PROGRAMMING MANUAL
for
MAZATROL FUSION 640MT Pro
- MAZATROL PROGRAMMING -

MANUAL No. : H737PA0014E

Serial No. :

Before using this machine and equipment, fully understand the contents of this manual to ensure proper operation. Should any questions arise, please ask the nearest Technical/Service 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/Service 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/Service 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/Service Center with the manual No. or the machine name, serial No. and manual name.

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

09. 2004

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

WARNING

- 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, solid check, 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

WARNING

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

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

Programming

WARNING

- 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 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 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 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. (This precaution applies only to the M640MT/MT 5X/T/T NEXUS/TN and M640M Pro/MT Pro.)
- The system of G-code and M-code commands differs between the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060) and the machines equipped with M640MT/MT 5X/T/T NEXUS/TN/MT Pro (such as the INTGEREX non e-Series, the SQT Series, the MPX Series and the QTN Series).
  Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

Sample program	Machine with M640M Pro	Machine with M640MT/MT 5X/T/ T NEXUS/TN/MT Pro
`S1000M3`	The milling spindle rotates at 1000 min^{-1}.	The turning spindle rotates at 1000 min^{-1}.
`S1000M203`	The turning spindle rotates at 1000 min^{-1}.	The milling spindle rotates at 1000 min^{-1}.

- For the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060), programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA 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 solid check function, and other functions, and confirm that the program operates properly. The modification of tool data may cause even a field-proven machining program to change in operational status.
  If the user operates the machine without being aware of any changes in program status, interference with the workpiece could arise from unexpected operation.
  For example, if the cutting edge of the tool during the start of automatic operation is present inside the clearance-including blank (unmachined workpiece) specified in the common unit of the MAZATROL program, care is required since the tool will directly move from that position to the approach point because of no obstructions being judged to be present on this path.
  For this reason, before starting automatic operation, make sure that the cutting edge of the tool during the start of automatic operation is present outside the clearance-including workpiece specified in the common unit of the MAZATROL program.

CAUTION

- If axis-by-axis independent positioning is selected and simultaneously rapid feed selected for each axis, movements to the ending point will not usually become linear. Before using these functions, therefore, make sure that no obstructions are present on the path.
- If the machine employs sliding surface structure, lubrication may prove to be insufficient during continuous microfeed machining (see Note 1 below), and in the worst case, seizure of the sliding surface could result. For these reasons, the sliding surface needs to be maintained in a well-lubricated condition during such machining by, for example, inserting an oil-film forming program (see Note 2 below).

List of applicable models and intended axes (Models that employ sliding surface structure)

Classification	Machine model	Axes with sliding surface structure
Lathes	INTEGREX 50Y	X-axis, Y-axis, Z-axis
	INTEGREX 50YB	X-axis, Y-axis, Z-axis
	INTEGREX 70Y	X-axis, Y-axis, Z-axis
	INTEGREX 70YB	X-axis, Y-axis, Z-axis
	SLANT TURN 450	X-axis, Z-axis
	SLANT TURN 50N	X-axis, Z-axis
	SLANT TURN 60N	X-axis, Z-axis
	SLANT TURN 80N	X-axis, Z-axis
	TURNING CENTER M-4N	X-axis, Z-axis
	TURNING CENTER M-5N	X-axis, Z-axis
	POWER MASTER	X-axis, Z-axis
	QUICK TURN 40	X-axis, Z-axis
	MEGA TURN series	X-axis, Z-axis
	SUPER QUADREX 200/250	Z2-axis
	SUPER QUICK TURN 200/250MY	Y-axis
	SUPER QUICK TURN 300MY	Y-axis
Vertical machining centers	FJV-35/50/60	Z-axis
	MTV-515/655/815	Z-axis
	V-40/60	Z-axis

For further details and more specific examples, refer to the relevant Machine Operating Manual, Part 4, Section 1-2, "Precautions for Microfeed Machining (Models that Employ Sliding Surface Structure)".

Note 1: Continuous microfeed machining refers to the operation in which the movement of the intended feed axis through strokes shorter than those required for lubrication is continuously repeated.

Note 2: The oil-film forming program refers to a program that creates an oil film on the sliding surface by moving the intended machining axis over a long stroke during machining.

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 100 times and the handle operated inadvertently, axial movement will become faster than that expected.

OPERATIONAL WARRANTY FOR THE NC UNIT

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: 5° to 40°C (41° to 104°F)

Note: When power is turned on, if the thermal sensor detects an ambient temperature under 5°C, the hard disk warm-up status indicator lamp will light up and the NC unit will not start operating at once. After automatic heating of the hard disk by its internal heater, the lamp will go out and the NC unit will start. It takes about 20 minutes for temperature to increase from 0 to 5°C in order to avoid condensation due to sudden changes in temperature.

2. Relative humidity

During machine operation: 30 to 75 % (without bedewing)

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

- NOTE -

CONTENTS

- NOTE -

1 MAZATROL PROGRAM CONFIGURATION

MAZATROL programs are each made up of a set of data referred to as unit.

The following types of units are prepared for this NC equipment:

- Common unit
- Materials shape unit
- Machining unit
  - Milling unit
    - Point machining unit
      - Drilling
      - Counterbore machining
      - Inversed faced hole machining
      - Reaming
      - Tapping
      - Boring
        - Through hole
        - Non-through hole
        - Stepped through hole
        - Stepped non-through hole
      - Back boring
      - Circular milling
      - Counterbore-tapping
    - Line machining unit
      - Central linear machining
      - Right-hand linear machining
      - Left-hand linear machining
      - Outside linear machining
      - Inside linear machining
      - Right-hand chamfering
      - Left-hand chamfering
      - Outside chamfering
      - Inside chamfering
    - Face machining unit
      - Face milling
      - End milling top
      - Pocket milling
      - End milling slot
    - Milling manual program machining
  - Turning unit
    - Bar-materials machining
    - Copy-machining
    - Corner-machining
    - Facing
    - Threading
    - Grooving
    - Drilling (turning)
    - Tapping (turning)
    - Turning manual program machining
- M-code unit
- Coordinate measuring unit
- Workpiece measuring unit
- Tool measuring unit
- Head selection unit
- Workpiece transfer unit
- Subprogram unit
- Simultaneous machining unit
- End unit

Data to be set in the units listed above is classified into the following three major types:

1. Unit data
   The data consists of data on the type of machining and the sections to be machined, etc.
2. Tool sequence data
   The tool sequence data consists of tool names and other data relating to the operation of the tools. This type of data exists for the milling (point, linear, and face machining) and turning units. For other units, data relating to tools exists with the unit data.
3. Shape sequence data
   The data consists mainly of data used to define machining patterns.
4. TPC data (Tool path control data)
   TPC data is the auxiliary data to be set on the **TPC** display. The data consists of tool path/tool change position adjustment data, M-codes, tool offset numbers, etc. Tool paths are automatically generated according to the data set on the **PROGRAM** display and various parameters. TPC data is intended to eliminate unnecessary paths by changing thus-generated tool paths on an unit-by-unit basis. Machining itself, therefore, will be executed even if TPC data is not set.

Example: PROGRAM display

```
UNo. MAT OD-MAX  ID-MIN LENGTH  WORK FACE   RPM
 0   FC   70.     0.     97.               3000             Unit data

UNo. UNIT   MODE  POS-B  POS-C    DIA   DEPTH   CHMF
 1  DRILLING ZC    ◆      ◆       10.    20.     0.         Unit data

SNo.TOOL NOM-ϕ  HOLE-ϕ HOLE-DEP PRE-DIA PRE-DEP RGH  DEPTH C-SP FR M M   Tool sequence data
 1 CTR-DR  12.   10.      ◆       ◆        ◆     90.    ◆    37   6.
 2  DRILL  10.   10.     20.      0.     100  DRIL T   5.    37  0.
FIG SHP   SPT-R/x  SPT-C/y  SPT-Z  SPT-Y    NUM.    ANGLE   Q     R   Shape sequence data
 1   PT      0.       0.      0.     0.      ◆        ◆     ◆
UNo. UNIT MODE    POS-B  POS-C SRV-A SLOT-WID BTM   WAL FIN-A FIN-R   Unit data
 2   SLOT  ZY      ◆      90.   31.    28.     2     2    0.    0.
SNo. TOOL    NOM-ϕ APRCH-1 APRCH-2 TYPE  AFD   DEP-A  DEP-R C-SP FR M M   Tool sequence data
 R1 END MILL  20.     ?       ?     CW   G01           ◆
 R2 END MILL   1.     ?       ?     CW   G01     3.    ◆
FIG SHP   SPT-R     Z      Y       RADIUS/th    I   J   P  CNR  RGH   Shape sequence data
 1  LINE   0.       0.    0.
 2  LINE   ◆       77.    0.
```

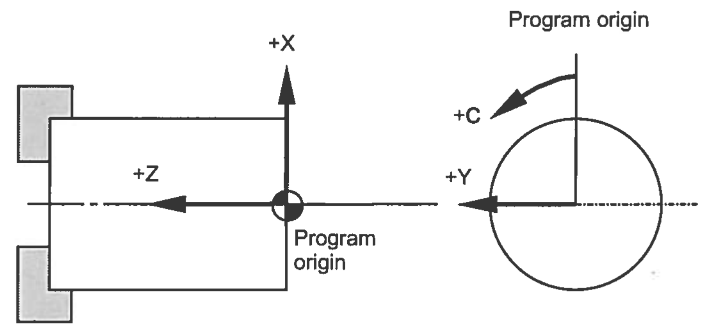
Specific details and setting procedures of each data are described in Chapter 3. Here (Chapter 1), you should understand what types of units and data constitute a program.

Note: Specify tools in program by their tool names, nominal diameters and suffixes.
Specify tools in the tool sequence data.
To operate the machine in the automatic operation mode, the tools that have been specified in the program must be registered on the **TOOL DATA** display.

2 PROGRAM COORDINATE SYSTEM

In general, machining dimensions on a drawing are indicated as the distances from a specific reference point. Likewise, within a program, a machining pattern is defined by setting the coordinates from a specific reference point. This reference point is referred to as the program origin and the coordinate system based on the program origin is referred to as the program coordinate system.

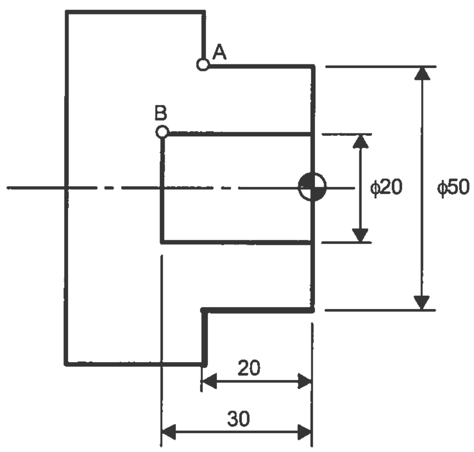
For MAZATROL programs, the following coordinate system is used to define machining patterns:



T3P001

The program origin of X-Z-coordinates system can be set anywhere on the center line of the workpiece. Usually, however, the crossing point of the center line of the workpiece and its finishing edge surface should be taken as the program origin. The program origin of C-axis (rotational axis) can be set at any position convenient for programming. For MAZATROL programs, set X-coordinates as diameter data. That is, the workpiece diameter indicated on the drawing must be set as it is.

Example: For the workpiece shape shown in the diagram below:
The coordinates (x, z) of point A are (50, 20), and the coordinates (x, z) of point B (20, 30).



T3P008

Note 1: For turning manual program machining units (**MANUAL. P**), milling manual program machining units (**M-MANUAL**) and facing units (**FACING**), the direction of Z-axis is opposite to the one shown in the diagram above. See the relevant items in Chapter 3 for further details.

Note 2: Refer to the sections of milling units for details on the C- and Y-axes.

- NOTE -

3 PROGRAM CREATION

Both the program data and sequence data within a MAZATROL program must be set on the **PROGRAM** display, and TPC data must be set on the **TPC** display. The TPC display is called up from the **PROGRAM** display.

This section first describes general procedures and precautions related to creating a MAZATROL program and then describes detailed procedures for setting each type of program data on a unit-by-unit basis.

Operating procedure for creating a MAZATROL program

(1) Select the **PROGRAM** display.

- Carry out the following operations to call up the **PROGRAM** display:

1) Press the display selector key.

➔ You will then see the following main-display selection menu in the menu display area of your screen:

POSITION	SET UP INFO	PROGRAM	TOOL DATA	CUTTING COND.	PARAM	DIANOS	DATA IN/OUT	TOOL LAYOUT	DISPLAY MAP

2) Press the **[PROGRAM]** menu key.

➔ The program last selected will be displayed on the **PROGRAM** display and the current menu will change over to this one:

WORK No.	FIND	PROGRAM	HELP	TPC		LAYOUT	TOOL PATH		PROGRAM FILE

(2) Press the **[WORK No.]** menu key.

➔ The display of **[WORK No.]** becomes highlighted and the work-Nos. listing window will be displayed.

* The work-Nos. listing window refers to a window that displays a list of work numbers of the programs that have already been registered in the NC equipment.

(3) Set the work number of the creating program.

- A work number is a number to be assigned to a program for identification of the program. Any number from 1 to 99999999 can be used as a work number.

- If a work number already registered in the NC unit is set, that program will be displayed on the screen. To create a new MAZATROL program, therefore, you must set a work number not used in other programs.
  You can check the work-Nos. listing window or the **PROGRAM FILE** display to see which work numbers are not yet used

- If you set a work number not used for the programs that have been registered in the NC unit, the current menu will change over to this one:

WORK No.	EIA/ISO PROGRAM *	MAZATROL PROGRAM							

* The EIA/ISO programming function is optional.

(4) Press the **[MAZATROL PROGRAM]** menu key.

➔ The following line will be displayed on the screen:

```
UNo.  MAT    OD-MAX   ID-MIN   LENGTH   WORK FACE   RPM
 0    ■
       \______ Cursor
```

This line denotes the common unit.

(5) Set data in each item of the common unit.

- See Section 3-1, "Common Unit" for details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item of the common unit, the cursor will move to the starting position of the next line and then the following menu A will be displayed, and pressing the >>> menu key changes A → B → A → B in order.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>	... A
	M M CODE	SUB PROGRAM	SEPARATE PROCESS	TRANSFER WORKPICE	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	SIMUL.	>>>	... B

(6) From the menus A and B, select a unit that is to follow the common unit.

➔ The unit data line of the selected unit will be displayed in the screen.

Example: If you have selected the bar-materials machining unit (**BAR**):

```
   UNo.  MAT   OD-MAX  ID-MIN  LENGTH  WORK FACE  RPM
    0    ***    ***     ***     ***      ***      ***

-> UNo.  UNIT  PART  CPT-X  CPT-Z  FIN-X  FIN-Z
    1    BAR    ■
                ↑
              Cursor
                     This line will be displayed.
```

If you have selected a unit that consists of only unit data (e. g. M-code unit):

(7) Set data in each item on the unit data line.

- See the relevant part of this section for further detail of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (unit data line).

If you have selected a unit that consists of unit data, tool sequence data, and shape sequence data of only one line (e. g. corner-machining unit):

(7)-1 Set data in each item on the unit data line.

- See the relevant part of this section for further detail of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (tool sequence data line).

(7)-2 Set data in each item on the tool sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (shape sequence data line).

(7)-3 Set data in each item on the shape sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (unit data line).

If you have selected a unit that consists of unit data, tool sequence data, and shape sequence data of multiple lines (e. g. bar-materials machining unit):

(7)-1 Set data in each item on the unit data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (tool sequence data line).

(7)-2 Set data in each item on the tool sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the cursor will move to the beginning of the next line (shape sequence data line).

(7)-3 Set data in each item on the shape sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.

(7)-4 After you have set the entire shape sequence data, press the **[SHAPE END]** menu key.

- The line that immediately succeeds the last shape sequence data line will be displayed as a unit data line.
- For a unit that permits you to set more than one line of shape sequence data, you cannot select the next unit unless you carry out this operation (pressing the **[SHAPED END]** menu key).

If you have selected a unit that consists of unit data, tool sequence data of multiple lines and shape sequence data of multiple lines (e. g. drilling unit):

(7)-1 Set data in each item on the unit data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.
- When you set data in the last item, the tool sequence data is made automatically and the cursor will move to the beginning of the tool sequence data line.

(7)-2 Set data in each item on the tool sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.

(7)-3 After you have set the entire tool sequence data, set data in each item on the shape sequence data line.

- See the relevant part of this section for further details of the data to be set.
- Each time you set data, the cursor moves to the next item automatically.

(7)-4 After you have set the entire shape sequence data, press the **[SHAPE END]** menu key.

- The line that immediately succeeds the last shape sequence data line will be displayed as a unit data line.
- For a unit that permits you to set more than one line of shape sequence data, you cannot select the next unit unless you carry out this operation (pressing the **[SHAPED END]** menu key).

(8) Select the units required for the intended machining operation by repeating steps (6) and (7) above (including steps (7)-1, (7)-2, (7)-3 and (7)-4), and then set data in each of the items displayed on the screen.

- A selectable unit differs according to the type of product to be machined. Select a unit in the most suitable order in accordance with your machining drawing, unit sheet, etc.
  After unit selection, the program can be generated just by setting data as guided by messages.

(9) Set the end unit at the end of the program.

- Press the **[END]** menu key.
- Without the end unit, the program will not be regarded as a complete one. Therefore, you must set the end unit at the last line of the program.

(10) Set data in each item of the end unit.

- See the section "End Unit (END)" for details of the data to be set.

Note 1: One MAZATROL program can contain a maximum of 99 units, including the common unit and the end unit. For units that allow you to set multiple lines of sequence data, up to a maximum of 25 lines of shape sequence data can be registered per unit (for turning manual program machining and milling manual program machining units only, up to a maximum of 250 lines of shape sequence data can be registered).

Note 2: The shape data that you have set can be checked for errors by calling up the **SHAPE CHECK** display while you are creating the program. See the Operating Manual for details.

Note 3: For the following units, TPC data can be set as required:
BAR unit, **CPY** unit, **CORNER** unit, **FACING** unit, **THREAD** unit, **T. GROOVE** unit, **T. DRILL** unit, **T. TAP** unit, **WPC MSR** unit, **WORK MES** unit, **TOOL MES** unit, **DRILLING** unit, **RGH CBOR** unit, **RGH BCB** unit, **REAMING** unit, **TAPPING** unit, **BK-CBORE** unit, **CIRC MIL** unit, **CBOR-TAP** unit, **BORE T1** unit, **BORE S1** unit, **BORE T2** unit, **BORE S2** unit, **LINE CTR** unit, **LINE RGT** unit, **LINE LFT** unit, **LINE OUT** unit, **LINE IN** unit, **CHMF RGT** unit, **CHMF LFT** unit, **CHMF OUT** unit, **CHMF IN** unit, **FCE MILL** unit, **TOP EMIL** unit, **POCKET** unit, **SLOT** unit, **TRANSFER** unit

See "TPC DATA SETTING" for further details of the data to be set.

3-1 Common Unit

The common unit is the first to be placed in a MAZATROL program, and always takes unit number 0.

Data that is set in this unit is referred to as common data, which becomes the base data for the entire program. When creating a MAZATROL program, therefore, you must first set data in this unit.

3-1-1 Setting unit data (common data)

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	TR2-DIA
0	[1]	[2]	[3]	[4]	[5]	[6]	[7]

[1] MAT

The following menu will be displayed when the cursor is placed at this item:

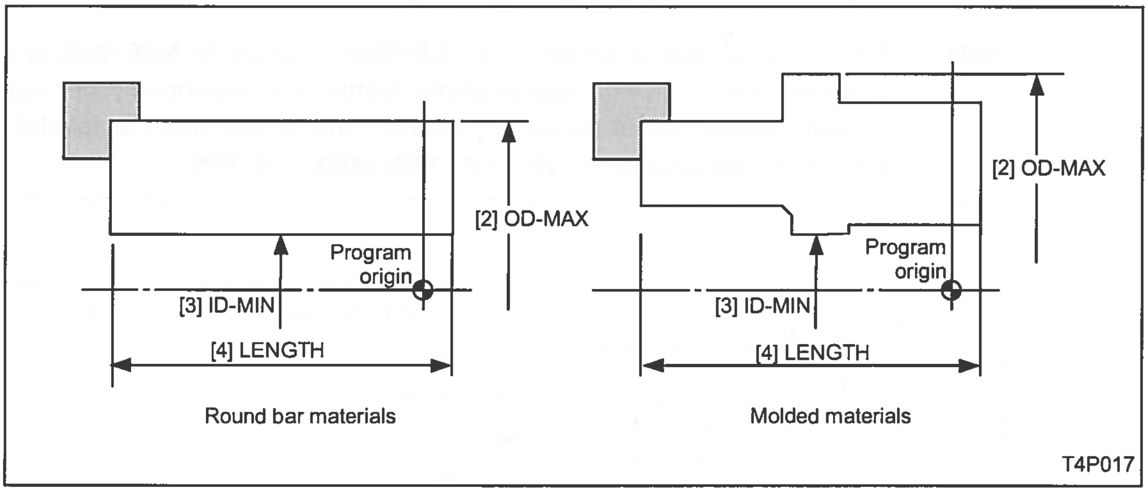
CST IRN	DUCT IRN	CBN STL	ALY STL	STNLESS	ALUMINUM	L.C.STL	AL CAST		

From the menu, select the materials type of the workpiece to be machined.

If the workpiece to be machined is of a materials type other than those listed above, pre-register that materials type on the **CUTTING CONDITION** display. See the Operating Manual for details. The data of this item is referred to by the system during automatic setting of cutting conditions.

[2] OD-MAX, [3] ID-MIN, [4] LENGTH

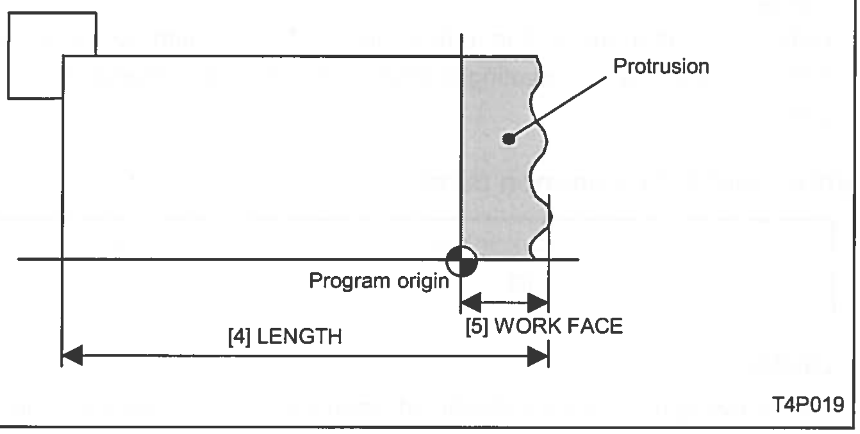
Set the maximum outside diameter, minimum inside diameter, and maximum length, respectively, of the workpiece.



- Set the workpiece length, including the edge protrusion (edge section to be cut), in item [4].

[5] WORK FACE

Set the length of the workpiece edge protrusion in the Z-axis direction.



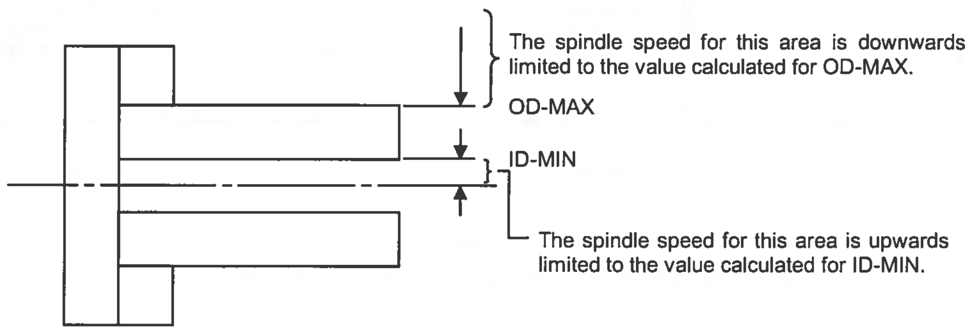
- The workpiece edge protrusion refers to a section to be cut during a facing unit (**FACING FACE**).
  For units other than facing units, the protrusion is not regarded as part of the workpiece.
  Therefore, if the workpiece edge is to be cut (that is, if a value other than 0 is set for this item), an facing unit must be selected before selecting a unit involving other machining operations.
  Either 0 or a plus value must always be set for this item.

[6] RPM

If the maximum spindle speed is to be limited, set that maximum value. Data does not need be set if the spindle speed is permitted to reach the maximum value provided for in the specifications.

This data has no relation to the milling axial velocity.

Note: For an X-axial tool-tip position over **OD-MAX** or under **ID-MIN** (both specified in the common unit), constant cutting speed control will opportunely be relieved by the constant spindle speed control for extra-workpiece area and the spindle will rotate at the speed calculated for the position of **OD-MAX** or **ID-MIN**.



* Constant peripheral (cutting) speed control is cancelled for extra-workpiece area in order to reduce the machining time.

[7] TR2-DIA

For a machine equipped with upper and lower turrets, enter a safe outside-diameter value for the lower turret. See Chapter 4, “LOWER-TURRET CONTROL FUNCTIONS”, for further details.

3-2 Materials Shape Unit (MATERIAL)

The shapes of cast materials or forged materials cannot be defined using the common unit alone. To machine such molded materials, the materials shape unit must be selected following the common unit and the shape data of the materials to be machined must be set.

Only the outside-diameter shape and inside-diameter shape of the intended workpiece can be defined using the materials shape unit. Data that has been set in this unit, as with common data, becomes base data for the entire program. This unit cannot be selected for round-bar materials. Press the **[WORKPICE SHAPE]** menu key to select the materials shape unit.

3-2-1 Setting unit data

UNo.	UNIT
*	MATERIAL [1]

[1] UNIT

The following menu will be displayed when the cursor is placed at this item.

OUT	IN								

- Select **[OUT]** to define the outside-diameter shape of the workpiece.
- Select **[IN]** to define the inside-diameter shape of the workpiece.

Both OUT and IN can be defined using a maximum of 25 sequences.
You must first select **[OUT]**, however, when defining both the outside-diameter and inside-diameter shapes of a workpiece. That is, after selecting the materials shape unit as both units No. 1 and No. 2, define the outside-diameter shape using unit No. 1 and then define the inside-diameter shape using unit No. 2.

3-2-2 Setting sequence data

UNo.	UNIT					
*	MATERIAL ***					
SEQ	SHP	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
1	[1]	[2]	[3]	[4]	[5]	[6]

[1] SHP

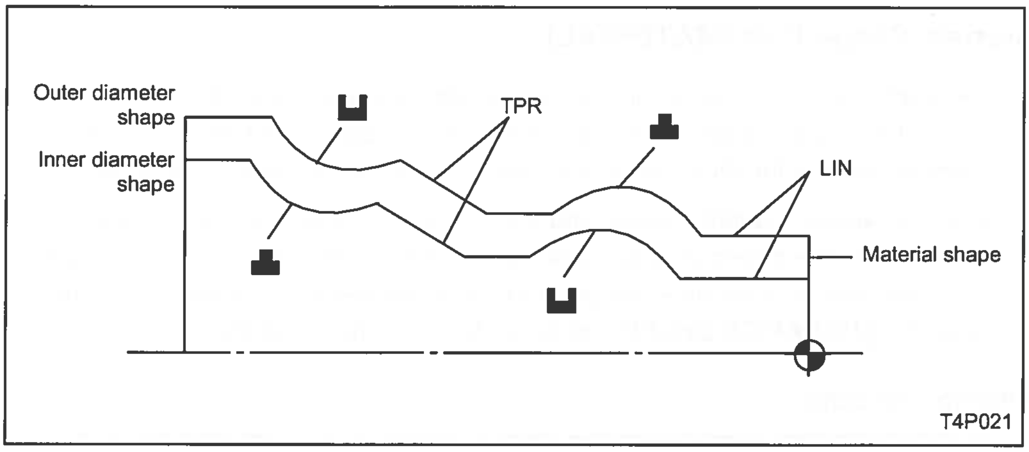
The following menu will be displayed when the cursor is placed at this item.

LIN	TPR								SHAPE END

Select the type of shape from the above menu.

The data of the displayed menu denote the following shapes:

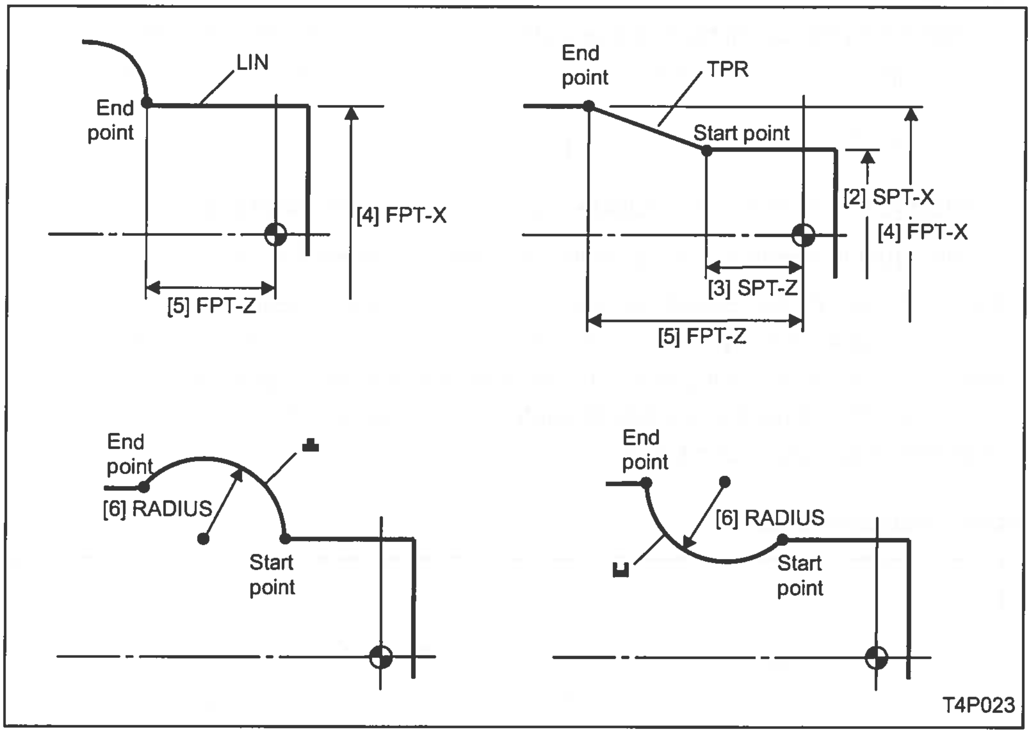
LIN : Line parallel to the center line of the workpiece
TPR : Line not parallel to the center line of the workpiece (Taper line)
: Convex arc
: Concave arc



T4P021

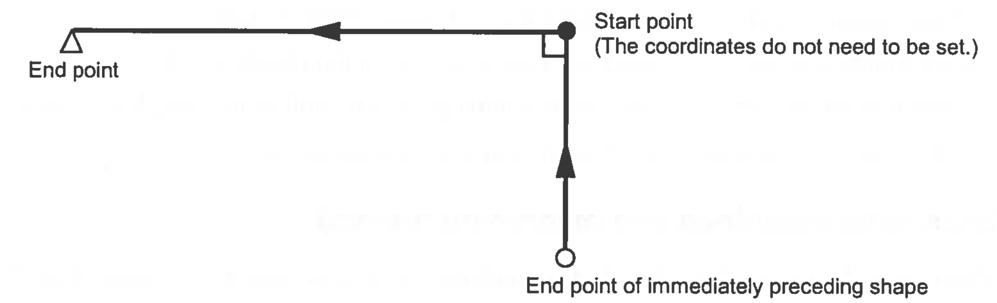
[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z, [6] RADIUS

Set the coordinates of the intended start point and end point of the shape you selected for item [1]. Also set the radius of the desired circle if you have selected or .



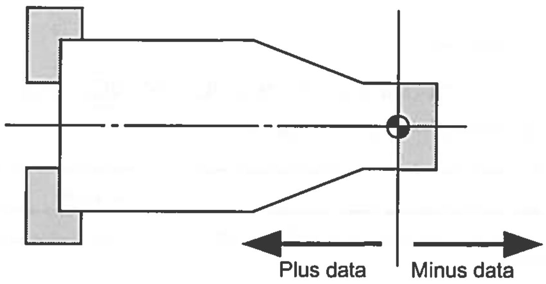
T4P023

- If you have selected **[LIN]** for item [1] above, it is good enough just to designate only the coordinates of the end point **(FPT-X** and **-Z)**. This is because the NC unit will then form automatically two orthogonal lines between the end point of the immediately preceding shape (or the program origin for an **LIN** as the first shape) and that end point.



T4P022

Note 1: The Z-coordinates of any points located to the right of the program origin must be set with a minus sign.



T4P024

Note 2: If the start point of a shape is present in the same position as that of the end point of the immediately preceding shape, those coordinates can be automatically set by pressing the **[NEXT]** menu key.

UNo.	UNIT					
1	MATERIAL OUT					
SEQ	SHP	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
1	LIN	◆	◆	20.	30.	◆
2	TPR	■ ← Cursor				◆

Pressing the **[NEXT]** menu key with the cursor at the position shown above sets the following data automatically:

UNo.	UNIT					
1	MATERIAL OUT					
SEQ	SHP	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
1	LIN	◆	◆	20.	30.	◆
2	TPR	20.	30.	■		◆

These values are set automatically.

You can use this function also for **BAR** and **CPY** units.

3-3 Types of the Milling Unit

The milling unit is available in the following three types :

- Point machining unit ...... used for drilling of holes (Section 3-4)
- Line machining unit .... used for a contour machining (Section 3-5)
- Face machining unit ...... used for machining an area and machining form (Section 3-6)

Each milling unit includes tool sequence and shape sequence.

3-3-1 Planes to be machined and machining methods

Data items for setting the plane to be machined and for setting the machining method exist in all point, linear, and face machining unit data. These data items are displayed as **MODE**, **POS-B**, and **POS-C**.

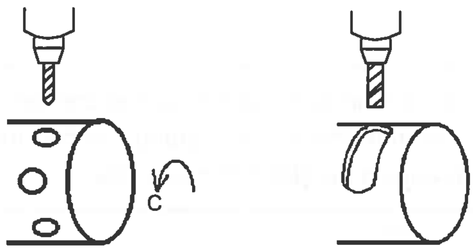
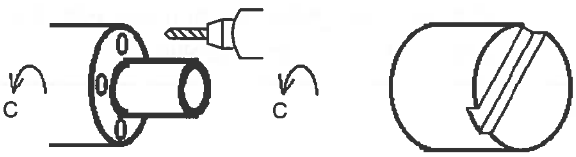
Specify the desired face and method under the **MODE**, **POS-B**, and **POS-C** columns.

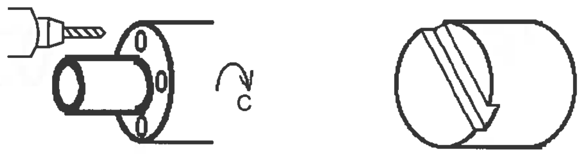
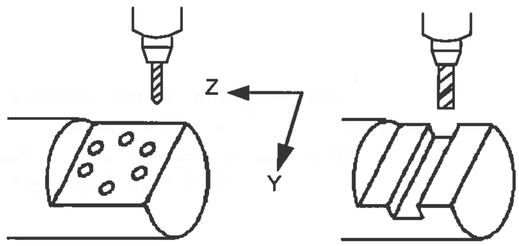
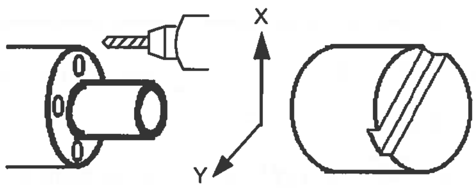
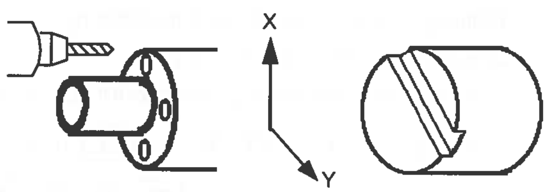
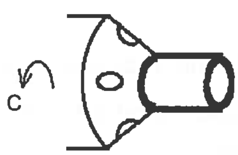
UNo.	UNIT	MODE [1]	POS-B [2]	POS-C [3]	DIA	DEPTH	CHMF
99	DRILLING	XY	180.0	-999.999	999.999	999.999	99.9

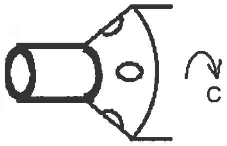
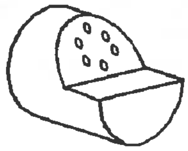
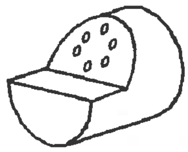
[1] MODE

Select the machining method.

Note: In oblique-face machining modes **/C**, **/Y**, $\boxed{/C}$, and $\boxed{/Y}$, be sure to select a tool horizontal-oriented (←) in the turret.

Mode	Description
ZC	Cylindrical sides can be machined into the desired shape as specified in the Z-C coordinate system. (C-axial machining)  **Note:** If C-axis function for No. 2 spindle is available, the line machining can be executed on the No. 2 spindle as well.
XC	Edges can be machined into the desired shape as specified in the R-C or X-Y coordinate system. (C-axial machining) 

<table>
<tr><th>Mode</th><th>Description</th></tr>
<tr><td>$\boxed{XC}$</td><td>Rear plane can be machined into the desired shape as specified in the R-C or X-Y coordinate system.
(C-axial machining)

Note: The line machining is possible only if the machine has C-axis function for No. 2 spindle.</td></tr>
<tr><td>ZY</td><td>Plane of cylinder can be machined into the desired shape as specified in the Z-Y coordinate system.
(Y-axial machining)
</td></tr>
<tr><td>XY</td><td>Edges can be machined into the desired shape as specified in the X-Y or R-C coordinate system.
(Y-axial machining)
</td></tr>
<tr><td>$\boxed{XY}$</td><td>Rear plane can be machined into the desired shape as specified in the X-Y or R-C coordinate system.
(Y-axial machining)
</td></tr>
<tr><td>/C</td><td>Holes can be machined on an oblique plane at the desired oblique positioning angle as specified in the B-axial direction. (C-axial machining)
This mode cannot be selected for the line or plane machining units.

The tool approaches from the edge side.</td></tr>
</table>

Mode	Description
$\boxed{\text{/C}}$	Holes can be machined on an oblique plane at the desired oblique positioning angle as specified in the B-axial direction. (C-axial machining) This mode cannot be selected for the line or plane machining units.  The tool approaches from the rear side.
/Y	Holes can be machined on an oblique plane at the desired oblique positioning angle as specified in the B-axial direction. (Y-axial machining)  The tool approaches from the edge side.
$\boxed{\text{/Y}}$	Holes can be machined on an oblique plane at the desired oblique positioning angle as specified in the B-axial direction. (Y-axial machining)  The tool approaches from the rear side.

The $\boxed{\text{XC}}$, $\boxed{\text{XY}}$, $\boxed{\text{/C}}$, $\boxed{\text{/Y}}$ mode can be selected for a machine model capable of back machining.

The **/C**, $\boxed{\text{/C}}$, **/Y**, $\boxed{\text{/Y}}$ mode can be selected for a machine model having a B-axis.

<u>Precautions for milling with the lower turret</u>

1. The machine operates in single-workpiece independent machining mode.
2. The machine operates only in point-machining mode.
   Drilling, inverse faced hole machining, reaming, tapping, and boring (see Note 2 below) are possible (see Note 1 below).
   Counterbore machining, back boring, circular milling, or counterbore-tapping is impossible.
3. It is possible to use **ZC**, **XC**, or $\boxed{\text{XC}}$ mode. (See Note 1.)
   It is not possible to use **/C**, $\boxed{\text{/C}}$, **ZY**, **XY**, $\boxed{\text{XY}}$, **/Y**, or $\boxed{\text{/Y}}$ mode.
4. The machine does not operate in line- or face-machining mode.
5. The lower turret cannot be used for the **M-MANUAL** unit that operates the Y-axis.
6. Simultaneous machining with the milling tools mounted in the upper and lower turrets is impossible.

Note 1: Machining that requires Y-axis operation results in an alarm (for chamfering cycle 2).

Note 2: Boring cycle 1 and 2 cannot be used (an alarm occurs for lower-turret milling spindle orientation).

[2] POS-B

When machining an oblique plane, specify angle B of the oblique plane with respect to a reference angle of 0 degrees of the edge.

This data item will become valid when the **/C**, **/Y**, /C, /Y mode is selected for a machine model having a B-axis.

[3] POS-C

Specify the position of the C-axis.

This data item will become valid when the **ZY**, **XY**, XY, **/Y**, /Y mode is selected.

3-3-2 Same-plane setting function

During unit selection, when the **[PLANE AUTO SET]** menu item shown below is made valid (reverse display), the **MODE**, **POS-B**, and **POS-C** data of the immediately preceding unit will be copied after selection of that unit.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

```
UNo.  MAT  OD-MAX  ID-MIN  LENGTH    WORK FACE  RPM
 0    FC    58.     0.      320.               10000
UNo.  UNIT   MODE    POS-B   POS-C   SRV-A    SRV-R    RGH    FIN-A    FIN-R
 1  LINE CTR  /Y      78.3    45.      2.       6.       1      0.2      ◆
SNo.  TOOL  NOM-φ  APRCH-1  APRCH-2  TYPE      AFD     DEP-A   DEP-R    C-SP   FR  M M
R1  END MILL  20.     ?        ?       ◆        G01      2.      ◆       130  0.751
F2  END MILL  20.     ?        ?       ◆        G01      ◆       ◆       130  2.56
FIG   SHP  SHIFT-Z  SHIFT-R    X    Y    RADIUS/th   I   J   P   CNR     RGH
 1   LINE    0.       3.       0.   0.
 2   LINE    ◆        ◆       30.   0.
UNo.  UNIT   MODE    POS-B    POS-C    NOM-   MAJOR-φ   PITCH   TAP-DEP   CHMF
 2    ■
```

↓ Select a tapping unit.

```
UNo.  UNIT     MODE    POS-B    POS-C    NOM-   MAJOR-φ   PITCH   TAP-DEP   CHMF
 2   TAPPING    /Y      78.3     45.
```

- If no data is specified in the **MODE**, **POS-B**, or **POS-C** items corresponding to the previous unit, the first **MODE**, **POS-B**, and **POS-C** data detected during a forward search will be copied. If, after the search, a unit does not exist that has data in its **MODE**, **POS-B**, or **POS-C** items, the cursor will move to **MODE** of the current unit to allow data entry using the normal method.
- Pressing once again the **[PLANE AUTO SET]** menu key clears the reverse display status and allows data entry using the normal method.

3-4 Point Machining Units

The point machining unit serves to determine the data concerning the machining method and machining form for the drilling of holes.

The unit includes the tool sequence determining the tool data used and the shape sequence determining the data concerning the machining dimensions on the drawing.

