| 9   | Upper turret — 1st spindle; Turning, Lower turret — 2nd spindle; Turning. | \[
\begin{align*}
&G109L1 \\
&M901 \\
&M202 \\
&M203 \text{M}00 \\
&M205 \\
&M950 \\
&M30
\end{align*}
\] | \[
\begin{align*}
&G109L2 \\
&M902 \\
&M302 \\
&M303 \text{M}00 \\
&M305 \\
&M950 \\
&M30
\end{align*}
\] |
| 10  | Upper turret — 1st spindle; Turning, Lower turret — 2nd spindle; Milling. | \[
\begin{align*}
&G109L1 \\
&M901 \\
&M202 \\
&M203 \text{M}00 \\
&M205 \\
&M950 \\
&M30
\end{align*}
\] | \[
\begin{align*}
&G109L2 \\
&M902 \\
&M300 \\
&M303 \text{M}00 \\
&M305 \\
&M950 \\
&M30
\end{align*}
\] |
| 11  | Upper turret — 1st spindle; Milling, Lower turret — 2nd spindle; Turning. | \[
\begin{align*}
&G109L1 \\
&M901 \\
&M200 \\
&M03 \text{M}00 \\
&M05 \\
&M202 \\
&M950 \\
&M30
\end{align*}
\] | \[
\begin{align*}
&G109L2 \\
&M902 \\
&M302 \\
&M303 \text{M}00 \\
&M305 \\
&M950 \\
&M30
\end{align*}
\] |
<table>
<thead>
<tr>
<th>No.</th>
<th>Machining pattern</th>
<th>Programming example</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>Upper turret — 1st spindle; Milling, Lower turret — 2nd spindle; Milling.</td>
<td>G109L1 M901 M200 M03 S000 M05 M202 M950 M30</td>
</tr>
<tr>
<td></td>
<td></td>
<td>: Machining data</td>
</tr>
<tr>
<td>13</td>
<td>Upper turret — 2nd spindle; Turning, Lower turret — 1st spindle; Turning.</td>
<td>G109L1 M902 M302 M303 S000 M305 M950 M30</td>
</tr>
<tr>
<td></td>
<td></td>
<td>: Machining data</td>
</tr>
<tr>
<td>14</td>
<td>Upper turret — 2nd spindle; Milling, Lower turret — 1st spindle; Turning.</td>
<td>G109L1 M902 M300 M03 S000 M05 M302 M950 M30</td>
</tr>
<tr>
<td></td>
<td></td>
<td>: Machining data</td>
</tr>
<tr>
<td>No.</td>
<td>Machining pattern</td>
<td>Programming example</td>
</tr>
<tr>
<td>-----</td>
<td>-------------------</td>
<td>---------------------</td>
</tr>
</tbody>
</table>
| 15  | Upper turret — 2nd spindle; Turning, Lower turret — 1st spindle; Milling. | ![Diagram](image1.png)  
|     |                   | G109L1              | G109L2              |
|     |                   | M902                 | M901                 |
|     |                   | M302                 | M200                 |
|     |                   | M303 S000            | M03 S000             |
|     |                   | : Machining data     | : Machining data     |
|     |                   | M305                 | M05                  |
|     |                   | M950                 | M202                 |
|     |                   | M30                  | M950                 |
|     |                   |                      | M30                  |
| 16  | Upper turret — 2nd spindle; Milling, Lower turret — 1st spindle; Milling. | ![Diagram](image2.png)  
|     |                   | G109L1              | G109L2              |
|     |                   | M902                 | M901                 |
|     |                   | G28XZ                | T003000              |
|     |                   | T014000T0 M6         | M200                 |
|     |                   | M300                 | M300                 |
|     |                   | M03 S000             | M03 S000             |
|     |                   | M950                 | M950                 |
|     |                   | G110C2M951           | G110C2M951           |
|     |                   | M952                 | G110C1M952           |
|     |                   | G00C90.              | G00C90.              |
|     |                   | G111                 | G111                 |
|     |                   | G00X100.Z0.          | G00X100.Z-10.        |
|     |                   | G01Z-50.F100         | G01X50.F100          |
|     |                   | G00X120.             | G00Z10.              |
|     |                   | Z0.                  | X100.                |
|     |                   |                      |                      |
|     |                   | M05                  | M05                  |
|     |                   | M202                 | M202                 |
|     |                   | M953                 | M953                 |
|     |                   | M30                  | M30                  |

**Note:** Give commands of cross machining control (G110) successively for the 1st and 2nd spindles.
<table>
<thead>
<tr>
<th>No.</th>
<th>Machining pattern</th>
<th>Programming example</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>Upper turret — 1st spindle; Turning, Lower turret — 1st spindle; Turning.</td>
<td>G109L1 M901 M202 M203 S OOO M950 : Machining data M951 M205 M952 M30</td>
</tr>
<tr>
<td>18</td>
<td>Upper turret — 2nd spindle; Turning, Lower turret — 2nd spindle; Turning.</td>
<td>G109L1 M902 M950 : Machining data M951 M305 M952 M30</td>
</tr>
<tr>
<td>19</td>
<td>Upper turret — 1st spindle; Milling, Lower turret — 1st spindle; Milling.</td>
<td>G109L1 M901 M950 M200 M951 M03 S OOO : Machining data M05 M202 M952 M30</td>
</tr>
<tr>
<td>No.</td>
<td>Machining pattern</td>
<td>Programming example</td>
</tr>
<tr>
<td>-----</td>
<td>--------------------------------------------------------</td>
<td>---------------------</td>
</tr>
<tr>
<td>20</td>
<td>Upper turret — 2nd spindle; Milling, Lower turret — 2nd spindle; Milling.</td>
<td></td>
</tr>
</tbody>
</table>

- G109L1
- M902
- M950
- M300
- M951
- M03 S000

  : Machining data

- M05
- M302
- M952
- M30
20 POLYGONAL MACHINING AND HOBBING (OPTION)

20-1 Polygonal Machining ON/OFF: G51.2/G50.2

1. **Function and purpose**

A workpiece is machined in a polygonal shape by turning the rotary tool at a constant rate to the workpiece at a given rotating speed. The shape to be machined depends on the following conditions:

- The number of the edges of a rotary tool
- The ratio of the rotating speed of a workpiece to that of a rotary tool

Polygonal machining has an advantage of machining polygonal workpieces in shorter time than polar coordinate interpolation. However, it has a disadvantage of not giving an accurate polygon. As a result, polygonal machining is usually used to machine bolt heads and nuts not requiring an accurate polygon.

D732S0036

Machining shape by polygonal machining (hatched section)
2. Programming format

Starting polygonal machining

G51.2 P_ Q_ D_; 

- Give a command so that addresses P and Q provide the following:
  (Address P): (Address Q) = (Workpiece rotational speed) : (Rotary tool speed)
- Command the rotational direction of rotary tool with the sign of address Q as follows.
  When the sign of Q is “+”, positive direction is selected.
  When the sign of Q is “−”, negative direction is selected.
- Use address D to specify the workpiece spindle to be used. The arguments D available, 1 to 4, refer to the settings of machine parameters (BA55 to BA58) as follows:

<table>
<thead>
<tr>
<th>Argument D</th>
<th>Workpiece spindle selected</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>As set by parameter BA55</td>
</tr>
<tr>
<td>2</td>
<td>As set by parameter BA56</td>
</tr>
<tr>
<td>3</td>
<td>As set by parameter BA57</td>
</tr>
<tr>
<td>4</td>
<td>As set by parameter BA58</td>
</tr>
</tbody>
</table>

- The default value of argument D is “1” (selection of workpiece spindle as set by BA55).
  An alarm will be caused (809 ILLEGAL NUMBER INPUT) if the specified argument D is not supported as appropriate by the parameter setting concerned. (For example: D3 or D4 is entered with the parameters BA57 and BA58 being set to “−1” [Invalid].)
- The command range of addresses P, Q and D is as follows:

<table>
<thead>
<tr>
<th>Address</th>
<th>Command range</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>1 to 9</td>
</tr>
<tr>
<td>Q</td>
<td>−9 to −1, 1 to 9</td>
</tr>
<tr>
<td>D</td>
<td>1 to 4</td>
</tr>
</tbody>
</table>

Command arguments P, Q and D with integers. They cannot be commanded with a value including decimal fraction.

The designation of an address other than P, Q and D in a G51.2 block leads to an alarm (806 ILLEGAL ADDRESS), which will also be caused by designating any address in the G50.2 block.

Canceling polygonal machining

G50.2;
3. Sample program

```
G91 G28 X0 Z0;
T11T00 M06;
G90 G94;
M260;
M203 S250;
G51.2 P1 Q-2 D1;
G0 X100.Z30.;
G0 X46.6 Z3.;
G1 Z-20.F50;
G1 X60.F100;
G0 Z3.;
G0 X46.0;
G1 Z-20.F30;
G1 X60.F100;
G0 X100.Z30.;
G50.2; M261; M205; M30;
```

Selection of tool No. 11 for polygonal machining
Mode of feed per minute
Polygonal machining mode selection
Normal rotation of turning spindle at 250 rpm
Reversed rotation of milling spindle at 500 rpm
Machining
Polygonal machining mode cancellation
Polygonal machining mode cancellation
Milling spindle stop
Turning spindle stop
End

4. Notes

1. Take care not to program in the following manners (otherwise the alarm 807 ILLEGAL FORMAT will be caused):
   - Giving G51.2/G50.2 with another command in the same block,
   - Giving G51.2 in the mode of G96 (control for constant surface speed),
   - Giving G51.2 in the mode of G51.2,
   - Selecting the milling mode in the G51.2 mode, or giving G51.2 in the milling mode,
   - Giving a synchronous tapping command in the mode of G51.2,
   - Giving a command for spindle synchronization, orientation, or linkage in the G51.2 mode,
   - Giving a fixed cycle command in the mode of G51.2, and vice versa.

2. Command a proper workpiece rotating speed and ratio of such workpiece rotating speed to the rotary tool speed so that the maximum rotating speed of rotary tool cannot be exceeded.

3. Move command such as one for general control axis cannot be given to the rotary tool axis except the command of reference point return.

4. A machine coordinate value of rotary tool axis is displayed within a range from 0 to “movement distance per rotation”. Relative coordinates and absolute coordinates are not renewed.

5. An absolute position detector cannot be mounted on the rotary tool axis.

6. Jogging feed and handle feed for the rotary tool axis are ineffective during polygonal machining.

7. Performing thread cutting during polygonal machining makes the start point of thread cutting irregularly shifted. Therefore, cancel the polygonal machining before thread cutting.

8. Rotary tool axis during polygonal machining is not counted as a synchronous control axis.

9. During polygonal machining, it is possible, indeed, but not advisable at all to apply feed hold or to change the override value for fear of deformation of the workpiece.
10. Avoid applying an emergency stop or resetting during polygonal machining for fear of deformation of the workpiece due to a sudden stop of axis motions.

11. The alarm **956 RESTART OPERATION NOT ALLOWED** will be caused when it is attempted to start modal restart operation from a block in the G51.2 mode.

12. The gear for rotary tool, if provided, must be taken into account in setting the ratio of milling spindle speed to spindle speed (Q : P).

13. Since phase matching between turning and milling spindles cannot be obtained, it is not possible to start machining with the workpiece in a specific angular position.

14. Give the required commands for multiple cutting steps (e.g. roughing and finishing) in one and the same program section of the G51.2 mode. If the mode is canceled, it is in general no more possible to obtain the same machining surface.

15. The spindle speed can be changed in the mode of polygonal machining without fear of disturbing the phase of machining surface.
20-2 Selection/Cancellation of Hob Milling Mode: G114.3/G113

1. Outline

A synchronization control of the milling spindle and the C-axis allows them to be used as the hob spindle and the workpiece spindle, respectively, and thus enables the turning machine to generate spur and helical gears on a level with a hob milling machine. The hob milling function, however, is only available to machines equipped with the control functions of the C-, B- and Y-axis.

2. Programming format

G114.3 D±_E_L_P_Q_R_; Start of hobbing

D .......Selection of workpiece spindle and its rotational direction
        ±1: C-axis of the first spindle
        ±2: C-axis of the secondary spindle
        “+” for a rotation of the workpiece spindle in the same direction as the hob spindle.
        “−” for a rotation of the workpiece spindle in the reverse direction to the hob spindle.

E .......Number of threads of the hob

L........Number of teeth on the gear

P .......Helix angle
        Specify the desired helix angle for a helical gear.
        Omit the argument, or specify 0 (degree) for a spur gear.

Q ......Module or Diametral pitch
        Specify the normal module, or diametral pitch, for a helical gear. Set a negative value (with a minus sign) to use a hob cutter with left-hand teeth.
        Enter the module for metric specification.
        Enter the diametral pitch for inch specification.

R ......Angle of phase shift
        Specify the angle for phase matching between the hob spindle (milling spindle) and the workpiece spindle (C-axis).
        The specified angle refers to the initial rotation (angular positioning) of the hob spindle after completion of the zero-point return of the hob and workpiece spindles as a preparation for the synchronization control.

G113; Cancellation of the hob milling mode
        The synchronization control of the hob spindle and the workpiece spindle is canceled.
- The setting range and default value for each argument are as follows:

<table>
<thead>
<tr>
<th>Address</th>
<th>Setting range</th>
<th>Default value</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td>±1, ±2</td>
<td>+1</td>
</tr>
<tr>
<td>E</td>
<td>0 to 20</td>
<td>1</td>
</tr>
<tr>
<td>L</td>
<td>1 to 9999</td>
<td>1</td>
</tr>
<tr>
<td>P</td>
<td>−90.000 to 90.000 [deg]</td>
<td>0 (Spur gear)</td>
</tr>
<tr>
<td>Q</td>
<td>±100 to ±25000 [0.001 mm or 0.0001 inch⁻¹]</td>
<td>Ommision of Q causes an alarm if a significant argument P is specified in the same block.</td>
</tr>
<tr>
<td>R</td>
<td>0 to 359.999 [deg]</td>
<td>No phase matching</td>
</tr>
</tbody>
</table>

- The argument D leads to an alarm if a value outside the setting range is specified.
- The workpiece spindle does not rotate with the argument E (Number of hob threads) set to “0”. Accordingly, the designation of argument R for phase matching is not effective.
- The argument Q is ignored if the argument P is not specified in the same block.