3-4-1 Types of point machining units

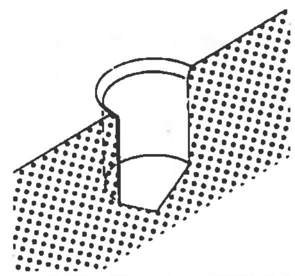
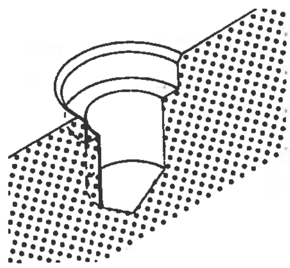
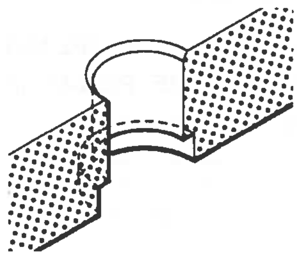
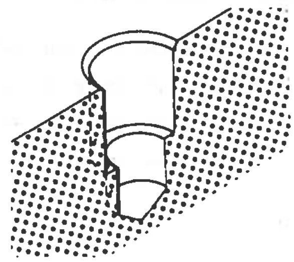
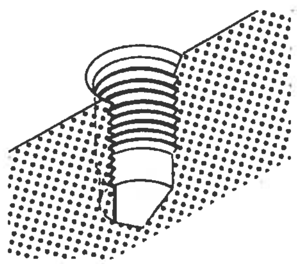
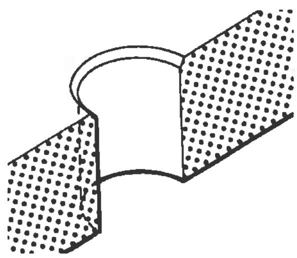
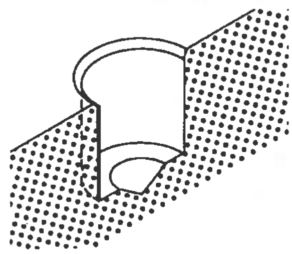
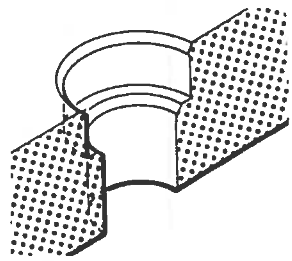
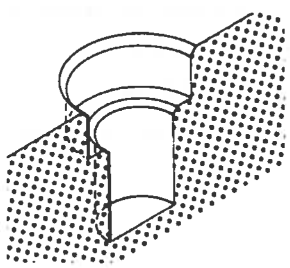
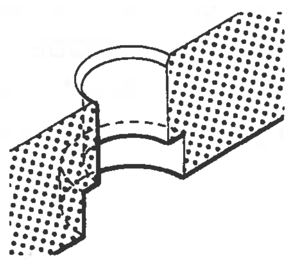
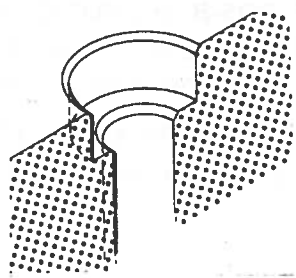
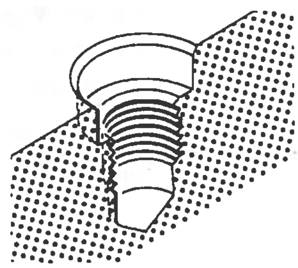
1. Drilling	2. Counterbore machining	3. Inversed faced hole machining	4. Reaming
 NM210-00532	 NM210-00533	 NM210-00534	 NM210-00535
5. Tapping	6-(1) Boring of through hole	6 (2) Boring of non-through hole	6 (3) Boring of stepped through hole
 NM210-00536	 NM210-00537	 NM210-00538	 NM210-00539
6-(4) Boring of stepped non-through hole	7. Back boring	8. Circular milling	9. Counterbore-tapping
 NM210-00540	 NM210-00541	 NM210-00542	 NM210-00543

Fig. 3-1 Types of point machining units

3-4-2 Procedure for selecting point machining unit

(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Pressing on the **[POINT MACH-ING]** menu key displays the following unit menu.

DRILLING	RGH CBOR	RGH BCB	REAMING	TAPPING	BORING	BK CBOR	CIRC MIL	CBOR TAP	HI SPD. DRL.USE

(3) Press the appropriate menu key of the desired machining unit.

When the **[BORING]** menu key is pressed, the menu of the four following machining sub-units is displayed.

BORING	BORING	BORING	BORING						

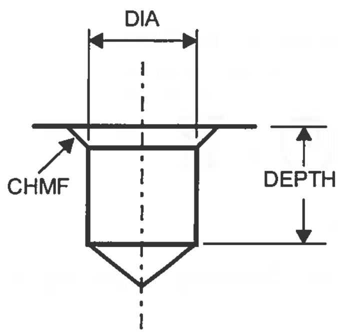
Remark: For the function of the **[HI SPD. DRL. USE]** menu key, refer to the Subsection 3-4-4, "Automatic tool development for cemented carbide drills".

3-4-3 Unit data and automatic tool development of the point machining unit

1. Drilling unit (DRILLING)

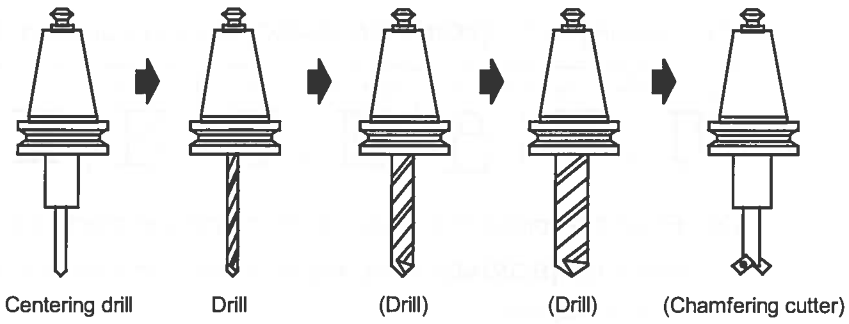
Select this drilling unit for machining of a hole with a drill.

DRILLING unit



M3P085

Tool sequence



M3P086

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tool.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0

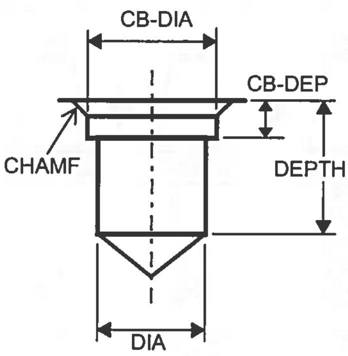
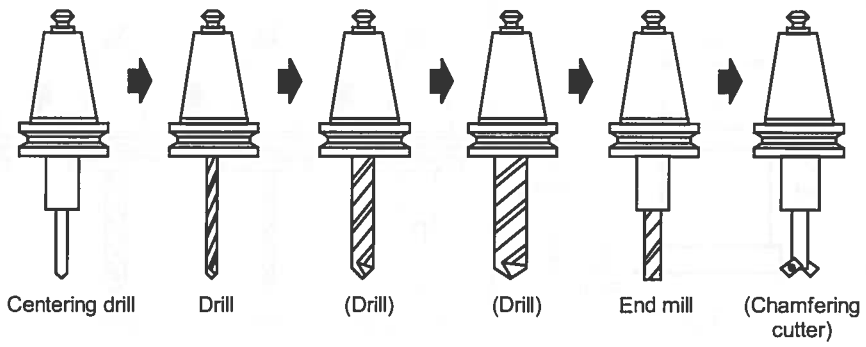
The bold codes represent parameter addresses.

Note: In the following cases the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.

- **DEPTH < CHMF**
- **DIA** ≤ 0
- **D10 < DIA**

2. Counterbore machining unit (RGH CBOR)

This unit is selected for machining a hole with a counterbore (faced hole).

RGH CBOR unit	Tool sequence
 M3P087	 M3P088

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tool.

<Development patterns>

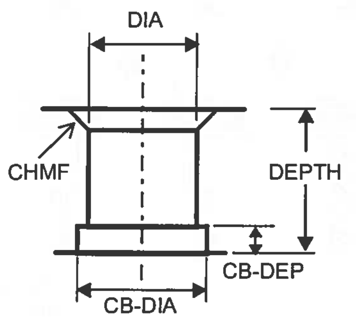
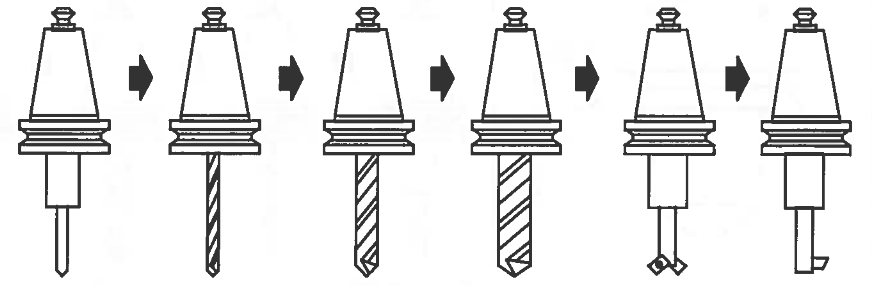
Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
End mill	Development is always executed.
Chamfering cutter	Development is not executed in the following casses: CHMF = 0 DIA + (DEPTH × 2) ≥ CB-DIA + (CHMF × 2) < **D13**

The bold codes represent parameter addresses.

Note: In the following cases the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **CB-DIA < DIA**
- **DEPTH < CB-DEP**
- **DEPTH < CHMF**

3. Inversed faced hole machining unit (RGH BCB)

This unit is selected for machining a hole with an inversed faced hole.

RGH BCB unit	Tool sequence
	

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tool.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2 – D4** CHMF = 0
Back facing tool	Development is always executed.

The bold codes represent parameter addresses.

Note: In the following cases the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.

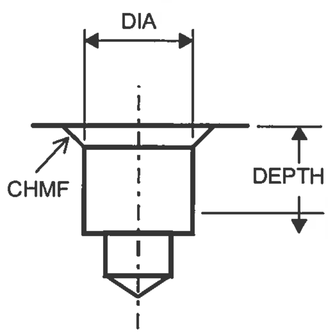
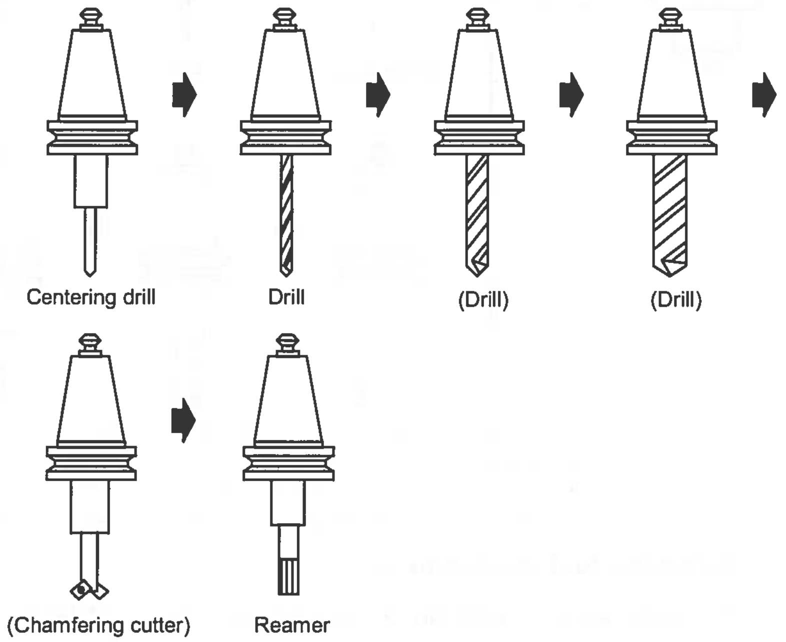
- **CB-DIA < DIA**
- **DEPTH < CB-DEP**
- **DEPTH < CHMF**

4. Reaming unit (REAMING)

Select this unit for performing finish machining with reamer.

In reaming, the content of the tool sequence to be set is different according to the preceding process.

A. Case of preceding process = drilling

REAMING unit	Tool sequence
 M3P091	 D734P0001

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tool.

<Development patterns>

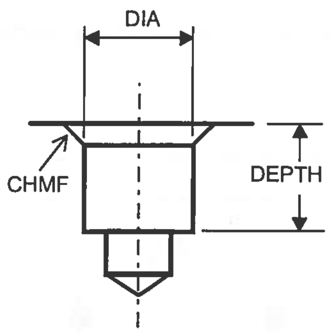
Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < DIA – **D35** ≤ **D8**: Development of one tool **D8** < DIA – **D35** ≤ **D9**: Development of two tools **D9** < DIA – **D35** ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Reamer	Development is always executed.

The bold codes represent parameter addresses.

Note: In the following case the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH < CHMF**

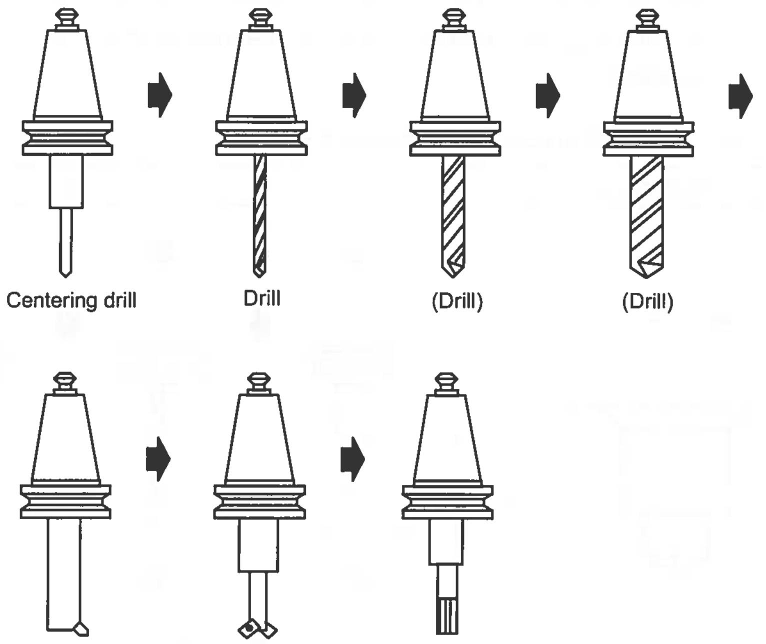
B. Case of preceding process = boring

REAMING unit



M3P093

Tool sequence



D734P0002

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

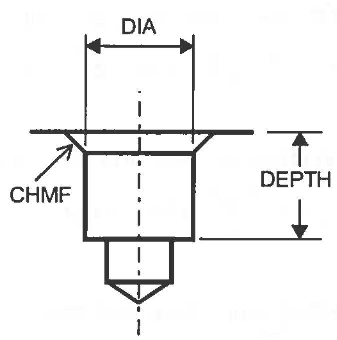
Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < DIA – **D36** ≤ **D8**: Development of one tool **D8** < DIA – **D36** ≤ **D9**: Development of two tools **D9** < DIA – **D36** ≤ **D10**: Development of three tools
Boring tool	Development is always executed.
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Reamer	Development is always executed.

The bold codes represent the parameter addresses.

Note: In the following case the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH < CHMF**

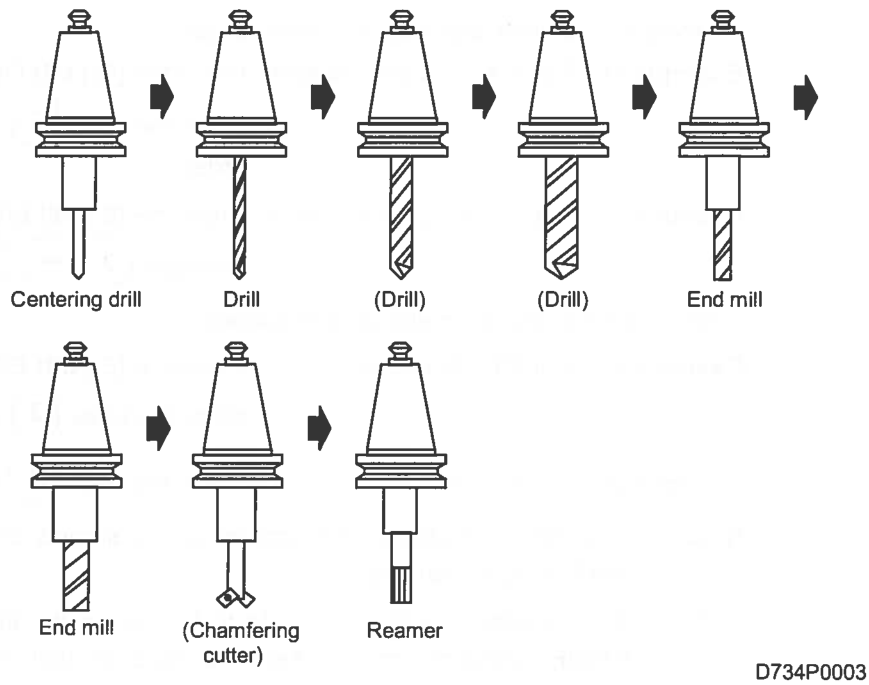
C. Case of preceding process = end mill

REAMING unit



M3P095

Tool sequence



D734P0003

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. $0 < DIA - \mathbf{D37} \le \mathbf{D8}$: Development of one tool $\mathbf{D8} < DIA - \mathbf{D37} \le \mathbf{D9}$: Development of two tools $\mathbf{D9} < DIA - \mathbf{D37} \le \mathbf{D10}$: Development of three tools
End mill	Development of two tools is executed.
Chamfering cutter	Development is not executed in the following cases: $DIA + (CHMF \times 2) \le \mathbf{D2} - \mathbf{D4}$ $CHMF = 0$
Reamer	Development is always executed.

The bold codes represent the parameter addresses.

Note: In the following case the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH < CHMF**

5. Tapping unit (TAPPING)

Select this unit for performing tapping.

<Setting the nominal diameter of unified screws>

Example 1: For 3/4-16 unified screws: Press the **[Q (1/4) QUARTER]** menu key, and then press the keys 3 – 1 6 and → in this order.

Example 2: For 1 1/8-7 unified screws: Press the **[E (1/8) EIGHTH]** menu key, and then press the keys 9 – 7 and → in this order.

<Setting the nominal diameter of pipe screws>

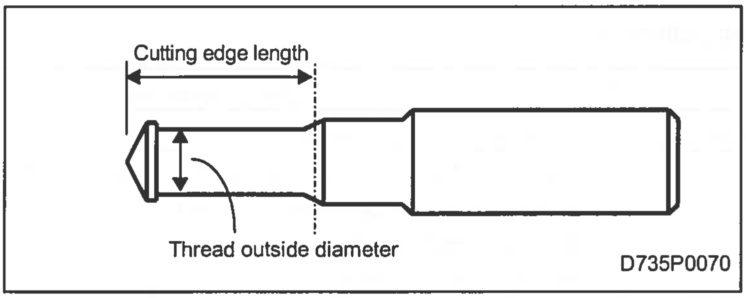
Example 1: For PT 3/8 screws: Press the **[E (1/8) EIGHTH]** menu key, and then press the keys 3 and → in this order.

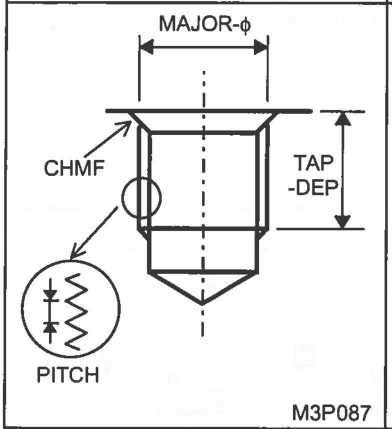
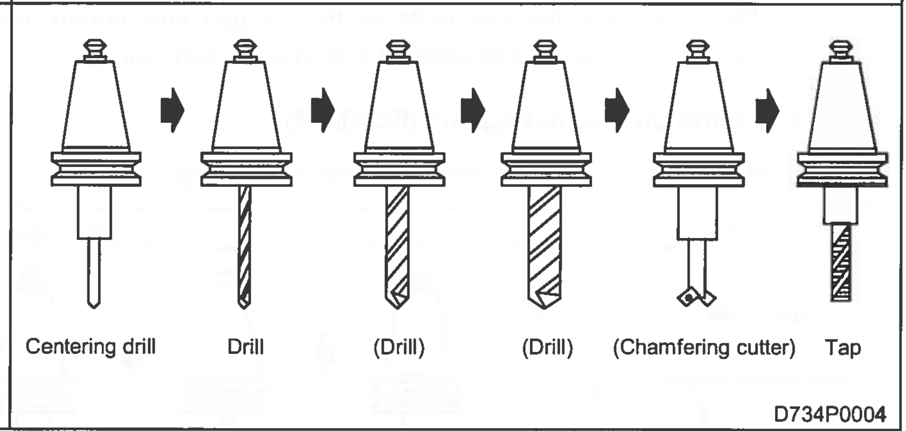
Example 2: For PF 1 screws: Press the keys 1 and → in this order.

Note 1: The thread depths of PT screws or PS screws are set automatically according to MAZAK specifications.

Note 2: For planetary tapping, the data to be set for the **MAJOR-ϕ**, **PITCH**, **TAP-DEP**, and **CHMF**, depends on the selected type of tool. Enter the data specified in the corresponding tool catalogue.
For **TAP-DEP**, enter the cutting edge length specified in the tool catalogue.
Also, set the tool data as follows.

- Enter the catalogued nominal diameter in the tool data item **ACT-ϕ**.
- Enter the catalogued thread outside diameter in the tool data item **DIAMETER**.
- Enter the catalogued cutting edge length in the tool data item **LENGTH**.



TAPPING unit	Tool sequence
	

The tools in parentheses (   ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0 < Diameter of pre-hole drilling ≤ **D8**: Development of one tool **D8** < Diameter of pre-hole drilling ≤ **D9**: Development of two tools **D9** < Diameter of pre-hole drilling ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: Diameter of hole + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Tap	Development always takes place.

The bold codes represent the parameter addresses.

In the following cases the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.

- **TAP-DEP < CHMF**
- Case of designation of threading other than the JIS standard threading (however, this can be used for forced insertion).

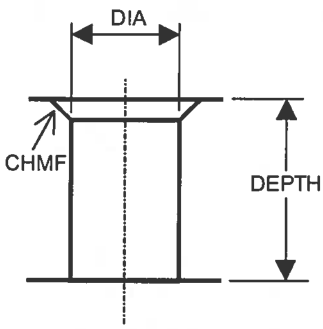
6. Boring unit (BORING)

The boring has the four units as the through hole boring, non-through hole boring, stepped through hole boring and stepped non-through hole boring.

A. Through hole boring unit (BORE T1)

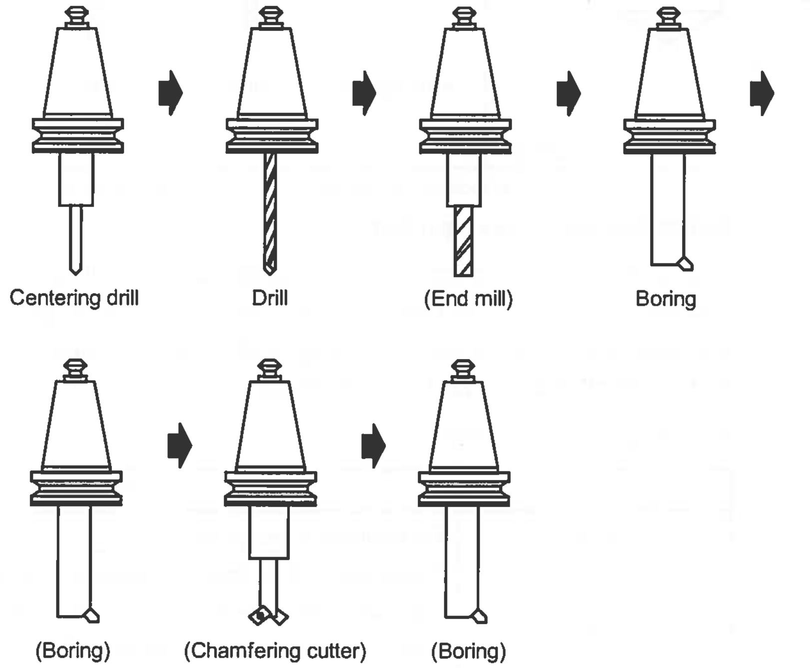
Select this unit for performing through-hole boring.

BORE T1 unit



M3P099

Tool sequence



M3P100

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

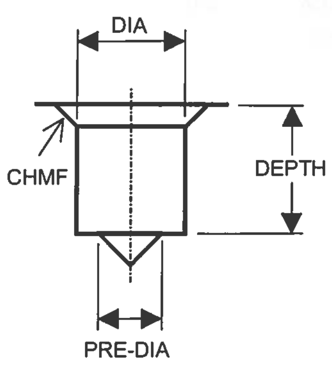
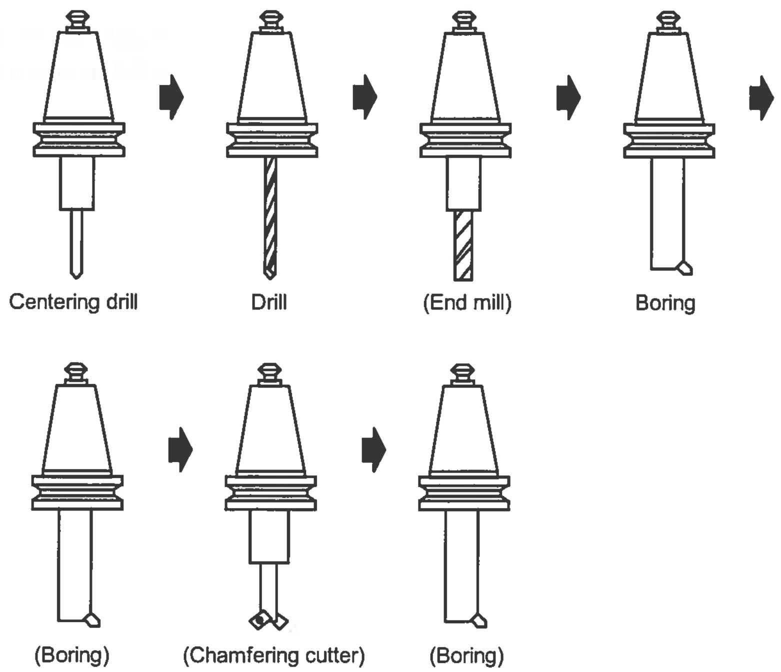
Tool	Development patterns
Centering drill	Development is always executed.
Drill	Development is always executed.
End mill	Development is not executed in the following case: DIA – 6.0 < **D8**
Boring tool	Development of a maximum of three tools is executed depending on the wall roughness. Wall roughness = 1, 2: Development of one tool Wall roughness = 3, 4: Development of two tools Wall roughness = 5, 6, 7, 8, 9: Development of three tools
Chamfering cutter	Development is not executed in the following case: CHMF = 0

The bold codes represent the parameter addresses.

Note: In the following cases the alarm **493 AUTO PROCESS IMPOSSIBLE** will be displayed.
- Diameter of faced hole < **DIA**
- **DEPTH** < Depth of faced hole
- **DEPTH** < **CHMF**

B. Non-through hole boring unit (BORE S1)

Select this unit for performing boring of non-through holes.

BORE S1 unit	Tool sequence
 M3P101	 M3P100

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tool.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	Development is always executed.
End mill	Development is not executed if the following three conditions are fulfilled: DIA – 6.0 < **D8** 10.0 < PRE-DIA DIA – PRE-DIA ≤ 6.0
Boring tool	The development of a maximum of three tools is executed according to the wall roughness. Wall roughness = 1, 2: Development of one tool Wall roughness = 3, 4: Development of two tools Wall roughness = 5, 6, 7, 8, 9: Development of three tools
Chamfering cutter	Development does not take place in the following case: CHMF = 0

The bold codes represent the parameter addresses.

Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

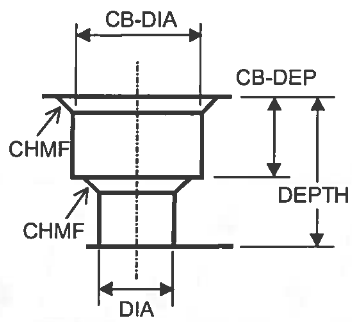
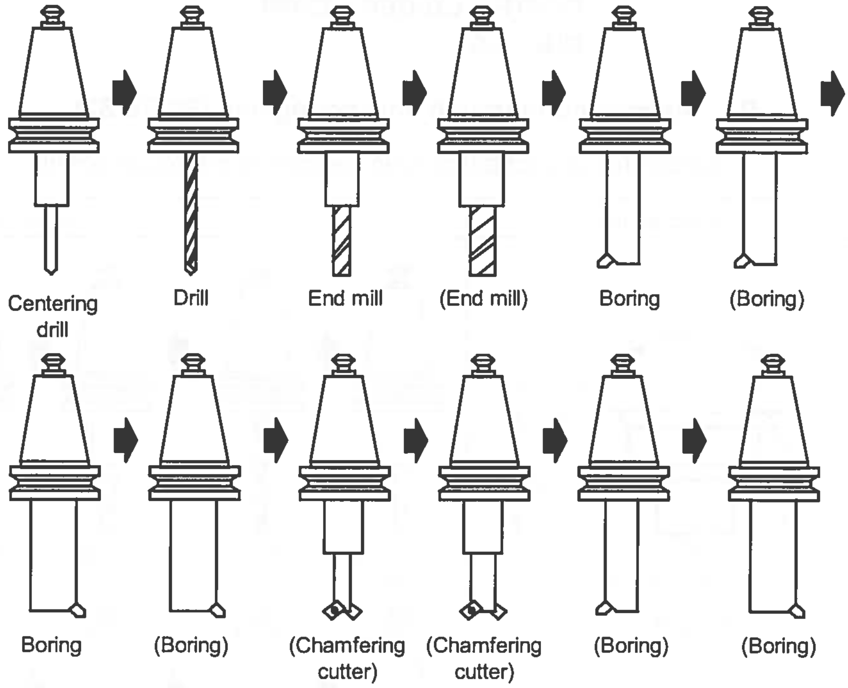
- **DIA** < **PRE-DIA**
- **DIA** ≤ 6.0
- **DEPTH** < **CHMF**
- **PRE-DIA** = 0 → **DEPTH** < (A/3.328558 – **D12**)
- **PRE-DIA** ≠ 0 → **DEPTH** < (A – **PRE-DIA**)/3.328558

A: **DIA** – 6.0 (incase of **DIA** – 6.0 < **D8**) or

A: **D8** (in case of **D8** ≤ **DIA** – 6.0)

C. Stepped through hole boring unit (BORE T2)

Select this unit for performing stepped through hole boring.

BORE T2 unit	Tool sequence
	
M3P102	M3P103

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	Development is always executed.
End mill	Development of a maximum of two tools is executed depending on the diameter of the hole. 0 < DIA – 6.0 < **D8**: Development of one tool **D8** < DIA – 6.0 ≤ 999.999: Development of two tools
Boring tool	The development of a maximum of three tools is executed depending on the wall roughness of the hole and depending on the wall roughness of the large hole, respectively. Wall roughness of hole = 1, 2: Development of one tool Wall roughness of hole = 3, 4: Development of two tools Wall roughness of hole = 5, 6, 7, 8, 9: Development of three tools Wall roughness of large hole = 1, 2: Development of one tool Wall roughness of large hole = 3, 4: Development of two tools Wall roughness of large hole = 5, 6, 7, 8, 9: Development of three tools

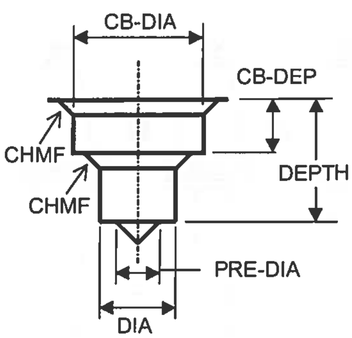
The bold codes represent the parameter addresses.

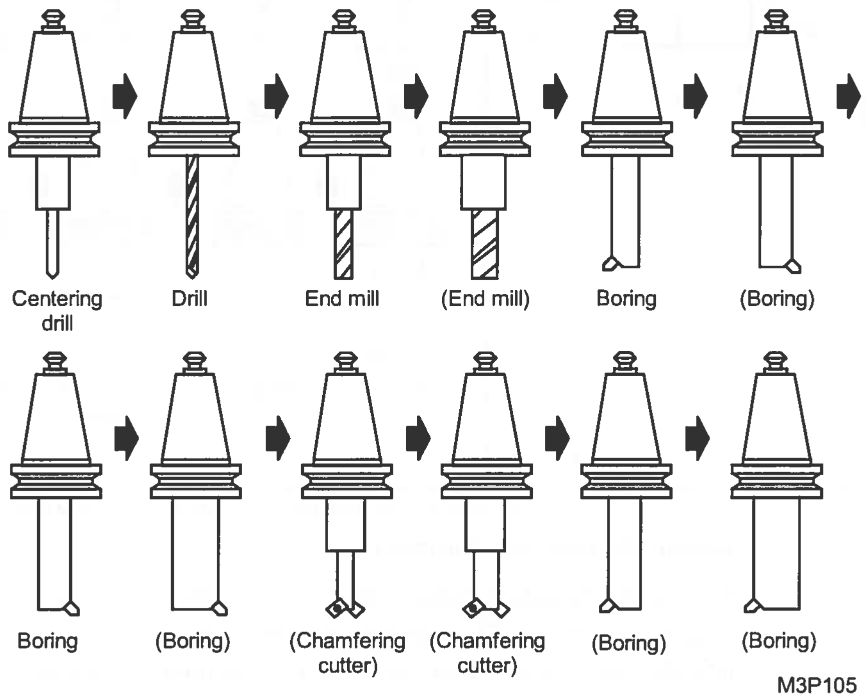
Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

- **CB-DEP < CHMF (CB)**
- **CB-DIA < DIA**
- **(CB-DIA – DIA)/2 < CHMF**
- **DEPTH – CB-DEP < CHMF**
- **DIA** ≤ 6.0

D. Stepped non-through hole boring unit (BORE S2)

Select this unit for performing stepped non-through boring.

<table>
<tr><th>BORE S2 unit</th><th>Tool sequence</th></tr>
<tr><td>


M3P104
</td><td>


M3P105
</td></tr>
</table>

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	Development is always executed.
End mill	Development of a maximum of two tools is executed depending on the diameter of the hole. 0 < DIA – 6.0 < **D8**, 10.0 < PRE-DIA and (DIA – PRE-DIA) ≤ 6.0: Development of one tool **D8** < DIA – 6.0 ≤ 999.999: Development of two tools
Boring tool	The development of a maximum of three tools is executed depending on the wall roughness of the hole and depending on the wall roughness of the large hole, respectively. Wall roughness of hole = 1, 2: Development of one tool Wall roughness of hole = 3, 4: Development of two tools Wall roughness of hole = 5, 6, 7, 8, 9: Development of three tools Wall roughness of large hole = 1, 2: Development of one tool Wall roughness of large hole = 3, 4: Development of two tools Wall roughness of large hole = 5, 6, 7, 8, 9: Development of three tools
Chamfering cutter	Development is not executed when the following two conditions are fulfilled: CHMF = 0 CHMF (CB) = 0

The bold codes represent the parameter addresses.

Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

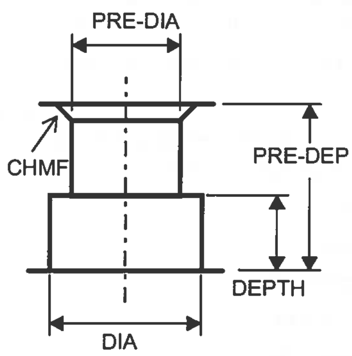
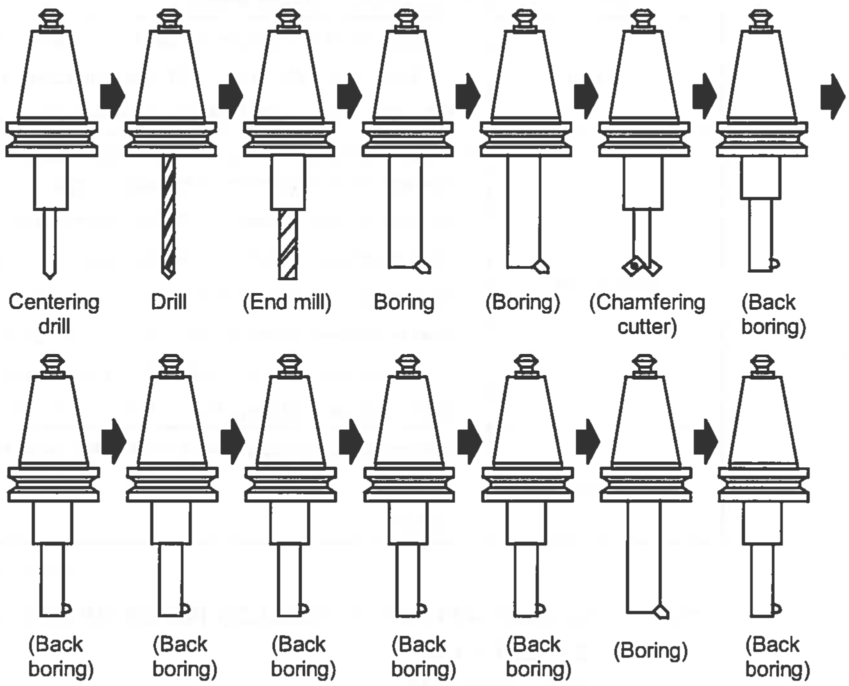
- **CB-DIA** < **DIA**
- **DIA** ≤ **PRE-DIA**
- **DEPTH** < **CB-DEP**
- **CB-DEP** < **CHMF (CB)**
- (**CB-DIA** – **DIA**)/2 < **CHMF**
- (**DEPTH** – **CB-DEP**) < **CHMF**
- **DIA** ≤ 6.0
- **DEPTH** < **CHMF**
- B ≤ 0

B: **DIA** – 6.0 (in case of **DIA** – 6.0 < **D8**) or

B: **D8** (in case of **D8** ≤ **DIA** – 6.0)

7. Back boring unit (BK-CBORE)

Select this unit for performing back boring.

BK-CBOR unit	Tool sequence
 M3P106	 M3P107

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	Development is always executed.
End mill	Development is not executed in the following case: PRE-DIA – 6.0 < **D8**
Boring tool	Development of a maximum of three tools is executed depending on the wall roughness. Wall roughness of pre-hole = 1, 2: Development of one tool (Roughing) Wall roughness of pre-hole = 3, 4: Development of two tools (Roughing, semi-finishing) Wall roughness of pre-hole = 5, 6, 7, 8, 9: Development of three tools (Roughing, semi-finishing, finishing)
Chamfering cutter	Development is not executed in the following case: CHMF = 0
Back boring tool	The development of a maximum of five tools is executed according to the value of N (See Note below.) N = 2: Development of two tools N = 3: Development of three tools N = 4: Development of four tools N = 5: Development of five tools
Back boring tool (Semi-finishing, finishing)	The development of a maximum of two tools is executed depending on the wall roughness. Wall roughness of hole = 1, 2: No development Wall roughness of hole = 3, 4: Development of one tool (Semi-finishing) Wall roughness of hole = 5, 6, 7, 8, 9: Development of two tools (Semi-finishing, finishing)

The bold codes represent the parameter addresses.

Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

- **DIA** < **PRE-DIA**
- **PRE-DEP** < **DEPTH**
- **PRE-DEP** < **CHMF**
- **PRE-DEP** ≤ **DIA**/2
- 5 < N

The value N is determined by the roughness and the number of times of back boring.

$$N = \frac{(\mathbf{DBBL} - \mathbf{DP})}{6} \quad \text{(Decimal fractions are rounded off.)}$$

Wall roughness of hole	**DBBL**
1, 2	**DIA**
3, 4	**DIA** – 1.0
5, 6, 7, 8, 9	**DIA** – 1.5

Wall roughness of pre-hole	**DP**
1, 2, 3, 4	**PRE-DIA**
5, 6, 7, 8, 9	**PRE-DIA** – 1.5

8. Circular milling unit (CIRC MIL)

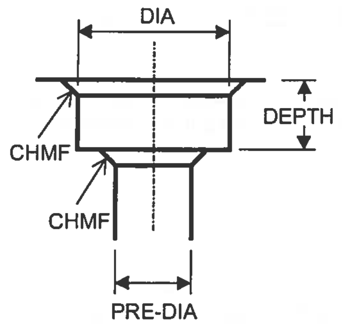
Select this unit for performing drilling with the end mill.

According to the set value in item **TORNA.**, one of the following two machining patterns is selected.

TORNA.: 0...................Circular milling cycle
1...................Tornado milling cycle

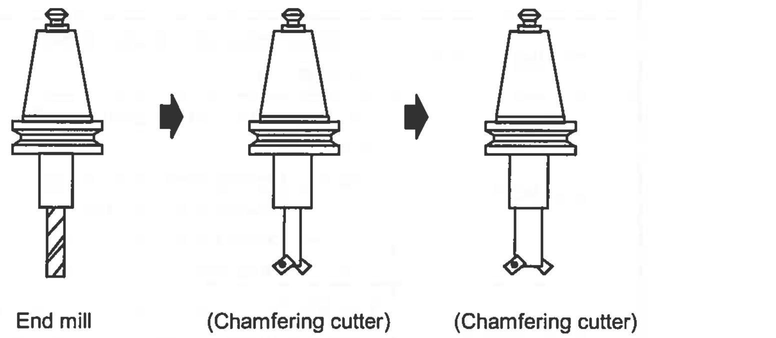
<u>Circular milling cycle</u>

CIRC MIL unit



M3P108

Tool sequence



M3P109

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are autromatically developed according to different patterns on the basis of the data entered in the unit. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

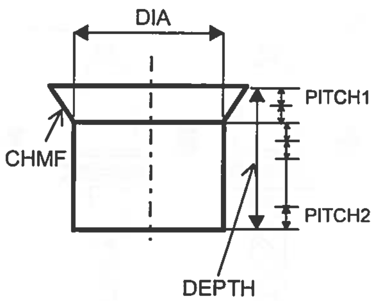
<Development patterns>

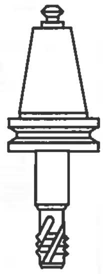
Tool	Development patterns
End mill	Development is always executed.
Chamfering cutter	Development is not executed under the following two conditions: CHMF = 0 CHMF (pre-hole) = 0

Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

- **DIA** < **PRE-DIA**
- **DEPTH** < **CHMF**
- (**DIA** – **PRE-DIA**)/2 < **CHMF** (pre-hole)

Tornado milling cycle

<table>
<tr><th>CIRC MIL unit</th><th>Tool sequence</th></tr>
<tr><td>

D735P0063</td><td>
End mill
(Thread mill)
D735P0064</td></tr>
</table>

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
End mill	Development is always executed.

Note 1: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

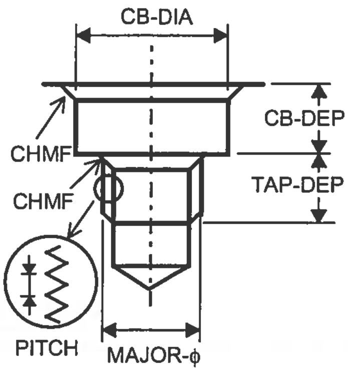
- **DIA** < **PRE-DIA**
- **DEPTH** < **CHMF**
- (**DIA** – **PRE-DIA**)/2 < **CHMF** (pre-hole)

Note 2: Set such a tool diameter in tool data that satisfies "**DIA** > tool diameter ≥ (**DIA**/2)".

9. Counterbore-tapping unit (CBOR-TAP)

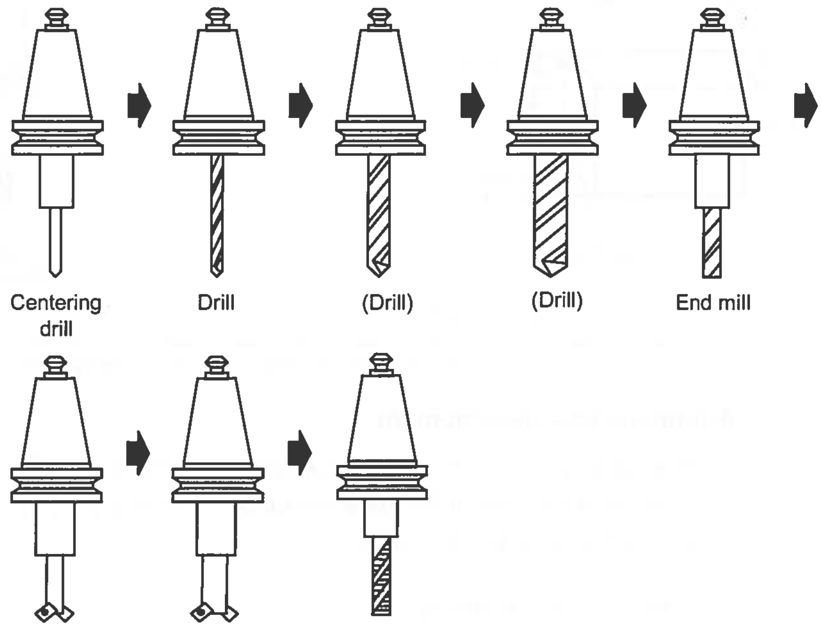
Select this unit for machining a tapped hole with a counterbore (faced hole).

CBOR-TAP unit



M3P110

Tool sequence



D734P0005

The tools in parentheses ( ) are developed or not developed depending on the particular case.

Automatic tool development

The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is performed on the basis of the tool sequence data and the unit data are not used for machining. If the data developed are inappropriate for the machining, edit by modifying the data or deleting the tools.

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	The development of a maximum of three tools is executed depending on the diameter of the hole. 0 < Hole diameter ≤ **D8**: Development of one tool **D8** < Hole diameter ≤ **D9**: Development of two tools **D9** < Hole diameter ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: **CHMF** (faced hole) = 0 **CHMF** (threaded hole) = 0
Tap	Development always takes place.