3. Sample program

A. Generating a spur gear (without phase matching)

```plaintext
M200;  Selection of the milling mode.
M03S0;  Start of milling spindle normal rotation at a speed of zero.
M250;   Unclamping of the B-axis.
G00B92.8;  B-axis rotation through the lead angle (92.8°) of the hob cutter.
M251;   Clamping of the B-axis.
G00X40.Z-5.;  Selection of the hob milling mode. Positive value of D for the same rotational direction (normal in this case) of the workpiece spindle as the hob spindle.
G114.3D+1E1L10;  Specification of the hob spindle rotation at 50 min⁻¹.
S50;    
G00X18.;
G01Z20.F10;
G00X40.;
Z-5.;
G113;   Cancellation of the hob milling mode.
M05;    Milling spindle stop.
M202;   Cancellation of the milling mode.
```


B. Generating a helical gear (with phase matching)

G94; Selection of asynchronous feed mode.
M200; Selection of the milling mode.
M03S0; Start of milling spindle normal rotation at a speed of zero.
M250; Unclamping of the B-axis.
G00B92.8; B-axis rotation through the lead angle (92.8°) of the hob cutter.
G00X40.Z-5.; Selection of the hob milling mode (with phase matching for zero shift angle).
G114.3D-1E1L10P45Q2.5R0; Helix angle 45° (for B-axis rotation), Module 2.5 (mm).
M251; Negative value of D for the reverse rotational direction of the workpiece spindle to the hob spindle.
S50; Clamping of the B-axis.
G00X18.; Specification of the hob spindle rotation at 50 min⁻¹.
G01Z20.F10;
G00X40.;
Z-5.;
G113; Cancellation of the hob milling mode.
M05; Milling spindle stop.
M202; Cancellation of the milling mode.

C. Gear cutting on the secondary spindle

M950; Waiting command.
G110C2; Changing to the C-axis of System 2 (for cross machining control).
M951; Waiting command.
G92C0; Setting coordinate system (for the C-axis).
M300; Selection of the milling mode for the 2nd spindle.
M03S50; Start of milling spindle normal rotation at a speed of zero.
M250; Unclamping of the B-axis.
G00B92.8; B-axis rotation through the lead angle (92.8°) of the hob cutter.
M251; Clamping of the B-axis.
G00X40.Z-5.; Selection of the hob milling mode. Positive value of D for the same rotational direction (normal in this case) of the workpiece spindle as the hob spindle.
G114.3D+2E1L10; Specification of the hob spindle rotation at 50 min⁻¹.
M03S50;
G00X18.;
G01Z20.F10;
G00X40.;
Z-5.;
G113; Cancellation of the hob milling mode.
M952; Waiting command.
G111; Cancellation of the cross machining control.
M953; Waiting command.
M05; Milling spindle stop.
M302; Cancellation of the milling mode for the 2nd spindle.
4. Detailed description

1. Give an S-code and M-code, respectively, to specify the rotational speed and direction of the spindle selected as the hob spindle.

2. The block of G114.3 must be preceded by a command of “0” speed and a selection of the rotational direction of the hob spindle. The synchronization cannot be established if a command of G114.3 is given with the hob spindle already rotating or without its rotational direction specified.

3. The rotational speed of the workpiece spindle is determined by the number of hob threads and that of gear teeth, both specified in the block of G114.3.

\[ Sw = Sh \times \frac{E}{L} \]

where:
- \( Sh \): Rotational speed of the hob spindle
- \( Sw \): Rotational speed of the workpiece spindle
- \( E \): Rotational ratio of the hob spindle (Number of hob threads)
- \( L \): Rotational ratio of the workpiece spindle (Number of gear teeth)

4. Once determined by the hob milling command (G114.3), the rotational relationship between the workpiece spindle and the hob spindle is maintained in all operation modes until a hob milling cancel command (G113) or spindle synchronization cancel command is given.

5. The synchronization of the workpiece spindle with the hob spindle is started by the hob milling command (G114.3) at a speed of 0 revolutions per minute.

6. In the mode of hob milling the C-axis counter on the POSITION display does not work as the indicator of actual motion.

7. Do not fail to give a milling mode cancel command (M202) after cancellation of the hob milling mode by G113.

8. Use the preparatory function for asynchronous feed (G94) to cut a helical gear.

5. Remarks

1. Gear cutting accuracy cannot be guaranteed if the milling spindle speed is changed by operating the override keys during execution of a feed block in the hob milling mode.

2. If a motion command for the C-axis (workpiece spindle) is given in the middle of the hob milling mode by a manual or MDI interruption, or even in the program, such a shifting motion will be superimposed on the synchronized C-axis movement. In this case, however, the synchronization between the C-axis and the milling spindle cannot be guaranteed.

3. The selection of the hob milling mode (G114.3) in the mode of polygonal machining (G52.1) will result in an alarm. The polygonal machining cannot be selected in the hob milling mode, either.

4. The designation of the secondary spindle by D±2 does not have any effect if it is not provided with the optional C-axis control function.

5. A faulty machining could occur if the axis movement should come to a stop in the hob milling mode by the activation of the single-block operation mode or the feed hold function.

6. A phase mismatching or an excessive error could occur if the milling spindle should be stopped in the hob milling mode by a command of M05, M00, or M01.

7. The C-axis offset settings are ignored appropriately in the hob milling mode.

8. If the specified speed of the milling spindle is in excess of its upper limit, the milling spindle speed will be set to that limit and the C-axis will rotate in accordance with the milling spindle limit speed and the rotational ratio.
9. If the calculated speed of the C-axis rotation exceeds its upper limit, the C-axis speed will be set to that limit and the milling spindle will rotate in accordance with the C-axis speed limit and the rotational ratio.

10. The hob milling function is not compatible with the geometry compensation function (G61.1). Cancel the geometry compensation mode as required to use the hob milling function.
- NOTE -
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_H_I_J_K_Q_E_M1 [or M0];
X [or Z] _Y_; (Setting of hole position)
G67;
```

![Diagram of Tornado Tapping](TEP300)

- 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.
- Give the command G10.9X0 (Radius data input for the X-axis) as required beforehand to use the tornado tapping function.
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.

![Diagram](tep303)

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.

![Diagram](tep304)
22 HIGH-SPEED MACHINING MODE FEATURE (OPTION)

1. Function and purpose

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 high-precision control function.

The microsegment machining capability in the high-speed machining mode is as follows:

<table>
<thead>
<tr>
<th>Operation mode</th>
<th>Max. speed</th>
<th>Conditions required</th>
</tr>
</thead>
<tbody>
<tr>
<td>Memory operation</td>
<td>None</td>
<td></td>
</tr>
<tr>
<td>HD operation</td>
<td>135 m/min (5315 IPM)</td>
<td>With the POSITION display selected on the screen (see Note 2)</td>
</tr>
<tr>
<td>Ethernet operation</td>
<td></td>
<td>Avoid unusual key operations (see Note 3)</td>
</tr>
<tr>
<td>IC card operation</td>
<td></td>
<td>None</td>
</tr>
</tbody>
</table>
The microsegment machining capability is restricted further by the functions used in, or applied to, the program as shown below:

<table>
<thead>
<tr>
<th>Preparatory functions</th>
<th>Fairing function</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Not applied</td>
</tr>
<tr>
<td>G01 Linear interpolation only</td>
<td>135 m/min (5315 IPM)</td>
</tr>
<tr>
<td>G02/G03 Circular interpolation included</td>
<td>33 m/min (1299 IPM)</td>
</tr>
<tr>
<td>G6.1 Fine-spline interpolation included</td>
<td>101 m/min (3976 IPM)</td>
</tr>
</tbody>
</table>

**Note 1:** The microsegment machining capabilities shown above refer to the case where three-axis simultaneous motion commands consist of 32 characters per block for a segment length of 1 mm.

**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.

### 2. 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.

### 3. Commands available in the high-speed machining mode

Only axis motion commands with the corresponding preparatory functions (G-codes) and feed functions (F-codes), and designation of sequence number 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 and G03.

The circular interpolation can be programmed with R (radius designation) as well as with I and J (center designation). If the machining program includes circular commands, however, make bit 2 of the F96 parameter valid.

- **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. Sample program
   
   G28 X0 Y0 Z0
   G90 G0 X -100 Y -100.
   G43 Z -5 H03
   G01 F3000
   G05 P2
   X0.1
   X0.1 Y0.001
   X0.1 Y0.002
   ...
   X0.1 F200
   G05 P0
   G49 Z0
   M02

   High-speed machining mode ON

   When F84 bit 5 = 0:
   Incremental motion under G01

   When F84 bit 5 = 1:
   Absolute motion under G01

   High-speed machining mode OFF

   M02

   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.

4. Additional functions in the high-speed machining mode

   A. 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

   Before fairing  
   After fairing
Fairing is also valid for a succession of protruding paths as shown below:

![Diagram showing fairing process](image)

B. 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

![Diagram showing deceleration at corners](image)

If axial movement at the section of a large curvature should be conducted without deceleration, excessive acceleration will be developed to cause a path error due to inner cornering.

C. 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

An adequate deceleration can be performed without suffering any effects of this microblock.

5. 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
M98 H001
G49 Z0
G28 X0 Y0 Z0
M02
```

Movement under the conditions of G90, G00 and G43

```
Movement under the conditions of G90 and G01
```

Subprogram (O001)

```
N001 F3000
G05 P2
G01 X0.1
X-0.1 Y-0.001
X-0.1 Y-0.002

```

M02

High-speed machining mode ON

When F84 bit 5 = 0:
Incremental motion

When F84 bit 5 = 1:
Motion under the current mode (G90 or G91)

High-speed machining mode OFF

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:

![Diagram showing the selection and cancellation of high-speed machining mode]

4. Restrictions on programming and machine operation are listed in the following table:

<table>
<thead>
<tr>
<th>Specification</th>
<th>Standard mode</th>
<th>High-speed mode (Designation in the mode)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Classification</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Control axes</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum controllable axis quantity</td>
<td>16</td>
<td>16</td>
</tr>
<tr>
<td>Effective controllable axis quantity</td>
<td>16</td>
<td>7</td>
</tr>
<tr>
<td>Simultaneously controllable axis</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Axis name</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>CT axis</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Units of control</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Unit of input</td>
<td>ABC</td>
<td>ABC</td>
</tr>
<tr>
<td>Unit of programming</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Unit-of-programming × 10</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Input formats</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tape code</td>
<td>ISO/EIA</td>
<td>ISO/EIA</td>
</tr>
<tr>
<td>Label skip</td>
<td>O</td>
<td>– (–)</td>
</tr>
<tr>
<td>ISO/EIA automatic identification</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Parity H</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Parity V</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Tape format</td>
<td>O</td>
<td>Refer to the programming format.</td>
</tr>
<tr>
<td>Program number</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Sequence number</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Control IN/OUT</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Optimal block skip</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Buffers</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tape input buffer</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Pre-read buffer</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Position commands</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Absolute/incremental data input</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Inch/metric selection</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Decimal point input</td>
<td>O</td>
<td>O (O)</td>
</tr>
</tbody>
</table>
### Interpolation functions

<table>
<thead>
<tr>
<th>Specification</th>
<th>Standard mode</th>
<th>High-speed mode</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classification</td>
<td>Subclassification</td>
<td>(Designation in the mode)</td>
</tr>
<tr>
<td>Positioning</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>One-way positioning</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Circular interpolation</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Helical cutting</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Spiral interpolation</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Virtual-axis interpolation</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Threading</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Plane selection</td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Fine-Axis interpolation</td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>NURBS interpolation</td>
<td>O</td>
<td>O</td>
</tr>
</tbody>
</table>

### Feed functions

<table>
<thead>
<tr>
<th>Specification</th>
<th>Limitation in cutting direction</th>
<th>Minimum limiting speed of feed axes/According to curvature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classification</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rapid feed rate</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Cutting feed rate</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Synchronous feed</td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Automatic acceleration/deceleration</td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Linear acceleration/deceleration before cutting interpolation</td>
<td>O</td>
<td>O (err)</td>
</tr>
</tbody>
</table>

### Dwell

<table>
<thead>
<tr>
<th>Specification</th>
<th>Limitation in cutting direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classification</td>
<td></td>
</tr>
<tr>
<td>Dwell in time</td>
<td>O</td>
</tr>
<tr>
<td>Dwell in number of revolutions</td>
<td>O</td>
</tr>
<tr>
<td>M-command</td>
<td>O</td>
</tr>
<tr>
<td>M independent output command</td>
<td>O</td>
</tr>
<tr>
<td>Optional stop</td>
<td>O</td>
</tr>
<tr>
<td>No. 2 miscellaneous functions</td>
<td>O</td>
</tr>
</tbody>
</table>

### Miscellaneous function

<table>
<thead>
<tr>
<th>Specification</th>
<th>Limitation in cutting direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classification</td>
<td></td>
</tr>
<tr>
<td>Tool functions</td>
<td></td>
</tr>
<tr>
<td>S-command</td>
<td>O</td>
</tr>
<tr>
<td>T-command</td>
<td>O</td>
</tr>
<tr>
<td>Tool operation time integration</td>
<td>O</td>
</tr>
<tr>
<td>Spare-tool selection</td>
<td>O</td>
</tr>
<tr>
<td>Tool-length offset</td>
<td>O</td>
</tr>
<tr>
<td>Tool-position offset</td>
<td>O</td>
</tr>
<tr>
<td>Tool radius compensation</td>
<td>O</td>
</tr>
<tr>
<td>3D tool radius compensation</td>
<td>O</td>
</tr>
<tr>
<td>Tool-offset memory</td>
<td>O</td>
</tr>
<tr>
<td>Number of tool offset data sets</td>
<td>O</td>
</tr>
<tr>
<td>Programmed tool-offset input</td>
<td>O</td>
</tr>
<tr>
<td>Tool-offset number auto selection</td>
<td>O</td>
</tr>
</tbody>
</table>