The bold codes represent the parameter addresses.

Note: The alarm **493 AUTO PROCESS IMPOSSIBLE** is given in the following cases:
- **CB-DIA < MAJOR-ϕ**
- (**CB-DIA – MAJOR-ϕ**)/2 < **CHMF** (threaded hole)
- **PRE-DEP < CHMF** (faced hole)
- **TAP-DEP < CHMF** (threaded hole)

3-4-4 Automatic tool development for carbide drills

The Subsection 3-4-3 describes automatic tool development for drilling using high speed steel drills. Automatic tool development for cemented carbide drills is described below. This function allows machining time and programming time to be reduced. Before using this function, thoroughly understand its usage, since mis-use causes tool damage.

After point machining unit selection, the following menu is displayed. Press the **[HI SPD DRL. USE]** menu key to make the function valid (reverse the display status of the menu item) before selecting a unit. Automatic tool development for cemented carbide drills will occur for the tool sequence:

DRILLING	RGH CBOR	RGH BCB	REAMING	TAPPING	BORING	BK CBOR	CIRC MIL	CBOR TAP	HI SPD. DRL.USE

Automatic tool development for drilling with cemented carbide drills is valid for all poit-machining units and described below using a drilling unit as an example.

UNo.	UNIT	MODE	POS-B	POS-C	DIA	DEPTH	CHMF					
2	DRILLING											
SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	DRILL	O	O	O	O	O	$\boxed{\text{DRIL}}$	O				
2	CHAMFER	O	O	O	O	O	◆	O				

1) Centering drill data for machining a center hole is not developed automatically.
2) Drilling cycle is developed at **RGH** in the drilling tool sequence, irrespective of the hole depth.
3) Only one drill data is developed automatically, even for a large hole diameter.
4) When the hole diameter is larger than the value of parameter **D2** (nominal diameter of a centering drill), chamfering cutter data is developed automatically. Tool data for chamfering with a centering drill is developed automatically for a hole diameter (**DIA**) smaller than or equal to the value of parameter **D2** (nominal diameter of a centering drill).

UNo.	UNIT	MODE	POS-B	POS-C	DIA	DEPTH	CHMF					
2	DRILLING											
SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	DRILL	O	O	O	O	O	DRIL	O				
2	CTR-DR	O	O	◆	◆	◆	90°	◆				

O : The data displayed here are automatically determined by automatic tool development function.

◆ : Data are not necessary to be set here.

3-4-5 New tapping auto-setting scheme

Any given value for tapping with the tapping unit/counterbore-tapping unit can be specified as an auto-set value by editing the required text file within the hard disk. (New tapping auto-setting scheme)

The items corresponding to the new tapping auto-setting scheme are listed below.

O: New tapping auto-setting scheme applicable
—: New tapping auto-setting scheme inapplicable

Type of thread to be tapped	Tapping/Counterbore-tapping unit				
	MAJOR-ϕ	PITCH	TAP-DEP	PRE-DIA	PRE-DEP
Metric thread	—	—	—	O	—
Unified thread	—	—	—	O	—
Pipe thread (PT)	O	O	O	O	O
Pipe thread (PF)	O	O	—	O	—
Pipe thread (PS)	O	O	O	O	O

1. Tapping for metric thread /unified thread

In the case of tapping for metric thread/unified thread, the new tapping auto-setting scheme is valid only when parameter **D95** is set as follows:

D95 bit 2 = 0: The text file is invalid and tapping for metric thread is subject to the conventional auto-setting scheme.

= 1: The text file is valid and tapping for metric thread is subject to auto-setting based on editing.

D95 bit 1 = 0: The text file is invalid and tapping for unified thread is subject to the conventional auto-setting scheme.

= 1: The text file is valid and tapping for unified thread is subject to auto-setting based on editing.

The text file format, the text data items, and the editing procedure are shown below.

A. Text file format

```
[M]
PRE_DIA_1=80000 ;<M1> Diameter of Prehole(1/100000mm)        ←—— Pre-hole diameter
PRE_DIA_2=90000 ;<M1.1> Diameter of Prehole(1/100000mm)      ←—— Pre-hole diameter
            :
            :
[UN]
PRE_DIA_1=150000;<No.1-64UN> Diameter of Prehole(1/100000mm) ←—— Pre-hole diameter
PRE_DIA_2=180000;<No.2-56UN> Diameter of Prehole(1/100000mm) ←—— Pre-hole diameter
            :
            :
```

B. Text data items

- Pre-hole diameter (Setting unit: 1/100000 mm)
  This item denotes the auto-setting values for **NOM-ϕ** and **HOLE-ϕ** in the last drill tool sequence whose automatic tool development will be conducted for the tapping unit/counterbore-tapping unit.

C. Editing procedure

(1) Click the Start button and select "Programs" from the Start menu option. Then click "Explorer".

(2) After copying "TapPrDia.org" (an auto-setting model file for metric thread/unified thread tapping) within the "C:\nm64tdata" directory into this directory, change the file name to "TapPrDia.txt".

(3) Open "TapPrDia.txt" using a commercially available editor.

(4) Edit the file seeing the above description of "Text file format" and "Text data items" and taking notice of each data unit. An example of editing is shown below.

Note 1: If data is not entered correctly, alarm **494 AUTO TAP PROCESS IMPOSSIBLE** will be displayed when auto-setting is executed.
Enter data within the following range:

Item	Keyword	Input unit	Minimum value	Maximum value
Pre-hole diameter	PRE_DIA	1/100000 mm	10000	99990000

Enter integral decimal numbers.
For this item always enter "0" as the least two significant digits (that is, the last two digits).

Note 2: Even within the above data range, the particular combination of data settings in each item may display an asterisk (∗) to indicate that the amount of chamfering cannot be calculated. In such a case, to ensure that the amount of chamfering will be calculated properly, enter data in each item so that the calculation results in the following calculation expressions range from "0" to "99.9":
[If parameter **D44** is set to "0"]
(Chamfering) = {(Tap outside diameter) + (Thread pitch) × 2 – (Prehole diameter)}/2
[If parameter **D44** is set to "1"]
(Chamfering) = {(Tap outside diameter) – (Prehole diameter)}/2

Note 3: Even when data within the above data range is entered, alarm **493 AUTO PROCESS IMPOSSIBLE** may be displayed during automatic development of the tool data.

Note 4: Entered prehole diameter value has its respective last two digits cut away.

(5) After editing the file, execute "Overwrite & Save".

(6) Close "Explorer".

D. Example of editing

For "M1 tapping", proceed as follows to auto-set 0.7 mm as the prehole diameter:

(1) Open the text file "TapPrDia.txt".

(2) Move the cursor to the masked item shown below and then edit data in the required units. Do not edit other items.

```
[M]
PRE_DIA_1=70000 ;<M1>     Diameter of Prehole(1/100000mm)
PRE_DIA_2=90000 ;<M1.1>   Diameter of Prehole(1/100000mm)
         :
         :
```

Note 1: Since the default settings of the text file data conform to the conventional scheme, auto-set data cannot be modified by merely changing the value of bit 1 or bit 2 in the **D95** parameter.

Note 2: When modifying the metric thread/unified thread tapping auto-set data, the user itself needs to edit and manage the text file.

Note 3: After text file editing, the new data is incorporated into the auto-set data immediately.

Note 4: Even for inch specifications, assign data in units of 1/100000 mm to the text file.

Note 5: Since auto-set data having an assigned decimal point and exceeding the minimum allowable number of digits cannot be displayed, text file modifications may not be displayed as auto-settings intact.

Example: Even if the value of PRE_DIA_1 is changed to 8600, a nominal drill diameter of 0.9 may be displayed as its auto-set value.

2. Tapping for pipe thread

In the case of tapping for pipe thread, the new tapping auto-setting scheme is valid only when parameter **D95** is set as follows:

D95 bit 0 = 0: The text file is invalid and tapping for pipe thread is subject to the conventional auto-setting scheme.

= 1: The text file is valid and tapping for pipe thread is subject to auto-setting based on editing.

The text file format, the text data items, and the editing procedure are shown below.

A. Text file format

```
[PT]
;PT 1/8
DIAMETER_1=972800 ;Diameter(1/100000mm)                ←—— Tap outside diameter
THREAD_1=2800     ;Number of Thread(1/100Thread)       ←—— Total threads
DEPTH_1=1560000   ;Depth(1/100000mm)                   ←—— Thread depth
PRE_DIA_1=820000  ;Diameter of Prehole(1/100000mm)     ←—— Pre-hole diameter
PRE_DEP_1=1841000 ;Depth of Prehole(1/100000mm)        ←—— Pre-hole depth
             :
             :
[PF]
;PF 1/8
DIAMETER_1=972800 ;Diameter(1/100000mm)                ←—— Tap outside diameter
THREAD_1=2800     ;Number of Thread(1/100Thread)       ←—— Total threads
PRE_DIA_1=886000  ;Diameter of Prehole(1/100000mm)     ←—— Pre-hole diameter
             :
             :
[PS]
;PS1/8
DIAMETER_1=972800 ;Diameter(1/100000mm)                ←—— Tap outside diameter
THREAD_1=2800     ;Number of Thread(1/100Thread)       ←—— Total threads
DEPTH_1=1550000   ;Depth(1/10000mm)                    ←—— Thread depth
PRE_DIA_1=850000  ;Diameter of Prehole(1/100000mm)     ←—— Pre-hole diameter
PRE_DEP_1=1831000 ;Depth of Prehole(1/100000mm)        ←—— Pre-hole depth
             :
             :
```

B. Text data items

- Tap outside diameter (Setting unit: 1/100000 mm)
This item denotes the auto-setting values for **MAJOR-ϕ** of the tapping unit/counterbore-tapping unit and **HOLE-ϕ** in the tool sequence for the tap. (PT, PF, and PS pipe threads)
- Total threads (Setting unit: 1/100 threads)
This item refers to the total number of threads per inch of a tap, and this value is used for auto-setting **PITCH** of the tapping unit/counterbore-tapping unit. (PT, PF, and PS pipe threads)
- Thread depth (Setting unit: 1/100000 mm)
This item denotes the auto-setting value for **TAP-DEP** of the tapping unit/counterbore-tapping unit. (PT and PS pipe threads)
- Pre-hole diameter (Setting unit: 1/100000 mm)
This item denotes the auto-setting values for **NOM-ϕ** and **HOLE-ϕ** in the last drill tool sequence whose automatic tool development will be conducted for the tapping unit/counterbore-tapping unit. (PT, PF, and PS pipe threads)
- Pre-hole depth (Setting unit: 1/100000 mm)
This item denotes the auto-setting value for **HOLE-DEP** in the last drilling tool sequence for which automatic tool development will be conducted for the tapping unit/counterbore-tapping unit. (PT and PS pipe threads)

C. Editing procedure

(1) Click the Start button and select "Programs" from the Start menu option. Then click "Explorer".

(2) After copying "Pipescdt.org" (an auto-setting model file for pipe thread tapping) within the "C:\nm64tdata" directory into this directory, change the file name to "Pipescdt.txt".

(3) Open "Pipescdt.txt" using a commercially available editor.

(4) Edit the file seeing the above description of "Text file format" and "Text data items" and taking notice of each data unit. An example of editing is shown below.

Note 1: If data is not entered correctly, alarm **494 AUTO TAP PROCESS IMPOSSIBLE** will be displayed when auto-setting is executed.
Enter data within the following range:

Item	Keyword	Input unit	Minimum value	Maximum value
Tap outside diameter*	DIAMETER	1/100000 mm	100	9999900
Total threads	THREAD	1/100 threads	260	21474836470
Thread depth*	DEPTH	1/100000 mm	100	99999900
Pre-hole diameter*	PRE_DIA	1/100000 mm	1000	99990000
Pre-hole depth*	PRE_DEP	1/100000 mm	1000	99990000

Enter integral decimal numbers.

*For these items always enter "0" as the least significant digit (that is, the last digit).

Note 2: Even within the above data range, the particular combination of data settings in each item may display an asterisk (∗) to indicate that the amount of chamfering cannot be calculated. In such a case, to ensure that the amount of chamfering will be calculated properly, enter data in each item so that the calculation results in the following calculation expressions range from "0" to "99.9":
[If parameter **D44** is set to "0"]
(Chamfering) = {(Tap outside diameter) + (Thread pitch) × 2 – (Prehole diameter)}/2
[If parameter **D44** is set to "1"]
(Chamfering) = {(Tap outside diameter) – (Prehole diameter)}/2

Note 3: Even when data within the above data range is entered, alarm **493 AUTO PROCESS IMPOSSIBLE** may be displayed during automatic development of the tool data.

Note 4: Entered prehole diameter and depth values have their respective last two digits cut away.

(5) After editing the file, execute "Overwrite & Save".

(6) Close "Explorer".

D. Example of editing

For "PT1/8", proceed as follows to auto-set 10.117 mm as the tap outside diameter, 27 as the number of threads, 11 mm as the thread depth, 8.43 mm as the prehole diameter, and 17 mm as the prehole depth:

(1) Open the text file "Pipescdt.txt" and move the cursor to "PT1/8".

(2) Move the cursor to each masked item shown below and then edit data in the required units. Do not edit other items.

```
[PT]
;PT 1/8
DIAMETER_1=1011700 ;Diameter(1/100000mm)
THREAD_1=2700      ;Number of Thread(1/100Thread)
DEPTH_1=1100000    ;Depth(1/100000mm)
PRE_DIA_1=843000   ;Diameter of Prehole(1/100000mm)
PRE_DEP_1=1700000  ;Depth of Prehole(1/100000mm)
        :
        :
```

Note 1: Since the default settings of the text file data conform to the conventional scheme, auto-set data cannot be modified by merely changing the value of bit 0 in the **D95** parameter.

Note 2: When modifying the thread tapping auto-set data, the user itself needs to edit and manage the text file.

Note 3: After text file editing, the new data is incorporated into the auto-set data immediately.

Note 4: Even for inch specifications, assign data in units of 1/100000 mm to the text file.

Note 5: Since auto-set data having an assigned decimal point and exceeding the minimum allowable number of digits cannot be displayed, text file modifications may not be displayed as auto-settings intact.

Example: Even if the value of PRE_DIA_1 is changed to 625000, a nominal drill diameter of 6.3 may be displayed as its auto-set value.

3-4-6 Tool sequence data of the point machining unit

The tool sequence data are automatically developed by entering the machining unit.

However, certain data must be set by means of menu keys or numeric keys on the basis of the tool used or the machining procedure.

List of tool sequence data

	TOOL	NOM-ϕ		#	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
CTR DR	O	O	O	O	O	◆	◆	◆	O	◆	O	O	O	O
DRILL	O	O	O	O	O	O	O	O	O	O	O	O	O	O
CHAMFER	O	O	O	O	O	O	O	O	◆	O	O	O	O	O
END MILL	O	O	O	O	O	O	O	O	O	O	O	O	O	O
BCK FACE	O	O	O	O	O	O	◆	O	◆	◆	O	O	O	O
REAMER	O	O	O	O	O	O	◆	◆	O	O	O	O	O	O
TAP	O	O	O	O	O	O	O	O	O	O	O	O	O	O
BOR BAR	O	O	O	O	O	O	O	O	O	O	O	O	O	O
B-B BAR	O	O	O	O	O	O	O	O	O	O	O	O	O	O
	1	2	3	13	4	5	6	7	8	9	10	11	12	12

O : Setting possible.
◆ : Not necessary to be set here.

Remark: If **[TAPPING CYCLE]** menu item is selected for **PRE-DIA**, there is no need to set data in **PRE-DEP**.

1. **Tool designation: TOOL**

Used to specify the name of the tool to be used for machining. The tool designation can be changed by means of menu keys.

CENTER DRILL	DRILL	CHAMFER CUTTER	ENDMILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR	

2. **Nominal diameter of tool: NOM-ϕ**

Used to specify the nominal diameter of the tool.

Setting range (0 to 999.9 mm)

Note 1: For the end mill and the chamfering cutter, their nominal diameter is entered by means of numeric keys.

Note 2: The alarm **495 NO ASSIGNED TOOL IN TOOL FILE** is given if the tool entered has not been previously recorded in the **TOOL FILE** display.

3. **Tool identification**

Select the identification symbol of tools of the same nominal diameter from the menu.

A	B	C	D	E	F	G	H		>>>

4. Diameter of machining hole: HOLE-ϕ

Used to specify the diameter of the hole to be machined.

The data for this article can be modified by means of numeric keys.

Note: For the chamfering cutter, this concerns a value equal to twice the distance from the centerline to the wall of the hole.
Enter 999 if there is no interference.

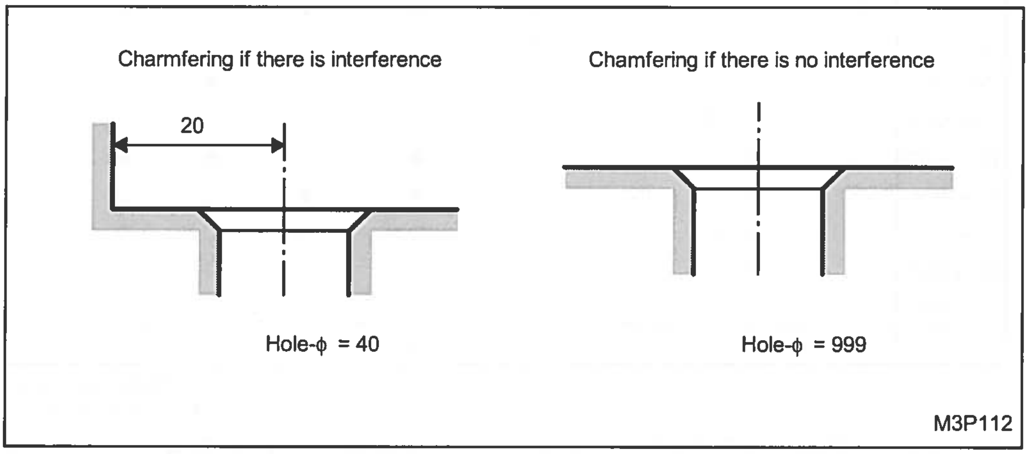


Fig. 3-2 Specification of diameter of machining hole

5. Depth of machining hole: HOLE-DEP

Used to specify the depth of the hole to be machined.

The data for this article can be modified by means of numeric keys.

Note 1: For the chamfering cutter, this article is specified as illustrated below.

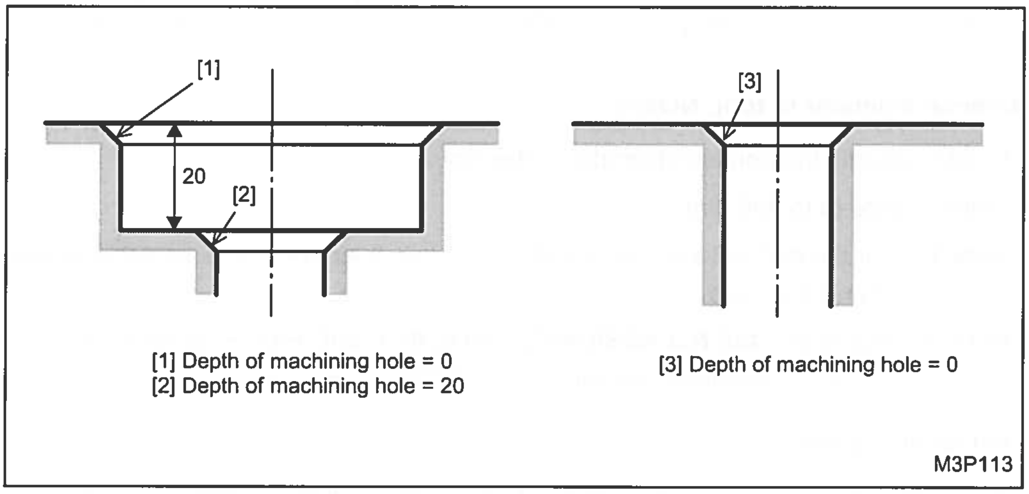
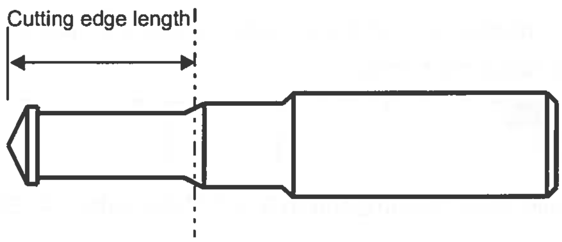


Fig. 3-3 Specification of depth of machining hole

Note 2: For planetary tapping, the appropriate data for the selected type of tool must be set. Enter the data specified in the corresponding tool catalogue. Enter the catalogued cutting edge length in **HOLE-DEP**.



D735P0072

6. **Diameter of pre-hole: PRE-DIA**

Used to specify the diameter of the pre-hole for the final hole to be machined.

The data for this article can be modified by means of numeric keys.

Note 1: In the case of boring, the boring cycle can be selected from the menu. **[CYCLE 1]** is selected at the time of automatic tool development.

CYCLE 1	CYCLE 2	CYCLE 3							

For details, refer to Subsection 3-4-7 “Tool path”, “8. Boring tool”.

Note 2: For back boring, enter the diameter of the through hole.

Note 3: In the case of tapping, the tapping cycle can be selected from the menu. **[TAPPING CYCLE]** is selected at the time of automatic tool development.

TAPPING CYCLE	PECKING CYCLE	PLANET CYCLE							

[TAPPING CYCLE] Conventional tappping cycle

[PECKING CYCLE] Pecking cycle using a synchronous tap

[PLANET CYCLE] Machining cycle using a planetary tapping tool (only for machines with the Y-axis)

For details, refer to Subsection 3-4-7 “Tool path”, “7. Tap”.

7. **Depth of the pre-hole: PRE-DEP**

Used to specify the depth of the pre-hole for the final hole to be machined.
The data for this article can be modified by means of numeric keys.

Note 1: Enter the depth of the through hole in the case of back facing or back boring for this article.

Note 2: Enter the depth of the faced hole in the case of boring for this article. Consequently, preset data of 0 is displayed for through hole boring and non-through hole boring.

Note 3: Enter the interference depth in the case of chamfering for this article.

Note 4: For the end mill, the direction of cutting can be selected from the menu. **[CCW CUT]** is selected at the time of automatic tool development.

CW CUT	CCW CUT								

For the tornado cycle of the circular milling unit, the direction of cutting can be selected from the following menu:

CW CUT	CCW CUT								

For details, refer to Subsection 3-4-7 "Tool path", "4. End mill".

Note 5: Data setting is not required for **[TAPPING CYCLE]**. Set "Cutting depth per peck" for **[PECKING CYCLE]**. The value of the **U64** parameter "Pre-hole machining feed" is set for **[PLANET CYCLE]** automatically.

8. **Cutting surface roughness: RGH**

Enter the cutting surface roughness by means of numeric keys or menu keys.

▼ 1	▼ 2	▼▼ 3	▼▼ 4	▼▼▼ 5	▼▼▼ 6	▼▼▼ 7	▼▼▼▼ 8	▼▼▼▼ 9	

Note 1: For the centering drill, the angle of tool tip can be selected from the menu.
In automatic tool development mode, 90° is selected.

90°	118°	60°							

Note 2: For the drill, the drilling cycle can be selected from the menu. In automatic tool development mode, these data are automatically determined on the basis of the machining depth, the drill diameter and the parameters concerned.

DRILLING CYCLE	PECKING CYCLE 1	PECKING CYCLE 2	PECKING CYCLE 3	AUTOPECK CYCLE		DECREME PECKING CYCLE 1	DECREME PECKING CYCLE 2	DECREME PECKING CYCLE 3	

For details, refer to Subsection 3-4-7 "Tool path", "2. Drill".

Note 3: Enter the duration of the stopping time for the tapping.
In automatic tool development mode, **FIX** is selected.

Note 4: For end mill (Tornado cycle)
During automatic tool development, the system sets the same value as for the **BTM** item of the circular milling unit. If the **BTM** item value of the circular milling unit is 0, bottom finishing will not occur. Unless the **BTM** item value is 0, bottom finishing will occur.

9. Cutting depth: DEPTH

Used to specify the cutting depth or the amount of chamfering at the time of the machining according to the type of tool:

- Cutting depth on Z-axis per pass in the case of drill.
- Amount of chamfering in the case of chamfering cutter.
- Radial cutting depth or amount of chamfering in the case of circular milling cycle or tornado milling cycle of the end mill, respectively.
- In the case of boring with a reamer, specify the return speed of the reamer (as feed per minute) by means of menu keys or numeric keys. In tool automatic development mode **[CUT G01]** (cutting feed) is selected.

CUT G01	RAPID G00								

Cutting feed speed is selected by parameter **D18**.

- Thread pitch in the case of tap.
- Cutting depth in the radial direction in the case of boring bar and back boring tool.

10. Circumferential speed: C-SP

To auto-set a circumferential speed (m/min) and feedrate (mm/rev), select the corresponding tool material type from the menu.

The tool material types in the menu are the same as those which have been set on the **CUTTING CONDITION (REGIST)** display.

To register new tool material types, refer to Section of “CUTTING CONDITION Display”, of the relevant Operating Manual.

HSS AUTO	CARBIDE AUTO								

Data can also be set using the numeric keys.

11. Feedrate: FR

Used to specify the feedrate of the tool. Same as the circumferential speed, the entry of data is done by means of menu keys or numeric keys.

12. M-code: M

Used to specify the M-code which is immediately output after the ATC of the tool involved by using the numeric keys.

Note: Two figures or three figures of M-code can be specified in the left or right data item respectively.

13. Retraction position of the lower turret: #

For a machine having upper and lower turrets, it is possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.

The following menu is displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

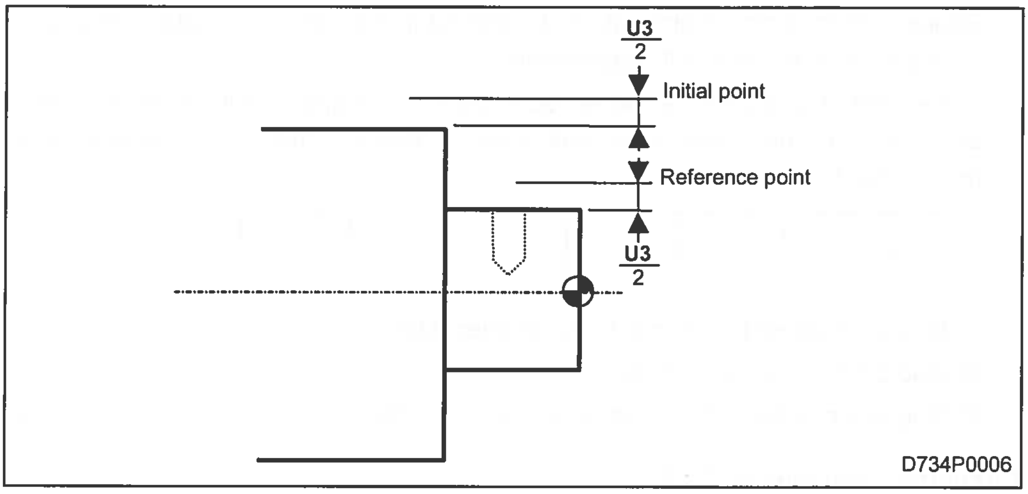
Note: For details refer to Chapter 4 “LOWER-TURRET CONTROL FUNCTIONS”.

3-4-7 Tool path of the point machining unit

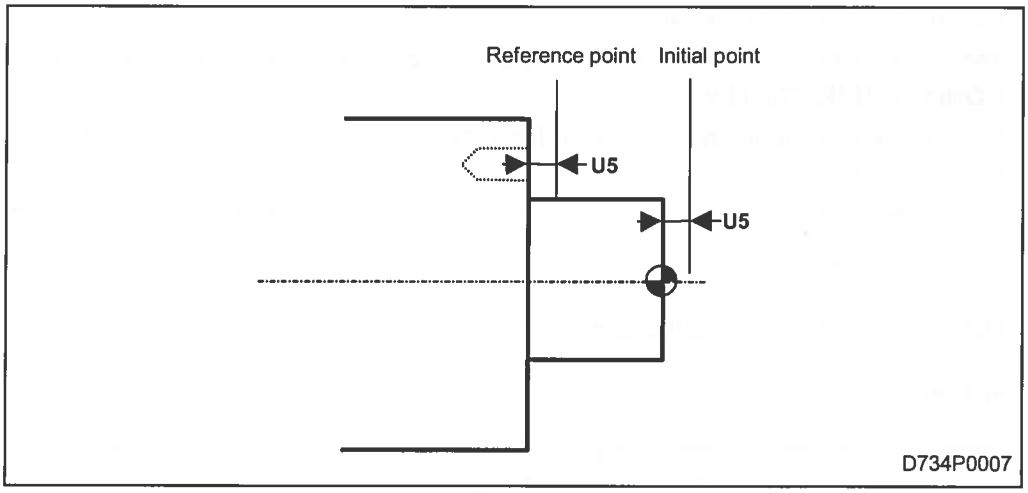
This section shows the path of each tool used during execution of a point machining unit.

The initial and reference points in each tool path are as shown below.

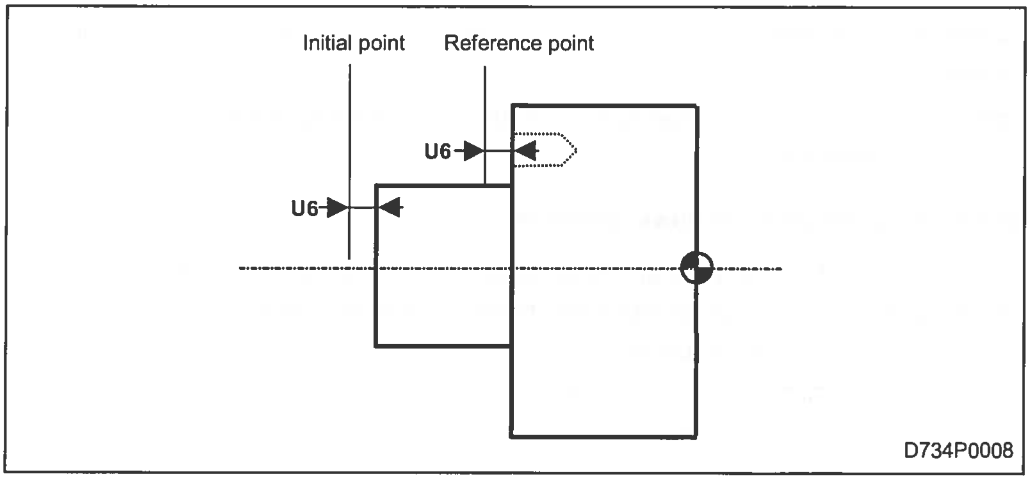
- When the selected mode is **ZC** or **ZY**



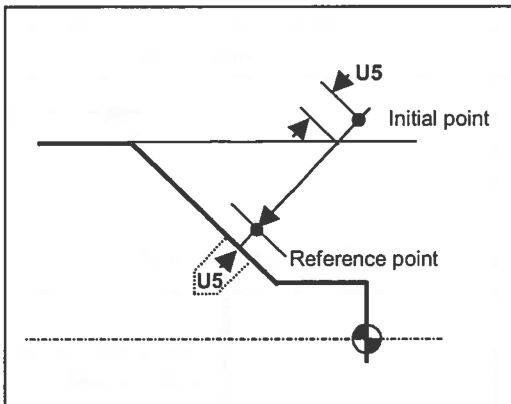
- When the selected mode is **XC** or **XY**



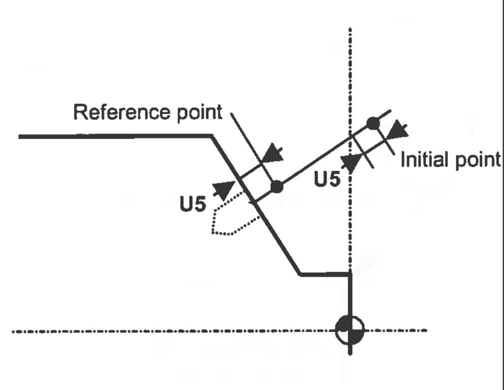
- When the selected mode is [XC] or [XY]



- When the selected mode is /C or /Y

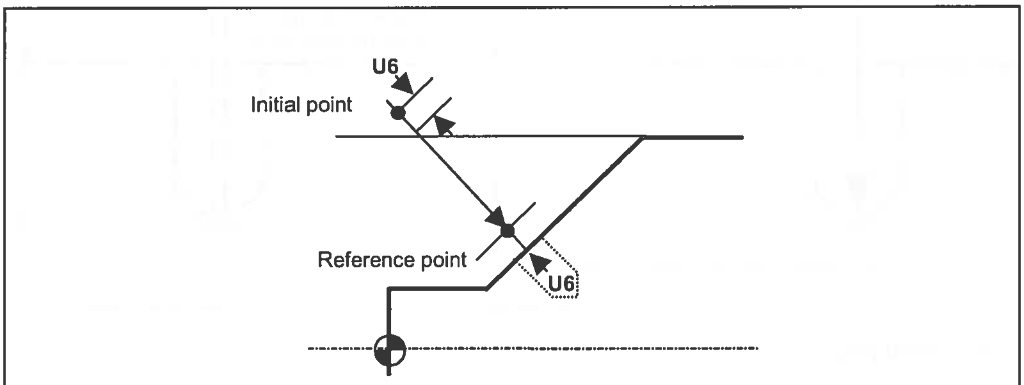


D734P0009



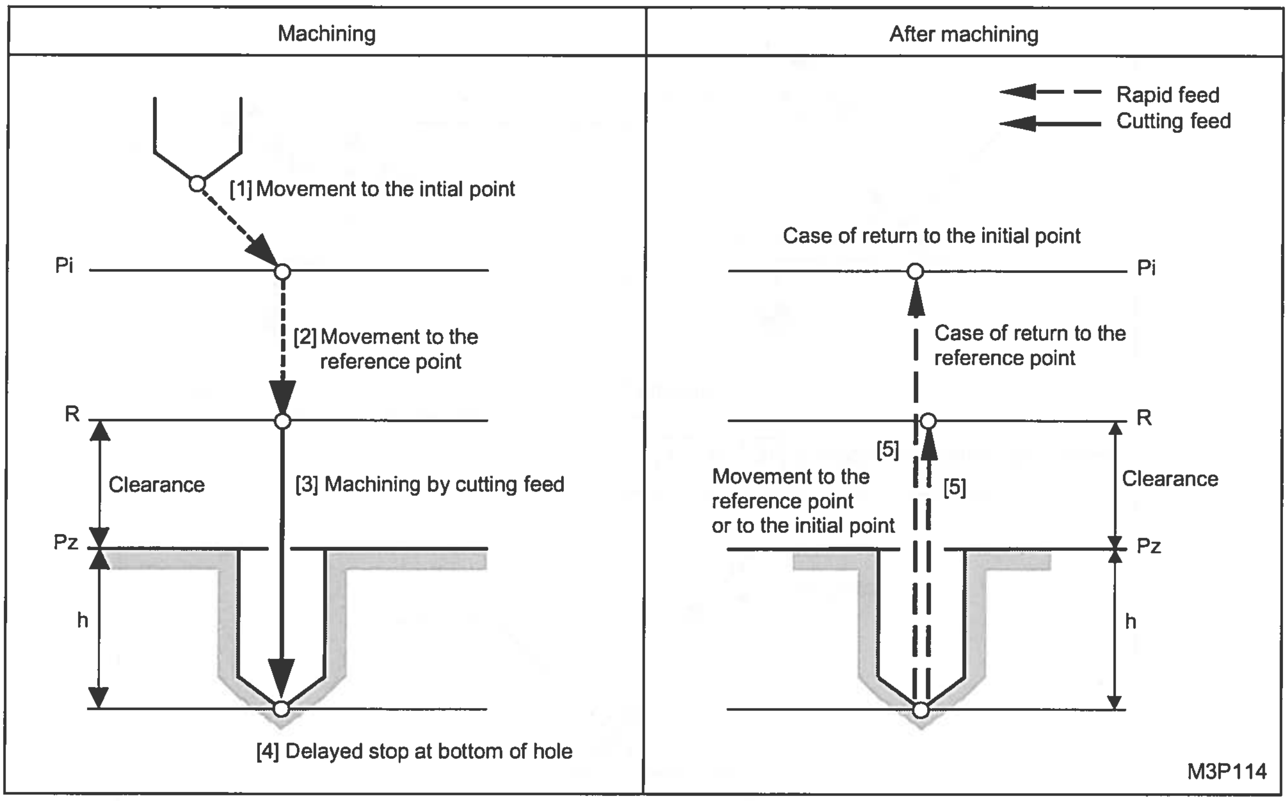
D734P0010

- When the selected mode is /C or /Y



D734P0011

1. Centering drill



Pi: Initial point

Pz: Coordinate value in the cutting direction to be entered in the shape sequence

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	U5

h: Depth of the hole to be calculated by the data **HOLE-ϕ** and **RGH** (angle of tool tip) entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display

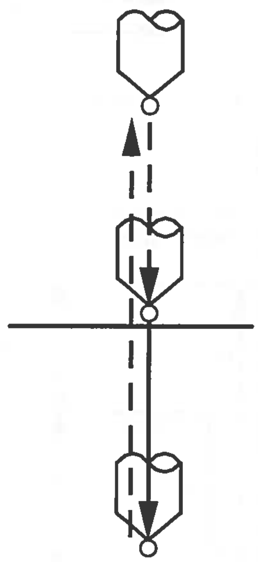
$$h = \frac{\dfrac{\text{Diameter of machining hole}}{2}}{\tan\left(\dfrac{\text{Angle of cutting tool tip}}{2}\right)} + \text{Tool correction}$$

Note: The time of delayed stop of the feed at bottom of hole is set by the parameter **D3**.

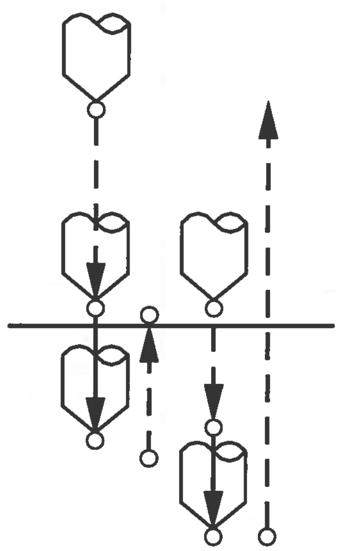
2. Drill

The cycle of machining with drill is available in the following 8 types.

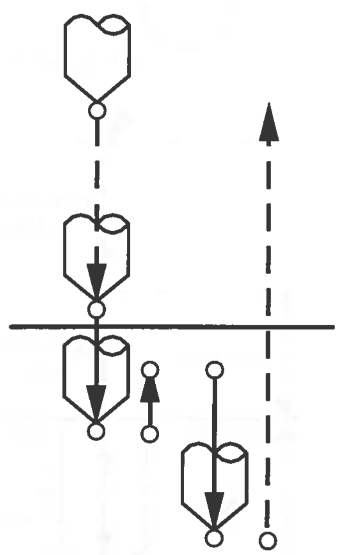
A. DRILLING CYCLE



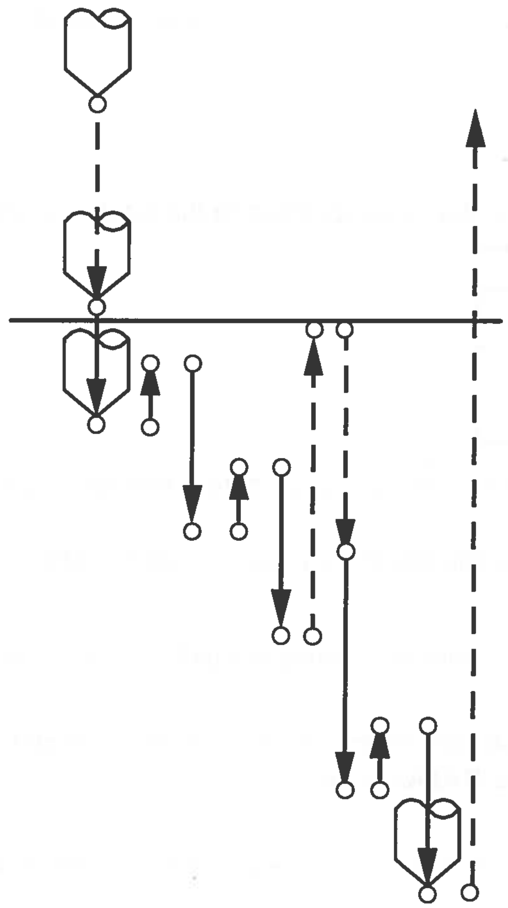
B. PECKING CYCLE 2



C. PECKING CYCLE 1



D. PECKING CYCLE 3



E. Automatic pecking cycle

While monitoring the cutting load torque applied during deep-hole machining with the drill, the system activates the pecking function, only when necessary. Thus, tool damage is avoided and the machining time saved.

F. Decremental deep-hole machining cycle

This cycle, compared with the normal deep-hole machining cycle, decrements the cutting depth in steps with each cutting operation.

G. Decremental high-speed deep-hole machining cycle

This cycle, compared with the normal high-speed deep-hole machining cycle, decrements the cutting depth in steps with each cutting operation.

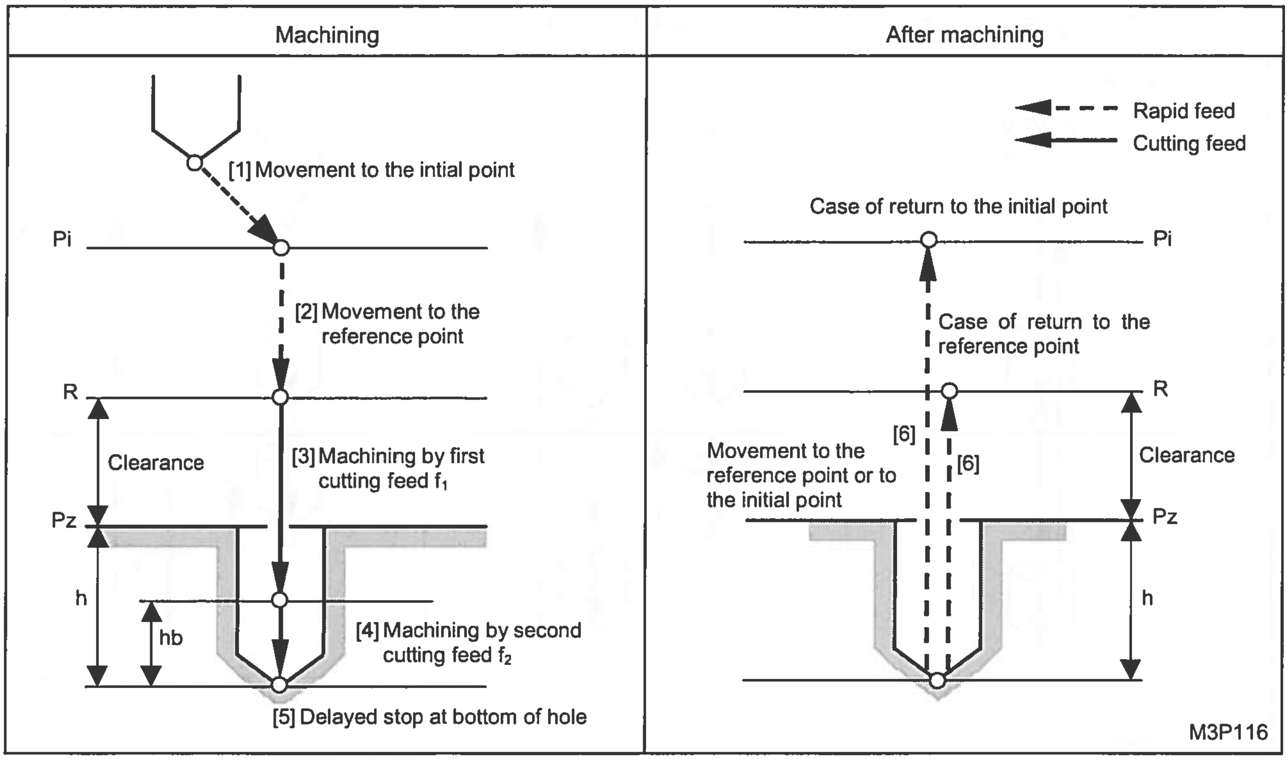
H. Decremental ultra-deep-hole machining cycle

This cycle, compared with the normal ultra-deep-hole machining cycle, decrements the cutting depth in steps with each cutting operation.

D734P0012

Remark: Use parameters **D45** and **D46** to set the decrement and the minimum cutting depth, respectively, for the intended decremental machining cycle.

A. Drilling cycle



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1** or **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display
h = Depth of machining hole + Tool correction

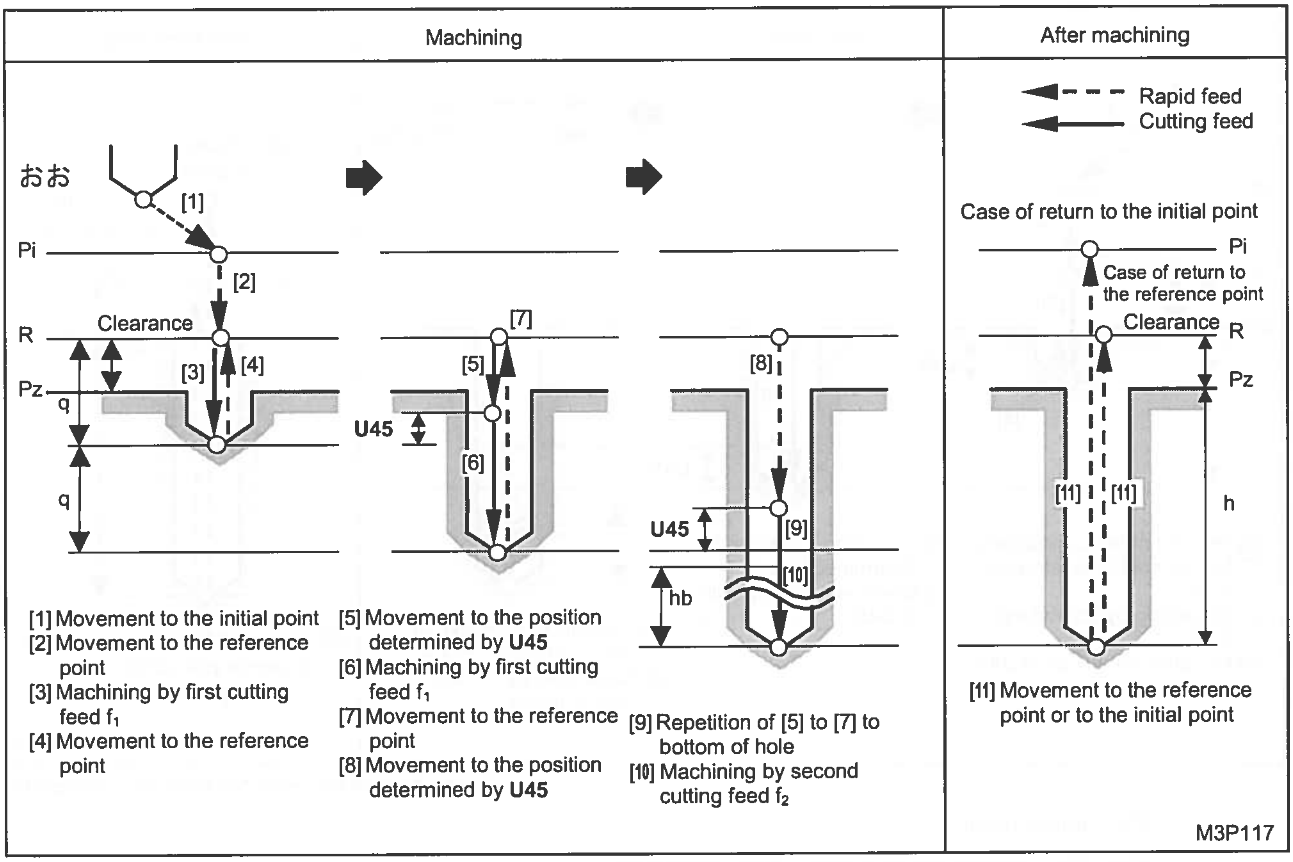
hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

f_1: Feedrate (**FR**) to be set for the tool sequence

f_2: Feedrate to be modified by the data **PRE-DEP** (feedrate updating rate)
$f_2 = f_1 \times$ Feedrate updating rate

Note: The time of delayed stop of the feed at bottom of hole is set by the parameter **K77**.

B. Deep hole drilling cycle (PECKING CYCLE 2)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\mathbf{U3}}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1** or **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display
h = Depth of machining hole + Tool correction

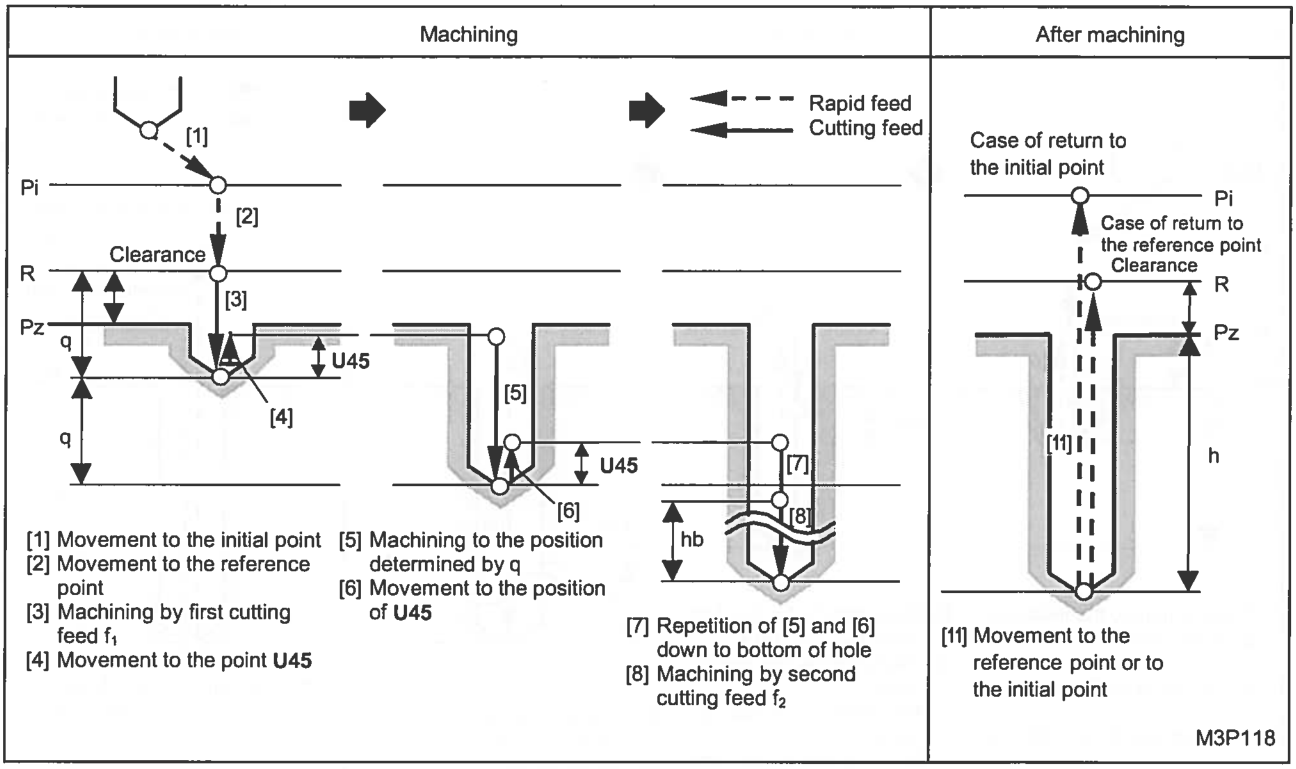
q: Cutting depth to be entered in the tool sequence data

hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

f_1: Feedrate (**FR**) to be set for the tool sequence

f_2: Feedrate to be modified by the data **PRE-DEP** (feedrate updating rate)
$f_2 = f_1 \times$ Feedrate updating rate

C. High-speed hole drilling cycle (PECKING CYCLE 1)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	**U3**/2
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1** or **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

q: Cutting depth to be entered in the tool sequence data

hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

f_1: Feedrate (**FR**) to be set for the tool sequence

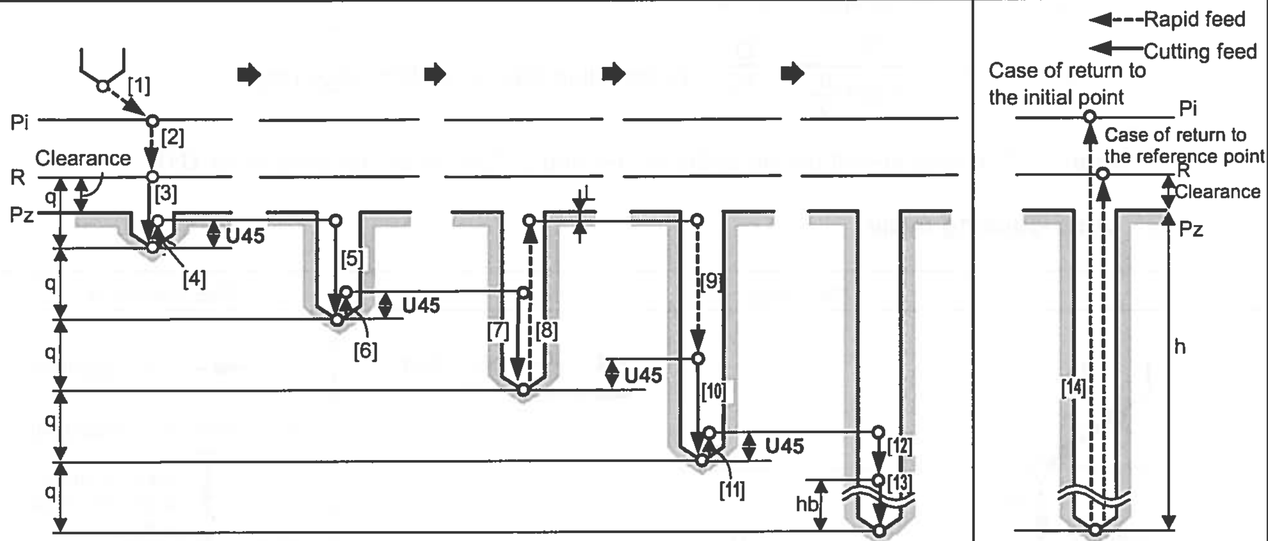
f_2: Feedrate to be modified by the data **PRE-DEP** (feedrate updating rate)

$f_2 = f_1 \times$ Feedrate updating rate

Note: The feed speed on the paths [4] and [6] is set by the parameter **U18**.

D. Ultra-deep drilling cycle (PECKING CYCLE 3)

Machining	After machining



[1] Movement to the initial point
[2] Movement to the reference point
[5] Machining by first cutting feed f_1
[4] Movement to the point **U45**
[5] Machining to the position determined by q
[6] The tool moves downward through the amount of relief of pecking.
[7] Machining to the position determined by q
[8] After pecking has been repeated **P73** times, the tool moves to the entrance of the hole.
[9] Movement to the position of **U45**
[10] Machining to the position determined by q
[11] The tool moves downward through the amount of relief of pecking.
[12] Repetition of [7] and [9] down to bottom of hole
[13] Machining by second cutting feed f_2

[14] Movement to the reference point or to the initial point

D734P0013

The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\mathbf{U3}}{2}$
XC, XY, /C, /Y XC, XY, /C, /Y	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1** or **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display
h = Depth of machining hole + Tool correction

q: Cutting depth to be entered in the tool sequence data

hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

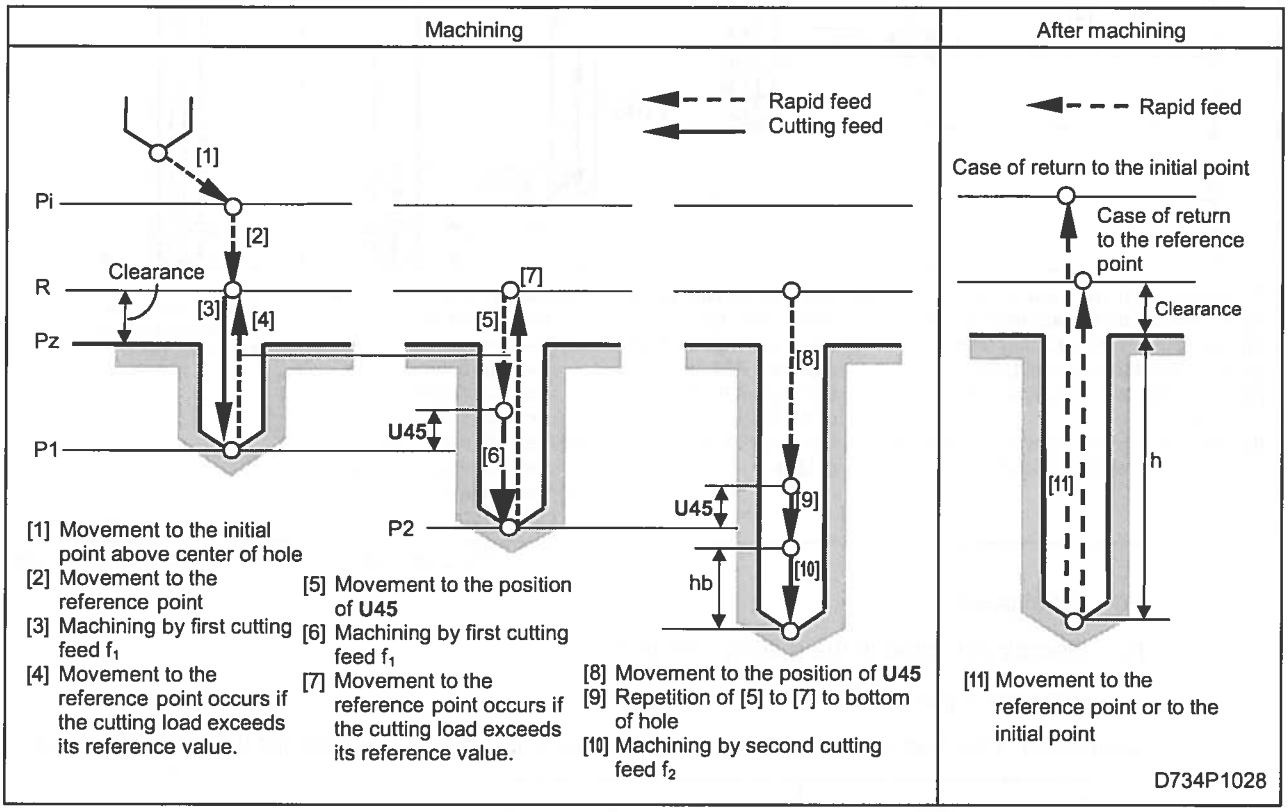
f_1: Feedrate (**FR**) to be set for the tool sequence

f_2: Feedrate to be modified by the data **PRE-DEP** (feedrate updating rate)
$f_2 = f_1 \times$ Feedrate updating rate

L : $L = \dfrac{D}{2 \times \tan\frac{\theta}{2}} + \dfrac{D}{10}$ (D: tool diameter, θ: cutting edge angle)

Note: The feed speed on the paths [4], [6] and [11] is set by the parameter **U18**.

E. Auto-pecking cycle



The bold codes represent the parameter addresses.

Note 1: Auto-pecking operates with the drilling tools that are 4 mm (0.16 in.) or more in diameter and of the high-speed steel specifications. Whether the auto-pecking function actually works depends on the particular status of the machine and/or tool. If this function does not work, use the decremental drill-pecking cycle.

Note 2: The auto-pecking function does not operate for specific machine specifications such as universal inverter specifications.

Pi: Initial point

Pz: Coordinate value in the cutting direction

P_1, P_2: The positions where autonomous pecking will occur if the cutting load exceeds its reference value

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	U5

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1** or **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

hb: **PRE-DIA** to be set for the tool sequence

f_1: Feedrate (**FR**) to be set for the tool sequence

f_2: f_1 × **PRE-DEP** to be set for the tool sequence

3. Chamfering cutter

Chamfering is classified into two types: Chamfering performed by the tool which only moves on the axis (Cycle 1) and chamfering performed by the tool which moves on the plane direction (Cycle 2).
The cycle used is selected automatically.

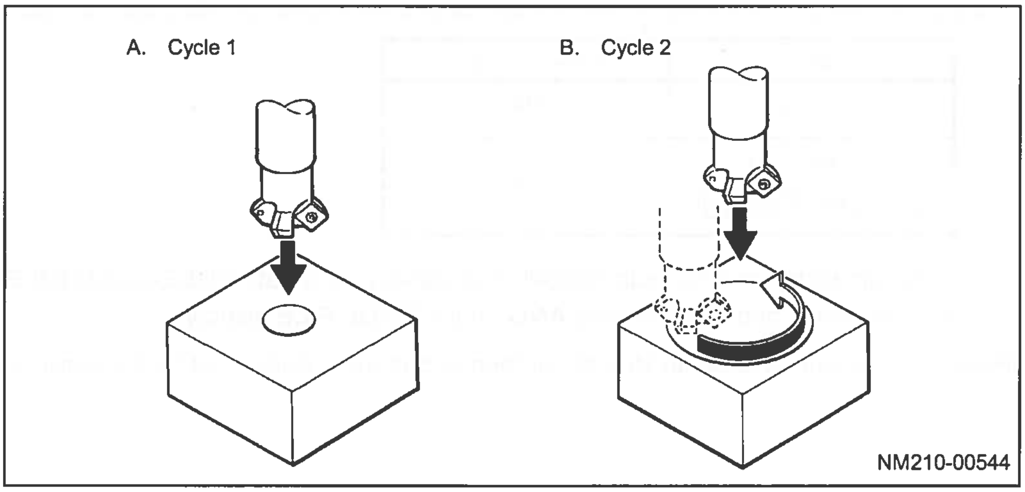
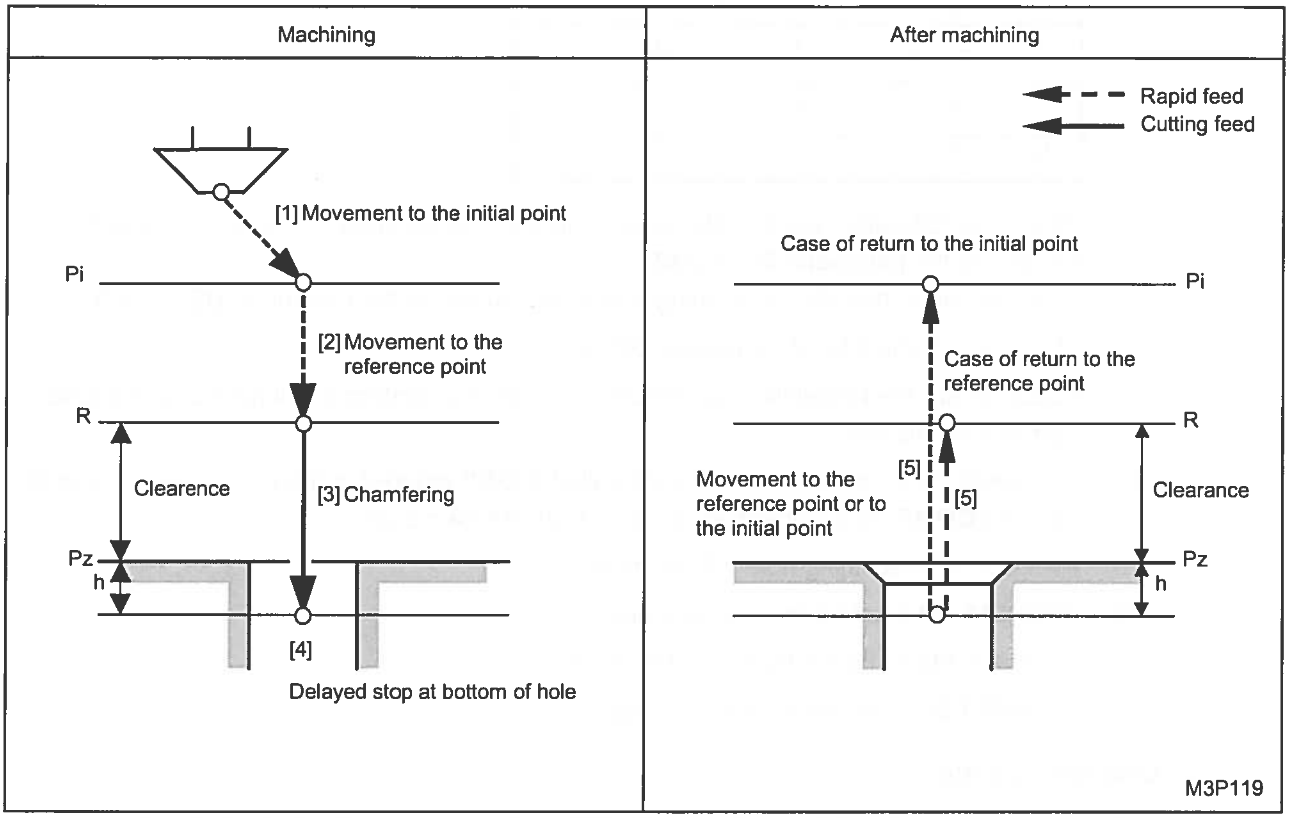


Fig. 3-4 Cycle 1 and cycle 2

A. Cycle 1



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

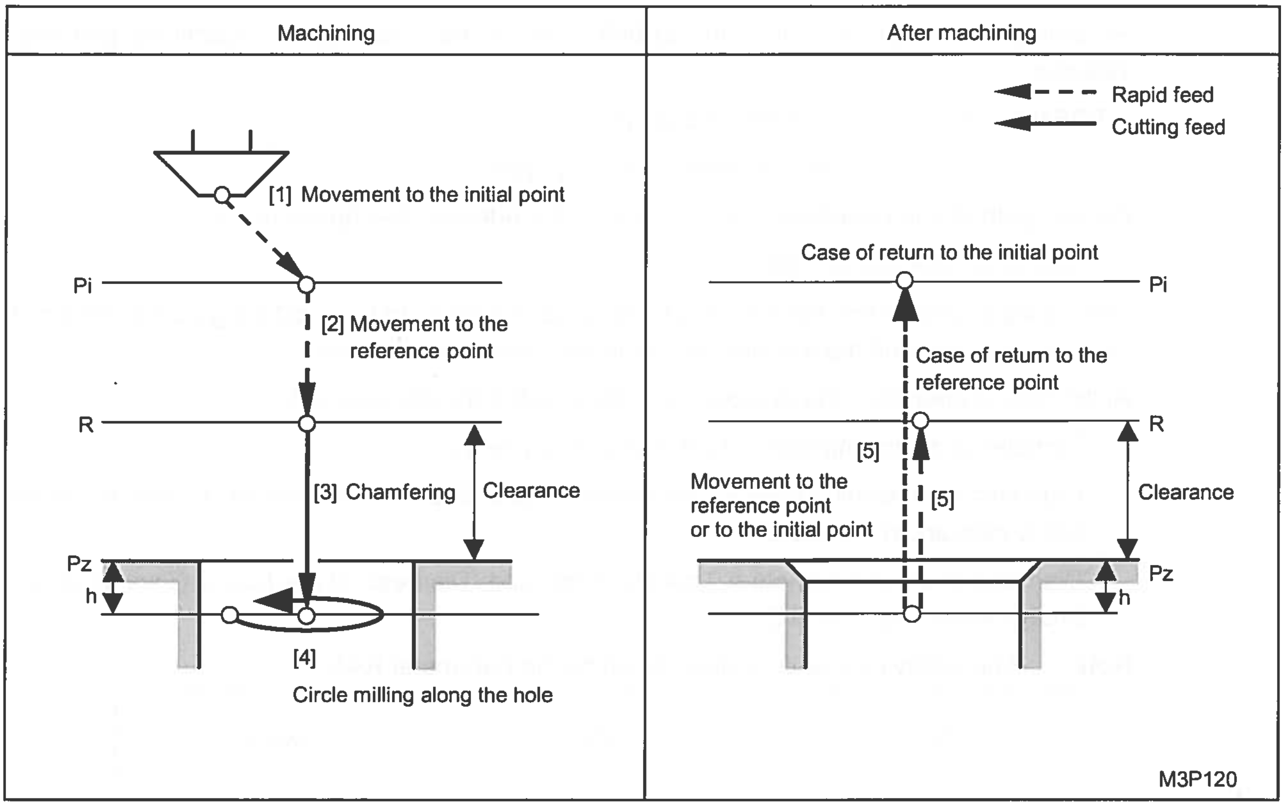
Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

h: Optimum distance to be automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** in the tool sequence and also the data **ANG** on the **TOOL FILE** display

Note: The time of delayed stop of the feed at bottom of hole is set by the parameter **D16**.

B. Cycle 2



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{\text{XC}}$, $\boxed{\text{XY}}$, $\boxed{\text{/C}}$, $\boxed{\text{/Y}}$	**U5**

When the following condition is fulfilled, the clearance before machining will be equaled to the parameter **D42**.
However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 7 of parameter **D91** is 1.

h: The optimum distance is automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** of the tool sequence and also the data **ANG** on the **TOOL FILE** display.

Note: For the circular milling, refer to the paragraph dealing with 4. End mill, 2. Circular milling-B.

4. End mill

According to the set value in item **TORNA.**, one of the following two machining patterns is selected.

TORNA.: 0 ....circular milling cycle

1 ....circular tornado milling cycle

For tool path of each machining pattern refer to the relevant description below.

<In case of circular milling cycle>

End milling is divided into the following three types according to the machining hole diameter, the pre-hole diameter and the tool diameter entered in the tool sequence.

At the time of operation, the appropriate cycle is automatically selected.

- Diameter of machining hole = Tool diameter (Cycle 1)
- "Diameter of machining hole > Tool diameter" and "Diameter of pre-hole > (Tool diameter + Safety clearance)" (Cycle 2)
- "Diameter of machining hole > Tool diameter" and "Diameter of pre-hole ≤ (Tool diameter + Safety clearance)" (Cycle 3)

Note: The safety clearance is determined by the parameter **K43**.

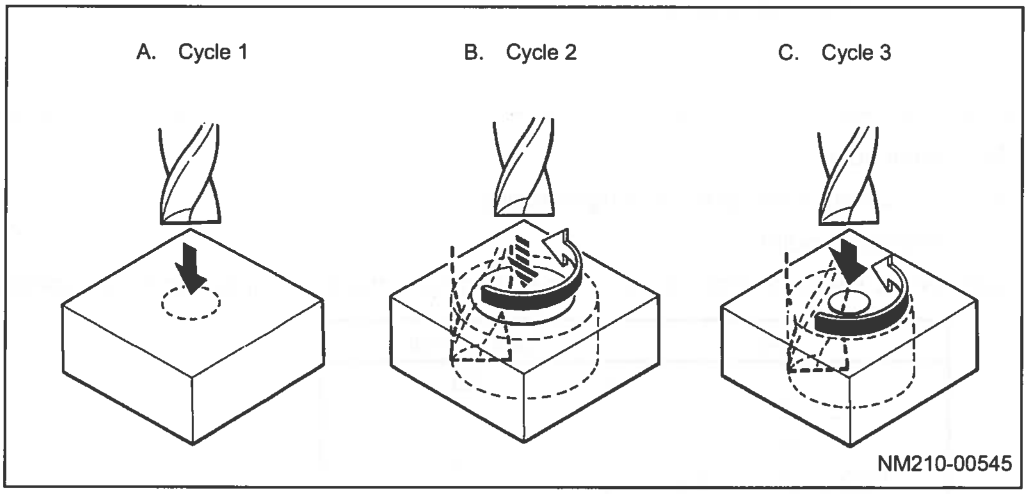
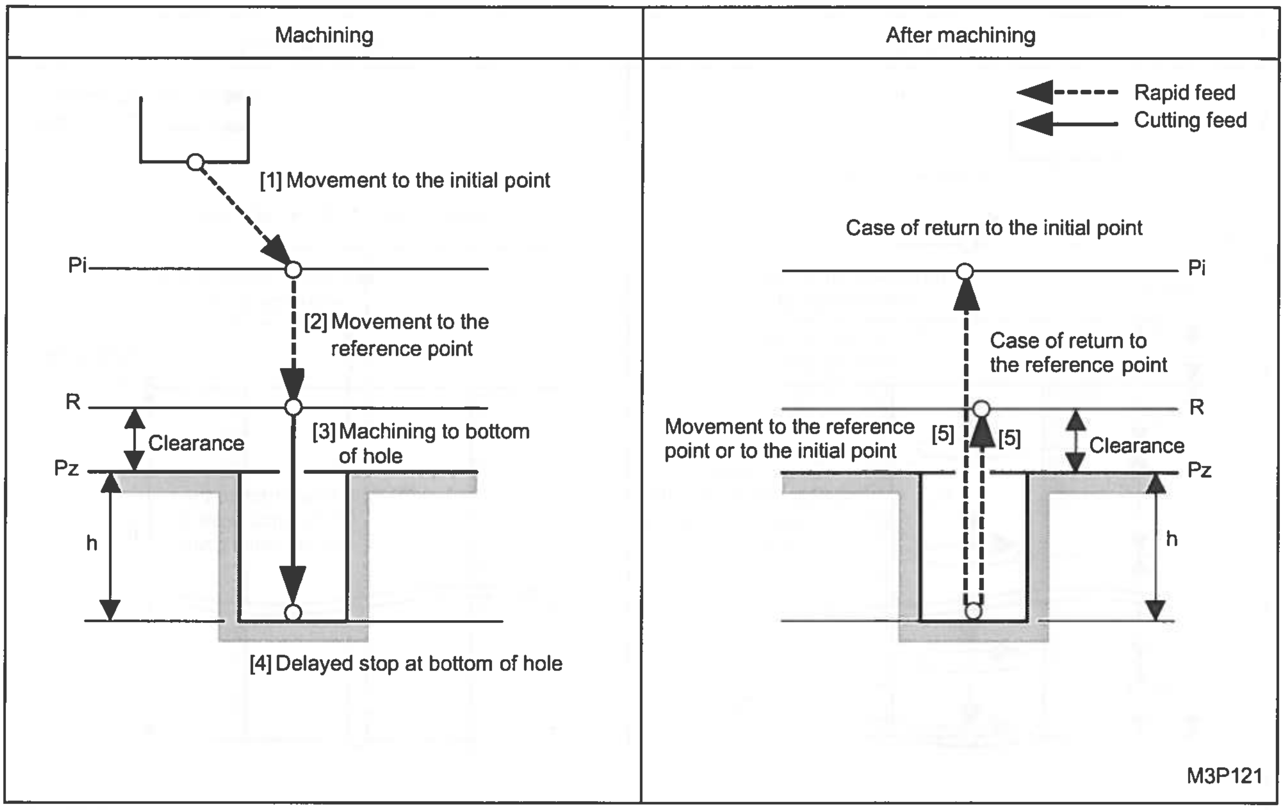


Fig. 3-5 Cycle 1, 2, 3

A. Cycle 1



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

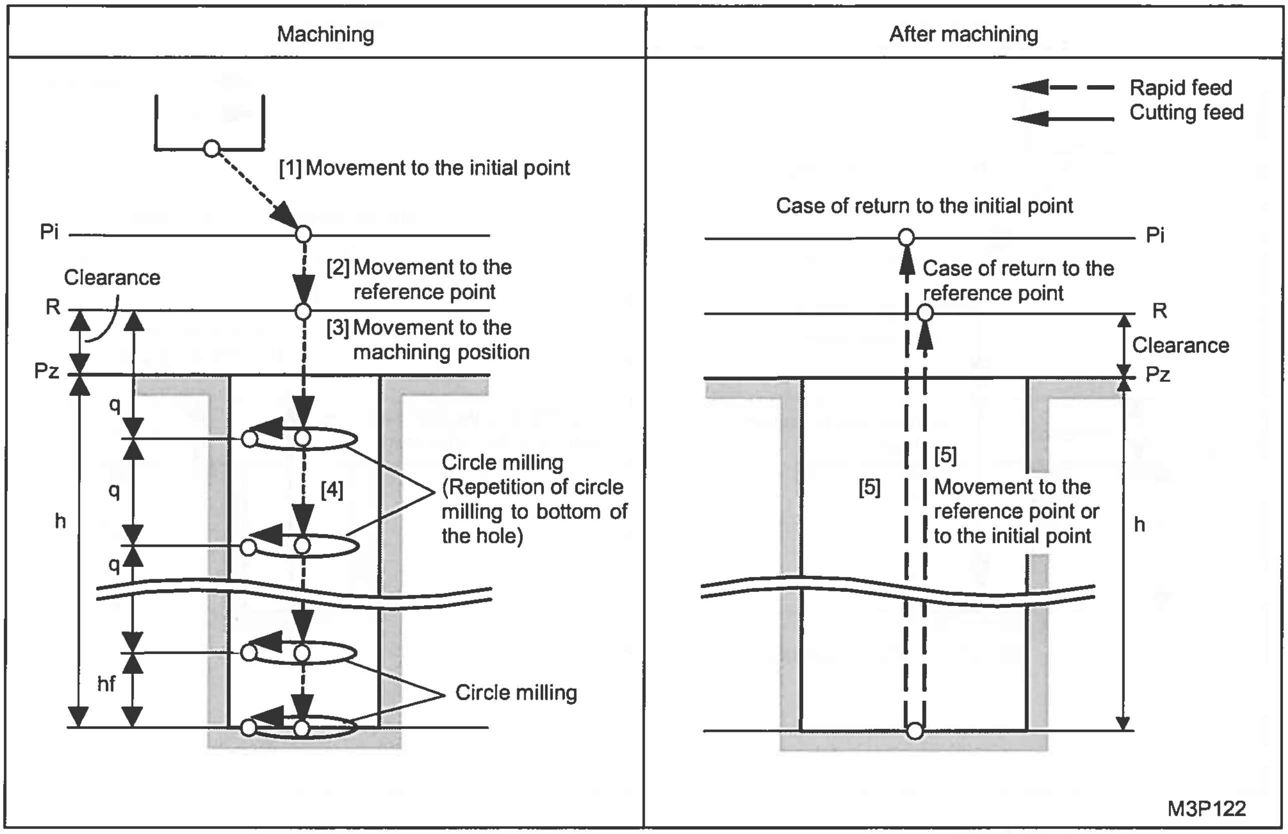
Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	U5

h: Depth of machining hole (**HOLE-DEP**) to be entered in the tool sequence

Note: The time of delayed stop of the feed at bottom of hole is set by the parameter **D19**.

B. Cycle 2



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y XC, XY, /C, /Y	U5

h: Optimum distance to be automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** in the tool sequence and also the data **ANG** on the **TOOL FILE** display

hf: Bottom finishing allowance to be determined by the data **RGH** entered in the tool sequence and also by the parameter **D21**

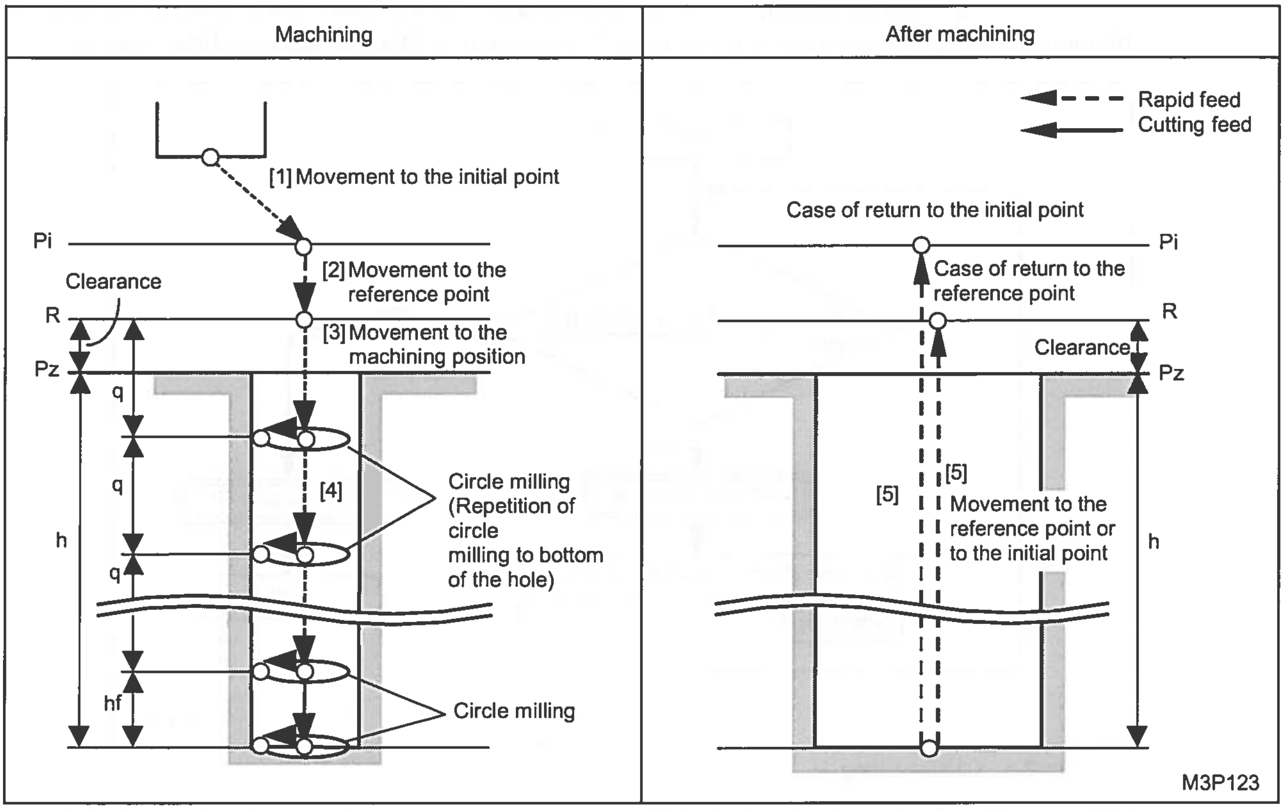
q: Cutting depth on Z per pass to be determined by:

$$\frac{h - hf}{(\text{Whole of } \frac{h - hf}{cmx}) + 1}$$

(cmx = Data **DEPTH** entered in **TOOL FILE** display)

Note: For the circular milling, see Cycle 3 below.

C. Cycle 3



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	**U5**

h: Optimum distance to be automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** in the tool sequence and also the data **ANG** on the **TOOL FILE** display

hf: Bottom finishing allowance to be determined by the data **RGH** entered in the tool sequence and also by the parameter **D21**

q: Cutting depth on Z per pass to be determined by:

$$\frac{h - hf}{(\text{Whole of } \frac{h - hf}{cmx}) + 1}$$

(cmx = Data **DEPTH** entered in **TOOL FILE** display)

Note: The feed speed on the tool paths [3] and [4] is equaled to the parameter **E17**, if bit 0 of parameter **D92** is set at 1.

Circular milling

Circular milling is automatically selected according to the diameter of the machining hole, the diameter of the pre-hole and the cutting depth entered in the tool sequence of the program.

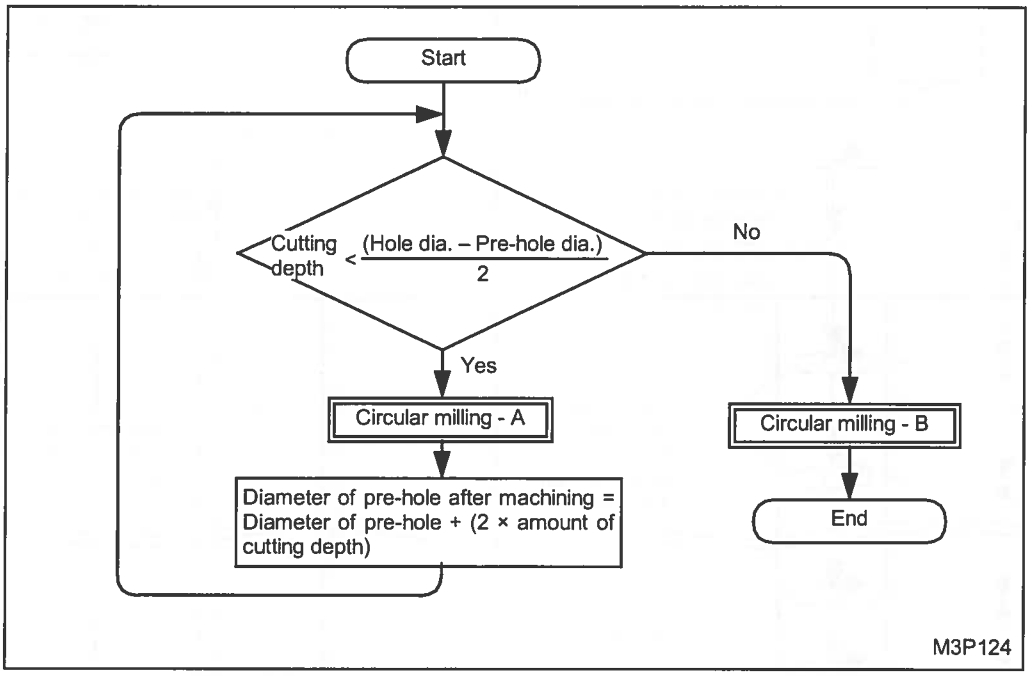


Fig. 3-6 Circular milling

Note: In the Cycle 3, the pre-hole diameter (data entered in tool sequence) is equal to the tool diameter (data entered in the **TOOL DATA** display).

1. Circular milling-A
   The movement of circular milling-A is as shown below.

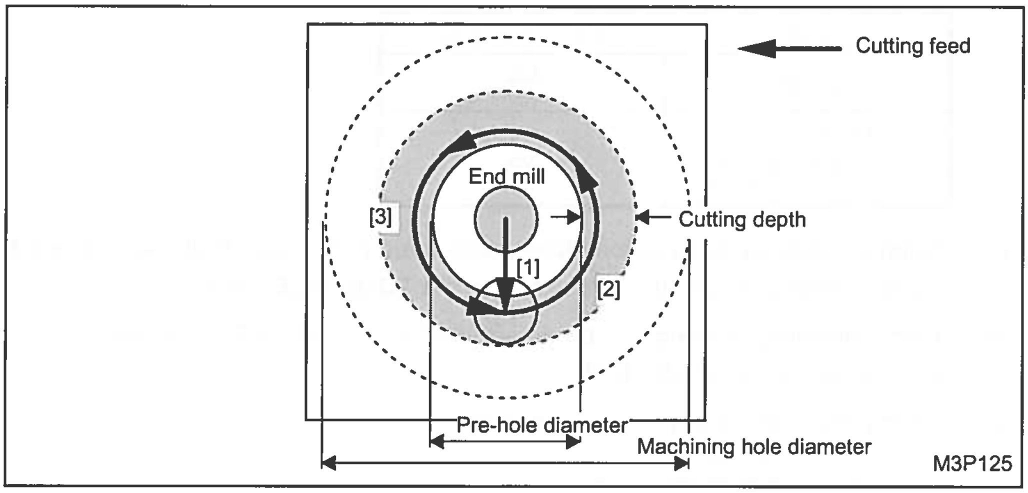


Fig. 3-7 Circular milling-A

- The cutting direction (CW or CCW) can be designated in the program.
- The movement is done in the order [1]→[2]→[3].
- The movement of [1] starts with the end point of the preceding circular milling-A.

2. Circular milling-B

The movement of circular milling-B is as shown below.

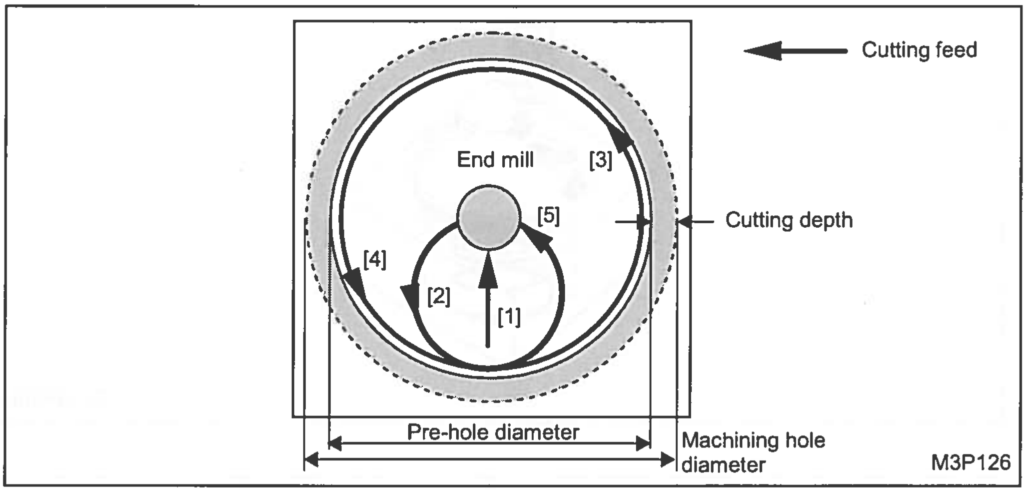


Fig. 3-8 Circular milling-B

- For end milling the cutting direction (CW or CCW) can be designated in the program. For chamfering the cutting direction is counterclockwise.
- The movement is done in the order [1]→[2]→[3]→[4]→[5].
- The movement of [1] starts with the end point of the preceding circular milling-A.