---

**Legend:**
- O: Valid
- –: Invalid
- err: Error
<table>
<thead>
<tr>
<th>Specification</th>
<th>Classification</th>
<th>Standard mode</th>
<th>High-speed mode (Designation in the mode)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Program auxiliary functions</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fixed cycle for drilling</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Pattern cycle</td>
<td></td>
<td>–</td>
<td>– (–)</td>
</tr>
<tr>
<td>Subprogram control</td>
<td></td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Variable command</td>
<td></td>
<td>–</td>
<td>(err)</td>
</tr>
<tr>
<td>Figure rotation</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Coordinate rotation</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>User macro</td>
<td></td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>User macro interruption</td>
<td></td>
<td>O</td>
<td>O</td>
</tr>
<tr>
<td>Scaling</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Mirror image</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Geometric function</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Geometric function</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Programmed parameter setting</td>
<td></td>
<td>O</td>
<td>err (err)</td>
</tr>
<tr>
<td><strong>Coordinate system setting</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Watchdog-based reference-point return</td>
<td></td>
<td>O</td>
<td>O (–)</td>
</tr>
<tr>
<td>Memory-based reference-point return</td>
<td></td>
<td>O</td>
<td>O (–)</td>
</tr>
<tr>
<td>Automatic reference-point return</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>#2/#3/#4 reference-point return</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Reference-point check</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Machine coordinate system offset</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Workpiece coordinate system offset</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Local coordinate system offset</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Coordinate system setting</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Coordinate system rotation setting</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Program restart</td>
<td></td>
<td>O</td>
<td>O (err)</td>
</tr>
<tr>
<td>Absolute data detection</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td><strong>Machine error correction</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Backlash correction</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Lost-motion correction</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Memory-based relative position correction</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Machine coordinate system correction</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td><strong>Protection functions</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Emergency stop</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Stroke end</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Software limit</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Programmed software limit</td>
<td></td>
<td>O</td>
<td>– (err)</td>
</tr>
<tr>
<td>Interlock</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>External deceleration</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Data protection</td>
<td></td>
<td>O</td>
<td>O (O)</td>
</tr>
<tr>
<td>Classification</td>
<td>Standard mode</td>
<td>High-speed mode (Designation in the mode)</td>
<td></td>
</tr>
<tr>
<td>------------------------</td>
<td>---------------</td>
<td>------------------------------------------</td>
<td></td>
</tr>
<tr>
<td><strong>Operation modes</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tape operation</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td>Memory operation</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td>MDI operation</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Jog feed</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>Incremental feed</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>Handle feed</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>Manual rapid feed</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>Handle interruption</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Auto/manual simultaneous</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>HD operation</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td>IC card operation</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td>Ethernet operation</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td><strong>External control signals</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automatic-operation start</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Automatic-operation halt</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Single-block stop</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>NC reset</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>External reset</td>
<td>O</td>
<td>- (O)</td>
<td></td>
</tr>
<tr>
<td>All-axis machine lock</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Axis-by-axis machine lock</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Dry run</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Miscellaneous-function lock</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Manual-absolute selection</td>
<td>O</td>
<td>O (-)</td>
<td></td>
</tr>
<tr>
<td><strong>Status output signals</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Control-unit ready</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Servo-unit ready</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Auto-run mode</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Auto-run in progress</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Auto-run halted</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Cutting feed in progress</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Tapping in progress</td>
<td>O</td>
<td>- (-)</td>
<td></td>
</tr>
<tr>
<td>Threading in progress</td>
<td>O</td>
<td>- (-)</td>
<td></td>
</tr>
<tr>
<td>Axis selected</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Axis-movement direction</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Rapid feed in progress</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Rewind</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>NC alarm</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Reset</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Movement-command completed</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td><strong>Measurement aid functions</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automatic tool-length measurement</td>
<td>O</td>
<td>- (err)</td>
<td></td>
</tr>
<tr>
<td>Skip</td>
<td>O</td>
<td>- (err)</td>
<td></td>
</tr>
<tr>
<td>Multi-step skip</td>
<td>O</td>
<td>- (err)</td>
<td></td>
</tr>
<tr>
<td>Manual skip</td>
<td>O</td>
<td>- (err)</td>
<td></td>
</tr>
<tr>
<td><strong>Axis control functions</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Servo off</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Follow-up</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
<tr>
<td>Control-axis removal</td>
<td>O</td>
<td>O (O)</td>
<td></td>
</tr>
</tbody>
</table>
### HIGH-SPEED MACHINING MODE FEATURE (OPTION)

<table>
<thead>
<tr>
<th>Specification</th>
<th>Standard mode</th>
<th>High-speed mode (Designation in the mode)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Classification</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Data input/output</td>
<td></td>
<td></td>
</tr>
<tr>
<td>External data input I/F</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>External data output I/F</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>External data input/output</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td><strong>Setting/display functions</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Setting/Display unit</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Settings display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Search</td>
<td>○</td>
<td>○ (err)</td>
</tr>
<tr>
<td>Check-and-stop</td>
<td>○</td>
<td>– (–)</td>
</tr>
<tr>
<td>MDI</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Program restart</td>
<td>○</td>
<td>○ (err)</td>
</tr>
<tr>
<td>Machining-time calculation</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>PC opening</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Program-status display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Integrated-time display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Graphics display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Multi-step skip</td>
<td>○</td>
<td>– (err)</td>
</tr>
<tr>
<td>Graphics check</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td><strong>Self-diagnostics</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Program-error display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Operation-error display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Servo-error display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Operation-stop-cause display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Servo monitor display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>NC-PC I/O signal display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>DIO display</td>
<td>○</td>
<td>○ (O)</td>
</tr>
<tr>
<td>Keyboard-operation record</td>
<td>○</td>
<td>○ (O)</td>
</tr>
</tbody>
</table>

O: Valid, –: Invalid, err: Error
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 shape correction being selected.
23-1 Programming Format

<table>
<thead>
<tr>
<th>G61.1 (G61.2);</th>
</tr>
</thead>
<tbody>
<tr>
<td>G5P2; ・・・・・・・High-speed machining mode (with smoothing control) ON</td>
</tr>
</tbody>
</table>

- 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.

23-2 Commands Available in the High-Speed Smoothing Control Mode

As is the case with the high-speed machining mode feature, only axis motion commands with the corresponding preparatory functions (G-codes) and feed functions (F-codes), and designation of sequence number 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 and G03. The circular interpolation 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.

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.
23-3 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: Faring function for the microsegment machining program
1: Faring for a protruding path
0: No faking

F103: Maximum length of a block to be removed for fairing

Fairing is also valid for a succession of protruding paths as shown below:
23-4 Related Parameters

The parameters related to this function are as follows:

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F3 bit 0</td>
<td>High-speed smoothing control valid/invalid</td>
<td>0: Invalid, 1: Valid.</td>
</tr>
<tr>
<td>F3 bit 1</td>
<td>Deceleration at stepped sections</td>
<td>0: No deceleration at very slightly stepped sections, 1: Deceleration at all stepped sections. Under the default setting (0), 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 1 to obtain precise feed control for the as-programmed shape.</td>
</tr>
</tbody>
</table>

23-5 Remarks

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.

23-6 Related Alarms

The alarms related to this function are as follows:

<table>
<thead>
<tr>
<th>Alarm No.</th>
<th>Alarm message</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
<tbody>
<tr>
<td>807</td>
<td>ILLEGAL FORMAT</td>
<td>An unavailable command code is given in the G5P2 mode.</td>
<td>Check the machining program and make corrections as required.</td>
</tr>
<tr>
<td>809</td>
<td>ILLEGAL NUMBER INPUT</td>
<td>The number of digits of the entered numerical data is too large.</td>
<td>Check the machining program and make corrections as required.</td>
</tr>
</tbody>
</table>
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):

<table>
<thead>
<tr>
<th>M-code</th>
<th>Accuracy level</th>
</tr>
</thead>
<tbody>
<tr>
<td>M821</td>
<td>1</td>
</tr>
<tr>
<td>M822</td>
<td>2</td>
</tr>
<tr>
<td>M823</td>
<td>3</td>
</tr>
<tr>
<td>M824</td>
<td>4</td>
</tr>
<tr>
<td>M825</td>
<td>5</td>
</tr>
<tr>
<td>M826</td>
<td>6</td>
</tr>
<tr>
<td>M827</td>
<td>7</td>
</tr>
<tr>
<td>M828</td>
<td>8</td>
</tr>
<tr>
<td>M829</td>
<td>9</td>
</tr>
<tr>
<td>M830</td>
<td>10</td>
</tr>
</tbody>
</table>

B. Programming example

```
G00G40G80G90G94G98
G91G00G28Z0.
G28X0.Y0.
T1T2M6
G00G90G54X182.15Y20.974S180M3
G43H1Z100.M8
Z5.
M825 ← Selection of Accuracy level 5.
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.
25 AUTOMATIC TOOL LENGTH MEASUREMENT: G37

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).

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

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F42</td>
<td>R-code command. Deceleration area</td>
</tr>
<tr>
<td>F43</td>
<td>D-code command. Measurement area</td>
</tr>
<tr>
<td>F44</td>
<td>F-code command. Measurement feed rate</td>
</tr>
<tr>
<td>F72</td>
<td>Conditions for skipping based on EIA G37</td>
</tr>
</tbody>
</table>

See the Parameter 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, $H01 = 99.99$

<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)

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, $H01 = 99.99$
5. **Detailed description**

1. Machine action based on command G37