Note: However, when bit 0 (bit 1 in the case of chamfering) of parameter **P10** is 1, the movement of [2] and [5] is done by the following shortened (rapid access) method.

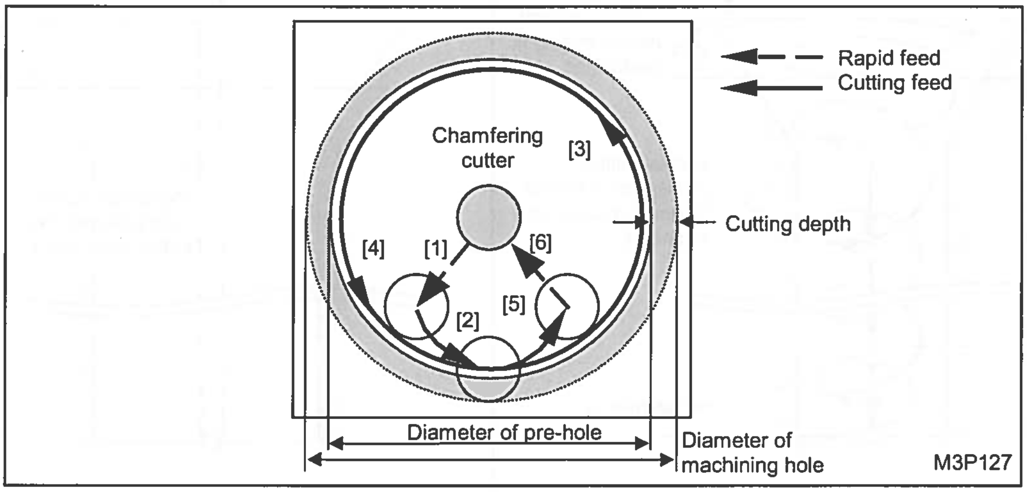
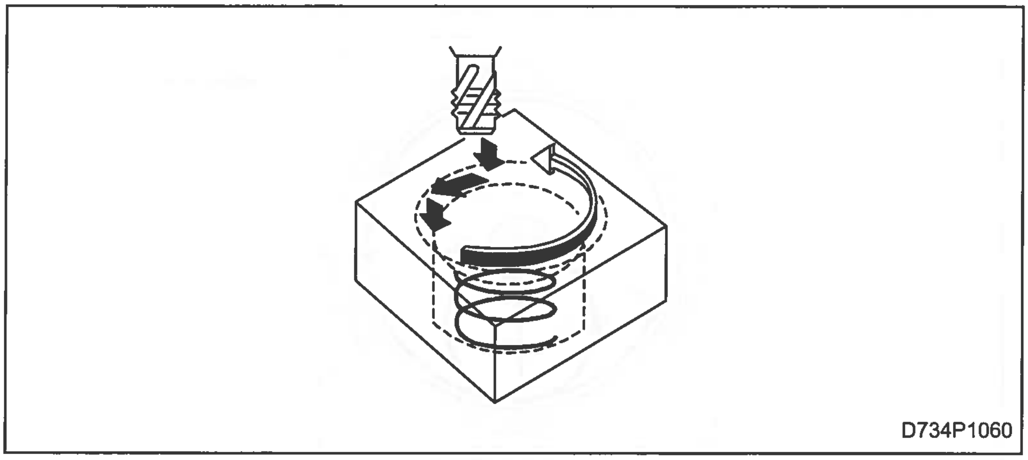
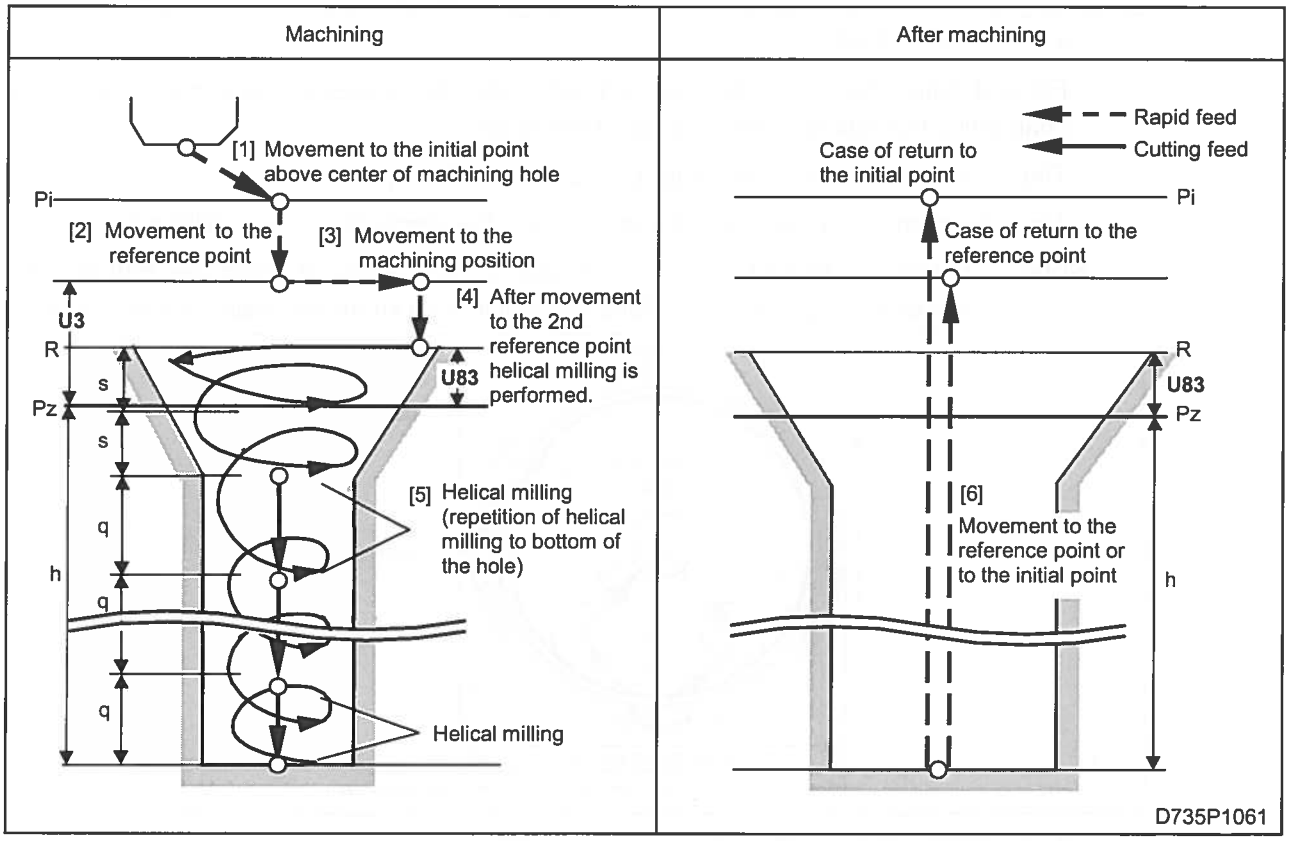


Fig. 3-9 Circular milling-B (case of shortening in chamfering)

- Case of shortening (rapid access) in chamfering is shown above
- The movement is done in the order [1]→[2]→[3]→[4]→[5]→[6].

<In case of tornado milling cycle>





The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coodinate value in the cutting direction

R: Safety clearance above the point Pz (parameter **U83**)

h: Optimum distance to be automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** in the tool sequence and also the data **ANG** on the **TOOL FILE** display

q: Pitch 2 to be entered in the **CIRC MIL** unit.

s: Pitch 1 to be entered in the **CIRC MIL** unit.

Note 1: The cutting direction (CW or CCW) can be designated in the program.

Note 2: The chamfering angle is 45 degrees.

Circular milling

(1) With bottom finishing (1 to 9 is set for **RGH** in the tool sequence)

The operation of the machine when it is programmed to perform bottom finishing operations is shown in Fig. 3-10.

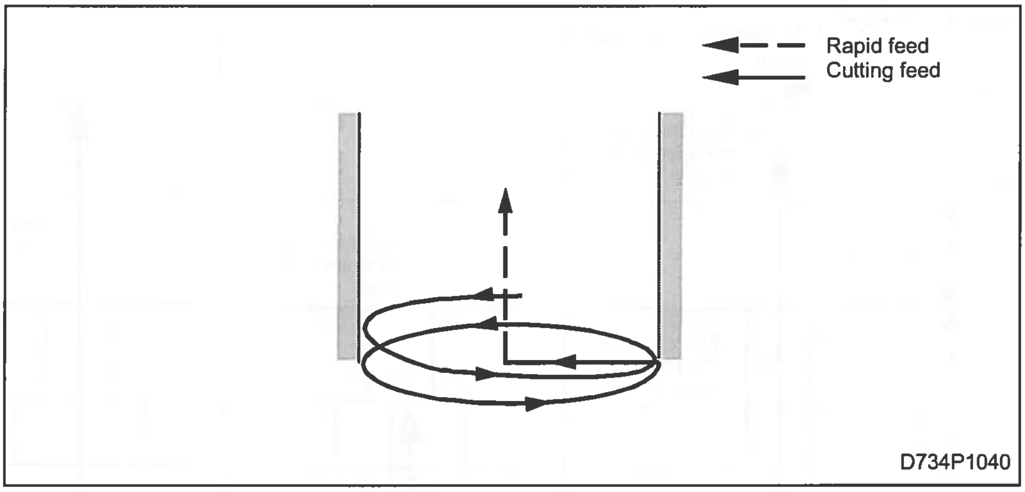


Fig. 3-10 Circular helical processing (with bottom finishing)

After helical interpolation down to the bottom of the hole, one entire circumference of arc interpolation occurs. Next, the tool moves to the center of the hole and then moves in the rapid feed rate to its initial point or to reference point in the axial direction.

(2) Without bottom finishing (0 is set for **RGH** in the tool sequence)

The operation of the machine when it is not programmed to perform bottom finishing operations is shown in the Fig. 3-11.

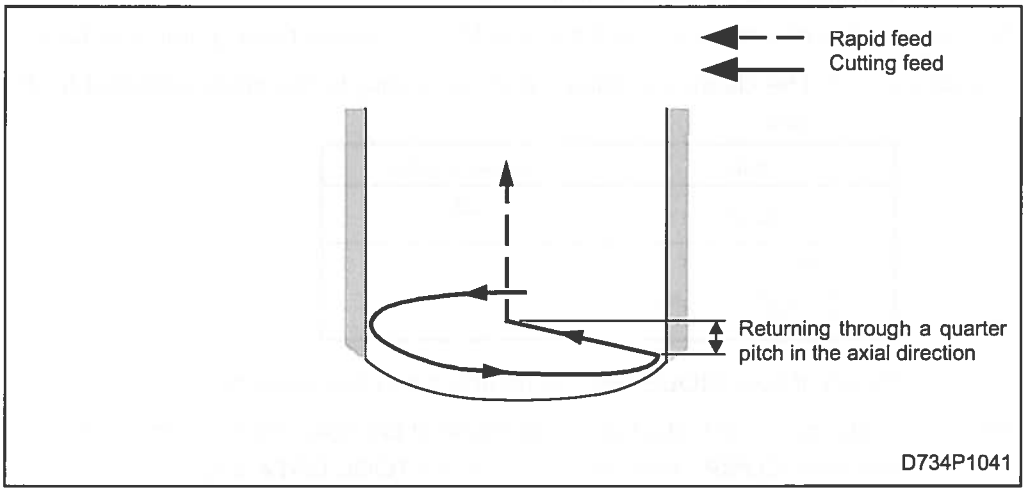
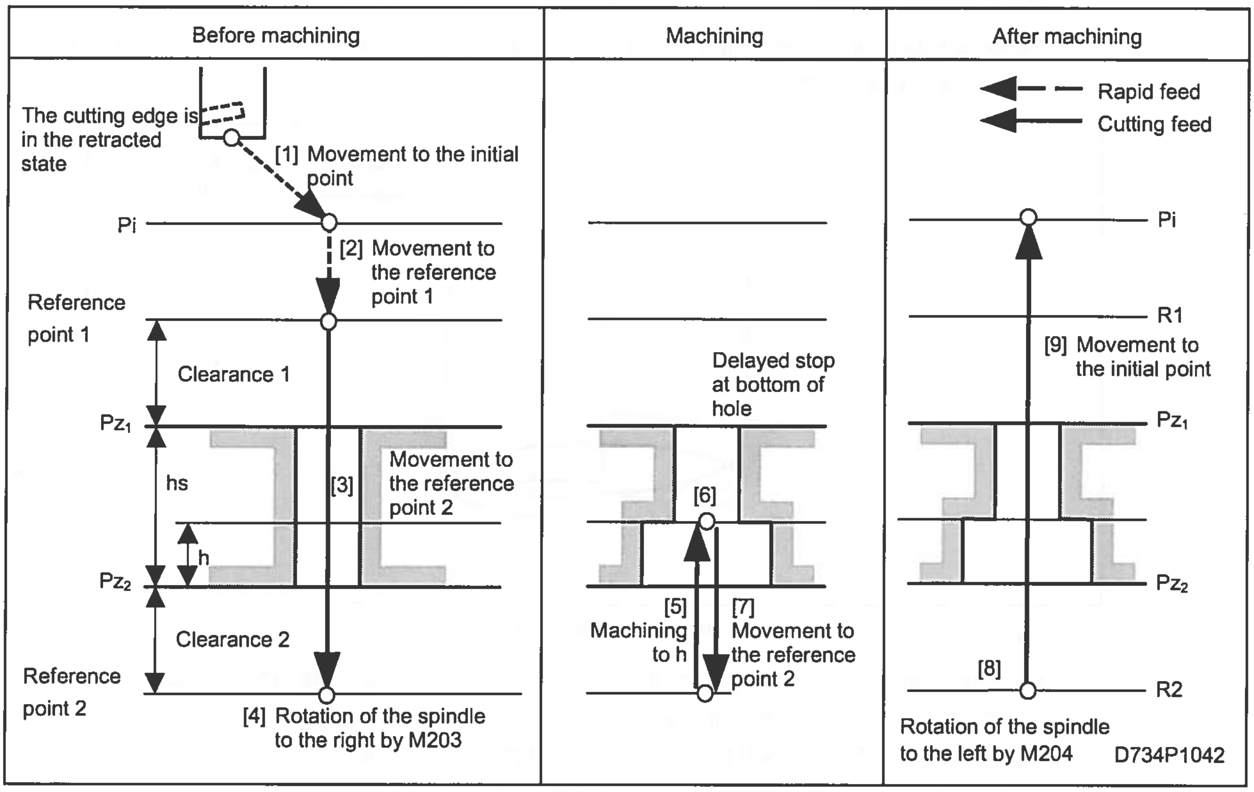


Fig. 3-11 Circular helical processing (without bottom finishing)

After helical interpolation down to the bottom of the hole, the tool moves to the center of the hole by returning through a quarter pitch in the axial direction and then moves in rapid feed rate to its initial point or to reference point in the axial direction.

The bottom of the hole does not undergo arc interpolation.

5. Back facing tool



Pi: Initial point

Pz_1: Coordinate value in the cutting direction

Pz_2: Position at a distance of hs from Pz_1

R1, R2: Safety clearance above the points Pz_1, Pz_2 respectively (parameter **U3**/2 or **U5**)

Clearance 1, 2: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\mathbf{U3}}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

h: Depth of hole (**HOLE-DEP**) to be entered in tool sequence

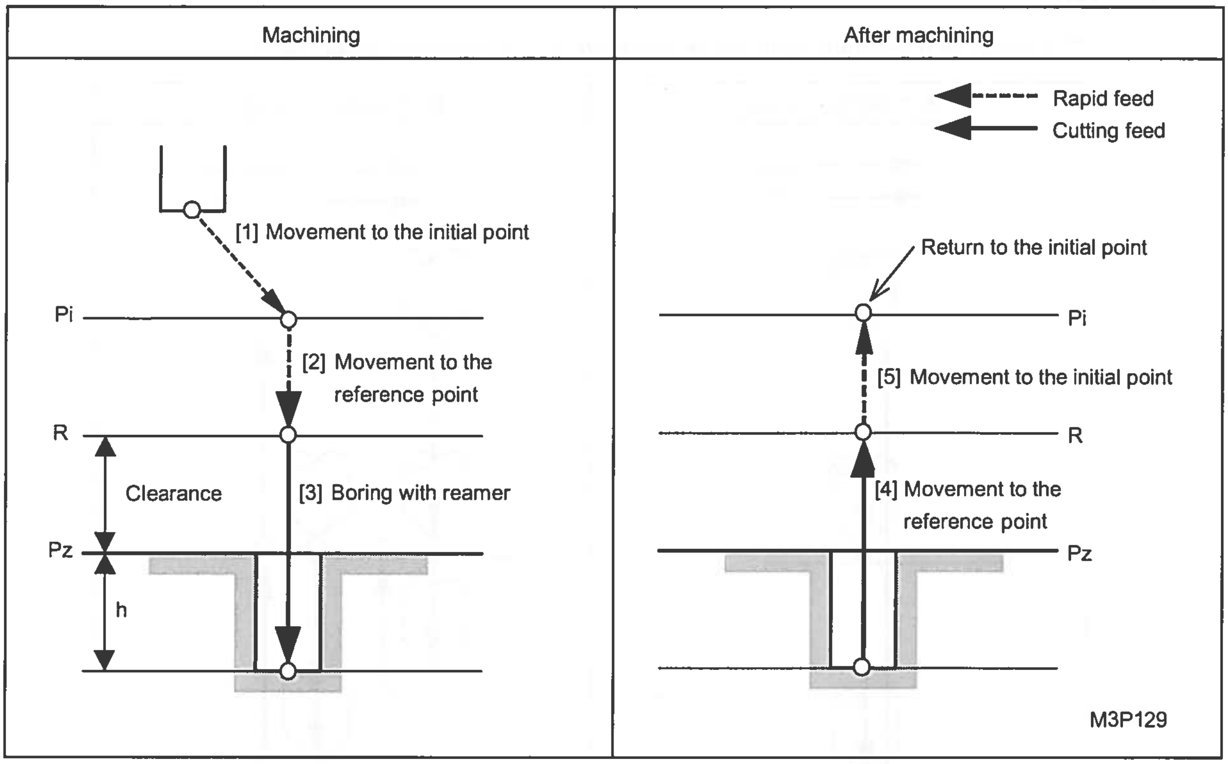
hs: Distance equal to the sum of the depth of pre-hole entered in the tool sequence and the tool data **COMP.** (tool correction) on the **TOOL DATA** display

Note 1: The time of delayed stop of the feed at bottom of hole is set by the parameter **D40**.

Note 2: Feed speed on the tool path [3] and [9] is set by the parameter **D5**.

Note 3: The rotation of the spindle to the right is performed by M203 entered in the tool sequence, whilst the rotation to the left is performed by M204.

6. Reamer



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1**.

However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 2 of parameter **D92** is 1.
- Case where the respective tool sequence contains a chamfering cutter as pre-machining tool.

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **COMP.** (tool correction) on the **TOOL DATA** display

Note: The feed speed of the tool path [4] is determined as follows by the data **DEPTH** in the tool sequence.

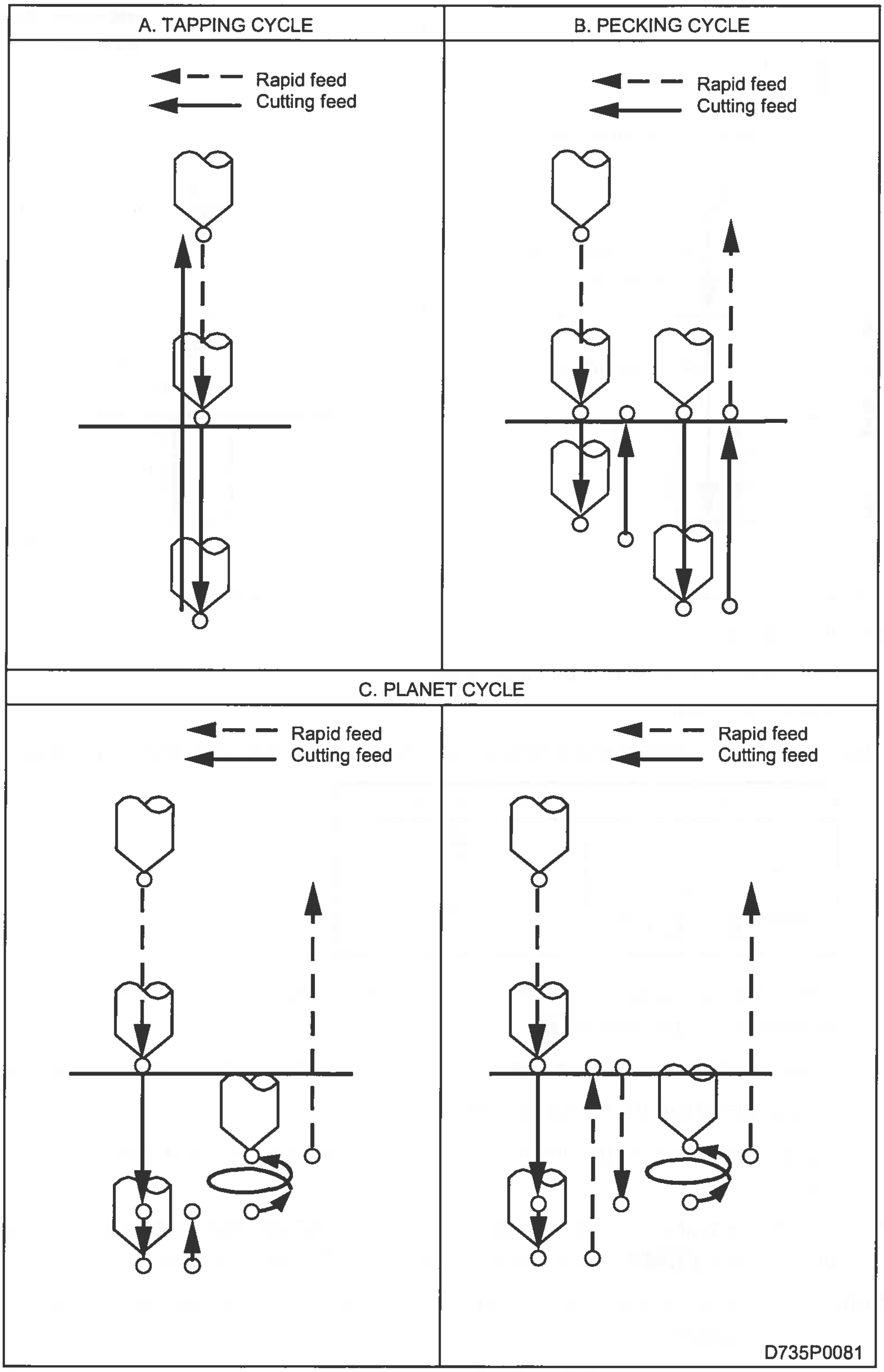
When menu key **[CUT G01]** is pressed .... Parameter **D18**

When menu key **[RAPID G00]** is pressed .... Rapid feed

When the value is entered in the article **DEPTH** .... Value entered (/min)

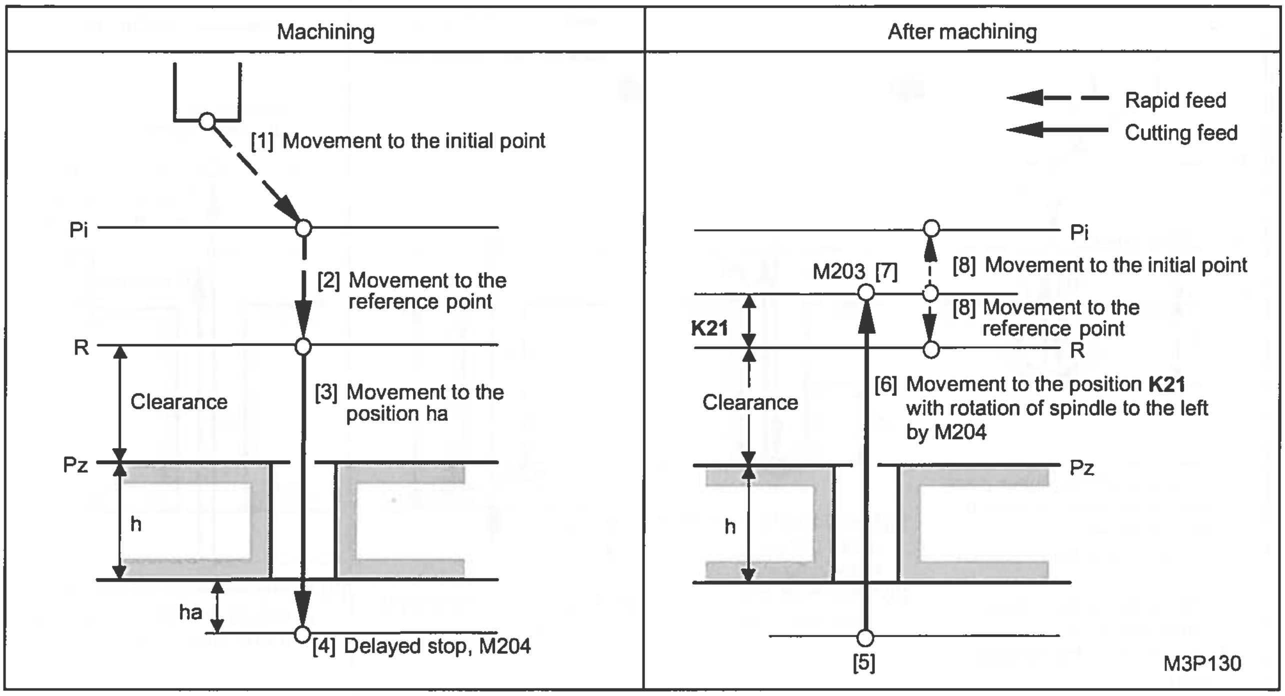
7. Tap

The cycle of machining with tap is available in the following three types.



See Items A to C for the tool paths in each cycle.

A. Tapping cycle



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	**U5**

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the tool correction (**COMP.**) in the **TOOL DATA** display

ha: Distance to be determined by (*A* – **D32**) × *Pt*

A .... **K20** when using metric and unified screws, **D43** when using pipe screws

Pt .... Pitch entered in the machining unit

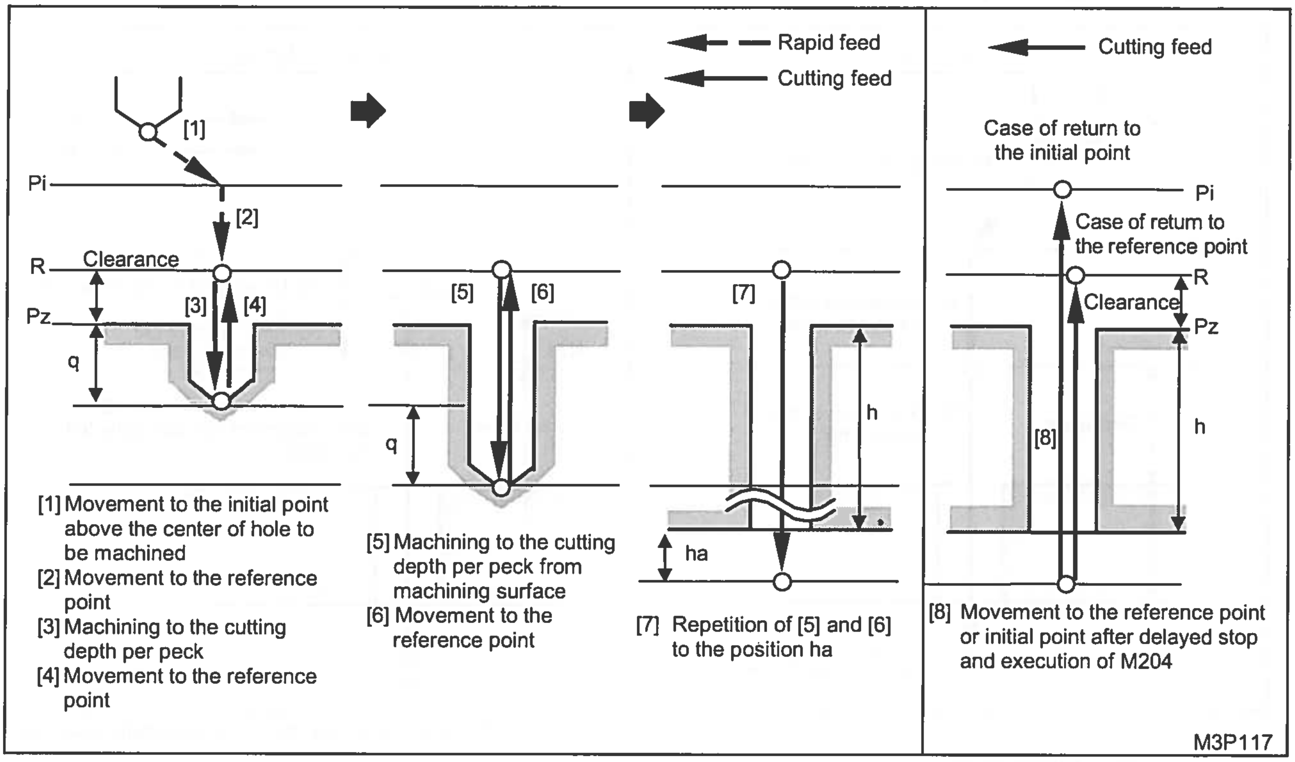
Note: The setting of parameter **K47** causes the following delayed stop for asynchronous tapping.

K47 = 0 .... Delayed stop after execution of M204 at bottom of hole [4]

K47 = 1 .... Delayed stop before execution of M204 at bottom of hole [5]

Moreover, the delayed stop is entered in the article **RGH** in the tool sequence. If **FIX** is selected for **RGH**, it will be determined by the parameter **D22**.

B. Deep hole drilling cycle (PECKING CYCLE)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\textbf{U3}}{2}$
XC, XY, /C, /Y XC, XY, /C, /Y	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1**.

However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 3 of parameter **D92** is 1.
- Case where the respective tool sequence contains a chamfering cutter as pre-machining tool.

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the tool correction (**COMP.**) in the **TOOL DATA** display

ha: Distance to be determined by (*A* – **D32**) × *Pt*

A .... **K20** when using metric and unified screws, **D43** when using pipe screws

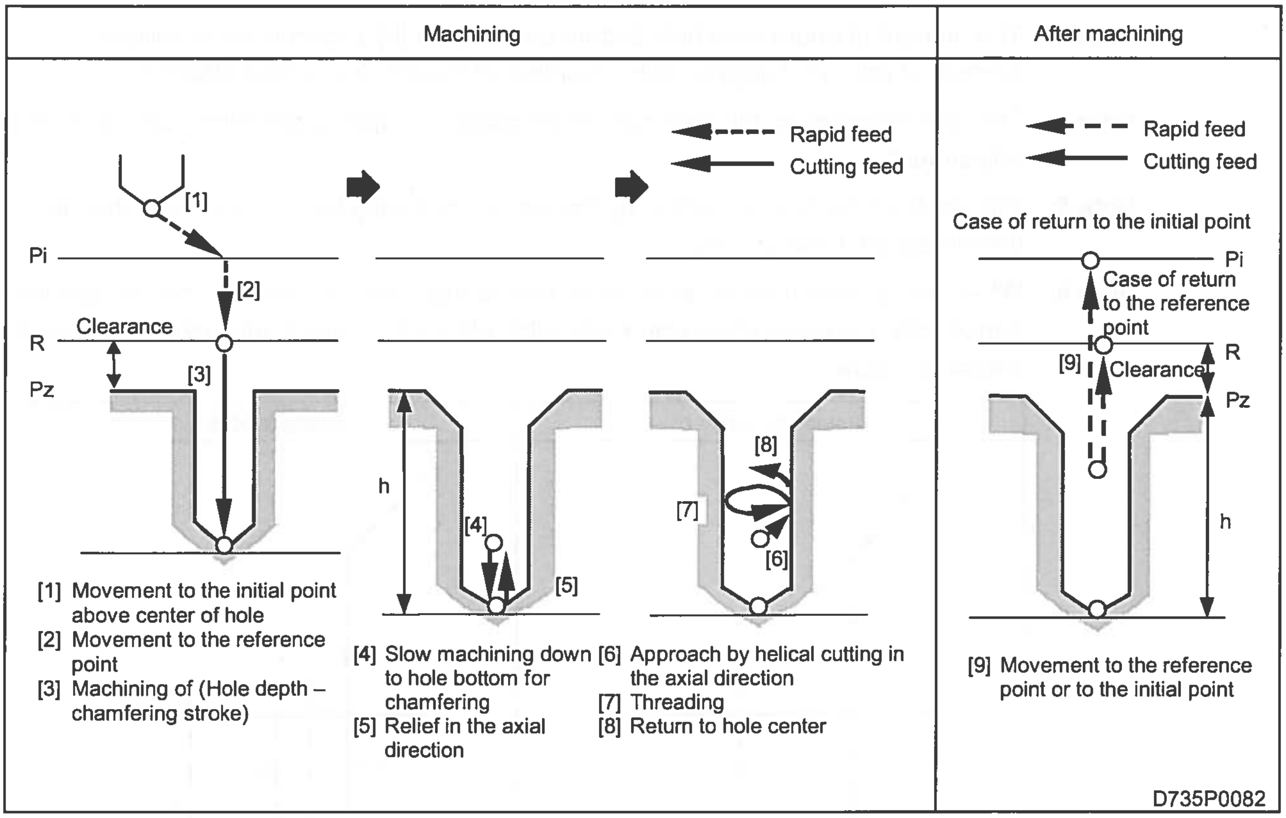
Pt .... Pitch entered in the machining unit

q: Cutting depth in one pass (**PRE-DEP**) to be entered in the tool sequence

Note: The delayed stop is entered in the article **RGH** in the tool sequence. If **FIX** is selected for **RGH**, it will be determined by the parameter **D22**.

C. Planetary tapping (PLANET CYCLE)

The planetary tapping cycle allows three types of machining (pre-hole machining, chamfering, and female threading) with one tool.



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\mathbf{U3}}{2}$
XC, XY, /C, /Y [XC], [XY], [/C], [/Y]	**U5**

When the following two conditions are fulfilled, the clearance before machining will be equaled to the parameter **D1**.

However, clearance after machining is always equaled to the parameter **U3**/2 or **U5**.

- Case where the bit 3 of parameter **D92** is 1.
- Case where the respective tool sequence contains a chamfering cutter as pre-machining tool.

h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence

Note 1: Inversed tapping cannot be executed.

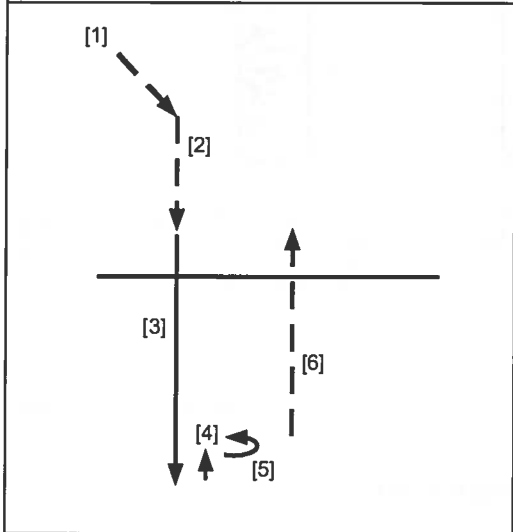
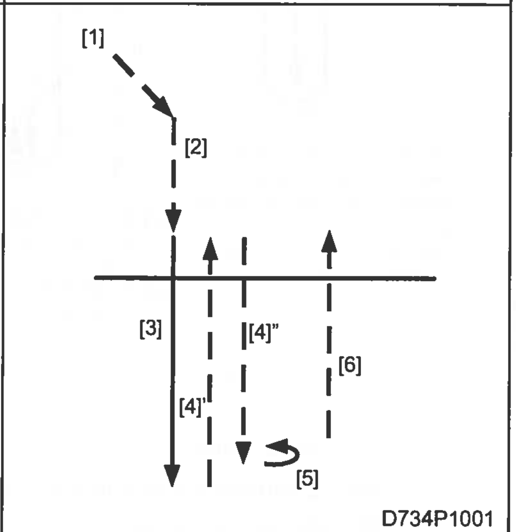
Note 2: The feed rate for chamfering on tool path [4] is calculated as follows:
Chamfering feed = Pre-hole machining feed × Chamfering feed override (parameter **U62**)/100

Note 3: The amount of return from hole bottom on tool path [5] is calculated as follows:
Amount of return = Tapping pitch × Number of threads (parameter **U63**)/10

Note 4: The tool diameter in the tool data is modified for tapping diameter correction (fine-adjustment).

Note 5: The depth of the tapped section by the actual machining becomes smaller than that of the thread set in the program.

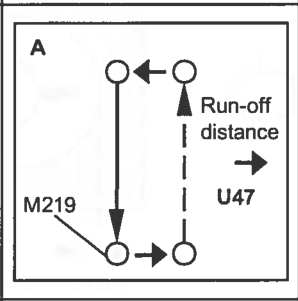
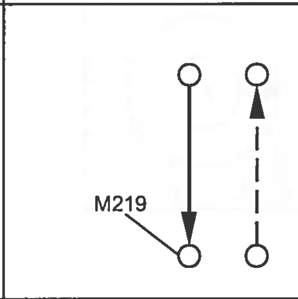
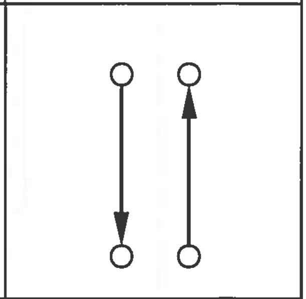
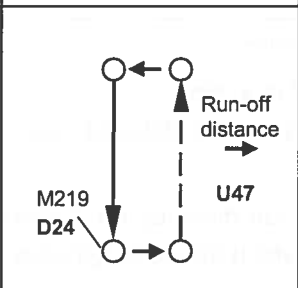
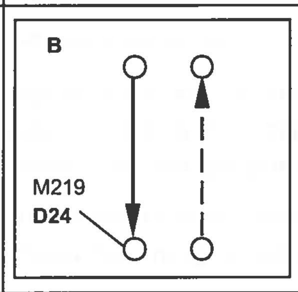
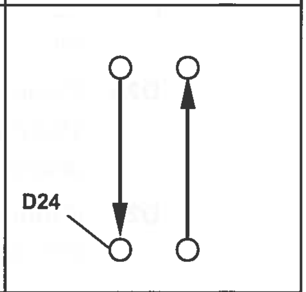
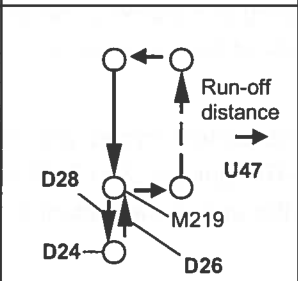
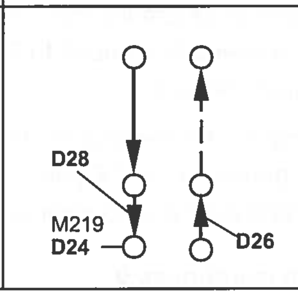
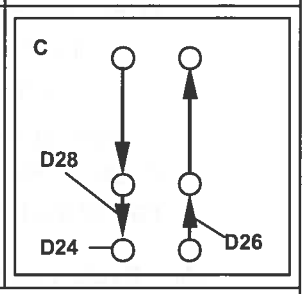
Note 6: When so parameterized, after hole machining, the tool moves out of the hole temporarily to remove chips where one pitch of helical cutting is then executed. See the diagrams below.

Parameter **K75** = 0	Parameter **K75** = 1
	

8. Boring tool

The path of the boring tool is classified in 9 types on the basis of the contents of the program, as shown in the figure below.

Table 3-2 Tool path of the boring tool

	Run-off on Z-axis Yes/No	Delayed stop Yes/No	Cycle 1	Cycle 2	Cycle 3
Roughness 0	No	No			
Roughness 1	No	Yes			
Roughness 2-9	Yes	Yes			
← – – Rapid feed		←—— Cutting feed			M3P131

The bold codes represent the parameter addresses.

In the following, M219, **D24**, **U47**, **D26** and **D28** shown on the figure above, are explained.

M219: M-code to stop the milling spindle in the pre-determined position
(Orientation of milling spindle)

U47: Parameter to determine the run-off distance on the plane. The spindle is oriented at the bottom of the hole. The tool moves to the initial point or to the reference point after clearance of the machining surface. This is used for the finish machining because any damage to the machining surface can be prevented at the time of the return of the tool.

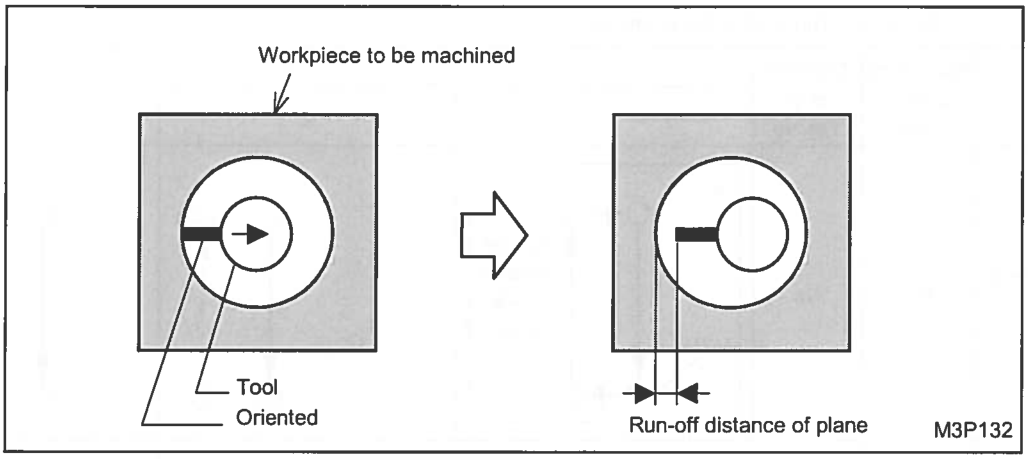


Fig. 3-12 Run-off distance on the X-Y plane

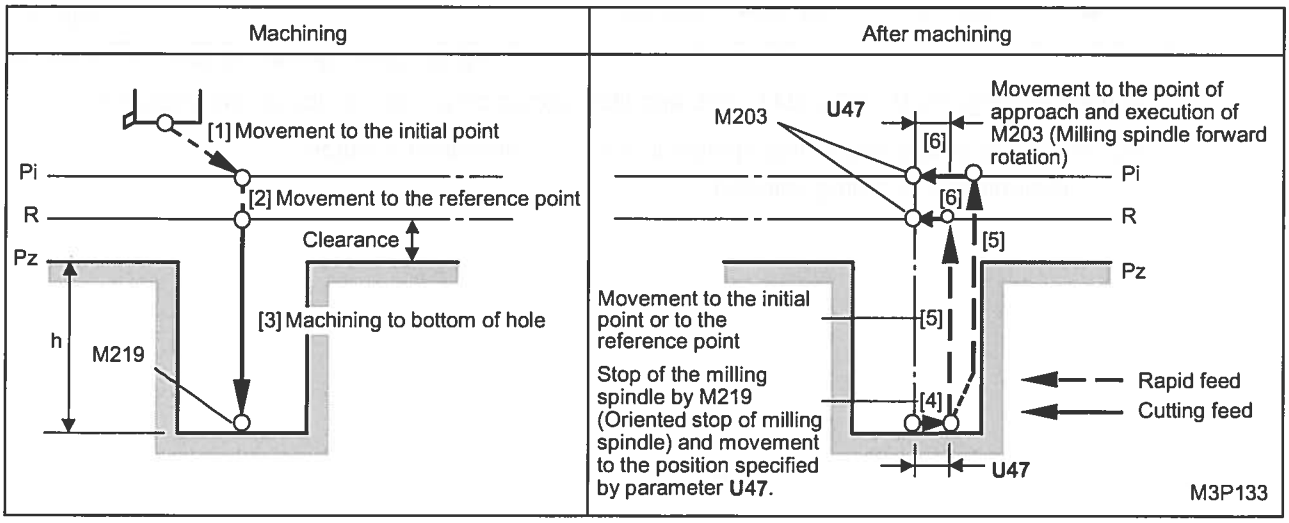
D24: Parameter to determine delayed stop time.
The machining is done in excess of the delayed stop time which serves to improve the precision of the hole machining.