```
G28X0Y0Z0
G90G0G43Zz0...........................................[1]
G37Zz0Rr0Dd0Ff0......................................[2],[3]
G0G90Zz1..................................................[4]
G28X0Y0Z0..................................................[5]
```

- \( h_0 \): Offset number
- \( z_0 \): Coordinate of the measurement point (measurement position)
- \( r_0 \): Starting point of movement at measurement feed rate
- \( d_0 \): The area where the tool is to stop moving
- \( f_0 \): Measurement feed rate

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 msec for the NC side alone. Accordingly, the following measurement error may occur:

   \[
   \text{Maximum measurement error [mm]} = \text{Measurement feed rate [mm/min]} \times \frac{1}{60} \times \frac{0.2 \text{ [ms]}}{1000}
   \]

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.

   \[
   \text{Maximum amount of overrun [mm]} = \text{Measurement feed rate [mm/min]} \times \frac{1}{60} \times \frac{30.3 \text{ [ms]}}{1000}
   \]

30.3 [msec] 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:

- G0 G90 G43 Z-200. H01
- G0 G90 Z-200.

Machine in its single-block stop status at block [1]

Start button on

Block [2] executed

Block [3] executed

Machine replaced in its single-block stop status

6. Precautions

1. 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.

2. 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.

3. Alarm **924 G37, H COMMANDS SAME BLOCK** will result if an H code exists in the block of G37.

4. Alarm **925 H CODE REQUIRED** will result if G43 H_ does not exist before the block of G37.

5. 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.

6. 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.

7. Set G37 data or parameter data so that the following condition is satisfied:

   \[ \text{Measurement point} - \text{Staring point} > R\text{-code value or parameter } r > D\text{-code value or parameter } d \]

8. 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.

9. 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.

10. Set G37 (automatic tool length measurement code) together with G43 H_ (offset number assignment code).

    G43 H_
    G37 Z_R_D_F_

11. 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.
12. 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.

13. 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.

14. 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.
AUTOMATIC TOOL LENGTH MEASUREMENT: G37

- NOTE -
26 DYNAMIC OFFSETTING: M173, M174 (Option)

1. Function and purpose

Machining with rotation of the rotary table (C-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 jig design, unless a very precise jig 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.

2. Detailed description

1. Automatic limitation by the software does not occur even if dynamic offsetting may cause the stroke limit to be overstepped.

2. 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).

3. The workpiece origin must be set on the axis of rotation of a workpiece when it is to be machined using dynamic offsetting.

M173 Dynamic offsetting ON
G01 C360. F500
M174 Dynamic offsetting OFF
4. The related parameters are as follows:

<table>
<thead>
<tr>
<th>Address</th>
<th>Name</th>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>S5</td>
<td>Axis of table rotation</td>
<td>Unit: 0.0001 mm/ 0.00001 in.</td>
<td>Set the machine coordinates of the axis of the table rotation for the</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Range: ±99999999</td>
<td>controlled axes concerned.</td>
</tr>
<tr>
<td>I11</td>
<td>Axis of workpiece rotation</td>
<td>Unit: 0.0001 mm/ 0.00001 in.</td>
<td>Set the machine coordinates of the axis of the workpiece rotation,</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Range: ±99999999</td>
<td>existing at a table angle of 0 degrees, for the controlled axes</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>concerned. (This parameter is valid only for manual operation.)</td>
</tr>
</tbody>
</table>

5. 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 programs

G55................. The workpiece axis is assumed to go through the origin of the G55 system.
G0X_Y_Z_............ Approach
M173................ Dynamic offsetting ON
G1X_F_............... Start of cutting
    C_F............... C-axis rotation
    X_F............... Relief on the X-axis
M174................ Dynamic offsetting OFF
M30.................... End of machining
27 DYNAMIC OFFSETTING II: G54.2P0, G54.2P1 - G54.2P8

1. Function and purpose

When a workpiece fixed on the turntable is to be machined with the rotation of the table, mis-matching 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.

With the measurement results of the reference dynamic offset (Gs) 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 Gs 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, “G1b1Cθ” 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

When the system is reset, the dynamic offsetting mode is normally canceled. It depends, however, on the setting of parameter F95 bit 7 whether or not dynamic offsetting is canceled on system reset operations.

F95 bit 7 = 0: The dynamic offset is cleared 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 cleared by resetting, the tool will not move on the path corresponding to the cleared vector (even if bit 0 of the F87 parameter described later is set to “0”).

C. Operation by the selection and cancellation 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.

The cancellation command (G54.2P0) moves the tool by a vector reverse to the current dynamic offset. 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).

D. 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 ・・・αα;

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 × 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
   - Mirror image (by G51.1 or control signal)
   - Scaling (G51)
   - Figure rotation (M98)
   - Coordinates rotation (G68)
   - 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.
   \[ L81 = \begin{array}{l}
   0: \text{Makes the dynamic offsetting function invalid.} \\
   1: \text{Two rotational axes (C-axis on A-axis)} \\
   2: \text{One rotational axis (A-axis)} \\
   3: \text{One rotational axis (C-axis)} \\
   4: \text{One rotational axis (B-axis)}
   \end{array} \]

B. Dynamic offset type
   Specify whether or not the tool is to be offset by each change only in the deviation vector.
   \[ F87 \text{ bit } 0 = \begin{array}{l}
   0: \text{Offset (the indication of both workpiece and machine coordinates changes.)} \\
   1: \text{Not offset (no change in the position indication at all)}
   \end{array} \]
   Normally set this parameter to “0”.

C. Center of table rotation
   Specify the center of rotation of the table in machine coordinates.
   The preset values refer to the factory adjustment at Mazak.
   \[ S5 \ X, \ Y \quad \text{Center of rotation of the turntable (Machine coordinates)} \]
   \[ S12 \ \ Y, \ Z \quad \text{Axis of rotation of the tilting table (Machine coordinates)} \]
   \[ S11 \ Z \quad \text{Distance (length) from the tilting axis to the turntable surface} \]
   \hspace{1cm} (The turntable center must be in the direction of –Z from the tilting axis.)
   \textbf{Note:} \quad \text{When } L81 = 2, 3, \text{ or } 4, \text{ the } S11 \text{ and } S12 \text{ 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.
   \[ F87 \text{ bit } 1 = \begin{array}{l}
   0: \text{The mismatch check is conducted.} \\
   1: \text{The mismatch check is not conducted.}
   \end{array} \]
   Normally set this parameter to “0”.

27-5
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

<table>
<thead>
<tr>
<th>No.</th>
<th>POSITION (workpiece coordinates)</th>
<th>MACHINE (machine coordinates)</th>
<th>Dynamic offset</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
<td>Y</td>
<td>Z</td>
</tr>
<tr>
<td>N1</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N2</td>
<td>315.000</td>
<td>315.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N3</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N4</td>
<td>0.000</td>
<td>-1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N5</td>
<td>0.000</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N6</td>
<td>10.000</td>
<td>1.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N7</td>
<td>0.000</td>
<td>11.000</td>
<td>0.000</td>
</tr>
<tr>
<td>N8</td>
<td>0.866</td>
<td>10.500</td>
<td>0.000</td>
</tr>
</tbody>
</table>
D. Illustration of the sample program

<table>
<thead>
<tr>
<th>N-No.</th>
<th>N3</th>
<th>N4</th>
<th>N5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Illustration</td>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td><img src="image1" alt="Illustration 1" /></td>
<td><img src="image2" alt="Illustration 2" /></td>
<td><img src="image3" alt="Illustration 3" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N-No.</th>
<th>N6, N7</th>
<th>N8</th>
<th>N9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Illustration</td>
<td>4</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td></td>
<td><img src="image4" alt="Illustration 4" /></td>
<td><img src="image5" alt="Illustration 5" /></td>
<td></td>
</tr>
</tbody>
</table>

<Explanation>
1. N3 turns the table on the C-axis to ● (C = 0) and positions the tool tip to the × 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 × 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 × 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 × point.
5. N8 turns the table on the C-axis to ● and causes the tool tip to be shifted by linear interpolation to the × point.
- NOTE -
28 FUNCTION OF ESTIMATING THE MOMENT OF INERTIA FOR SPINDLE AND SERVO AXIS (OPTION)

The moment of inertia depends upon the weight and shape of the workpiece turned, including its fixture. For the purpose of reducing the vibration in general and the overshooting at speed change, there are preparatory functions designed to automatically set, or modify, the parameters concerned with the servo-controlled axes as well as the workpiece spindle in accordance with the particular moment of inertia.

Machining with small moment of inertia
- Time constant for (pos. & neg.) acceleration is small.
- Gain can be enhanced.
- It takes an unnecessarily long time to change the speed.
- There is a fear of vibration due to inappropriate gain.

Machining with large moment of inertia
- There is a fear of overshooting at speed change.
- There is a fear of vibration due to inappropriate gain.
- Time constant for (pos. & neg.) acceleration is large.
- Gain is set to a lower value.

28-1 Estimating the Moment of Inertia for Setting Parameters: G297

1. Outline

The moment of inertia can be estimated for the designated spindle or servo axis, the weight in question is judged to fall under three levels (large, middle, and small), and the judgment is indicated on the POSITION display by displaying the spindle’s or axis’ name in the corresponding colors (red, yellow, and blue) in the display area of load meter.

<table>
<thead>
<tr>
<th>Weight level</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small</td>
<td>Blue</td>
</tr>
<tr>
<td>Middle</td>
<td>Yellow</td>
</tr>
<tr>
<td>Large</td>
<td>Red</td>
</tr>
</tbody>
</table>

Finally the parameters concerned, gain and time constant among others, will be set or modified according to the above judgment.

2. Programming format

A. Spindle

G297 R_(P_); Estimating the moment of inertia for setting parameters
R .......Selection of the spindle for the estimation of the moment of inertia.
Set 1 and 2 respectively as the argument for the main and sub-spindle of the system in whose program section the G297 command is given.
1: Main spindle of turning
2: Sub-spindle of turning
FUNCTION OF ESTIMATING THE MOMENT OF INERTIA FOR SPINDLE AND SERVO AXIS (OPTION)

P ....... Selection of the system for the setting of parameters.
  0: All systems
  1: System in question
  2: Spindle in question

B. Servo axis

G297 Q_ (P_); Estimating the moment of inertia for setting parameters
  Q ....... Selection of the servo axis for the estimation of the moment of inertia.
    Set the number of the desired servo axis within the system in question.
  P ....... Selection of the system for the setting of parameters.
    0: All systems
    1: System in question
    2: Axis in question

C. Supplement
- A command of G297 without argument R or Q will lead to an alarm (807 ILLEGAL FORMAT).
- A command of G297 with multiple settings of argument R or Q is processed with the last setting
  as the definitive one.
- A command of G297 with both arguments R and Q is processed as a command with argument
  R only.
- Argument P can be omitted when P0 is desired.

3. Programming examples
G297R1P2; ........... Estimating the moment of inertia of the main spindle, and setting
parameters.
G297Q5P2; ........... Estimating the moment of inertia of the 5th axis within the system in
question, and setting parameters.
G297Q1P2; ........... Estimating the moment of inertia of the 1st axis within the system in
question, and setting parameters.

Estimating the moment of inertia, and setting parameters, for the turning mode
M202; .................. Selection of turning mode
G297R1P2; ........... Estimating the moment of inertia of the main turning spindle, and setting
parameters.

Estimating the moment of inertia, and setting parameters, for the milling mode
M200; .................. Selection of milling mode
G297R1P2; ........... Estimating the moment of inertia of the main turning spindle under C-axis
control, and setting parameters.

4. Remarks
1. A turning spindle capable of C-axis control can undergo estimation of the moment of inertia
   both as a turning spindle and as the C-axis. In the application of G297 to the turning spindle
   as the C-axis, however, it may not be possible to set parameters appropriate to the
   particular moment of inertia when the difference between the maximum and minimum of the
   moment of inertia is great at a considerable extent. In that case, give a G297 command for
   the turning spindle in the turning mode.
2. Select the turning mode beforehand to apply G297 to the turning spindle.
3. Select the milling mode beforehand to apply G297 to the turning spindle as the C-axis.
4. Designate the desired side beforehand with a headstock selection command to apply G297 to the turning spindle.
5. A command of G297 will lead to an alarm (807 ILLEGAL FORMAT) when argument Q refers to an axis in another system.
6. A command of G297 will lead to an alarm (807 ILLEGAL FORMAT) when argument R, Q, or P falls outside their setting range.
7. Give axis movement commands as required to set the component concerned to a safe position before a G297 command whose execution includes an actual rotary or linear motion for the purpose of estimating the moment of inertia for the designated spindle or servo axis.

28-2 Setting Parameters Related to the Moment of Inertia: G298

1. Outline

As compared with G297, a G298 command is used to automatically set or modify the gain, time constant, and other parameters for the spindle or servo axis, without estimating the moment of inertia, according to the designated weight level (large, middle, or small), which is indicated on the POSITION display by displaying the spindle’s or axis’ name in the corresponding color (red, yellow, or blue) in the display area of load meter.

<table>
<thead>
<tr>
<th>Weight level</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small</td>
<td>Blue</td>
</tr>
<tr>
<td>Middle</td>
<td>Yellow</td>
</tr>
<tr>
<td>Large</td>
<td>Red</td>
</tr>
</tbody>
</table>

2. Programming format

A. Spindle

G298 R_(D_) (P_) ;  Setting parameters

R ......Selection of the spindle for the setting of parameters.
Set 1 and 2 respectively as the argument for the main and sub-spindle of the system in whose program section the G298 command is given.
1: Main spindle of turning
2: Sub-spindle of turning

D ......Selection of the level of the moment of inertia.
0: for standard parameters
1: for the parameters of pattern 1 (small weight)
2: for the parameters of pattern 2 (middle weight)
3: for the parameters of pattern 3 (large weight)

P ......Selection of the system for the setting of parameters.
0: All systems
1: System in question
2: Spindle in question
B. Servo axis

G298 Q_ (D_) (P_);  Setting parameters

Q ........ Selection of the servo axis for the setting of parameters.
Set the number of the desired servo axis within the system in question.

D ........ Selection of the level of the moment of inertia.
0: for standard parameters
1: for the parameters of pattern 1 (small weight)
2: for the parameters of pattern 2 (middle weight)
3: for the parameters of pattern 3 (large weight)

P ........ Selection of the system for the setting of parameters.
0: All systems
1: System in question
2: Axis in question

C. Supplement

- A command of G298 without argument R or Q will lead to an alarm (807 ILLEGAL FORMAT).
- A command of G298 with multiple settings of argument R or Q is processed with the last setting as the definitive one.
- A command of G298 with both arguments R and Q is processed as a command with argument R only.
- Argument D can be omitted when it is desired to set the parameters concerned with reference to the judgment resulting from the previous estimation of the moment of inertia (by G297).
- Argument P can be omitted when P0 is desired.

3. Programming examples

G298R1D0P0; ...... Setting standard parameters for all axes in all systems.
G297R1P2; .......... Estimating the moment of inertia of the main spindle, and setting parameters.
G298R1D3P2; ...... Setting the parameters of pattern 3 (large weight) for the main spindle.
G297Q5P2; .......... Estimating the moment of inertia of the 5th axis within the system in question, and setting parameters.
G298Q5D0P2; ...... Setting standard parameters for the 5th axis within the system in question.

4. Remarks

1. A command of G298 without argument D will cause the standard parameters to be set when no command for estimating the moment of inertia (with G297) has yet been executed after switching-on.
2. Give a command of G298D0 to reset the parameters modified by G298 to the standard values.
3. A command of G298 will lead to an alarm (177 ONE-TOUCH TUNING IMPOSSIBLE) when the component concerned has not been set at rest (rotation of the designated spindle, or movement on a servo axis which is designated or belongs to the same system as the designated one).
4. A command of G298 will lead to an alarm (807 ILLEGAL FORMAT) when argument R, Q, D, or P falls outside their setting range.
5. A command for parameter setting also concerns some parameters used commonly in all systems even when argument P is not set to 0 (for setting parameters in all systems).

5. **Parameters concerned**

The table below enumerates the main parameters to be set or modified by the function in question.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Servo axis</th>
<th>Spindle (C-axis)</th>
<th>Spindle</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>Rapid traverse speed</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>M3</td>
<td>Rate of cutting feed</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>N1</td>
<td>Time constant for rapid traverse</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>N2</td>
<td>Time constant for cutting feed</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>L74</td>
<td>Pre-interpolation – Rate of cutting feed</td>
<td>○</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>L75</td>
<td>Pre-interpolation – Time constant for cutting feed</td>
<td>○</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>L129</td>
<td>Time constant for G1 – S-shaped filter</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>L130</td>
<td>Time constant for G0 – S-shaped filter</td>
<td>○</td>
<td>○</td>
<td>—</td>
</tr>
<tr>
<td>SA33-36</td>
<td>Acceleration time constant for synchronous tapping</td>
<td>—</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>SA89-91</td>
<td>Spindle speed operating time constant</td>
<td>—</td>
<td>○</td>
<td>○</td>
</tr>
<tr>
<td>SA99</td>
<td>Time constant for orientation</td>
<td>—</td>
<td>○</td>
<td>○</td>
</tr>
</tbody>
</table>

In addition to the above parameters, some parameters relating to the servo axis and spindle are subject to setting or modification.
- NOTE -
29  FIVE-AXIS MACHINING FUNCTION

29-1 Tool Tip Point Control for Five-Axis Machining (Option)

29-1-1 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

<table>
<thead>
<tr>
<th>Tool rotating type</th>
<th>Table rotating type</th>
<th>Mixed type</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Tool rotating type diagram" /></td>
<td><img src="image2" alt="Table rotating type diagram" /></td>
<td><img src="image3" alt="Mixed type diagram" /></td>
</tr>
</tbody>
</table>

* The INTEGREX e-series machines belong to the mixed type.
29-1-2 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.
The angular position of the table at the fixing of the workpiece coordinate system depends on the setting of the parameter concerned as follows:

<table>
<thead>
<tr>
<th>Reference angular position</th>
<th>Starting position (F86 bit 6 = 0)</th>
<th>0° position (F86 bit 6 = 1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Method of fixing the workpiece coordinate system to the table</td>
<td>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.</td>
<td>The workpiece coordinate system is fixed to the table in its angular position of 0°.</td>
</tr>
<tr>
<td>C-axis origin = 45°</td>
<td>Reference position</td>
<td>Reference position</td>
</tr>
<tr>
<td>C = 0.</td>
<td>Y</td>
<td>Y</td>
</tr>
<tr>
<td>Fixing the workpiece coordinate system to the table occurs at the start of tool tip point control.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Example: Workpiece origin setting for the C-axis = 45°</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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.

[Diagram of table rotation and 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. 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).

9. The cancellation command G49 must be given with no other instruction codes.
### 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.)

<table>
<thead>
<tr>
<th>Commands for rotational axis motion or tool-axis direction</th>
<th>Motion commands for orthogonal axes</th>
</tr>
</thead>
<tbody>
<tr>
<td>not given (*1)</td>
<td></td>
</tr>
<tr>
<td>F162 bit 0 = 0</td>
<td>F162 bit 0 = 1</td>
</tr>
<tr>
<td>(Motion by the offset amount)</td>
<td>(No motion by the offset amount)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>G43.4Bh</th>
<th>G43.4Bh</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Control point</strong></td>
<td><strong>Control point</strong></td>
</tr>
<tr>
<td><strong>Tool tip point</strong></td>
<td><strong>Tool tip point</strong></td>
</tr>
<tr>
<td>A shifting motion occurs for the tip point to be at the</td>
<td>A shifting motion occurs for the tip point to be at the</td>
</tr>
<tr>
<td>current control point, i.e. in the tool-axis direction</td>
<td>current control point, i.e. in the tool-axis direction</td>
</tr>
<tr>
<td>through the length offset distance.</td>
<td>through the length offset distance.</td>
</tr>
<tr>
<td>No motions occur at all.</td>
<td>No motions occur at all.</td>
</tr>
<tr>
<td>(A motion by the offset amount does not occur.)</td>
<td>(A motion by the offset amount does not occur.)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>G43.4Bbh</th>
<th>G43.4Bbh</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Control point</strong></td>
<td><strong>Control point</strong></td>
</tr>
<tr>
<td><strong>Tool tip point</strong></td>
<td><strong>Tool tip point</strong></td>
</tr>
<tr>
<td>A linear and rotational motion occurs for the tip point</td>
<td>A linear and rotational motion occurs for the tip point</td>
</tr>
<tr>
<td>to be at the current control point, as in the programming</td>
<td>to be at the current control point, as in the programming</td>
</tr>
<tr>
<td>coordinate system (*2), through the length offset</td>
<td>coordinate system (*2), through the length offset</td>
</tr>
<tr>
<td>distance.</td>
<td>distance.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>G43.4Bbh</th>
<th>G43.4Bbh</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Control point</strong></td>
<td><strong>Control point</strong></td>
</tr>
<tr>
<td><strong>Tool tip point</strong></td>
<td><strong>Tool tip point</strong></td>
</tr>
<tr>
<td>A linear and rotational motion occurs for the tip point</td>
<td>A linear and rotational motion occurs for the tip point</td>
</tr>
<tr>
<td>to be at a point of the specified coordinates, inclusive of offset amount.</td>
<td>to be at a point of the specified coordinates, inclusive of offset amount.</td>
</tr>
</tbody>
</table>

(*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: In both cases giving the cancellation command G49 with any other instruction code 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.

\[
\begin{align*}
&G43.5X_1Z_1I_1J_1K_1 \newline
&X_2Z_2I_2J_2K_2 \newline
&X_3Z_3I_3J_3K_3
\end{align*}
\]

<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.

\[
\begin{align*}
&G43.5I_0J_0K_0H_0 \newline
&I_1J_1K_1H_1
\end{align*}
\]
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.

![Tool tip point](D740PB0045)

The feed is controlled so that the center of the tool tip may move at the specified rate.

![Control point](D740PB0046)

- Interpolation occurs in the continuous lines determined by the specified positions of the tool tip point.
- Specified rate of feed ($F$) = Speed of the tool tip point's motion
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.)

<table>
<thead>
<tr>
<th>Method</th>
<th>Uniaxial rotation interpolation</th>
<th>Joint interpolation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>F85 bit 3 = 0</td>
<td>F85 bit 3 = 1</td>
</tr>
<tr>
<td><strong>Operation</strong></td>
<td><strong>&lt;Tool axis direction relative to the workpiece&gt;</strong></td>
<td><strong>&lt;Tool axis direction relative to the workpiece&gt;</strong></td>
</tr>
<tr>
<td></td>
<td><strong>&lt;Machine action&gt;</strong></td>
<td><strong>&lt;Machine action&gt;</strong></td>
</tr>
<tr>
<td></td>
<td><em>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.</em></td>
<td><em>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.</em></td>
</tr>
<tr>
<td>Feature</td>
<td><em>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.</em></td>
<td><em>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.</em></td>
</tr>
</tbody>
</table>

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.

A linear interpolation with a constant rotational speed is applied to each rotational axis (as is normally the case with G01).
<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 uprighter [nearer to a position of $B = 0$] the tool.)

<table>
<thead>
<tr>
<th>Uniaxial rotation interpolation</th>
<th>Joint interpolation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial rotation by $90^\circ$ on the C-axis is required for the tool-axis inclination to always occur in a plane in this direction.</td>
<td>As a result of uniform motions on the B- and C-axis, the plane of tool-axis inclination pivots continuously.</td>
</tr>
</tbody>
</table>

Starting point
B0. C0.
(Attitude vector is upright.)

Ending point
B45. C90.

B36.
B27.
B18.
B9.

Starting point
B0. C0.
(Attitude vector is upright.)

Ending point
B45. C90.

B36. C72.
B27. C54.
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

<table>
<thead>
<tr>
<th>Tool rotating type</th>
<th>Table rotating type</th>
<th>Mixed type</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="D740PB0034" alt="Tool rotating type diagram" /></td>
<td><img src="D740PB0034" alt="Table rotating type diagram" /></td>
<td><img src="D740PB0034" alt="Mixed type diagram" /></td>
</tr>
</tbody>
</table>

(D740PB0034)
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.

```
I0.707
J0K0.707

B = 45°
C = 0°

B = –45°
C = 180°
```

Programmed command

Pairs of angles on the rotational axes for obtaining the specified tool's attitude relative to the workpiece

Solution with a positive angle of the B-axis

Solution with a negative angle of the B-axis

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 \textbf{F162} bit 1 to select the desired type.

<table>
<thead>
<tr>
<th>Type 1 of passage through singular point \textbf{F162} bit 1 = 0</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Operation</strong></td>
</tr>
<tr>
<td><strong>Example</strong></td>
</tr>
<tr>
<td><img src="image1" alt="Example Diagram" /></td>
</tr>
<tr>
<td><img src="image2" alt="Example Diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Type 2 of passage through singular point \textbf{F162} bit 1 = 1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Operation</strong></td>
</tr>
<tr>
<td><img src="image3" alt="Example Diagram" /></td>
</tr>
<tr>
<td><img src="image4" alt="Example Diagram" /></td>
</tr>
<tr>
<td><img src="image5" alt="Example Diagram" /></td>
</tr>
<tr>
<td><img src="image6" alt="Example Diagram" /></td>
</tr>
<tr>
<td><strong>Example</strong></td>
</tr>
<tr>
<td><img src="image7" alt="Example Diagram" /></td>
</tr>
<tr>
<td><img src="image8" alt="Example Diagram" /></td>
</tr>
</tbody>
</table>
29-1-3 Relationship to other functions

1. Commands usable in the same block with G43.4/G43.5

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Function</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Incremental data input</td>
<td>G91</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Feed function</td>
<td>F</td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td></td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td></td>
</tr>
<tr>
<td>Circular interpolation</td>
<td>G02/G03</td>
<td>Note 1</td>
</tr>
<tr>
<td>Dwell</td>
<td>G04</td>
<td></td>
</tr>
<tr>
<td>High-speed machining mode</td>
<td>G05</td>
<td></td>
</tr>
<tr>
<td>Exact-stop</td>
<td>G09</td>
<td></td>
</tr>
<tr>
<td>Plane selection</td>
<td>G17/G18/G19</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation OFF</td>
<td>G40</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the left)</td>
<td>G41.2/G41.4/G41.5</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right)</td>
<td>G42.2/G42.4/G42.5</td>
<td></td>
</tr>
<tr>
<td>Tool length offset (+/−)</td>
<td>G43/G44</td>
<td></td>
</tr>
<tr>
<td>Tool position offset OFF</td>
<td>G49</td>
<td></td>
</tr>
<tr>
<td>Scaling</td>
<td>G50</td>
<td></td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td></td>
</tr>
<tr>
<td>Geometry compensation mode</td>
<td>G61.1</td>
<td>Note 3</td>
</tr>
<tr>
<td>Cutting mode</td>
<td>G64</td>
<td></td>
</tr>
<tr>
<td>Macro call</td>
<td>G65</td>
<td></td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
</tr>
<tr>
<td>Incremental data input</td>
<td>G91</td>
<td></td>
</tr>
<tr>
<td>Inverse time feed</td>
<td>G93</td>
<td></td>
</tr>
<tr>
<td>Feed per minute</td>
<td>G94</td>
<td></td>
</tr>
<tr>
<td>M, S, T, B output to opposite system</td>
<td>G112</td>
<td>Note 2</td>
</tr>
<tr>
<td>Subprogram call/End of subprogram</td>
<td>M98/M99</td>
<td></td>
</tr>
<tr>
<td>Feed function</td>
<td>F</td>
<td></td>
</tr>
<tr>
<td>M, S, T, B function</td>
<td>MSTB</td>
<td>Note 2</td>
</tr>
<tr>
<td>Local variables, Common variables, Operation commands (arithmetic operations, trigonometric functions, square root, etc), Control commands (IF ∼ GOTO ∼, WHILE ∼ DO ∼)</td>
<td>Macro instructions</td>
<td></td>
</tr>
</tbody>
</table>

Note 1: A command for helical/spiral interpolation will lead to an alarm (972 ILLEGAL CMD TOOL TIP PT CTRL).

Note 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).

Note 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).
### B. Modes in which tool tip point control is selectable

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td></td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>G13.1</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate input OFF</td>
<td>G15</td>
<td></td>
</tr>
<tr>
<td>Plane selection</td>
<td>G17/G18/G19</td>
<td></td>
</tr>
<tr>
<td>Inch data input</td>
<td>G20</td>
<td></td>
</tr>
<tr>
<td>Metric data input</td>
<td>G21</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23</td>
<td></td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation OFF</td>
<td>G40</td>
<td></td>
</tr>
<tr>
<td>Control for normal-line direction OFF</td>
<td>G40.1</td>
<td></td>
</tr>
<tr>
<td>Tool length offset (+/-)</td>
<td>G43/G44</td>
<td></td>
</tr>
<tr>
<td>Tool position offset OFF</td>
<td>G49</td>
<td></td>
</tr>
<tr>
<td>Head offset for five-surface machining OFF</td>
<td>G49.1</td>
<td></td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>G50</td>
<td></td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>G50.1</td>
<td></td>
</tr>
<tr>
<td>Polygonal machining mode OFF</td>
<td>G50.2</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system/additional workpiece coordinate system</td>
<td>G54-59/G54.1</td>
<td></td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td></td>
</tr>
<tr>
<td>Geometry compensation mode</td>
<td>G61.1</td>
<td>Note 1</td>
</tr>
<tr>
<td>Cutting mode</td>
<td>G64</td>
<td></td>
</tr>
<tr>
<td>Macro call</td>
<td>G65</td>
<td></td>
</tr>
<tr>
<td>Modal macro call OFF</td>
<td>G67</td>
<td></td>
</tr>
<tr>
<td>Inclined-plane machining</td>
<td>G68.2</td>
<td></td>
</tr>
<tr>
<td>3-D coordinate conversion OFF</td>
<td>G69</td>
<td></td>
</tr>
<tr>
<td>Fixed cycle OFF</td>
<td>G80</td>
<td></td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
</tr>
<tr>
<td>Incremental data input</td>
<td>G91</td>
<td></td>
</tr>
<tr>
<td>Inverse time feed</td>
<td>G93</td>
<td></td>
</tr>
<tr>
<td>Feed per minute</td>
<td>G94</td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control OFF</td>
<td>G97</td>
<td></td>
</tr>
<tr>
<td>Initial point level return in hole-machining fixed cycles</td>
<td>G98</td>
<td></td>
</tr>
<tr>
<td>R-point level return in hole-machining fixed cycles</td>
<td>G99</td>
<td></td>
</tr>
<tr>
<td>Single program multi-system control</td>
<td>G109</td>
<td></td>
</tr>
<tr>
<td>Cross machining control OFF</td>
<td>G111</td>
<td></td>
</tr>
<tr>
<td>Hob milling mode OFF</td>
<td>G113</td>
<td></td>
</tr>
<tr>
<td>Radius data input for X-axis ON</td>
<td>G10.9X0</td>
<td></td>
</tr>
</tbody>
</table>

**Note 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).*
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 29-1 Tool offset data items used according to the parameter settings

<table>
<thead>
<tr>
<th>Pattern</th>
<th>Data items used (Display and Data item names)</th>
<th>Parameter</th>
<th>Programming method</th>
</tr>
</thead>
<tbody>
<tr>
<td>[1]</td>
<td>TOOL OFFSET Offset data items</td>
<td>F94 bit 7 0</td>
<td>G43.4/G43.5 with H-code</td>
</tr>
<tr>
<td>[2]</td>
<td>TOOL DATA LENGTH</td>
<td>F93 bit 3 1</td>
<td>T-code</td>
</tr>
<tr>
<td></td>
<td>LENGTH + LENG. No.</td>
<td></td>
<td>T-code + H-code</td>
</tr>
<tr>
<td></td>
<td>LENGTH + LENG. CO.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[3]</td>
<td>TOOL DATA LENG. No.</td>
<td>F94 bit 7 1</td>
<td>G43.4/G43.5 with H-code</td>
</tr>
<tr>
<td></td>
<td>LENG. CO.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>[4]</td>
<td>TOOL OFFSET + TOOL DATA Offset data items + LENGTH</td>
<td>F93 bit 3 1</td>
<td>G43.4/G43.5 with H-code + T-code</td>
</tr>
</tbody>
</table>
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) |
|     | ...       | Machining with length offset function (G43) |
| N10 | G43.4 Xx1 Yy1 Zz1 Bb1 | Tool tip control ON (Note 2) |
| N11 | G01 X_Y_Z_B_C_ | Machining with tip point control (G43.4) |
|     | ...       | 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) |
|     | ...       | Machining with length offset function (G43) |

**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 29-1 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

   ![Diagram](image1)

   In the G43 mode the POSITION value (of the tip position as recognized by the NC) corresponds with the real tip point when B-axis = 0°.

   ![Diagram](image2)

   The POSITION value [1] in the G43 mode does not correspond with the real tip point [2] unless B-axis = 0°.

   ![Diagram](image3)

   Upon change of G43 for G43.4 the real tip point is indicated correctly by the POSITION value.

2. Change of G43.4/G43.5 for G43/G44

   ![Diagram](image4)

   In the G43.4 mode the POSITION value (of the tip position as recognized by the NC) corresponds with the real tip point, irrespective of B-axis position.

   ![Diagram](image5)

   Change of G43.4 for G43 makes the POSITION value [1] different from the real tip point [2] unless B-axis = 0°.

   ![Diagram](image6)

   Correct POSITION indication will not occur in the G43 mode until the B-axis is indexed to the 0° position.
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)
...                                           ...
```

**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 29-1 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.
D. 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. Tool tip point control is not available if the C-axis control of the turning spindle No. 2 is concerned (on accordingly executed machines).