D26: Parameter to determine the run-off distance on the axis. The feed speed is reduced to 70% of the programmed value, which allows to improve the machining precision.

D28: Parameter to determine the finishing allowance at the bottom of the hole.
The feed speed is reduced to 70% of the programmed value, which allows to improve the machining precision.

In order to simplify the description, three pattern cycles are described: Cycle 1—Roughness 0, Cycle 2—Roughness 1, and Cycle 3—Roughness 2 to 9. "General precautions concerning the path of the boring tool" is also given at the end of the description of cycles.

A. Cycle 1 with roughness 0



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

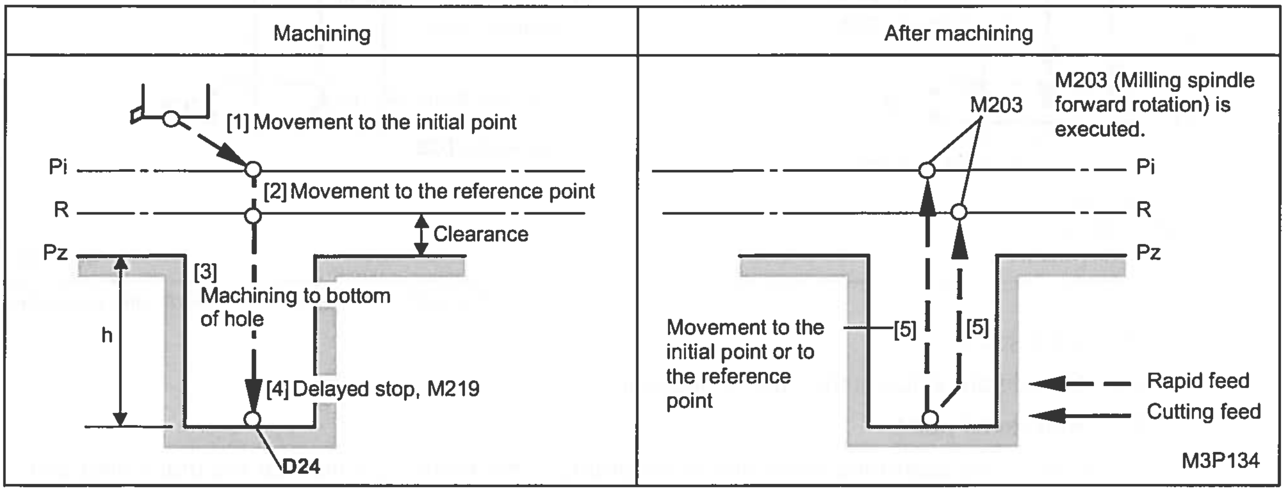
Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	U3 / 2
XC, XY, /C, /Y XC, XY, /C, /Y	U5

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **COMP.** (tool correction) in the **TOOL DATA** display

Note: When M204 is entered in the tool sequence, the spindle rotates in the reverse direction.

B. Cycle 2 with roughness 1



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

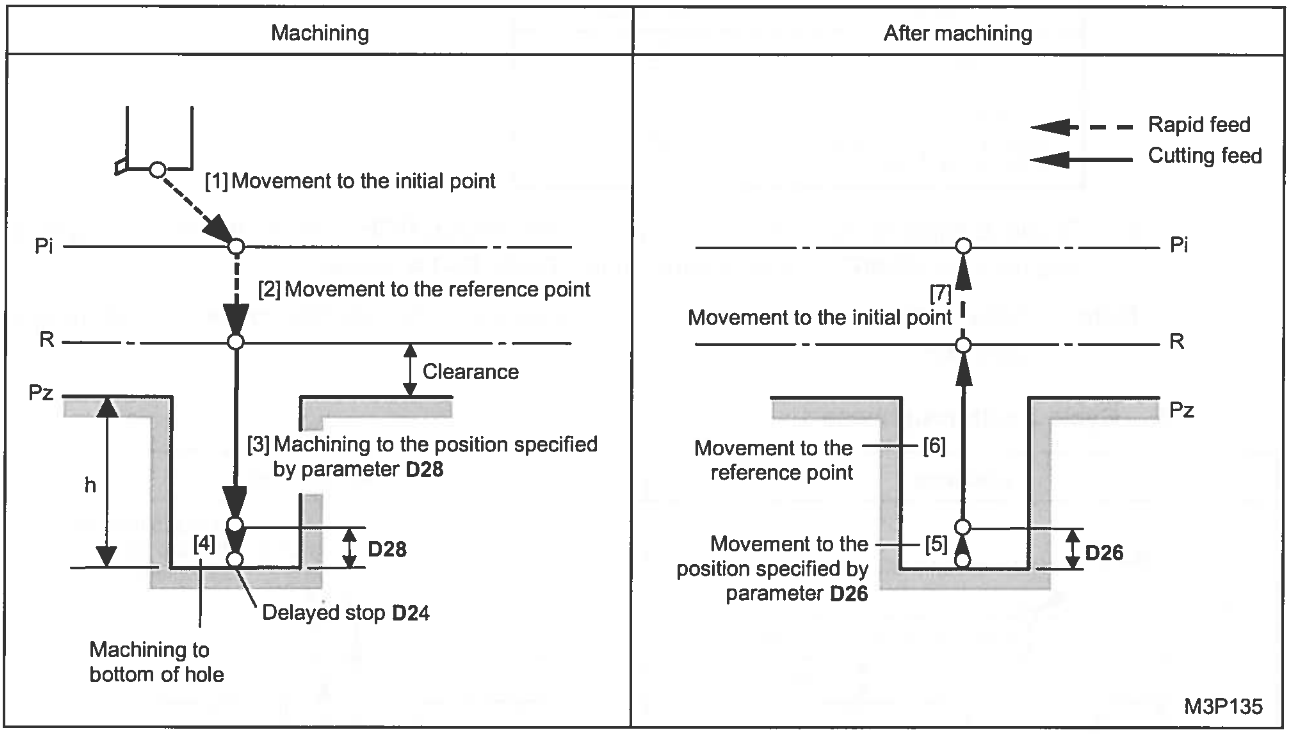
Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	U3 / 2
XC, XY, /C, /Y XC, XY, /C, /Y	U5

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **COMP.** (tool correction) in the **TOOL DATA** display

Note: The delayed stop time of the feed at bottom of hole is set by the parameter **D24**.

C. Cycle 3 with roughness 2 to 9



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y XC, XY, /C, /Y	U5

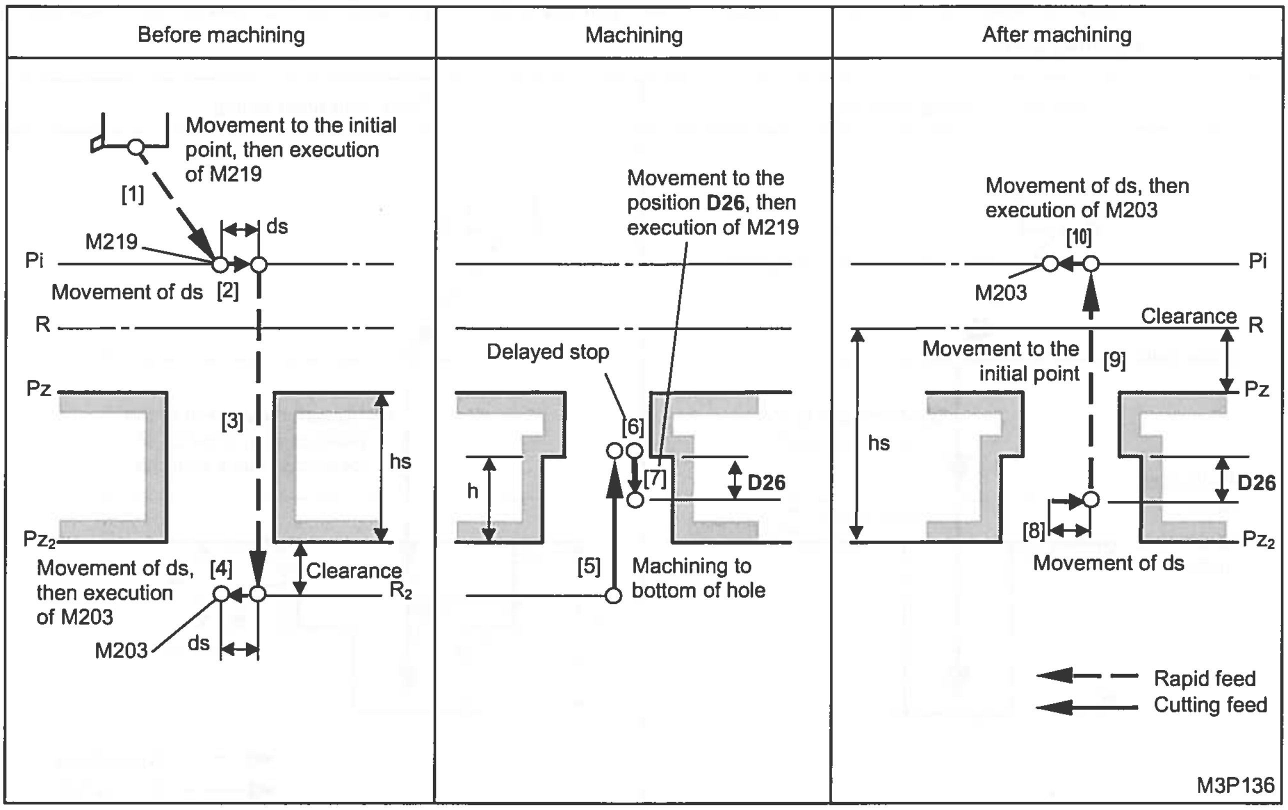
h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **COMP.** (tool correction) in the **TOOL DATA** display

Note 1: The feed speed [4] and [5] is 70% on the programmed value.

Note 2: The feed speed [6] is set by the parameter **D18**.

Note 3: The delayed stop time of the feed at bottom of hole is set by the parameter **D24**.

9. **Back boring tool**



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Coordinate value in the cutting direction

Pz_2: Position at a distance of hs from Pz

R, R_2: Safety clearance above the points Pz, Pz_2 (parameter **U3**/2 or **U5**)

h: Distance equal to the sum of the depth of hole **(HOLE-DEP)** entered in the tool sequence and the data **COMP.** (tool correction) on the **TOOL DATA** display

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{\mathbf{U3}}{2}$
XC, XY, /C, /Y XC, XY, /C, /Y	**U5**

hs: Depth of pre-hole (**PRE-DEP**) to be entered in the tool sequence

ds: Run-off distance on the plane determined by $\frac{d_1 - d_2}{2}$ + **D33**

d_1 .... Diameter of hole (**HOLE-ϕ**) entered in the tool sequence

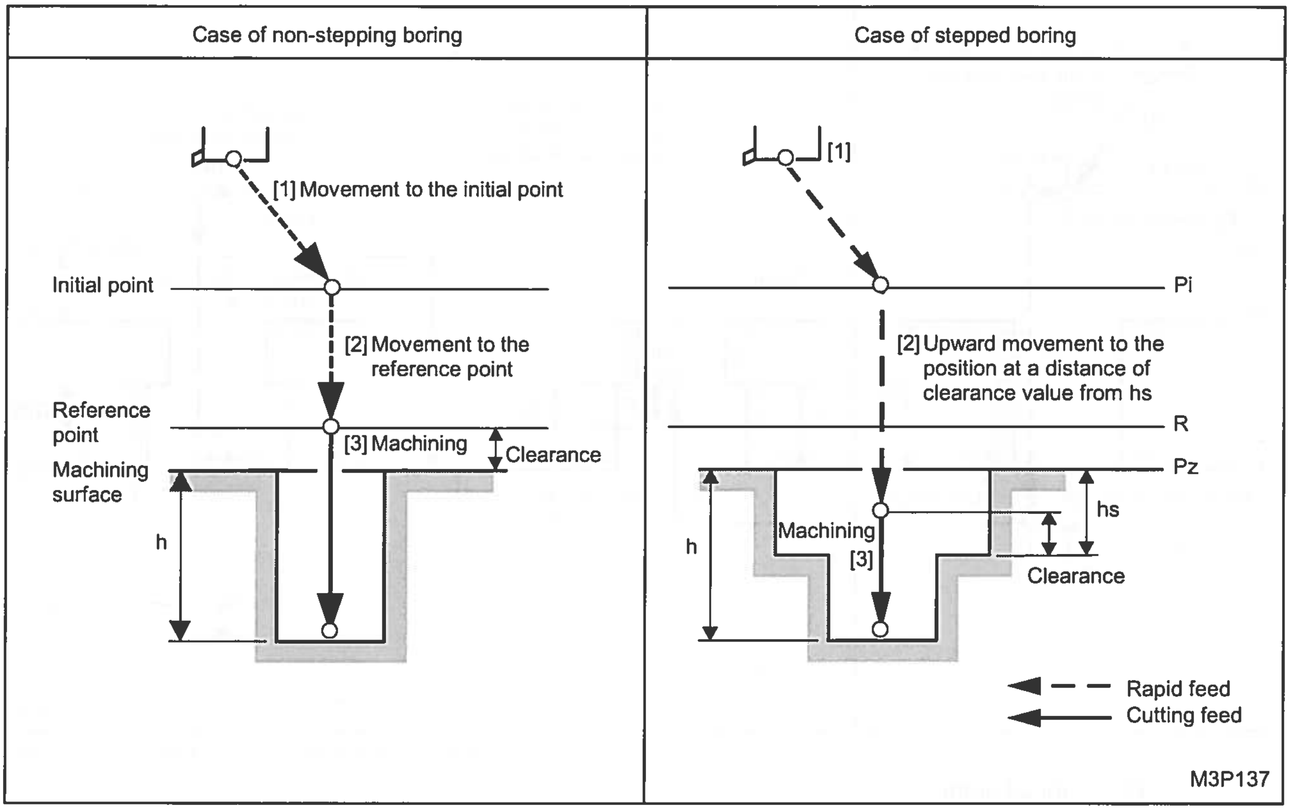
d_2 .... Diameter of pre-hole (**PRE-DIA**) entered in the tool sequence

D33 .... Movement on the plane entered in the parameter

Note: The delayed stop time of the feed is set by the parameter **D40**.

General precautions concerning the path of the boring tool

Stepped hole boring and non-stepped hole boring differ in the path of the tool to the machining starting point.



Pi: Initial point

Pz: Coordinate value in the cutting direction

R: Reference point

Clearance: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	U5

h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **COMP.** (tool correction) in the **TOOL DATA** display

hs: Depth of pre-hole (**PRE-DEP**) to be entered in the tool sequence

Note: Cutting start point is moved from reference point to a distance specified in hs (depth of pre-hole).

3-4-8 Setting the shape sequence data of the point machining unit

The machining unit and tool sequence data has been set above. Next, set the shape sequence data for the point machining unit.

1. Types of point maching shape

Five types of point machining patterns are provided, and the shape that can be selected differs according to the mode specified for the unit.

O: Selection possible, ×: Selection impossible

Shape	Mode									
	ZC	XC	$\boxed{\text{XC}}$	/C	$\boxed{\text{/C}}$	ZY	XY	$\boxed{\text{XY}}$	/Y	$\boxed{\text{/Y}}$
PT	O	O	O	O	O	O	O	O	O	O
ARC	O	O	O	O	O	O	O	O	O	O
LIN	×	×	×	×	×	O	O	O	O	O
SQR	×	×	×	×	×	O	O	O	O	O
GRD	×	×	×	×	×	O	O	O	O	O

Refer to the section 3-3-1 “Planes to be machined and machining methods“ for the detail of the modes.

2. Entry of shape sequence data

A. When the selected mode in the unit is ZC

1. Point (**PT**)

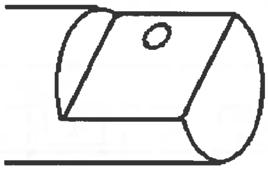
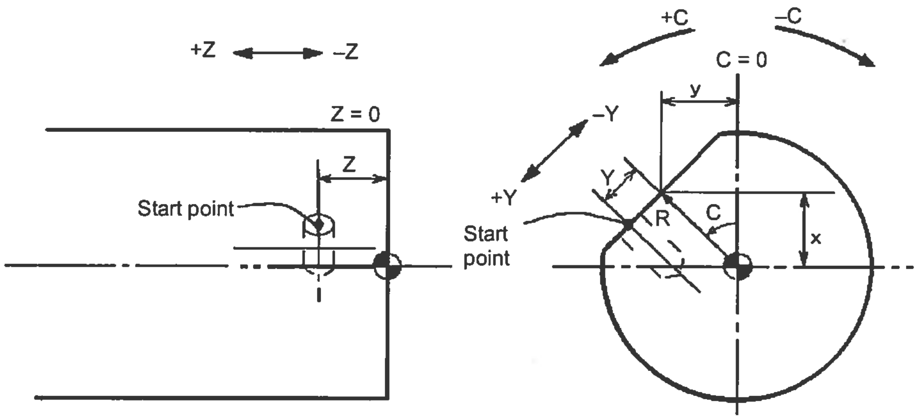
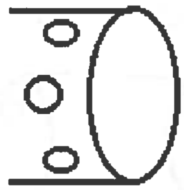


FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	SPT-Y	NUM.	ANGLE	Q	R
1	PT	[1]	[2]	[3]	[4]	◆	◆	◆	[5]

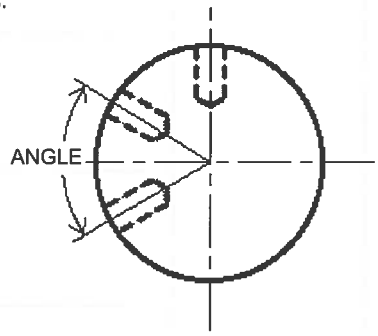
Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z [4] SPT-Y	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. 
[5] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)



D734P0015

FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	SPT-Y	NUM.	ANGLE	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z [4] SPT-Y	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. (See "1 Point (PT)" for further details.)
[5] NUM.	Specify the number of holes to be drilled.
[6] ANGLE	Specify the angle between two adjacent holes. 
[7] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[8] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

B. When the selected mode in the unit is XC, $\boxed{XC}$, /C or $\boxed{/C}$

1. Point (PT)

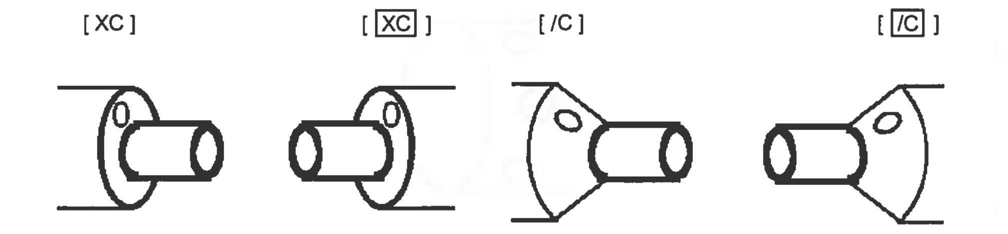
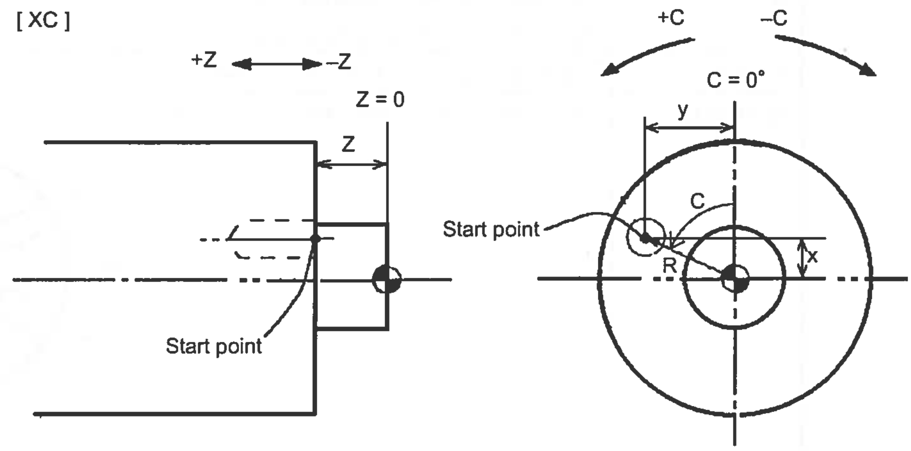
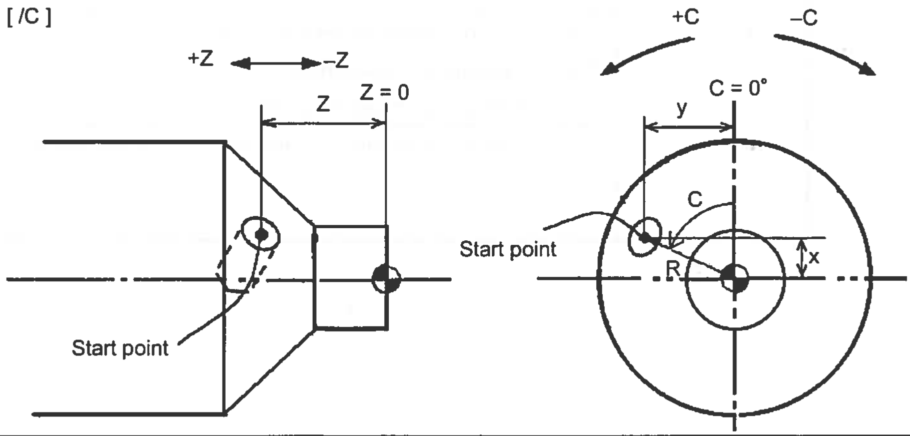


FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANGLE	Q	R
1	PT	[1]	[2]	[3]	◆	◆	◆	[4]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data.  
[4] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)

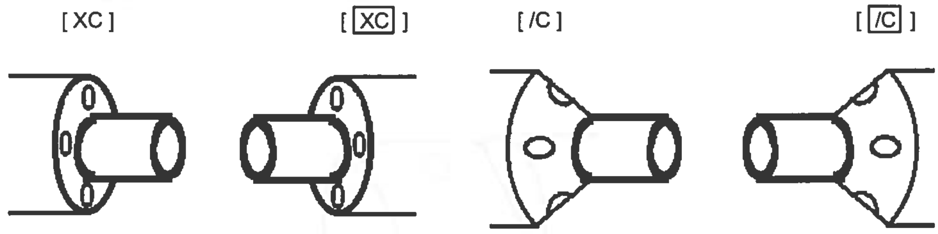
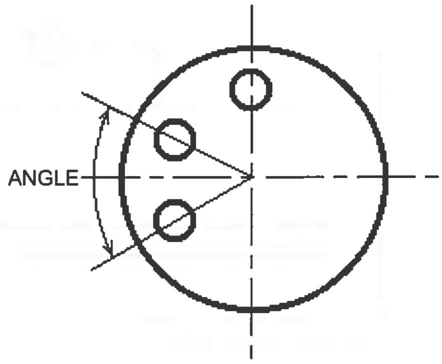
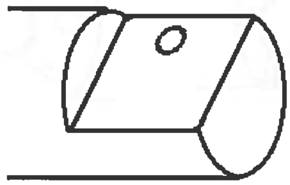


FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANGLE	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	[6]	[7]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. (See "1 Point (PT)" for further details.)
[4] NUM.	Specify the number of holes to be drilled.
[5] ANGLE	Specify the angle between two adjacent holes. 
[6] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[7] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

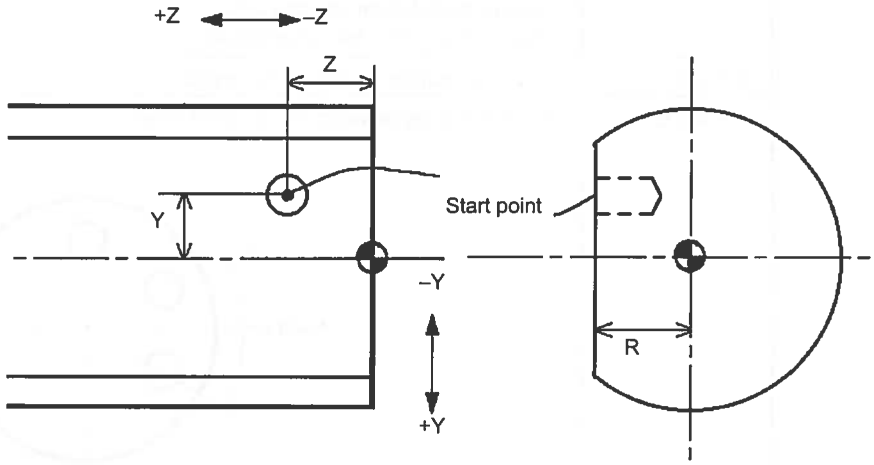
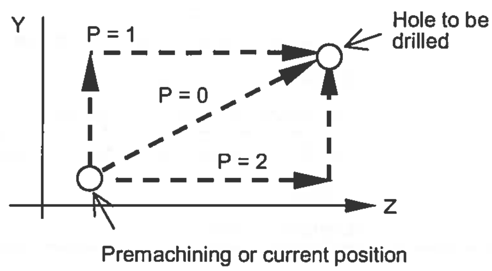
C. When the selected mode in the unit is ZY

1. Point (**PT**)

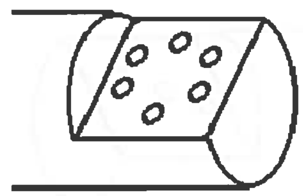


D734P0014

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
1	PT	[1]	[2]	[3]	◆	◆	◆	◆	◆	◆	[4]	◆	[5]

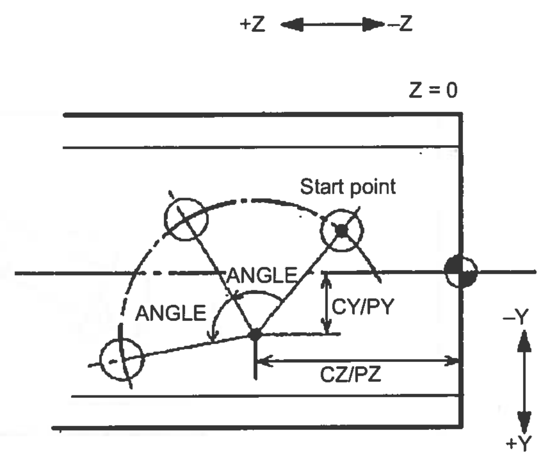
Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be drilled. 
[4] P	Specify the tool path. 
[5] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (ARC)

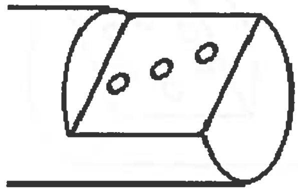


D734P0018

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	◆	[6]	◆	[7]	◆	[8]	[9]

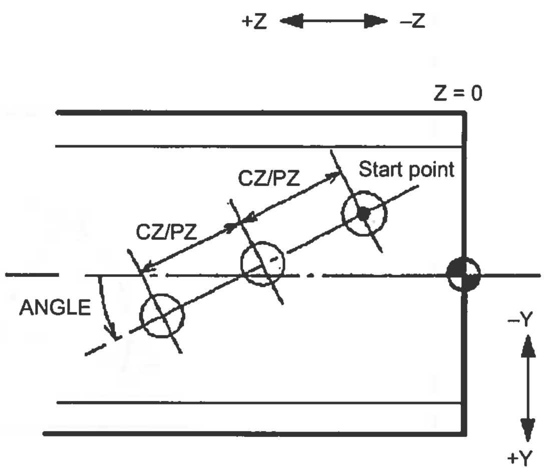
Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[4] CZ/PZ [5] CY/PY	Specify the coordinate of the center of the arc. 
[6] M	Specify the number of holes to be drilled.
[7] ANGLE	Specify the angle between two adjacent holes. (See the figure of item [4] CZ/ PZ, [5] CY/ PY.)
[8] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[9] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

3. Line (**LIN**)



D734P0019

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
1	LIN	[1]	[2]	[3]	[4]	◆	◆	[5]	◆	[6]	◆	[7]	[8]

Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[4] CZ/PZ	Specify the pitch between two adjacent holes in the line of holes. 
[5] M	Specify the number of holes to be drilled.
[6] ANGLE	Specify the angle formed by the line of holes and the Z-axis. (See the figure of the item [4] CZ/PZ.)
[7] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[8] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

4. Square (**SQR**)

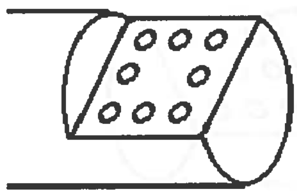
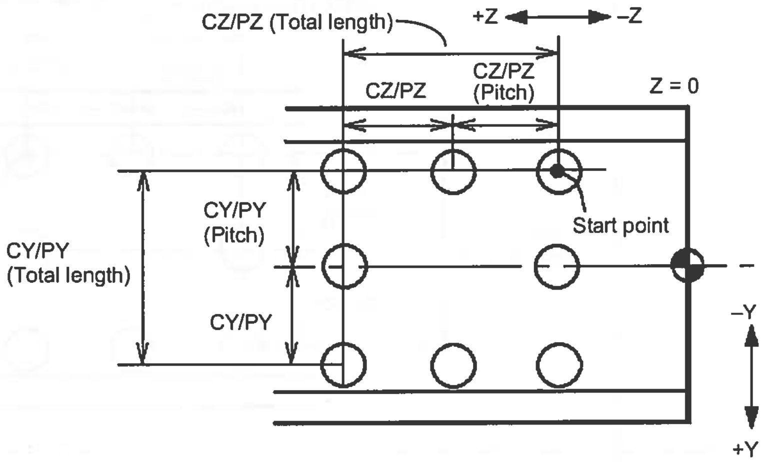
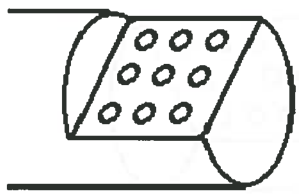

D734P0020

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	◆	[9]	[10]	[11]

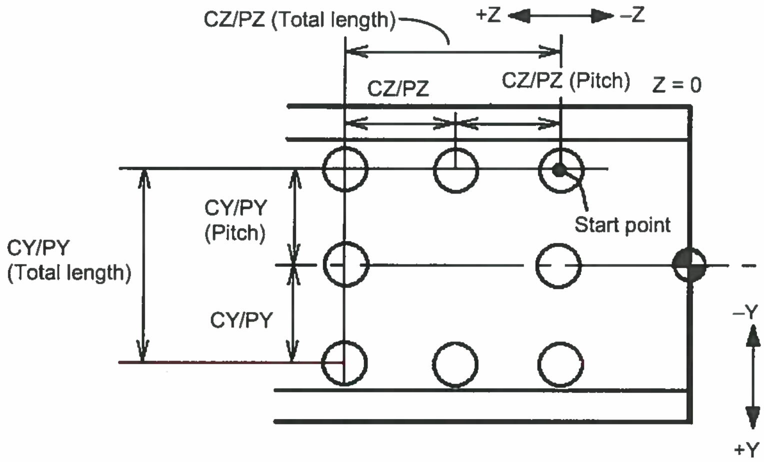
Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[4] CZ/PZ	Specify the pitch between holes or the total length of the Z-axis. 
[5] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [4] CZ/ PZ.)
[6] F	Specify whether the data entered in **CZ/PZ** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[7] M	Specify the number of holes on the line of holes of the Z-axis.
[8] N	Specify the number of holes on the line of holes of the Y-axis.
[9] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[10] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[11] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

5. Grid (**GRD**)



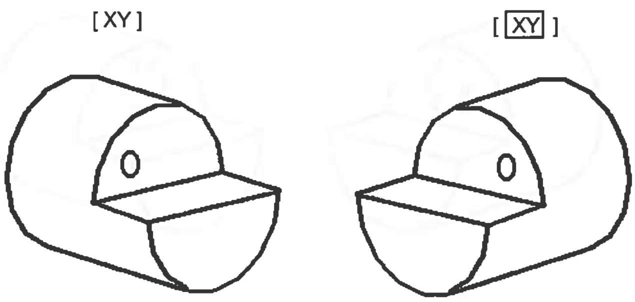
D734P0021

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
1	GRD	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	◆	[9]	[10]	[11]

Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[4] CZ/PZ	Specify the pitch between holes or the total length of the Z-axis. 
[5] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [4] CZ/ PZ.)
[6] F	Specify whether the data entered in **CZ/PZ** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[7] M	Specify the number of holes on the line of holes of the Z-axis.
[8] N	Specify the number of holes on the line of holes of the Y-axis.
[9] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[10] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[11] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

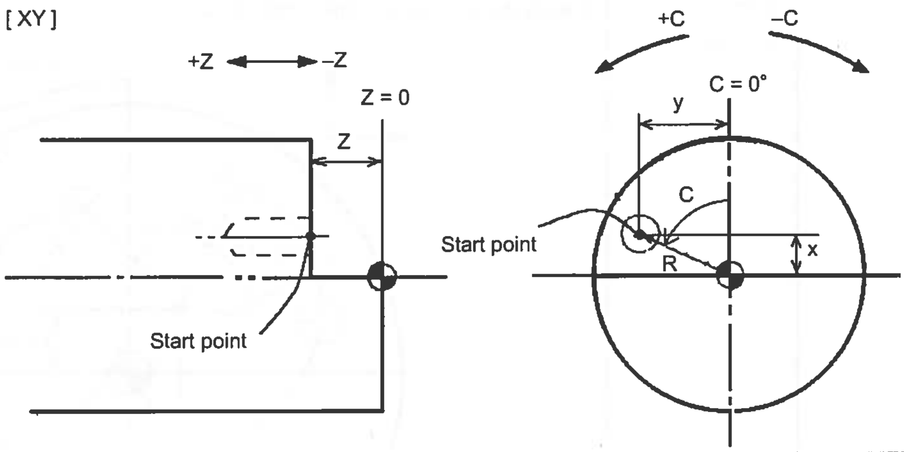
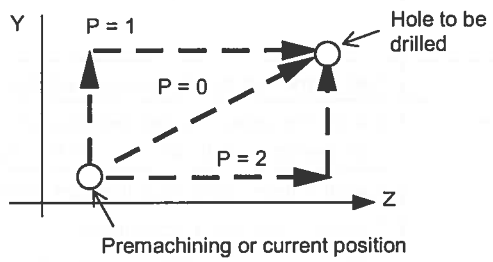
D. When the selected mode in the unit is XY or XY

1. Point (PT)



D734P0022

FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	PT	[1]	[2]	[3]	◆	◆	◆	◆	◆	◆	[4]	◆	[5]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. 
[4] P	Specify the tool path. 
[5] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)

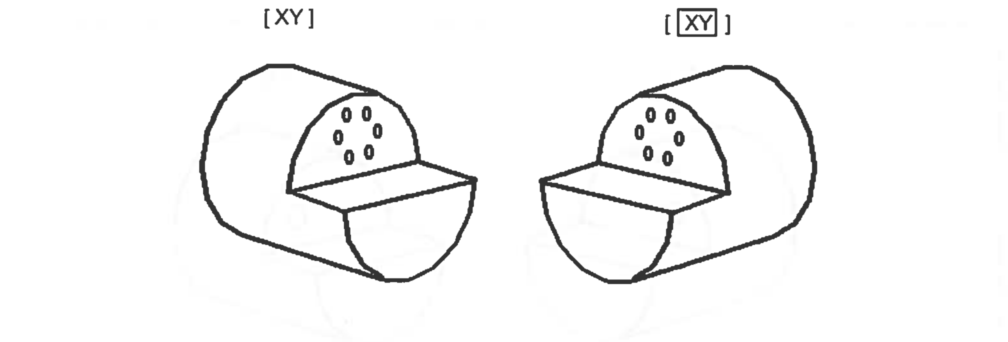
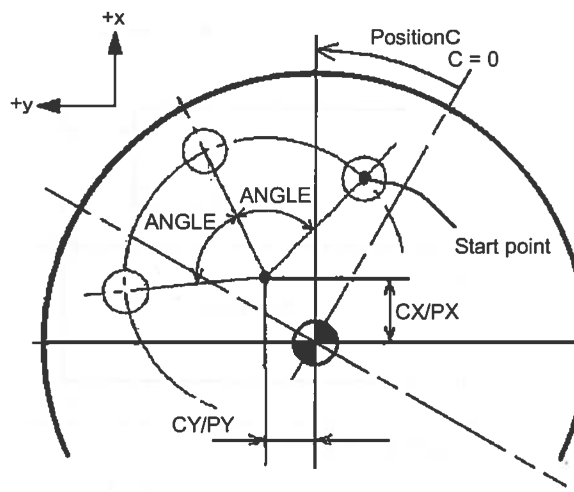
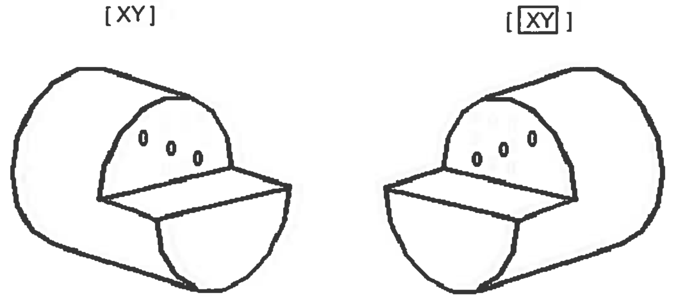


FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	◆	[6]	◆	[7]	◆	[8]	[9]

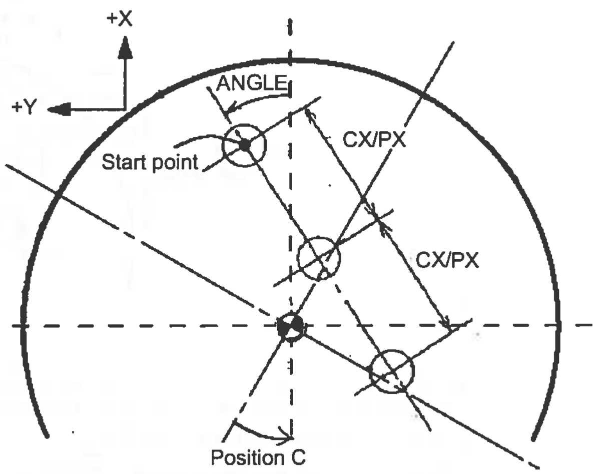
<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>[1] SPT-R/x
[2] SPT-C/y
[3] SPT-Z</td><td>Specify the start point of the hole to be machined.
- To set the start point in R-C coordinates, enter the radius and the angle as they are.
- To set the start point in x-y coordinates, change the [x-y INPUT] menu item to the reverse display mode before entering data.
(See “1 Point (PT)” for further details.)</td></tr>
<tr><td>[4] CX/PX
[5] CY/PY</td><td>Specify the coordinate of the center of the arc.

</td></tr>
<tr><td>[6] M</td><td>Specify the number of holes to be drilled.</td></tr>
<tr><td>[7] ANGLE</td><td>Specify the angle between two adjacent holes.
(See the figure of the item [4] CX/ PX, [5] CY/ PY.)</td></tr>
<tr><td>[8] Q</td><td>Specify if the machining at the start point is executed or not.
0: Actual execution of machining
1: Only positioning without machining</td></tr>
<tr><td>[9] R</td><td>Specify the position to which the tool returns after machining.
0: Initial point
1: Reference point</td></tr>
</table>

3. Line (**LIN**)



D734P0024

FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	LIN	[1]	[2]	[3]	[4]	◆	◆	[5]	◆	[6]	◆	[7]	[8]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. (See "1 Point (PT)" for further details.)
[4] CX/PX	Specify the pitch between two adjacent holes in the line of holes. 
[5] M	Specify the number of holes to be drilled.
[6] ANGLE	Specify the angle formed by the line of holes and the Z-axis. (See the figure of the item [4] CX/ PX.)
[7] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[8] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

4. Square (**SQR**)

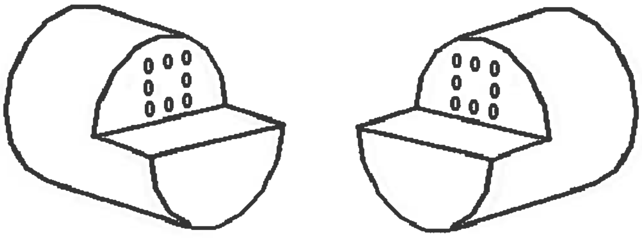
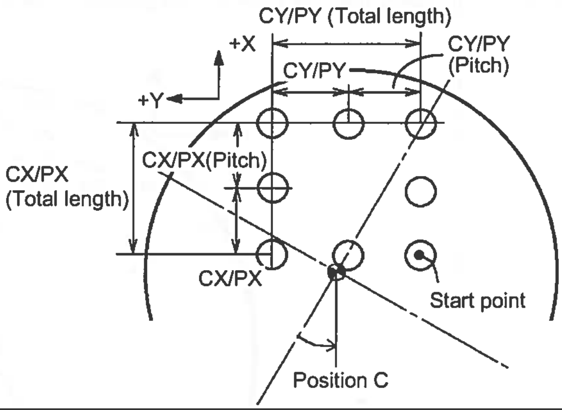
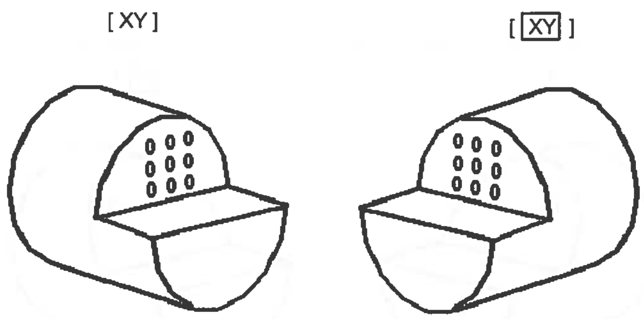


FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	◆	[9]	[10]	[11]

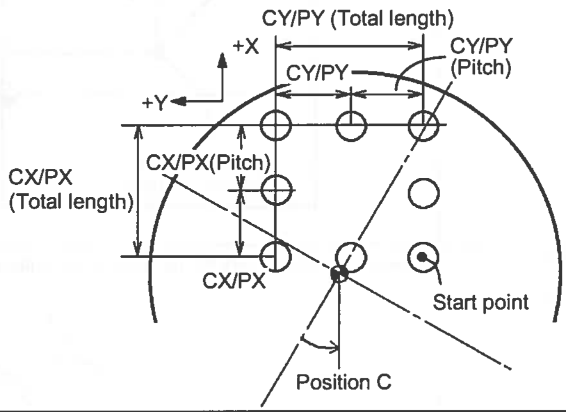
Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. (See "1 Point (PT)" for further details.)
[4] CX/PX	Specify the pitch between holes or the total length of the X-axis. 
[5] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [4] CX/PX.)
[6] F	Specify whether the data entered in **CX/PX** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[7] M	Specify the number of holes on the line of holes of the X-axis.
[8] N	Specify the number of holes on the line of holes of the Y-axis.
[9] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[10] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[11] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

5. Grid (**GRD**)



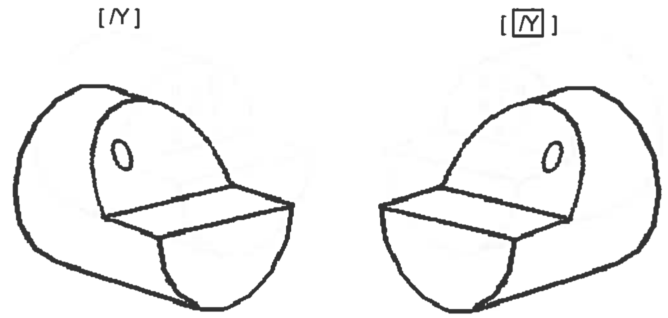
D734P0026

FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	GRD	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	◆	[9]	[10]	[11]

Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. (See "1 Point (PT)" for further details.)
[4] CX/PX	Specify the pitch between holes or the total length of the X-axis. 
[5] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [4] CX/PX.)
[6] F	Specify whether the data entered in **CX/PX** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[7] M	Specify the number of holes on the line of holes of the X-axis.
[8] N	Specify the number of holes on the line of holes of the Y-axis.
[9] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[10] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[11] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

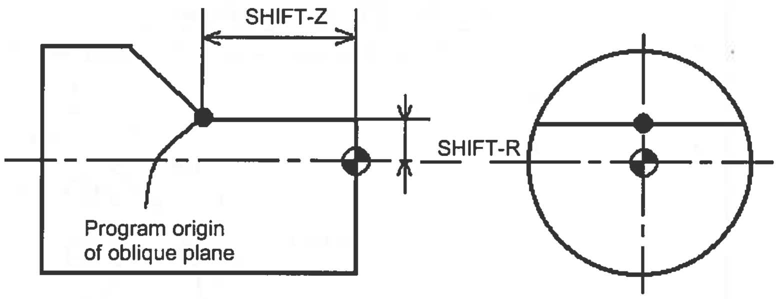
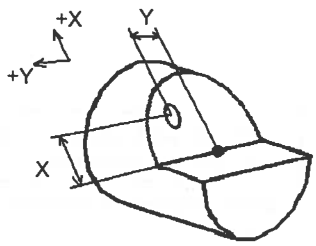
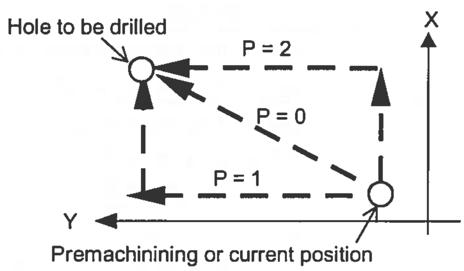
E. When the selected mode in the unit is /Y or $\boxed{/Y}$

1. Point (PT)



D734P0027

FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	PT	[1]	[2]	[3]	[4]	◆	◆	◆	◆	◆	◆	[5]	◆	[6]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). 
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be drilled. 
[5] P	Specify the tool path. 
[6] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)

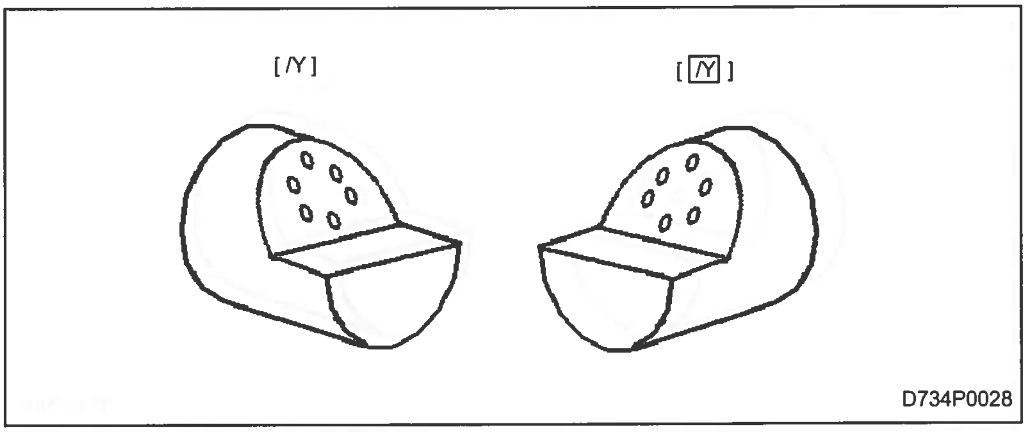
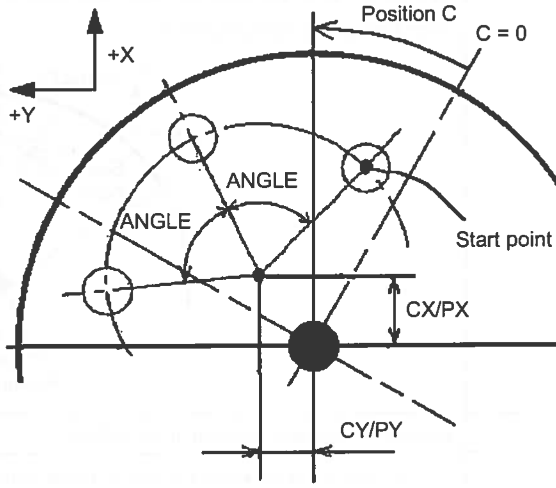
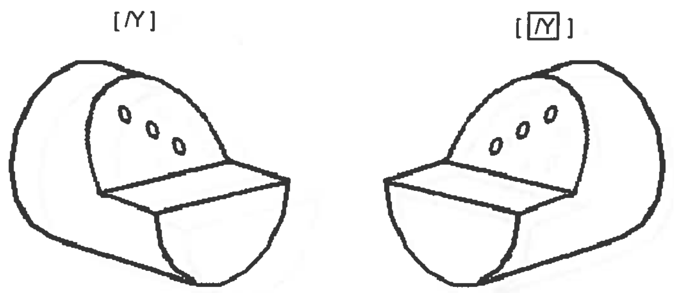


FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	[6]	◆	[7]	◆	[8]	◆	[9]	[10]

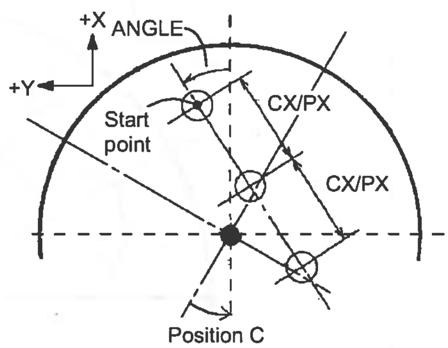
Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See "1 Point (PT)" for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[5] CX/PX [6] CY/PY	Specify the central coordinates of the arc. 
[7] M	Specify the number of holes to be drilled.
[8] ANGLE	Specify the angle between two adjacent holes. (See the figure of the item [5] CX/ PX, [6] CY/ PY.)
[9] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[10] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

3. Line (**LIN**)



D734P0029

FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	LIN	[1]	[2]	[3]	[4]	[5]	◆	◆	[6]	◆	[7]	◆	[8]	[9]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See “1 Point (PT)” for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See “1 Point (PT)” for further details.)
[5] CX/PX	Specify the pitch between two adjacent holes in the line of hole. [Top view of the oblique plane] 
[6] M	Specify the number of holes to be drilled.
[7] ANGLE	Specify the angle between the line of holes and the X-axis on the oblique plane. (See the figure of the item [5] CX/PX.)
[8] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[9] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

4. Square (**SQR**)

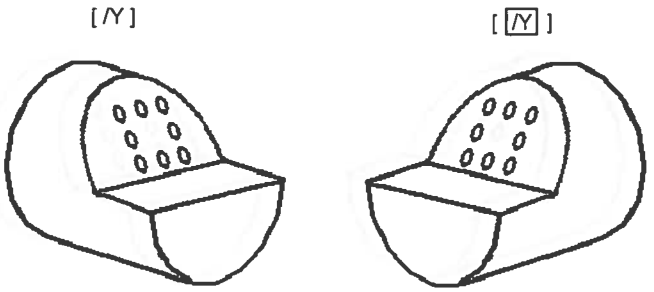
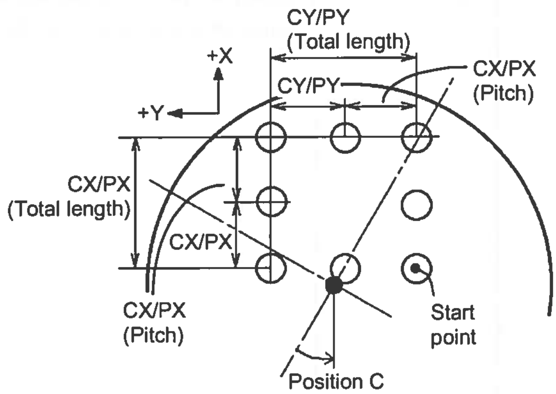


FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	◆	[10]	[11]	[12]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See "1 Point (PT)" for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[5] CX/PX	Specify the pitch between holes or the total length of the X-axis. [Top view of the oblique plane] 
[6] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [5] CX/PX.)
[7] F	Specify whether the data entered in **CX/PX** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[8] M	Specify the number of holes on the line of holes of the X-axis.
[9] N	Specify the number of holes on the line of holes of the Y-axis.
[10] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[11] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[12] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

5. Grid (**GRD**)

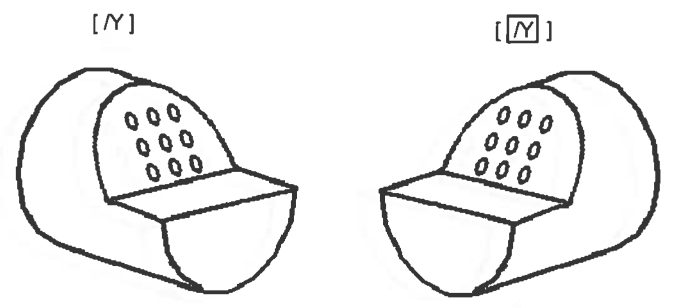
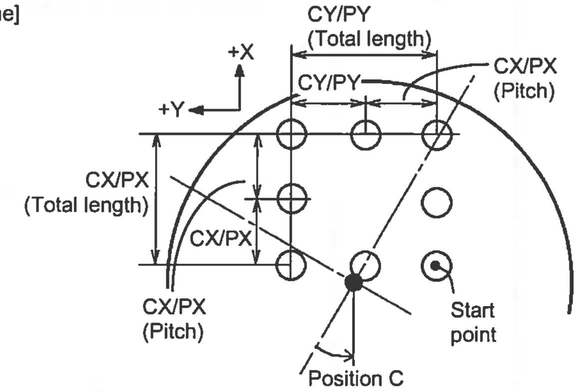


FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
1	GRD	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	◆	[10]	[11]	[12]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See "1 Point (PT)" for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See "1 Point (PT)" for further details.)
[5] CX/PX	Specify the pitch between holes or the total length of the X-axis. [Top view of the oblique plane] 
[6] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (See the figure of the item [5] CX/PX.)
[7] F	Specify whether the data entered in **CX/PX** and **CY/PY** concern the pitch or the total length. 0: Pitch 1: Total length
[8] M	Specify the number of holes on the line of holes of the X-axis.
[9] N	Specify the number of holes on the line of holes of the Y-axis.
[10] P	Specify if the machining at the four corners is executed or not. 0: Machining at the four corners 1: No machining at the four corners
[11] Q	Specify if the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
[12] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

3-5 Line Machining Units

Line machining units are used to enter a contour machining method and the data relating to a form to be machined. The unit includes two sequences: One is the tool sequence, subject to which data are entered in relation to the operational details of tool and the other the shape sequence, subject to which data are entered in relation to the machining dimensions specified on drawing.

3-5-1 Types of line machining units

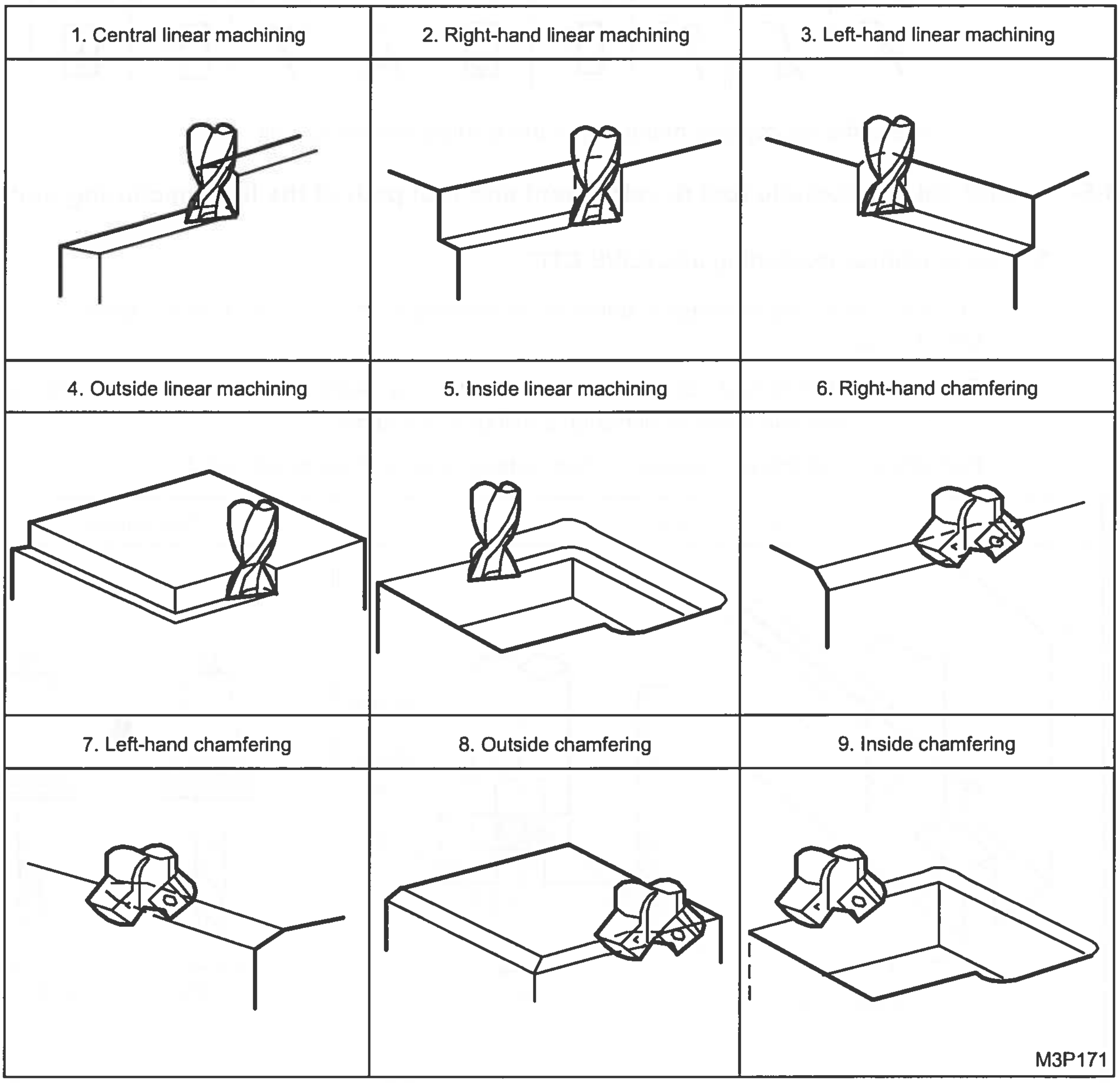


Fig. 3-13 Types of line machining units

3-5-2 Procedure for selecting line machining unit

(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Pressing the **[LINE MACH-ING]** menu key displays the following line machining unit menu.

LINE CTR	LINE RGT	LINE LFT	LINE OUT	LINE IN	CHMF RGT	CHMF LFT	CHMF OUT	CHMF IN	

(3) Press the appropriate menu key of the desired machining unit.

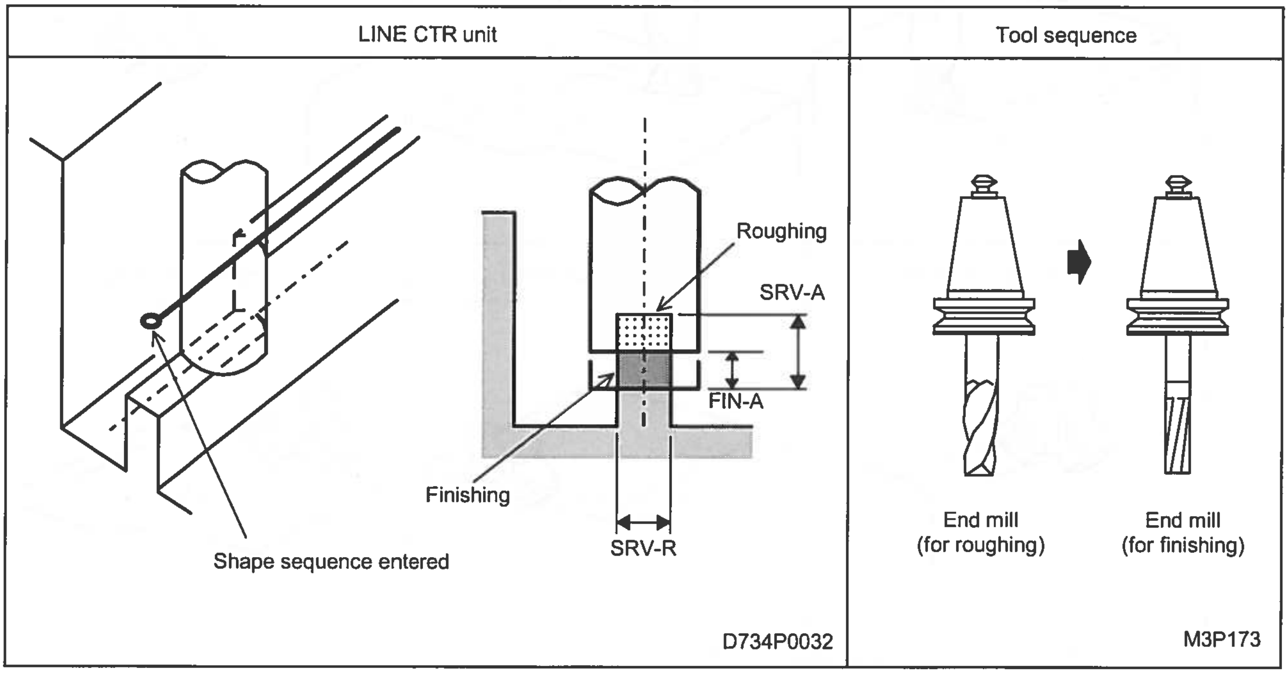
3-5-3 Unit data, automatic tool development and tool path of the line machining unit

1. Central linear machining unit (LINE CTR)

This unit should be selected to carry out machining so that the tool has its center move on the line of a form.

Remark 1: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to either face mill or ball end mill.

Remark 2: For the tool sequence data setting, refer to Subsection 3-5-4.

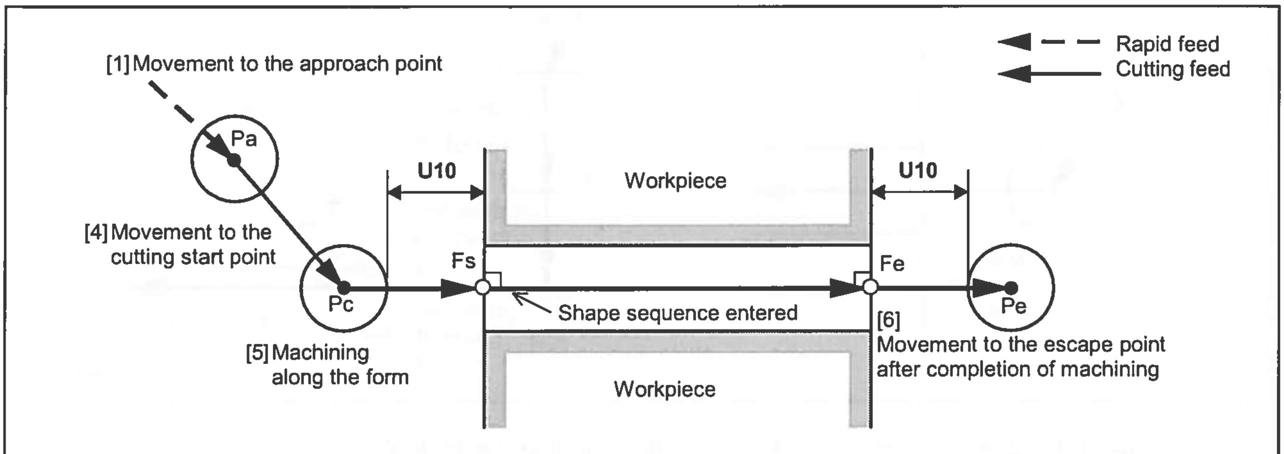


RGH: A roughness code should be selected out of the menu.

FIN-A: An axial finishing allowance is automatically entered once a roughness code has been selected.

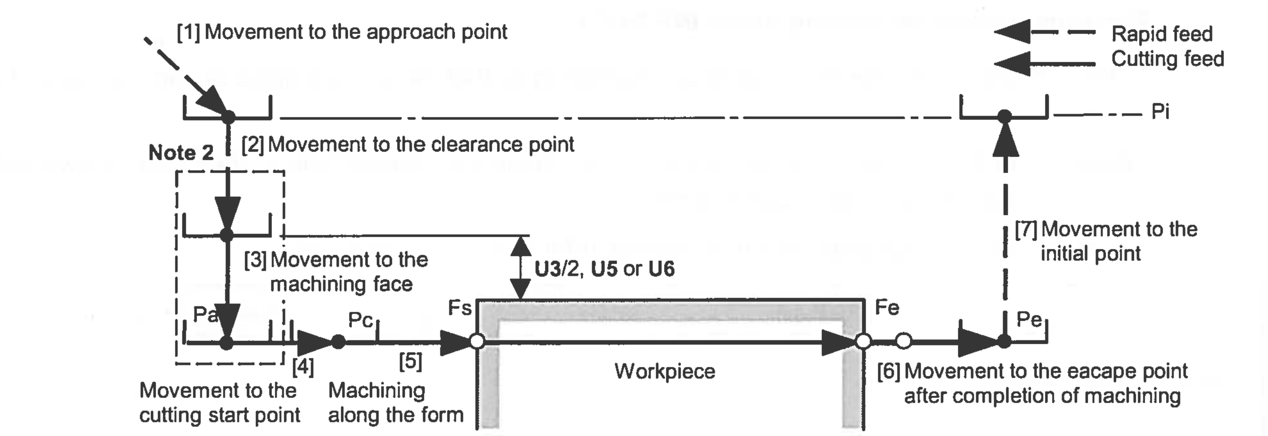
Tool path

Top view of the shape



M3P174

Side view of the shape



M3P175

The bold codes represent parameter addresses.

Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

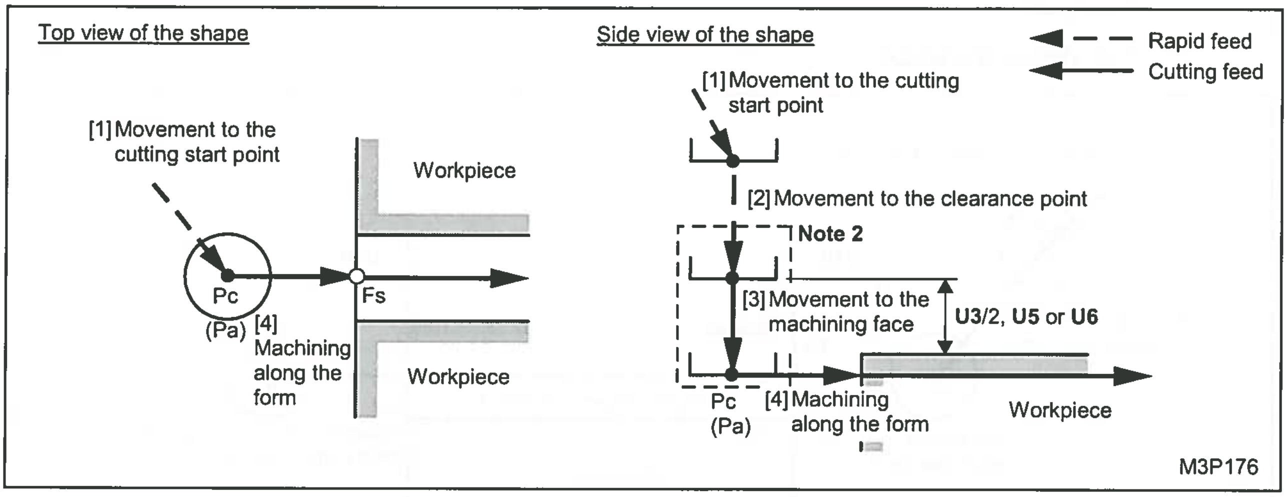
Pc: Cutting start point to be automatically established

Fs: Start point of form to be entered in the shape sequence

Fe: End point of form to be entered in the shape sequence

Pe: Escape point to be determined automatically

Note 1: When ? is displayed in the articles **APRCH-1**, **-2** by pressing the **[AUTO SET]** menu key, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate value of the cutting start point will be entered in these articles.



Note 2: See Subsection 3-5-6, "Precautions in line machining".

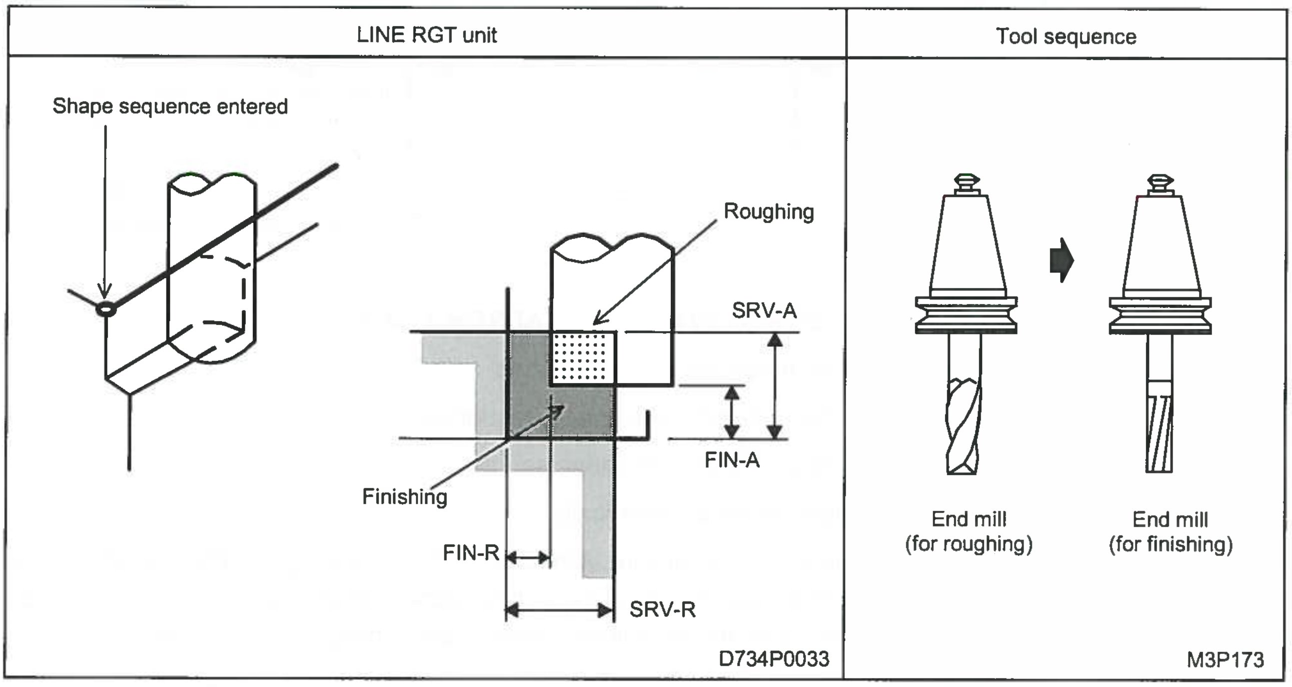
Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

2. **Right-hand linear machining unit (LINE RGT)**

This unit should be selected to carry out machining so that the tool will move on the right side of a form.

Note 1: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Note 2: For the tool sequence data setting, refer to Subsection 3-5-4.



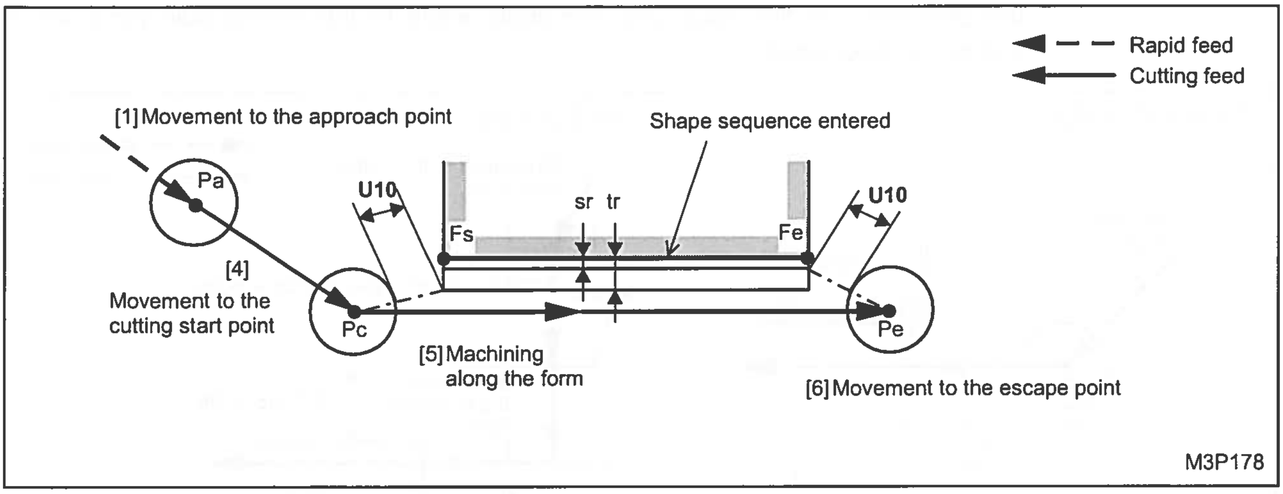
RGH: A roughness code should be selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a roughness code has been selected.

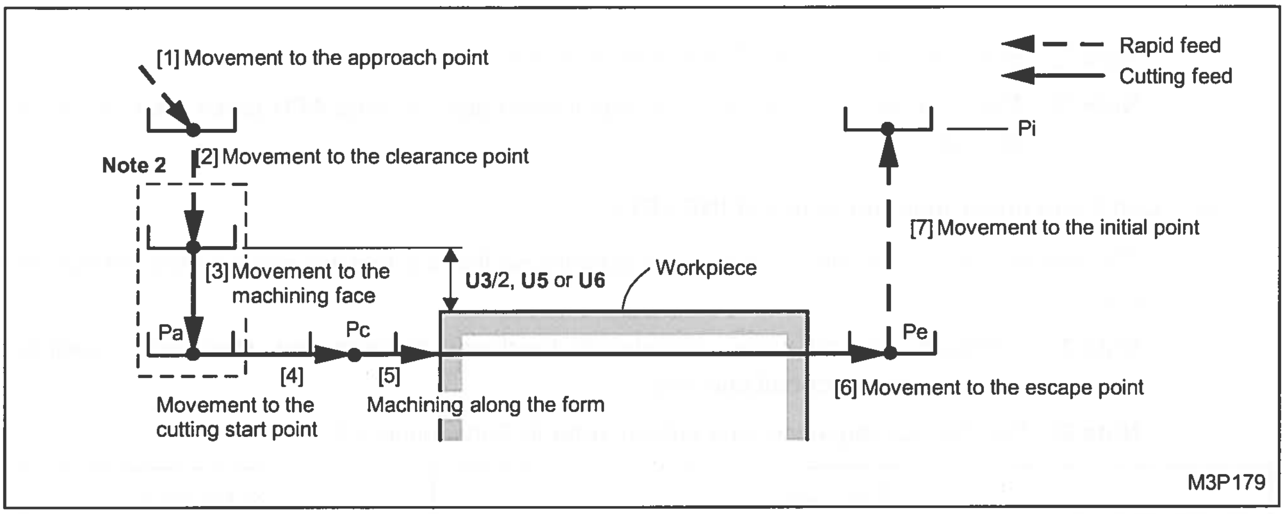
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Fs: Start point of form to be entered in the shape sequence

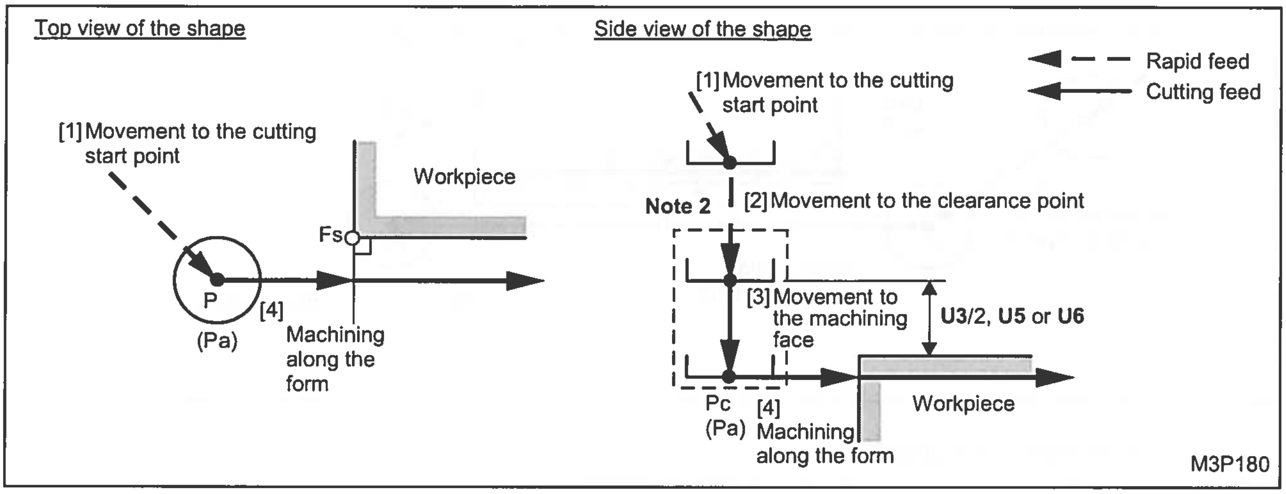
Fe: End point of form to be entered in the shape sequence

Pe: Escape point to be automatically established

tr: Radial cutting allowance to be determined by the data **SRV-R** in the machining unit

sr: Radial finishing allowance to be determined by the data **FIN-R** in the machining unit

Note 1: When ? is displayed in the articles **APRCH-1**, **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate value of the cutting start point will be entered in these articles.



M3P180

Note 2: See Subsection 3-5-6 "Precautions in line machining".

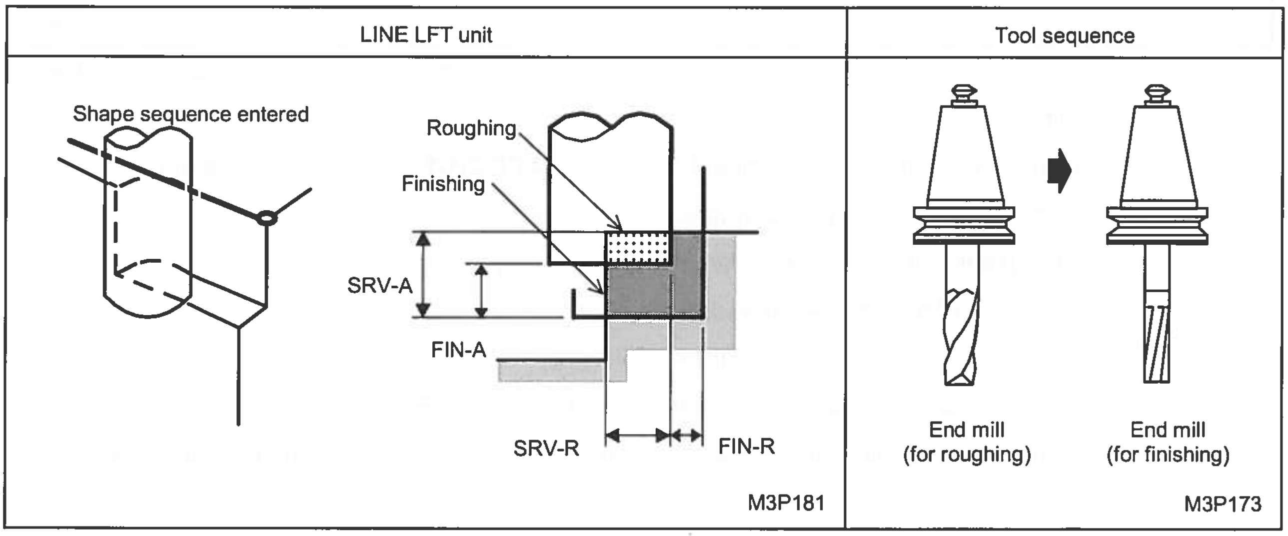
Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

3. Left-hand linear machining unit (LINE LFT)

This unit should be selected to carry out machining so that the tool will move on the left side of a form.

Note 1: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Note 2: For the tool sequence data setting, refer to Subsection 3-5-4.



M3P181 M3P173

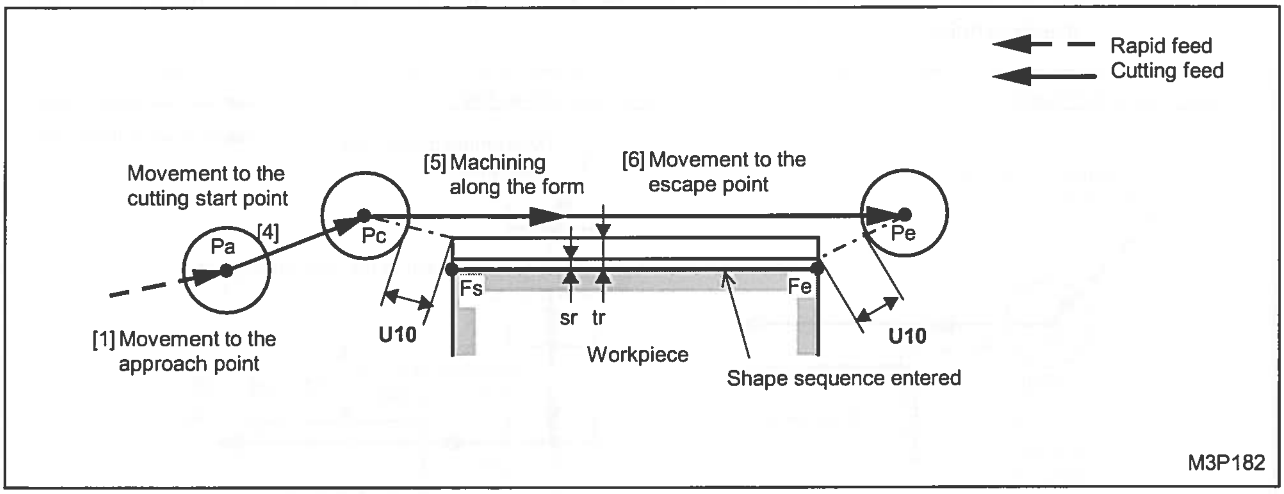
RGH: A roughness code should be selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a roughness code has been selected.

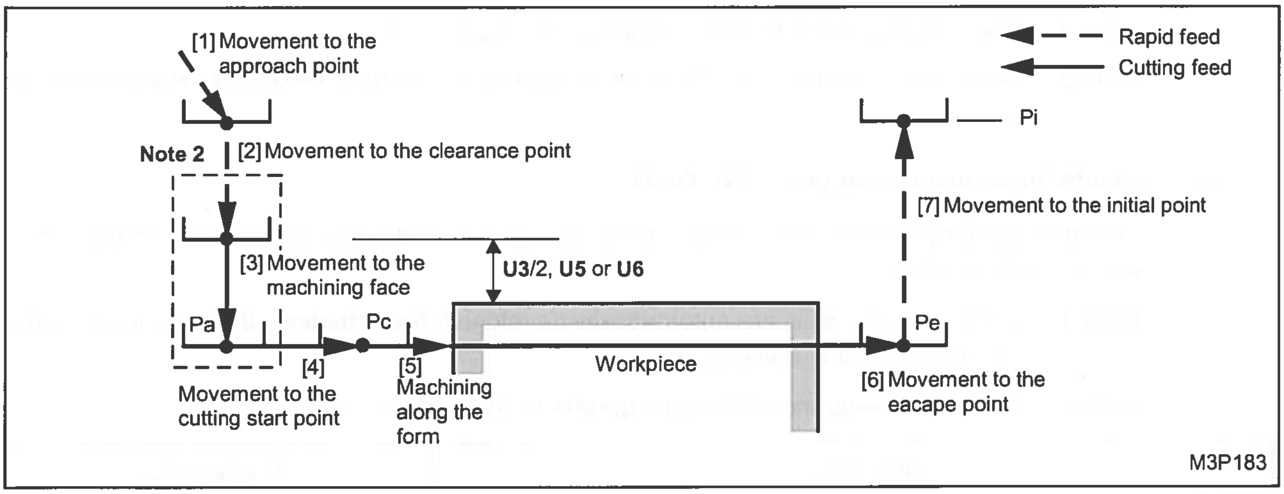
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Fs: Start point of form to be entered in the shape sequence

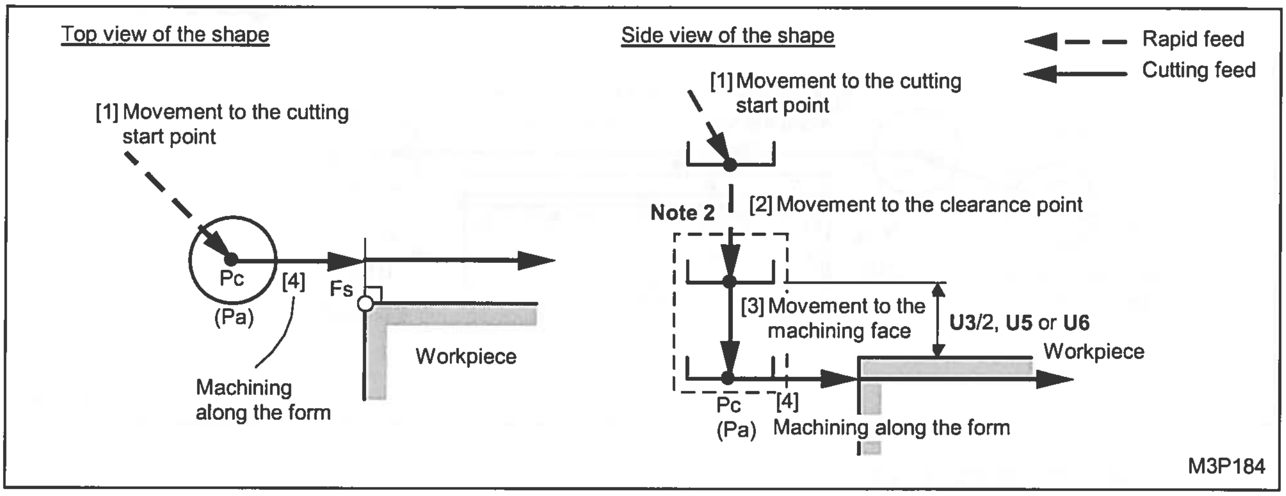
Fe: End point of form to be entered in the shape sequence

Pe: Escape point to be automatically established

tr: Radial cutting allowance to be determined by the data **SRV-R** in the machining unit

sr: Radial finishing allowance to be determined by the data **FIN-R** in the machining unit

Note 1: When ? is displayed in the articles **APRCH-1**, **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate value of the cutting start point will be entered in these articles.



Note 2: See Subsection 3-5-6, "Precautions in line machining".

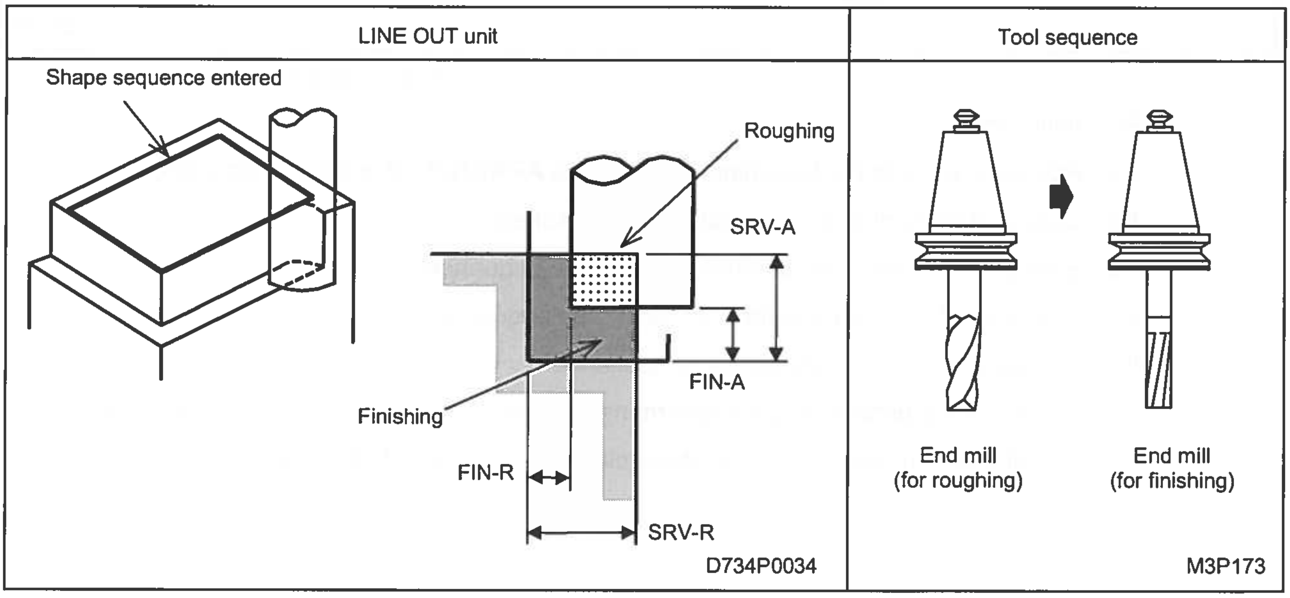
Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

4. Outside linear machining unit (LINE OUT)

This unit should be selected to carry out machining so that the tool will move to make a turn-around outside a form.