13. 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
14. 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).

15. 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).

16. 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.
29-1-4 Related parameters

1. Offset for the axis of rotation of the rotational axis

   A. Offset for the axis of rotation of the first rotational axis

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Contents</th>
<th>Setting range</th>
<th>Setting unit</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>K122</td>
<td>Axis-of-rotation position on the horizontal axis (X)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td>To be omitted</td>
</tr>
<tr>
<td>K123</td>
<td>Axis-of-rotation position on the vertical axis (Y)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td>To be omitted</td>
</tr>
<tr>
<td>K124</td>
<td>Axis-of-rotation position on the height axis (Z)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td></td>
</tr>
</tbody>
</table>

   B. Offset for the axis of rotation of the second rotational axis

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Contents</th>
<th>Setting range</th>
<th>Setting unit</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>K126</td>
<td>Axis-of-rotation position on the horizontal axis (X)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td></td>
</tr>
<tr>
<td>K127</td>
<td>Axis-of-rotation position on the vertical axis (Y)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td></td>
</tr>
<tr>
<td>K128</td>
<td>Axis-of-rotation position on the height axis (Z)</td>
<td>0 - 9999999999</td>
<td>0.0001 mm</td>
<td>To be omitted</td>
</tr>
</tbody>
</table>

Fig. 29-1 Vertical type
29  FIVE-AXIS MACHINING FUNCTION

2. Selection of programming coordinate system

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
<tbody>
<tr>
<td>F85 bit 2</td>
<td>0: Selection of the table coordinate system</td>
</tr>
<tr>
<td></td>
<td>1: Selection of the workpiece coordinate system</td>
</tr>
</tbody>
</table>

3. Selection of interpolation method for tool tip point control

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
<tbody>
<tr>
<td>F85 bit 3</td>
<td>0: Uniaxial rotation interpolation</td>
</tr>
<tr>
<td></td>
<td>1: Joint interpolation</td>
</tr>
</tbody>
</table>

4. Selection of override scheme for tool tip point control

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
<tbody>
<tr>
<td>F86 bit 2</td>
<td>Overriding the rapid traverse in the mode of tool tip point control is applied</td>
</tr>
<tr>
<td></td>
<td>0: to the speed of motion of the tool tip point</td>
</tr>
<tr>
<td></td>
<td>1: to the speed limit for the axis control’s reference point</td>
</tr>
<tr>
<td>F86 bit 5</td>
<td>Overriding the cutting feed in the mode of tool tip point control is applied</td>
</tr>
<tr>
<td></td>
<td>0: to the speed of motion of the tool tip point</td>
</tr>
<tr>
<td></td>
<td>1: to the speed limit for the axis control’s reference point</td>
</tr>
</tbody>
</table>

5. Selection of reference position on the rotational axis for tool tip point control

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
<tbody>
<tr>
<td>F86 bit 6</td>
<td>0: Angular position at the start of tool tip point control</td>
</tr>
<tr>
<td></td>
<td>1: Position of 0°</td>
</tr>
</tbody>
</table>
6. **Selection of canceling operation by G49 in the mode of tool tip point control**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
</table>
| F114 bit 1 | 0: Cancellation with axis movement for clearing the length offset amount  
  1: Cancellation without axis movement |

7. **Selection of operation for starting up the tool tip point control without motion commands**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
</table>
| F162 bit 0 | 0: Startup with axis movement by the length offset amount  
  1: Startup without axis movement |

8. **Selection of type of passage through singular point for tool tip point control**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
</tr>
</thead>
</table>
| F162 bit 1 | 0: 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).  
  1: 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.) |

9. **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.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
<th>Setting unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>S22</td>
<td>0 - 200000</td>
<td>1 mm/min</td>
</tr>
<tr>
<td></td>
<td>Rotational axis</td>
<td>1 deg/min</td>
</tr>
</tbody>
</table>

10. **Criterion for the neighborhood of singular point**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting range</th>
<th>Setting unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>K110</td>
<td>0 - 360</td>
<td>1 deg</td>
</tr>
</tbody>
</table>
29-2 Inclined-Plane Machining: G68.2, 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.

29-2-1 Function outline

1. Programming format

   A. Inclined-plane machining ON

   \[ \text{G68.2 } Xx \ Yy \ Zz \ I\alpha \ J\beta \ K\gamma \]

   G68.2: Inclined-plane machining ON
   X, Y, Z: Coordinates of the origin of a feature coordinate system. The origin needs to be specified with its absolute values in the workpiece coordinate system.
   I, J, K: Eulerian angles that determine the direction of a feature coordinate system.
   A counterclockwise rotation when viewed from the positive side of the axis of rotation to the origin is defined as positive rotation.

   1. The X-, Y- and Z-coordinates are to be specified with their absolute values in the original workpiece coordinate system. The designation of an axis can be omitted when no shift is required along the particular axis.
   2. The designation of I, J, or K can be omitted when the argument is zero (0: no rotation around the axis concerned).
B. Tool-axis direction control

\texttt{G53.1 \ p}

- **G53.1:** Tool-axis direction control
- **P:** Selection of a solution for the axis of rotation. (See 3 in 29-2-1.)
  - 0: Processed as “P = 1” or “P = 2”, depending on the construction of the machine.
  - Works the same as “P = 1” on this machine system.
  - 1: Solution with a positive angle of rotation on the B-axis.
  - 2: Solution with a negative angle of rotation on the B-axis.

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 instructions, an alarm is caused (\texttt{1808 CANNOT USE G53.1}).
3. The motion caused by a G53.1 code 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 (\texttt{809 ILLEGAL NUMBER INPUT}).

C. Inclined-plane machining OFF (cancellation)

\texttt{G69}  \hspace{1cm} \text{Inclined-plane machining OFF}

2. Setting a feature coordinate system

Enter a block of G68.2 (with arguments for translating and rotating the current workpiece coordinate system) to set a feature coordinate system. Use Eulerian angles to specify the rotation.

\texttt{G68.2 \ Xx \ Yy \ Zz \ I\alpha \ J\beta \ K\gamma} 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 \(\alpha^\circ\).
3) The turned coordinate system is then rotated around the \(X\)-axis by an angle of \(\beta^\circ\).
4) Finally, the coordinate system is rotated around the \(Z\)-axis by an angle of \(\gamma^\circ\).

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.
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. Detailed 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 rotation of the tool-axis or the table. 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. 20. I-90. J-45. K0.
   M98 P100
   G69
   G0 X300. Y0. Z200. B0. C0.
   M98 P100
   G69
   G0 X300. Y0. Z200. B0. C0.
   M98 P100
   G69
   G0 X300. Y0. Z200. B0. C0.
   M98 P100
   G69
   G0 X300. Y0. Z200. B0. C0.
   M98 P100
   G69
   G0 X300. Y0. Z200. B0. C0.
   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
29-2-2 Restrictions

The restrictions on the inclined-plane machining are in general similar to those imposed on the three-dimensional coordinate conversion (G68). Shown below are only the restrictions proper to the inclined-plane machining.

1. Commands usable in the same block with G68.2

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Function</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Incremental data input</td>
<td>G91</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Feed function</td>
<td>F</td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Giving any other command than those enumerated above in the same block as G68.2 will lead to an alarm (1806 ILLEGAL CMD TILTED PLANE CMD).

2. Commands usable in the same block with G53.1

No other commands can be given at all in the block of selecting the tool-axis direction control (G53.1). If a block of G53.1 should contain any other instructions, an alarm is caused (1808 CANNOT USE G53.1).
3. Relationship to other G-codes

A. Commands available in the mode of inclined-plane machining (G68.2)

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Function</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Fixed cycle (Boring 1)</td>
<td>G75</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Fixed cycle (Boring 2)</td>
<td>G76</td>
</tr>
<tr>
<td>Circular interpolation (CW)</td>
<td>G02</td>
<td>Fixed cycle (Back spot facing)</td>
<td>G77</td>
</tr>
<tr>
<td>Circular interpolation (CCW)</td>
<td>G03</td>
<td>Fixed cycle (Boring 3)</td>
<td>G78</td>
</tr>
<tr>
<td>Dwell</td>
<td>G04</td>
<td>Fixed cycle (Boring 4)</td>
<td>G79</td>
</tr>
<tr>
<td>Exact-stop</td>
<td>G09</td>
<td>Fixed cycle OFF</td>
<td>G80</td>
</tr>
<tr>
<td>Selection of XY-plane</td>
<td>G17</td>
<td>Fixed cycle (Deep-hole drilling)</td>
<td>G83</td>
</tr>
<tr>
<td>Selection of ZX-plane</td>
<td>G18</td>
<td>Fixed cycle (Tapping)</td>
<td>G84</td>
</tr>
<tr>
<td>Selection of YZ-plane</td>
<td>G19</td>
<td>Fixed cycle (Synchronous tapping)</td>
<td>G84.2</td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation OFF</td>
<td>G40</td>
<td>Fixed cycle (Synchronous reverse tapping)</td>
<td>G84.3</td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation (left)</td>
<td>G41</td>
<td>Fixed cycle (Reaming)</td>
<td>G85</td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation (right)</td>
<td>G42</td>
<td>Fixed cycle (Boring 5)</td>
<td>G86</td>
</tr>
<tr>
<td>Tool length offset (+)</td>
<td>G43</td>
<td>Fixed cycle (Back boring)</td>
<td>G87</td>
</tr>
<tr>
<td>Tool tip point control type 1</td>
<td>G43.4</td>
<td>Fixed cycle (Boring 6)</td>
<td>G88</td>
</tr>
<tr>
<td>Tool tip point control type 2</td>
<td>G43.5</td>
<td>Fixed cycle (Boring 7)</td>
<td>G89</td>
</tr>
<tr>
<td>Tool length offset (−)</td>
<td>G44</td>
<td>Absolute data input</td>
<td>G90</td>
</tr>
<tr>
<td>Tool length offset OFF</td>
<td>G49</td>
<td>Incremental data input</td>
<td>G91</td>
</tr>
<tr>
<td>Tool position offset, extension</td>
<td>G45</td>
<td>Inverse time feed</td>
<td>G93</td>
</tr>
<tr>
<td>Tool position offset, reduction</td>
<td>G46</td>
<td>Feed per minute (asynchronous)</td>
<td>G94</td>
</tr>
<tr>
<td>Tool position offset, double extension</td>
<td>G47</td>
<td>Feed per revolution (synchronous)</td>
<td>G95</td>
</tr>
<tr>
<td>Tool position offset, double reduction</td>
<td>G48</td>
<td>Initial point level return in hole-machining fixed cycles</td>
<td>G98</td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>G50.1</td>
<td>R-point level return in hole-machining fixed cycles</td>
<td>G99</td>
</tr>
<tr>
<td>Mirror image ON</td>
<td>G51.1</td>
<td>Single program multi-system control</td>
<td>G109</td>
</tr>
<tr>
<td>Tool-axis direction control</td>
<td>G53.1</td>
<td>M, S, T, B output to opposite system</td>
<td>G112</td>
</tr>
<tr>
<td>One-way positioning</td>
<td>G60</td>
<td>Face drilling cycle</td>
<td>G283</td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td>Face tapping cycle</td>
<td>G284</td>
</tr>
<tr>
<td>Geometry compensation</td>
<td>G61.1</td>
<td>Face synchronous tapping cycle</td>
<td>G284.2</td>
</tr>
<tr>
<td>Cutting mode</td>
<td>G64</td>
<td>Face boring cycle</td>
<td>G285</td>
</tr>
<tr>
<td>User macro single call</td>
<td>G65</td>
<td>Outside drilling cycle</td>
<td>G287</td>
</tr>
<tr>
<td>User macro modal call A</td>
<td>G66</td>
<td>Outside tapping cycle</td>
<td>G288</td>
</tr>
<tr>
<td>User macro modal call B</td>
<td>G66.1</td>
<td>Outside synchronous tapping cycle</td>
<td>G288.2</td>
</tr>
<tr>
<td>User macro modal call OFF</td>
<td>G67</td>
<td>Outside boring cycle</td>
<td>G289</td>
</tr>
<tr>
<td>3-D coordinate conversion OFF</td>
<td>G69</td>
<td>Subprogram call/End of subprogram</td>
<td>M98/M99</td>
</tr>
<tr>
<td>Fixed cycle (Chamfering cutter 1, CW)</td>
<td>G71.1</td>
<td>Feed function</td>
<td>F</td>
</tr>
<tr>
<td>Fixed cycle (Chamfering cutter 2, CCW)</td>
<td>G71.2</td>
<td>M, S, T, B function (Note)</td>
<td>MSTB</td>
</tr>
<tr>
<td>Fixed cycle (High-speed deep-hole drilling)</td>
<td>G73</td>
<td>Local variables, Common variables, Operation commands</td>
<td>Macro instructions</td>
</tr>
<tr>
<td>Fixed cycle (Reverse tapping)</td>
<td>G74</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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).

Note: 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).
### B. Modes in which inclined-plane machining (G68.2) is selectable

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Exact stop mode</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Geometry compensation mode</td>
</tr>
<tr>
<td>High-speed machining OFF</td>
<td>G05P0</td>
<td>Automatic corner override</td>
</tr>
<tr>
<td>Radius data input for X-axis ON</td>
<td>G10.9</td>
<td>Tapping mode</td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>G13.1</td>
<td>Cutting mode</td>
</tr>
<tr>
<td>Polar coordinate input OFF</td>
<td>G15</td>
<td>User macro single call</td>
</tr>
<tr>
<td>Selection of XY-plane</td>
<td>G17</td>
<td>User macro modal call OFF</td>
</tr>
<tr>
<td>Selection of ZX-plane</td>
<td>G18</td>
<td>Programmed coordinate rotation OFF</td>
</tr>
<tr>
<td>Selection of YZ-plane</td>
<td>G19</td>
<td>Fixed cycle OFF</td>
</tr>
<tr>
<td>Inch data input</td>
<td>G20</td>
<td>Absolute data input</td>
</tr>
<tr>
<td>Metric data input</td>
<td>G21</td>
<td>Incremental data input</td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23</td>
<td>Inverse time feed</td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation OFF</td>
<td>G40</td>
<td>Feed per minute (asynchronous)</td>
</tr>
<tr>
<td>Tool length offset (+)</td>
<td>G43</td>
<td>Feed per revolution (synchronous)</td>
</tr>
<tr>
<td>Tool length offset (−)</td>
<td>G44</td>
<td>Constant surface speed control OFF</td>
</tr>
<tr>
<td>Tool length offset OFF</td>
<td>G49</td>
<td>Initial point level return in hole-machining fixed cycles</td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>G50</td>
<td>R-point level return in hole-machining fixed cycles</td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>G50.1</td>
<td>Single program multi-system control</td>
</tr>
<tr>
<td>Polygonal machining mode OFF</td>
<td>G50.2</td>
<td>Cross machining control OFF</td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 1</td>
<td>G54</td>
<td>Hob milling mode OFF</td>
</tr>
<tr>
<td>Selection of add. workpiece coordinate systems</td>
<td>G54.1</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 2</td>
<td>G55</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 3</td>
<td>G56</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 4</td>
<td>G57</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 5</td>
<td>G58</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 6</td>
<td>G59</td>
<td></td>
</tr>
</tbody>
</table>

Giving a G68.2 command in a mode other than those enumerated above will lead to an alarm \((1807 \text{ CANNOT USE G68.2}).\)
## Modes in which tool-axis direction control (G53.1) is selectable

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Function</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Exact stop mode</td>
<td>G61</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Geometry compensation mode</td>
<td>G61.1</td>
</tr>
<tr>
<td>High-speed machining OFF</td>
<td>G05P0</td>
<td>Cutting mode</td>
<td>G64</td>
</tr>
<tr>
<td>Radius data input for X-axis ON</td>
<td>G10.9</td>
<td>User macro single call</td>
<td>G65</td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>G13.1</td>
<td>User macro modal call OFF</td>
<td>G67</td>
</tr>
<tr>
<td>Polar coordinate input OFF</td>
<td>G15</td>
<td>Inclined-plane machining</td>
<td>G68.2</td>
</tr>
<tr>
<td>Selection of XY-plane</td>
<td>G17</td>
<td>Fixed cycle OFF</td>
<td>G80</td>
</tr>
<tr>
<td>Selection of ZX-plane</td>
<td>G18</td>
<td>Absolute data input</td>
<td>G90</td>
</tr>
<tr>
<td>Selection of YZ-plane</td>
<td>G19</td>
<td>Incremental data input</td>
<td>G91</td>
</tr>
<tr>
<td>Inch data input</td>
<td>G20</td>
<td>Inverse time feed</td>
<td>G93</td>
</tr>
<tr>
<td>Metric data input</td>
<td>G21</td>
<td>Feed per minute (asynchronous)</td>
<td>G94</td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23</td>
<td>Feed per revolution (synchronous)</td>
<td>G95</td>
</tr>
<tr>
<td>Nose radius/Tool radius compensation OFF</td>
<td>G40</td>
<td>Constant surface speed control OFF</td>
<td>G97</td>
</tr>
<tr>
<td>Tool length offset (+)</td>
<td>G43</td>
<td>Initial point level return in hole-machining fixed cycles</td>
<td>G98</td>
</tr>
<tr>
<td>Tool length offset (−)</td>
<td>G44</td>
<td>R-point level return in hole-machining fixed cycles</td>
<td>G99</td>
</tr>
<tr>
<td>Tool length offset OFF</td>
<td>G49</td>
<td>Single program multi-system control</td>
<td>G109</td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>G50</td>
<td>Cross machining control</td>
<td>G111</td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>G50.1</td>
<td>Hob milling mode OFF</td>
<td>G113</td>
</tr>
<tr>
<td>Polygonal machining mode OFF</td>
<td>G50.2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 1</td>
<td>G54</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of add. workpiece coordinate systems</td>
<td>G54.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 2</td>
<td>G55</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 3</td>
<td>G56</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 4</td>
<td>G57</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 5</td>
<td>G58</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system 6</td>
<td>G59</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Giving a G53.1 command in a mode other than those enumerated above will lead to an alarm (*1808 CANNOT USE G53.1*).
4. Supplementary precautions

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 G68.2, system variables #5041 to #5054 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 G68.2 mode cancels the mode of inclined-plane machining and makes the G69 code modal in the group concerned.

4. External deceleration signal works only on the actually controlled axes of the machine coordinate system, not on those of the feature coordinate system.

5. Giving a positioning command with G28, G30, G53, etc., which is to be executed with reference to the machine coordinate system, will only lead to an alarm (1806 ILLEGAL CMD TILTED PLANE CMD).

6. Tool radius compensation (G41, G42, G40), mirroring by G-code (G51.1, G50.1), and fixed cycle must be selected and cancelled within the mode of inclined-plane machining.

   \[
   \begin{align*}
   & G68.2 \ X_0\ Y_0\ Z_0\ I_0\ J_0\ K_0 \quad \text{(Inclined-plane machining ON)} \\
   & G41 \ D1 \quad \text{(Tool radius compensation ON)} \\
   & G40 \quad \text{(Tool radius compensation OFF)} \\
   & G69 \quad \text{(Inclined-plane machining OFF)}
   \end{align*}
   \]

7. The execution of a G68.2 command 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.

Before the G68.2 command is entered, the real position of the tip point is in agreement with the positional indication.

If a new coordinate system is set by a G68.2 command, the tool length offset is internally applied to the Z-axis direction of the new coordinate system, resulting in a disagreement between the real tip point and its positional indication.

The G53.1 command really adjusts the direction of the tool axis to the Z-axis of the new coordinate system and, as a result, clears the disagreement in question.

D740PB0052
8. Do not use a G68.2 command in a subprogram of the eighth nesting level; otherwise an alarm will be caused (842 SUB PROGRAM NESTING EXCEEDED).

9. A G53.1 command block is in reality composed of two processes: motions on the rotational axes and redefinition of the feature coordinate system. In the single-block operation mode, therefore, press the start button twice to complete a G53.1 block.

10. Manual interruption is always performed on the basis of the machine coordinate system (without any coordinate conversion).

11. 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).

12. Tool path check can only be performed on the basis of the original workpiece coordinate system (without coordinate conversion taken into consideration).

13. Inclined-plane machining is not available if the C-axis control of the turning spindle No. 2 is concerned (to be used as the first or second rotational axis).

14. As for restarting operation, do not specify a block within the G68.2 mode (nor a block of cancellation [G69]) as a restarting position; otherwise an alarm will be caused (956 RESTART OPERATION NOT ALLOWED).

---

**Sample program**

```
N10 G00 X_Y_Z_
N11 G00 X_Y_Z_B_C
N20 G68.2 X_Y_Z_I_J_K
N21 G01 X_Y_Z_F_
N22 G01 X_Y_Z_F_
N23 G69
N30 G90 G00 X_Y_
N31 G90 G00 Z_
N32 T_T_M6
```

The program can be restarted from blocks Nos. 10, 11 and 32.

The program cannot be restarted from blocks Nos. 20 to 23.

* The program cannot be restarted from the G69 block, either.

The program cannot be restarted from blocks Nos. 30 and 31. (The program cannot be restarted even after G69 until motions on the X-, Y- and Z-axes are specified with absolute coordinates.)

---

### 29-2-3 Related alarms

<table>
<thead>
<tr>
<th>No.</th>
<th>Message</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1805</td>
<td>G68.2 OPTION NOT FOUND</td>
<td>The system of machining is not equipped with the optional function for inclined-plane machining.</td>
</tr>
<tr>
<td>1806</td>
<td>ILLEGAL CMD TILTED PLANE CMD</td>
<td>The command given is not compatible with the mode of inclined-plane machining.</td>
</tr>
<tr>
<td>1807</td>
<td>CAN NOT USE G68.2</td>
<td>The mode of inclined-plane machining is not selectable under the current modal conditions.</td>
</tr>
<tr>
<td>1808</td>
<td>CAN NOT USE G53.1</td>
<td>The command of tool-axis direction control is not available under the current modal conditions.</td>
</tr>
</tbody>
</table>
29-3 Tool Radius Compensation for Five-Axis Machining (Option)

29-3-1 Function outline

The function described in this section refers to three-dimensional tool radius compensation on a five-axis control machine (with the B- and C-axis for rotating the tool axis and the table, respectively) by computing the offset vector in a plane perpendicular to the tool axis.

![Diagram of tool radius compensation](D736P0522)

Fig. 29-3  Tool radius compensation for five-axis machining

29-3-2 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 29-3-4. Refer to Section 12-5 for a detailed description of the general tool radius compensation.

1. Programming format

   A. Tool radius compensation for five-axis machining ON

```
G41.5 (G41.2) (X_ Y_ Z_ B_ C_ D_);
G42.5 (G42.2) (X_ Y_ Z_ B_ C_ D_);
```

   G41.5 : Tool radius compensation (to the left)
   G42.5 : Tool radius compensation (to the right)
   XYZBC : Axis motion commands
   D : Tool offset data No. for radius compensation

   B. Tool radius compensation for five-axis machining OFF (cancellation)

```
G40 (X_ Y_ Z_ 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).

<table>
<thead>
<tr>
<th>Parameter Data in the TOOL DATA display</th>
<th>Data in the TOOL OFFSET display</th>
</tr>
</thead>
<tbody>
<tr>
<td>F92 bit 7</td>
<td>F94 bit 7</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

O: Used for tool radius compensation.
×: Not used.

29-3-3 Operation of tool radius compensation for five-axis machining

1. Startup of the tool radius compensation

The G41.5 or 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 G41.5 or G42.5 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.

Take care to give the startup code, G41.5 or G42.5, under the appropriate conditions of G-codes (see the table concerned in Subsection 29-3-5); otherwise an alarm will be caused (962 CANNOT USE 5X RADIUS COMP.).

2. Operation in the mode of tool radius compensation

In the mode of G41.5 or G42.5 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 29-3-5); 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,
2. Zero is specified as the number of offset data for radius compensation (D00), or
3. 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.
29-3-4 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. 29-4  Table coordinate system](image)

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. 29-5  Conversion of points into the compensation plane](image)
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. 29-6  Compensation in the compensation plane
29-3-5 Relationship to other functions

1. Commands usable in the same block with G41.5/G42.5

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Function</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td>Incremental data input</td>
<td>G91</td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td>Feed function</td>
<td>F</td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Giving any other command than those enumerated above in the same block as G41.5/G42.5 will lead to an alarm (961 ILLEGAL COMMAND 5X RADIUS COMP.).

2. Relationship to other commands

The compatibility of tool radius compensation for five-axis machining with other commands is as follows.

A. Commands available in the mode of tool radius compensation for five-axis machining

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td></td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td></td>
</tr>
<tr>
<td>Dwell</td>
<td>G04</td>
<td></td>
</tr>
<tr>
<td>Exact-stop</td>
<td>G09</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check ON</td>
<td>G22</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation OFF</td>
<td>G40</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right/left)</td>
<td>G41.2/G42.2</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right/left)</td>
<td>G41.5/G42.5</td>
<td></td>
</tr>
<tr>
<td>Tool length offset (+/-)</td>
<td>G43/G44</td>
<td></td>
</tr>
<tr>
<td>Tool tip point control type 1</td>
<td>G43.4</td>
<td></td>
</tr>
<tr>
<td>Tool length offset OFF</td>
<td>G49</td>
<td></td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td></td>
</tr>
<tr>
<td>Geometry compensation mode</td>
<td>G61.1</td>
<td></td>
</tr>
<tr>
<td>Cutting mode</td>
<td>G64</td>
<td></td>
</tr>
<tr>
<td>Macro call</td>
<td>G65</td>
<td></td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
</tr>
<tr>
<td>Incremental data input</td>
<td>G91</td>
<td></td>
</tr>
<tr>
<td>Inverse time feed</td>
<td>G93</td>
<td></td>
</tr>
<tr>
<td>Feed per minute</td>
<td>G94</td>
<td></td>
</tr>
<tr>
<td>Feed per revolution</td>
<td>G95</td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control ON</td>
<td>G96</td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control OFF</td>
<td>G97</td>
<td></td>
</tr>
<tr>
<td>M, S, T, B output to opposite system</td>
<td>G112</td>
<td>Note</td>
</tr>
<tr>
<td>Subprogram call/End of subprogram</td>
<td>M98/M99</td>
<td></td>
</tr>
<tr>
<td>Feed function</td>
<td>F</td>
<td></td>
</tr>
<tr>
<td>M, S, T, B function</td>
<td>MSTB</td>
<td>Note</td>
</tr>
<tr>
<td>Local variables, Common variables, Operation commands (arithmetic operations, trigonometric functions, square root, etc), Control commands (IF – GOTO –, WHILE – DO –)</td>
<td>Macro instructions</td>
<td></td>
</tr>
</tbody>
</table>

Note: 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.).
### B. Modes in which tool radius compensation for five-axis machining is selectable

<table>
<thead>
<tr>
<th>Function</th>
<th>Code</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning</td>
<td>G00</td>
<td></td>
</tr>
<tr>
<td>Linear interpolation</td>
<td>G01</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate interpolation OFF</td>
<td>G13.1</td>
<td></td>
</tr>
<tr>
<td>Plane selection</td>
<td>G17, G18, G19</td>
<td></td>
</tr>
<tr>
<td>Inch data input</td>
<td>G20</td>
<td></td>
</tr>
<tr>
<td>Metric data input</td>
<td>G21</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check ON</td>
<td>G22</td>
<td></td>
</tr>
<tr>
<td>Pre-move stroke check OFF</td>
<td>G23</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation OFF</td>
<td>G40</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right/left)</td>
<td>G41.2/G42.2</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right/left)</td>
<td>G41.4/G42.4</td>
<td></td>
</tr>
<tr>
<td>Tool radius compensation for five-axis machining (to the right/left)</td>
<td>G41.5/G42.5</td>
<td></td>
</tr>
<tr>
<td>Tool length offset (+/-)</td>
<td>G43/G44</td>
<td></td>
</tr>
<tr>
<td>Tool tip point control type 1</td>
<td>G43.4</td>
<td></td>
</tr>
<tr>
<td>Tool length offset OFF</td>
<td>G49</td>
<td></td>
</tr>
<tr>
<td>Scaling OFF</td>
<td>G50</td>
<td></td>
</tr>
<tr>
<td>Mirror image OFF</td>
<td>G50.1</td>
<td></td>
</tr>
<tr>
<td>Polygonal machining mode OFF</td>
<td>G50.2</td>
<td></td>
</tr>
<tr>
<td>Selection of workpiece coordinate system/additional workpiece coordinate system</td>
<td>G54-59/G54.1</td>
<td></td>
</tr>
<tr>
<td>Exact stop mode</td>
<td>G61</td>
<td></td>
</tr>
<tr>
<td>Geometry compensation mode</td>
<td>G61.1</td>
<td></td>
</tr>
<tr>
<td>Cutting mode</td>
<td>G64</td>
<td></td>
</tr>
<tr>
<td>Macro call</td>
<td>G65</td>
<td></td>
</tr>
<tr>
<td>Modal macro call OFF</td>
<td>G67</td>
<td></td>
</tr>
<tr>
<td>3-D coordinate conversion OFF</td>
<td>G69</td>
<td></td>
</tr>
<tr>
<td>Fixed cycle OFF</td>
<td>G80</td>
<td></td>
</tr>
<tr>
<td>Absolute data input</td>
<td>G90</td>
<td></td>
</tr>
<tr>
<td>Incremental data input</td>
<td>G91</td>
<td></td>
</tr>
<tr>
<td>Inverse time feed</td>
<td>G93</td>
<td></td>
</tr>
<tr>
<td>Feed per minute</td>
<td>G94</td>
<td></td>
</tr>
<tr>
<td>Feed per revolution</td>
<td>G95</td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control ON</td>
<td>G96</td>
<td></td>
</tr>
<tr>
<td>Constant surface speed control OFF</td>
<td>G97</td>
<td></td>
</tr>
<tr>
<td>Initial point level return in hole-machining fixed cycles</td>
<td>G98</td>
<td></td>
</tr>
<tr>
<td>R-point level return in hole-machining fixed cycles</td>
<td>G99</td>
<td></td>
</tr>
<tr>
<td>Single program multi-system control</td>
<td>G109</td>
<td></td>
</tr>
<tr>
<td>Cross machining control OFF</td>
<td>G111</td>
<td></td>
</tr>
<tr>
<td>Hob milling mode OFF</td>
<td>G113</td>
<td></td>
</tr>
<tr>
<td>Selection for radius data input</td>
<td>G10.9</td>
<td></td>
</tr>
<tr>
<td>Polar coordinate input OFF</td>
<td>G15</td>
<td></td>
</tr>
</tbody>
</table>

### 3. 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 inter-
  polate 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. Tool radius compensation for five-axis machining is not available if the C-axis control of the
   turning spindle No. 2 is concerned (on accordingly executed machines).

7. The function in question cannot be used at all in the mode of operation for turning.

8. 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>
   
   G43.4 H1  (Tool tip point control ON)
   ...
   G41.5 D2  (Tool radius compensation ON)
   ...
   ...
   G40       (Tool radius compensation OFF)
   ...
   G49       (Tool tip point control OFF)

   - The tool tip point control must have the workpiece coordinate system selected (with F85
     bit 2 = 1) for describing the tool path in the program. Otherwise (i.e. when the parameter
     F85 bit 2 is set to 0 to use the table coordinate system in programming) an alarm will be
     caused (962 CANNOT USE 5X RADIUS COMP.) by the selection of the mode of tool
     radius compensation for five-axis machining even with the above condition being satisfied.

9. 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)

29-3-6 Related alarms

<table>
<thead>
<tr>
<th>No.</th>
<th>Message</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>936</td>
<td>OPTION NOT FOUND</td>
<td>The system of machining is not equipped with the optional function of tool radius compensation for five-axis machining.</td>
</tr>
<tr>
<td>961</td>
<td>ILLEGAL COMMAND 5X RADIUS COMP.</td>
<td>The command given is not compatible with the mode of tool radius compensation for five-axis machining.</td>
</tr>
<tr>
<td>962</td>
<td>CANNOT USE 5X RADIUS COMP.</td>
<td>Tool radius compensation for five-axis machining is not selectable under the current modal conditions.</td>
</tr>
</tbody>
</table>
Example 1:

3 × φ12 (0.47) hole to be drilled 15 (0.59) deep

N001 (SPOT D16)
G90G80G40G49G00 Initial setting
G91G28Z0 Zero point return (Z-axis)
G28X0Y0 Zero point return (X- and Y-axes)
T01T00M06 Tool change
G90G54S1590M03 Coordinate system setting, Spindle start
G43Z50.H01M08 Movement to initial point, Tool length offset
G80G00250.M09 Fixed-cycle cancellation, Movement to initial point
G91G28Z0 Zero point return (Z-axis)
G28X0Y0 Zero point return (X- and Y-axes)
M01 Optional stop

N002 (DRL D12)
G90G80G40G49G00 Initial setting
G91G28Z0 Zero point return (Z-axis)
G28X0Y0 Zero point return (X- and Y-axes)
T02T00M06 Tool change
G90G54S1590M03 Coordinate system setting, Spindle start
G43Z50.H02M08 Movement to initial point, Tool length offset
G80G00250.M09 Fixed-cycle cancellation, Movement to initial point
G91G28Z0 Zero point return (Z-axis)
G28X0Y0 Zero point return (X- and Y-axes)
M30
Example 2:

N001 (F-MIL D100)
G55G90G94G40G49
G91G28Y0Z0
T01T00M06
G90G00S480M03
G43Z50.H01M08
G00Z.3
G01X110.F290
Y-30.
X-110.
G00Z50.
Z0
G01X110.
G00Z50.
Z0
G01X110.
G00Z50.
G91G28Y0Z0
M01

N002 (SPOT D16)
G55G90G94G40G49
G91G28Y0Z0
T02T00M06
G90G00S1500M03
G43Z50.H02M08
G99G81Z-5.R3.F127
X35.
X0Y0
G80M09
G91G28Y0Z0
M01

N003 (DRL D6.8)
G55G90G94G40G49
G91G28Y0Z0
T03T00M06
G90G00S935M03
G43Z50.H03M08
G99G82Z-12.R3.F75
X35.
G80M09
G91G28Y0Z0
M01

N004 (DRL D39.5)
G55G90G94G40G49
G91G28Y0Z0
T04T00M06
G90G00S185M03
X0Y0
G43Z50.H04M08
G99G82-22.R3.F56
G80M09
G91G28Y0Z0
M01

N005 (BOR D40.)
G55G90G94G40G49
G91G28Y0Z0
T05T00M06
G90G00S720M03
G43Z50.H05M08
G99G76Z-10.R3.F80
G80M09
G91G28Y0Z0
M01

N006 (TAP M8)
G55G90G94G40G49
G91G28Y0Z0
T06T00M06
M00
G90G00S280M03
G43Z50.H06
G99G84Z-16.25R3.F350H0
X35.
G80M09
G91G28Y0Z0
G28X0
T00T00M06
M30
**<Tool offset>**

Set tool length offset data and tool radius compensation data.

![Tool Offset Diagram](https://via.placeholder.com/150)

**<Work offset>**

Set in the workpiece coordinate system (G54 to G59) the distance from the machine zero point to the workpiece zero point.

![Work Offset Diagram](https://via.placeholder.com/150)
Example 3:

4 × φ9 through hole
φ14 counterbore, 10 deep

Unit: mm

MEP239
N001 (E-MIL D20)
G90G80G40G00G49
G91G28Y020
T01T00M06
G90G54G00X40.Y-20.
S320M03
G43Z50.H01M08
G002-10.
G42G01X303Y10.D11F64
Y70.
X89.7F1000
Y-20.F64
X30.F1000
S340M03
G01Y70.F40
X90.F1000
Y-20.F40
G40G00Z50.M09
G91G28Y020
M01

N002 (SPOT D16)
G90G80G40G00G49
G91G28Y020
T02T00M06
G90G54G00X15.Y40.
S1590M03
G43Z50.H02M08
G99G81-5.R3.F127
M98H008
G80G0250.
X60.Y25.
G98G81-25.R-17.F127
G80G0250.M09
G91G28Y020
M01

N003 (DRL D9.)
G90G80G40G00G49
G91G28Y020
T03T00M06
G90G54G00X15.Y40.
S750M03
G43Z50.H03M08
M98H008
G80G00Z50.M09
G91G28Y020
M01

N004 (DRL D29.5)
G90G80G40G00G49
G91G28Y020
T04T00M06
G90G54G00X60.Y25.
S250M03
G43Z50.H04M08
G80G0250.M09
G91G28Y020
M01

N005 (E-MIL D14)
G90G80G40G00G49
G91G28Y020
T05T00M06
G90G54G00X15.Y40.
S420M03
G43Z50.H05M08
M98H008
G80G0250.
G91G28Y020
M01

N006 (BOR D30.)
G90G80G40G00G49
G91G28Y020
T06T00M06
G90G54G00X60.Y25.
S900M03
G43Z50.H06M08
G98G76-30.R-17.F72
G80G0250.M09
G91G28Y020
M01

N007 (MENTORI D30)
G90G80G40G00G49
G91G28Y020
T07T00M06
G90G54G00X15.Y40.
S2100M03
G43Z50.H07M08
G99G81-2.5R3.F530
M98H008
G80G00Z50
G00X60.Y25.
G71.1Z-25.R-17.Q1.F530
G80G0250.M09
G91G28Y020
G28X0
M30

N008 (SUB PRO)
Y10.
X105.
Y40.
M99
- NOTE -
31 EIA/ISO PROGRAM DISPLAY

This chapter describes general procedures for and notes on constructing an EIA/ISO program newly, and then editing functions.

31-1 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 displayed in reverse to show the WORK No. SELECT window.

   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.
    - 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.
31-2 Editing Function of EIA/ISO PROGRAM Display

31-2-1 General

Establishing a constructing mode on the PROGRAM (EIA/ISO) display allows the following menu to be displayed as an initial one.

<table>
<thead>
<tr>
<th>PROGRAM COMPLETE</th>
<th>SEARCH</th>
<th>COPY</th>
<th>ALTER</th>
<th>ERASE</th>
<th>MOVE</th>
<th>FIND &amp; REPLACE</th>
<th>CHANGE PROGRAM</th>
<th>MACRO INPUT</th>
<th>MACRO VARIABLE</th>
</tr>
</thead>
</table>

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.

31-2-2 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 displayed in the reverse status 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 reverse-display status.

   (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 reverse the display status.

(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 displayed in reverse and the [ERASE] menu item is also displayed in reverse.

(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 displayed in reverse, which indicates that the reversed portion provides the object of erasure.

Example:

```
N001  G00 X10.
      IZ10.;
      G00 X100.
      G00 Z20.
N002  M08
      M03
```

(4) Press the input key.
  ➔ The character string displayed in reverse in (3) is erased.

Example:

```
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 displayed in reverse.
   (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 displayed in reverse.
   (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.
         ➔ The WORK No. SELECT window is displayed and the [PROGRAM COPY] menu item is displayed in reverse.
      (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 displayed in reverse and the [LINE(S) COPY] menu item is also displayed in reverse.

(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 displayed in reverse, which indicates that the reversed portion provides the object of copying.

Example:

(5) Press the input key.
   ➔ The area displayed in reverse 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 displayed in reverse does not change.

Example:

(7) Press the input key.
   ➔ The character string displayed in reverse is copied at the cursor position.

Example: (Continued)

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 displayed in reverse 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 and the [MOVE] menu item is also displayed in reverse.

(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 displayed in reverse, which indicates that the reversed portion provides the object of moving.

N001  G00 X10.  Z10.
N002  M08
M03

(4) Press the input key.

→ The area displayed in reverse 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 displayed in reverse does not change.

Example: (Continued)

N001  G00 X10.  Z10.
N002  M08
M03
G00 X100.
G00 Z20.

(6) Press the input key.

→ The character string displayed in reverse is moved to the cursor position.

Example: (Continued)

N001  G00 X10.
N002  M08
M03
G00 X100.
G00 Z20. M03

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 displayed in reverse.

(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.
31-3 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 will be opened.

- 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.
31-4 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 will be highlighted and the WORK No. SELECT window will appear.

   (3) Select the work number of the program to be displayed.
      
      ➔ The screen will be divided into the left and right part. One and the same section of the program is initially displayed in both parts.

   - 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 highlighted display of the menu item will be released and the division of the screen cancelled.

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.

(1) Press the [CHANGE PROGRAM] menu key.

- The highlighting of the title will be 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.).
31-5 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.

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.