Note 1: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Note 2: For the tool sequence data setting, refer to Subsection 3-5-4.



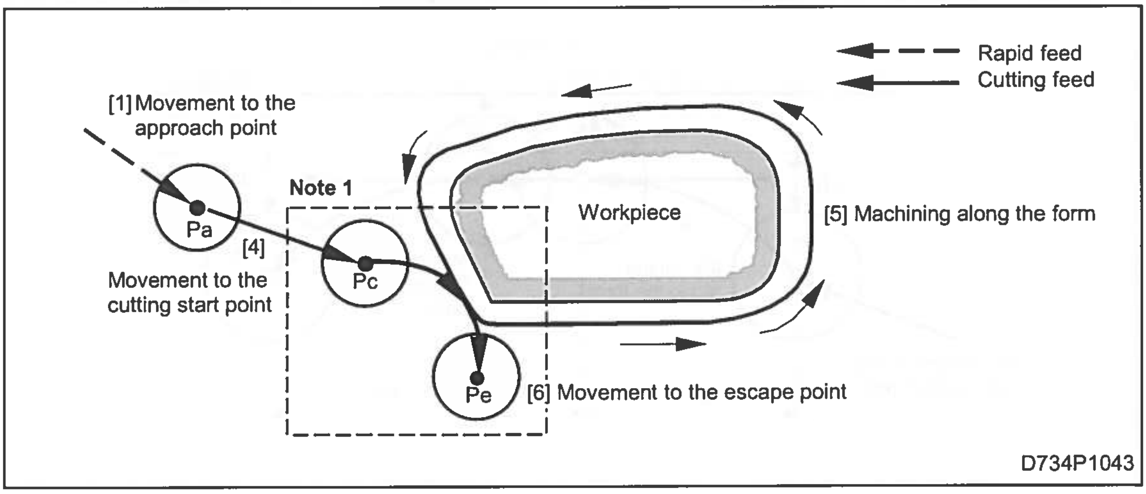
RGH: A roughness code should be selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a roughness code has been selected.

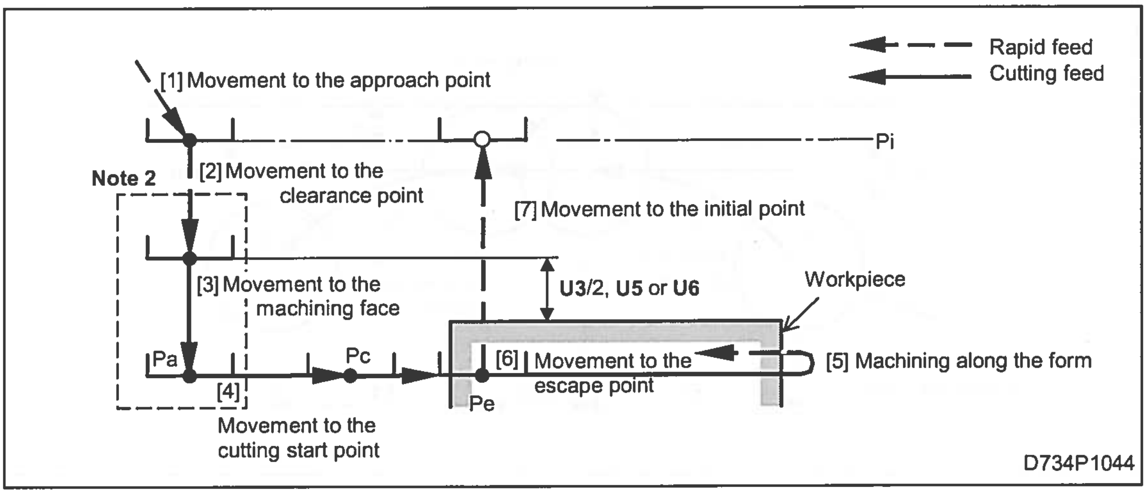
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

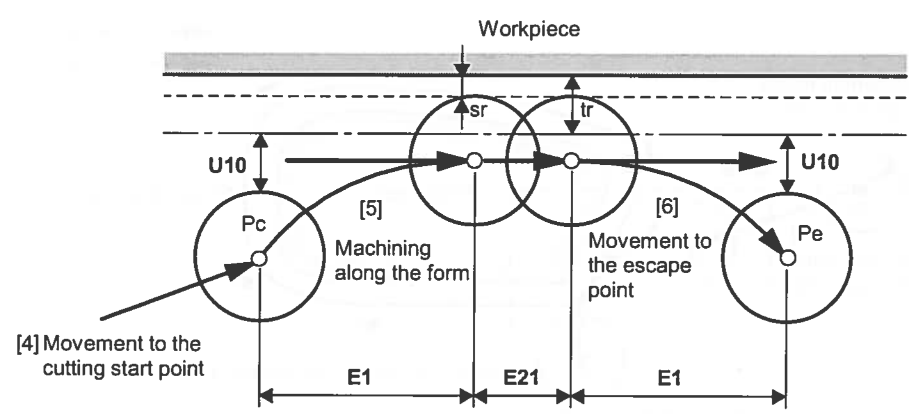
Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

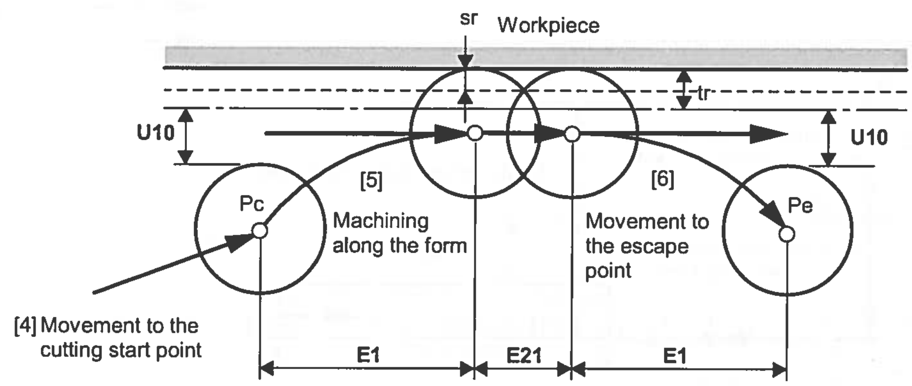
Note 1: Detail description of tool path near approach point and escape point

- In case of roughing



M3P190

- In case of finishing



M3P191

The bold codes represent parameter addresses.

tr: Radial cutting allowance to be determined by the data **SRV-R** in the machining unit

sr: Radial finishing allowance to be determined by the data **FIN-R** in the machining unit

Note 2: See Subsection 3-5-6, "Precautions in line machining".

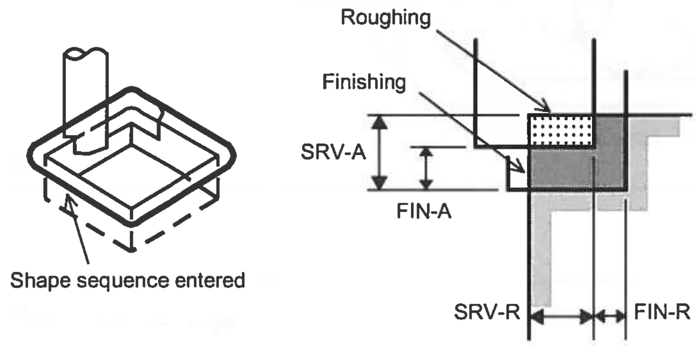
Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

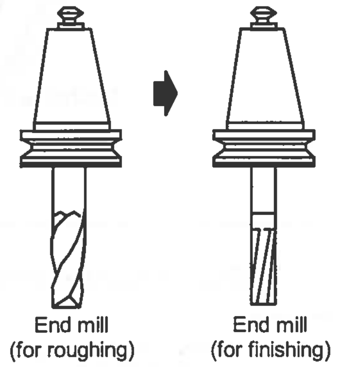
5. **Inside linear machining unit (LINE IN)**

This unit should be selected to carry out machining so that the tool will make a turn-around inside of a form.

Note 1: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Note 2: For the tool sequence data setting, see Subsection 3-5-4.

<table>
<tr><th>LINE IN unit</th><th>Tool sequence</th></tr>
<tr><td>


M3P196
</td><td>


M3P173
</td></tr>
</table>

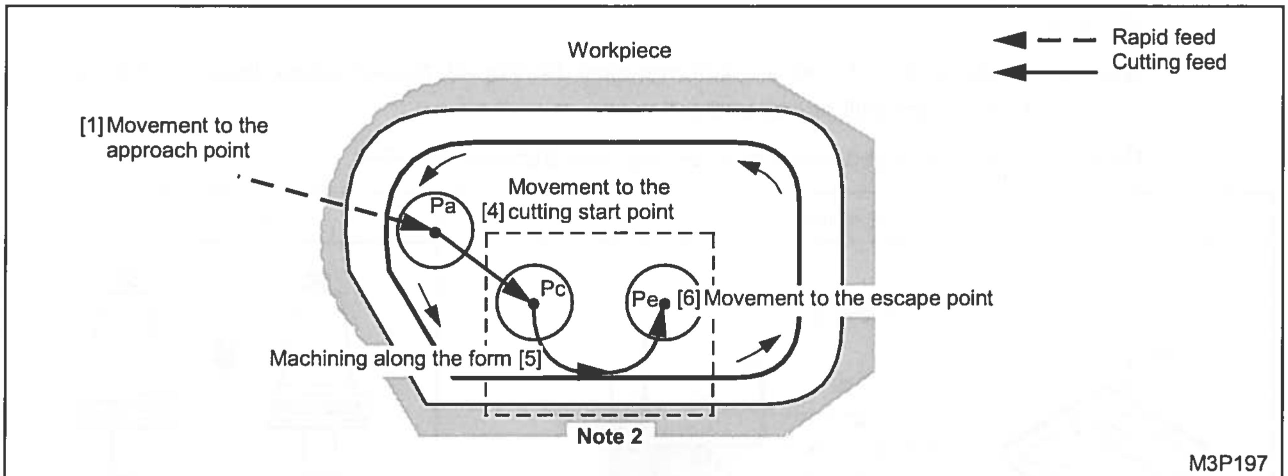
RGH: A roughness code should be selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a roughness code has been selected.

FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

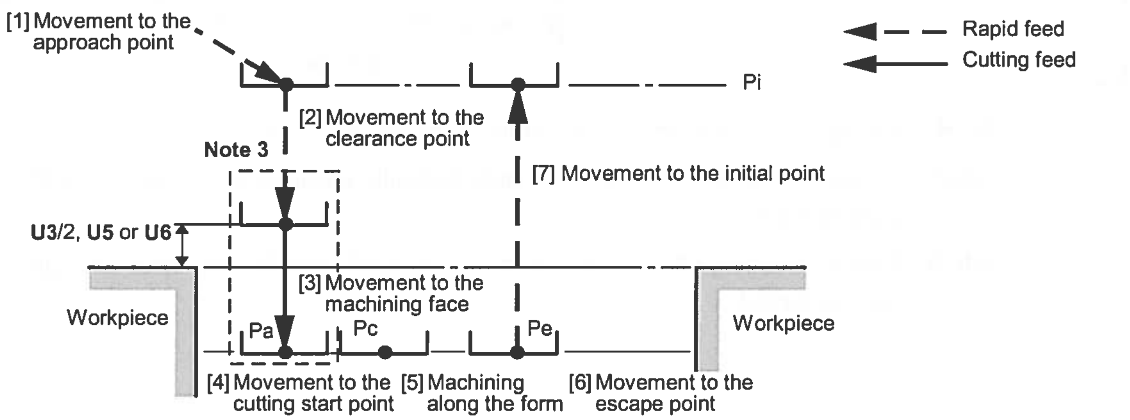
Tool path

Top view of the shape



M3P197

Side view of the shape



M3P198

The bold codes represent parameter addresses.

Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

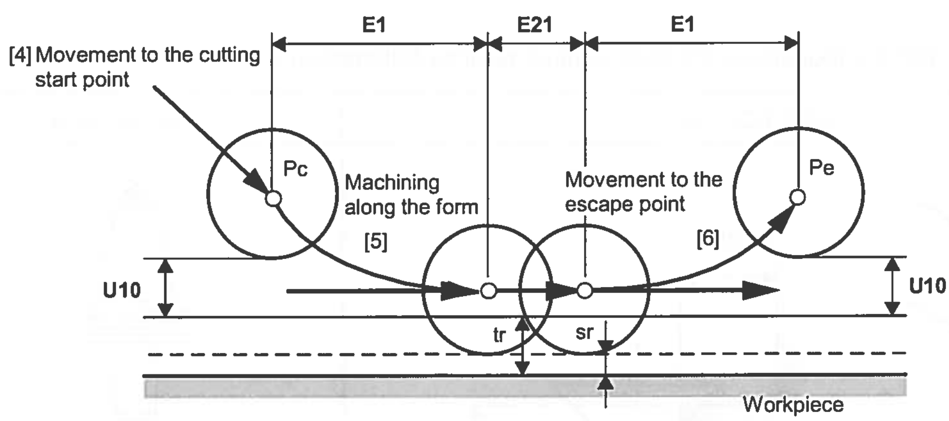
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatcially established

Note 1: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

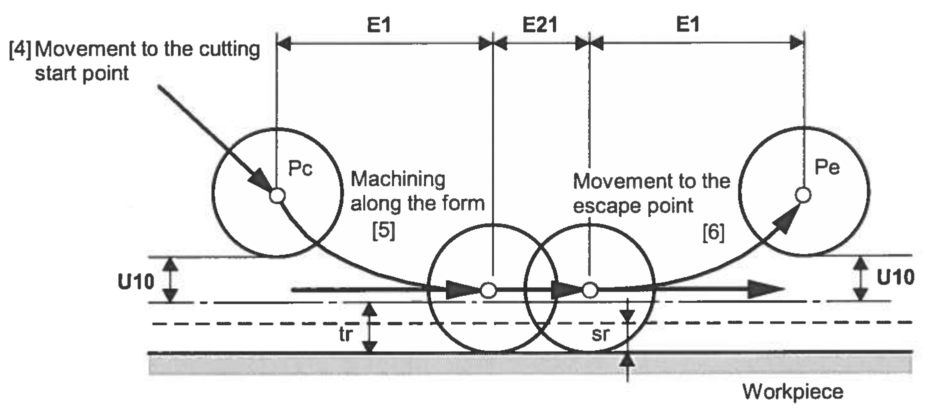
Note 2: Detail description of tool path near approach point and escape point

- In case of roughing



M3P201

- In case of finishing



M3P202

The bold codes represent parameter addresses.

tr: Radial cutting allowance to be determined by the data **SRV-R** in the machining unit

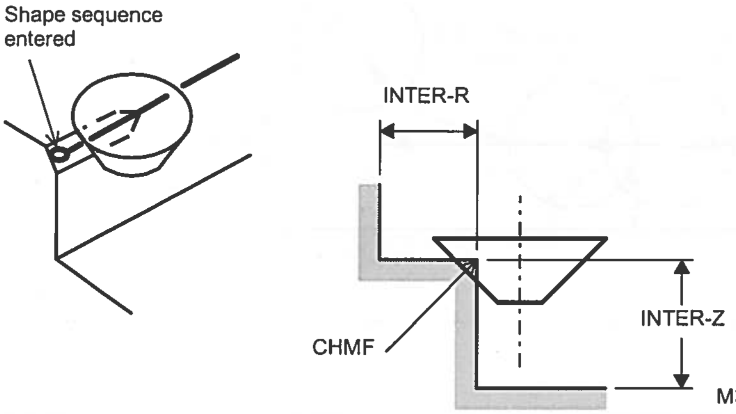
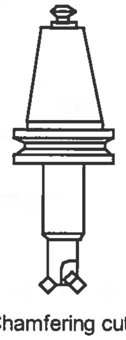
sr: Radial finishing allowance determined by the data **FIN-R** in the machining unit

Note 3: See Subsection 3-5-6, “Precautions in line machining”.

6. Right-hand chamfering unit (CHMF RGT)

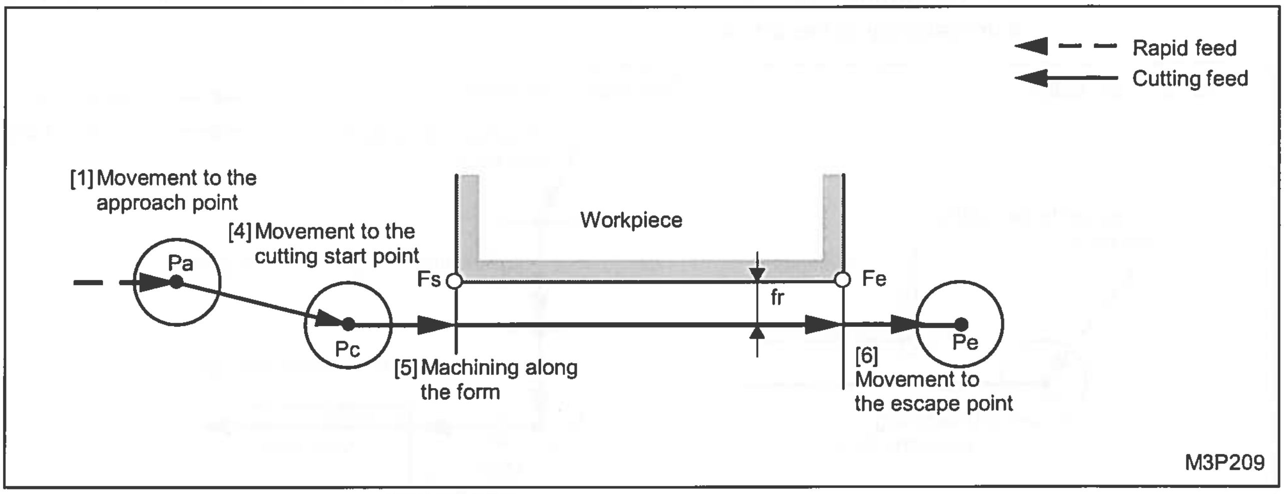
This unit should be selected to carry out chamfering so that a tool will move on the right side of a form.

Note: For the tool sequence data setting, refer to Subsection 3-5-4.

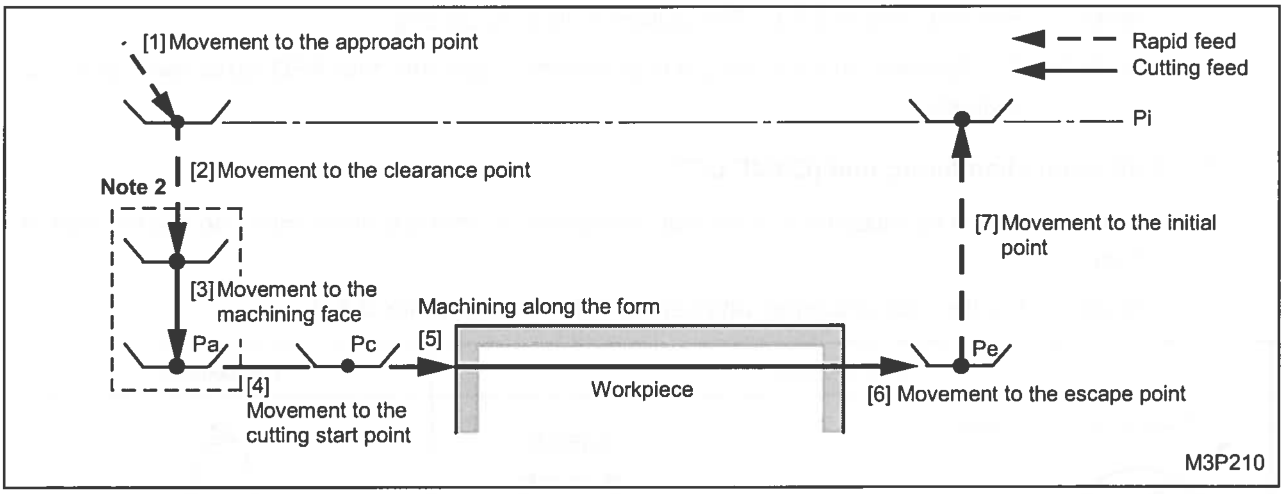
CHMF RGT unit	Tool sequence
 M3P207	 M3P208

Tool path

Top view of the shape



Side view of the shape



Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

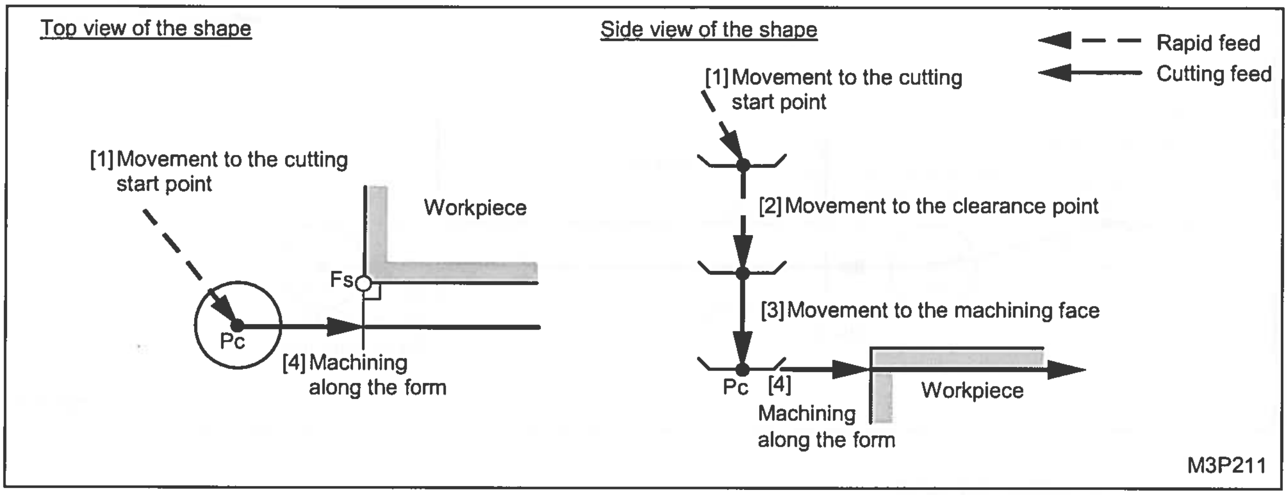
Fs: Start point of form to be entered in the shape sequence

Fe: End point of form to be entered in the shape sequence

Pe: Escape point to be automatically established

fr: Optimum distance to be automatically established, from the data entered in the **PROGRAM** and **TOOL FILE** displays

Note 1: When ? is displayed in the articles **APRCH-1** and **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, a coordinate of the cutting start point is entered automatically in the articles.



M3P211

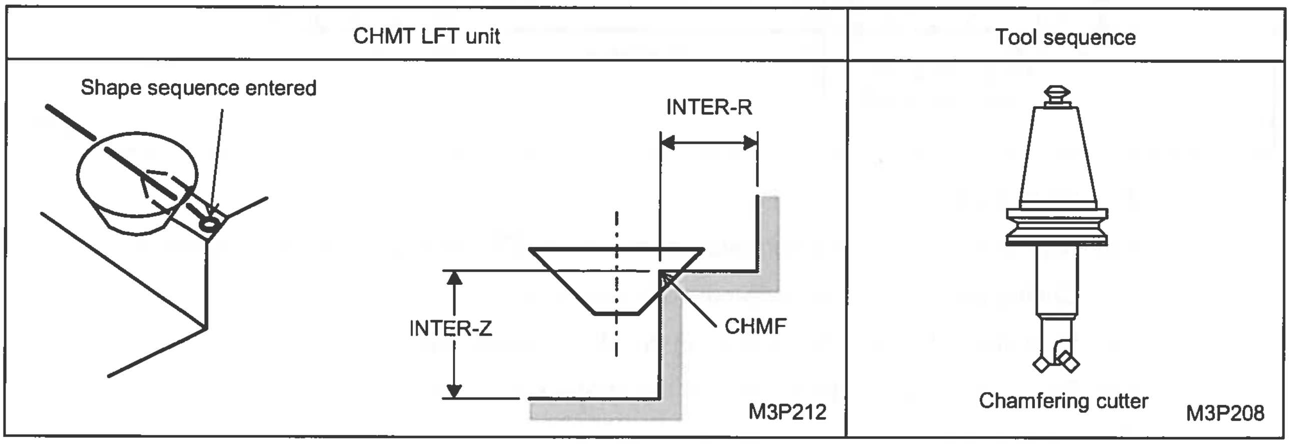
Note 2: See Subsection 3-5-6, "Precautions in line machining".

Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

7. **Left-hand chamfering unit (CHMF LFT)**

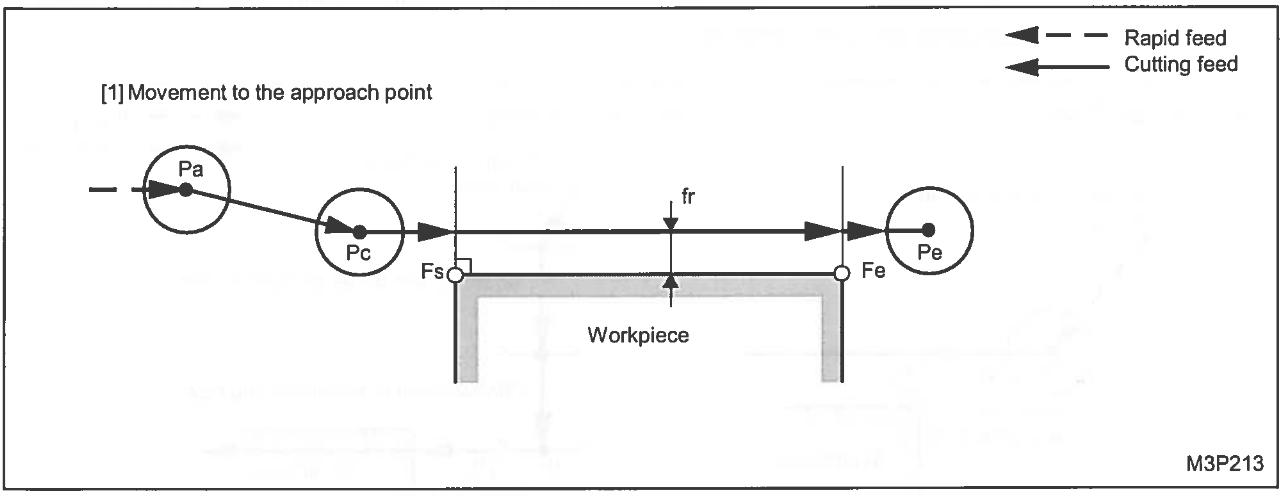
This unit should be selected to carry out chamfering so that a tool will move on the left side of a form.

Note: For the tool sequence data setting, refer to Subsection 3-5-4.

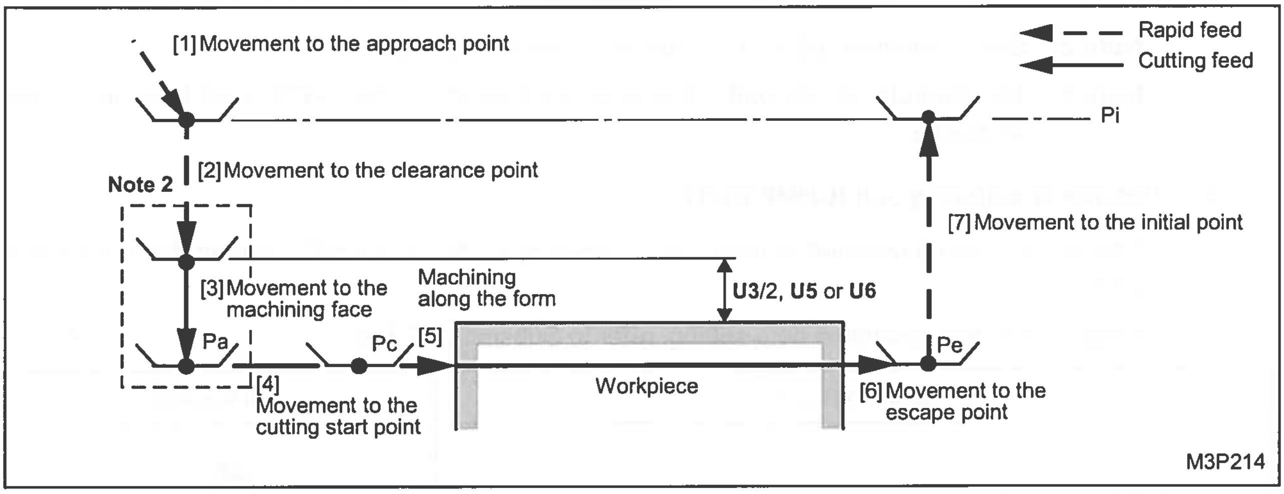


Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

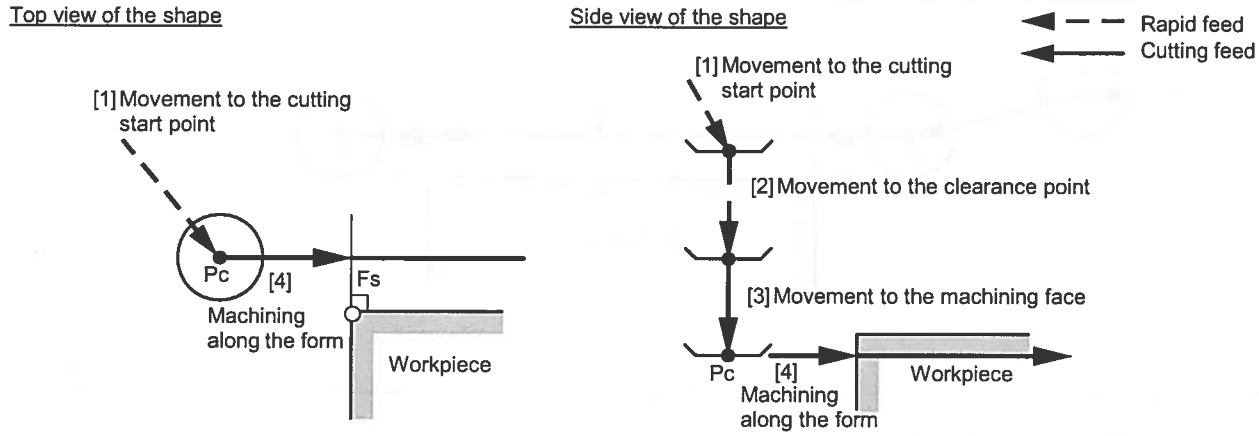
Fs: Start point of form to be entered in the shape sequence

Fe: End point of form to be entered in the shape sequence

Pe: Escape point to be automatically established

fr: Optimum distance to be automatically established, from the data entered in the **PROGRAM** and **TOOL FILE** displays

Note 1: When ? is displayed in the articles **APRCH-1** and **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, a coordinate of the cutting start point is entered automatically in the articles.



M3P215

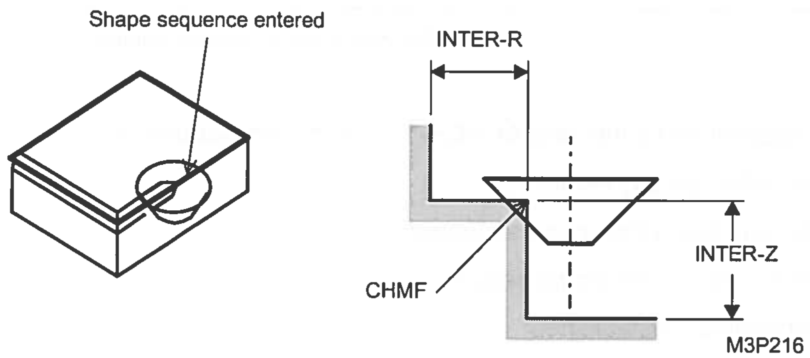
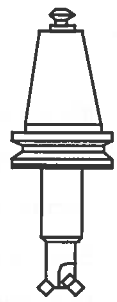
Note 2: See Subsection 3-5-6, "Precautions in line machining".

Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

8. Outside chamfering unit (CHMF OUT)

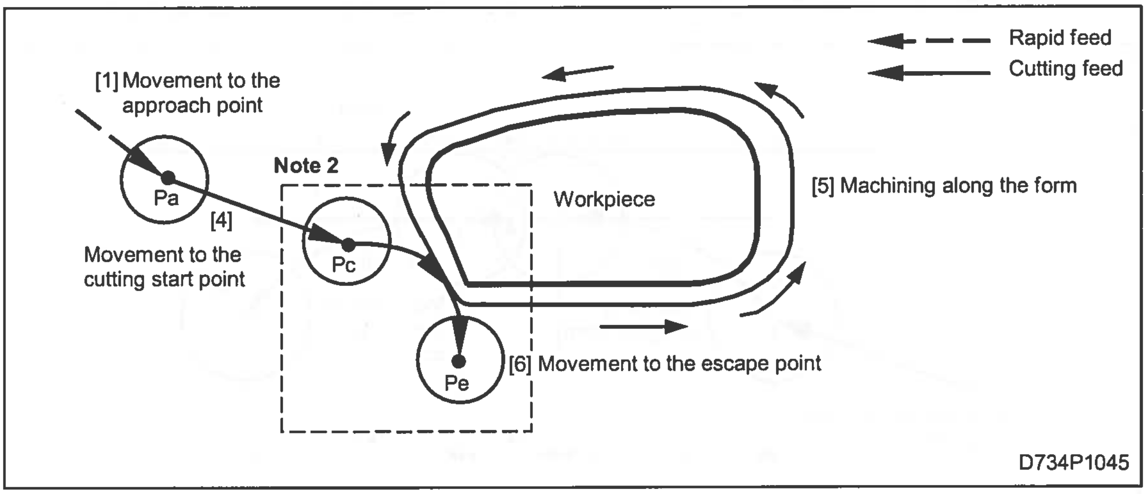
This unit should be selected to carry out chamfering so that a tool will move on the outside of a form.

Note: For tool sequence data setting, refer to Subsection 3-5-4.

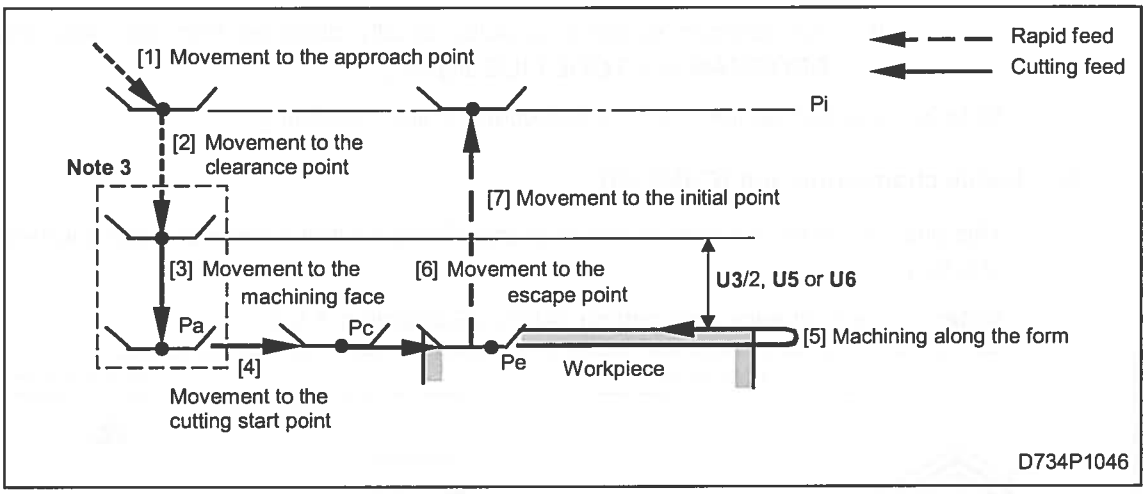
CHMF OUT unit	Tool sequence
 M3P216	 Chamfering cutter M3P208

Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

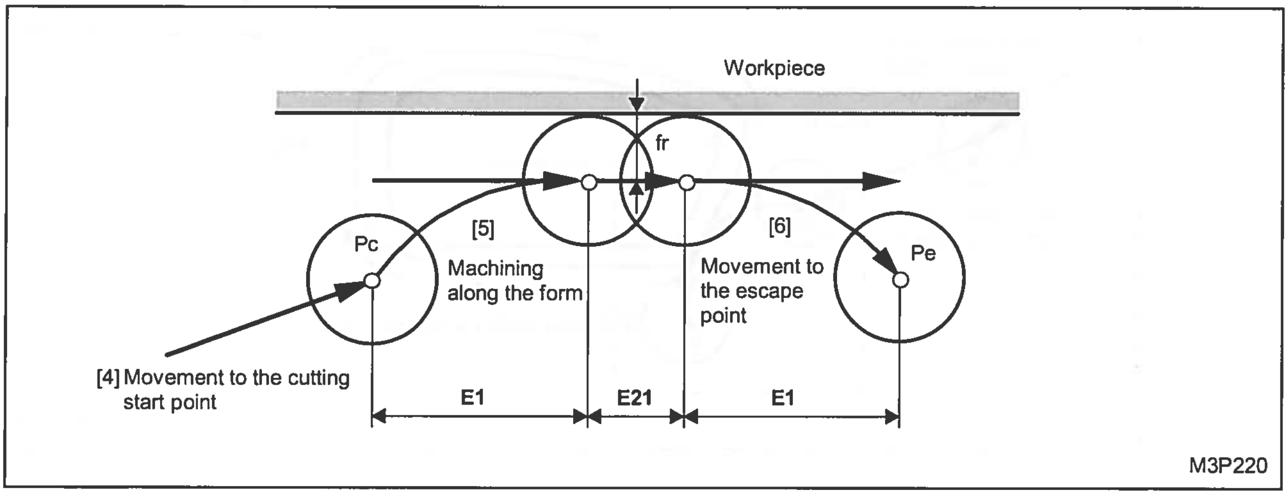
Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Note 1: The feedrate on tool path [3] is dependent upon the **AFD** (axial feed) in the tool sequence.

Note 2: Detail description of tool path near approach point and escape point



The bold codes represent parameter addresses.

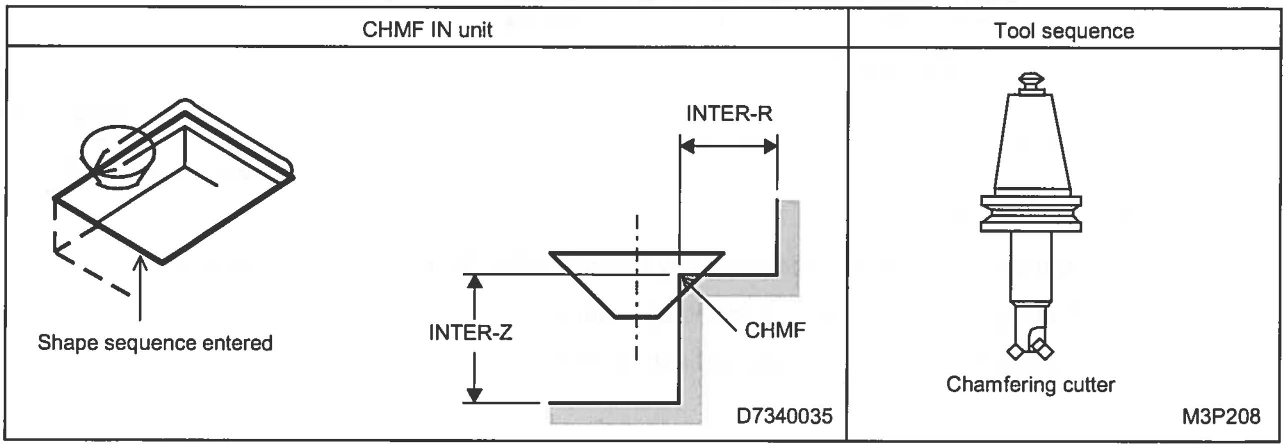
fr: An optimum distance is automatically obtained from the data entered in the **PROGRAM** and **TOOL FILE** displays

Note 3: See Subsection 3-5-6, "Precautions in line machining".

9. Inside chamfering unit (CHMF IN)

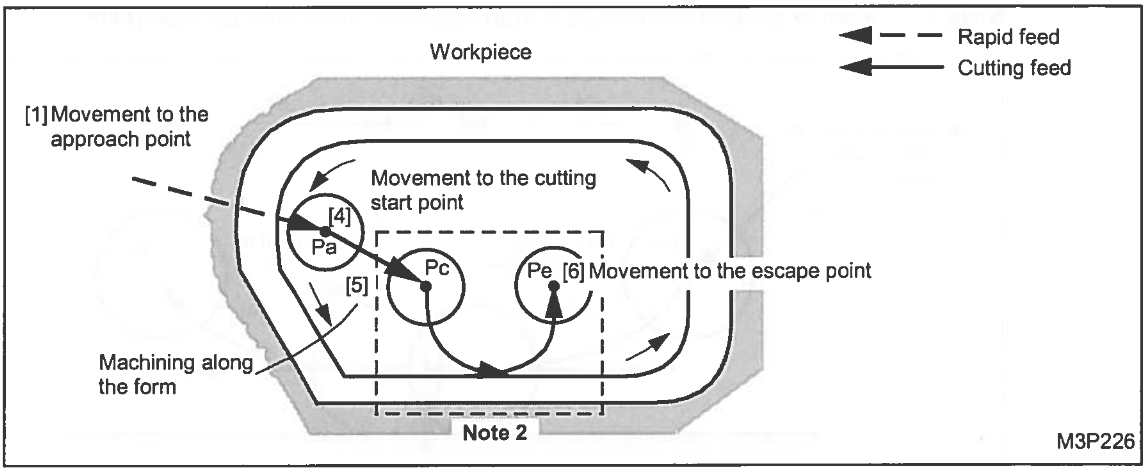
This unit should be selected to carry out chamfering so that a tool will make a turn-around inside of a form.

Note: For sequence data setting, refer to Subsection 3-5-4.

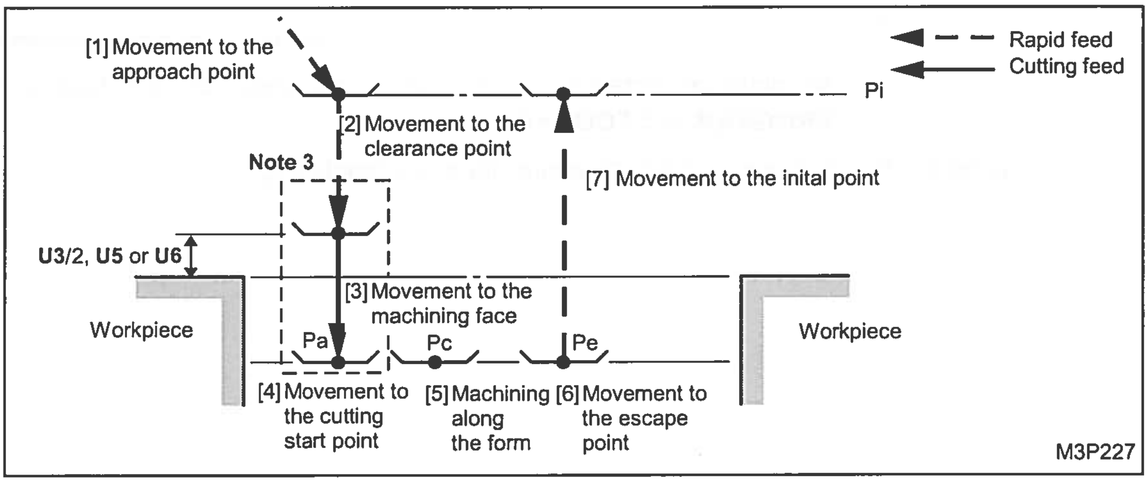


Tool path

Top view of the shape



Side view of the shape



The bold codes represent parameter addresses.

Pi: Initial point

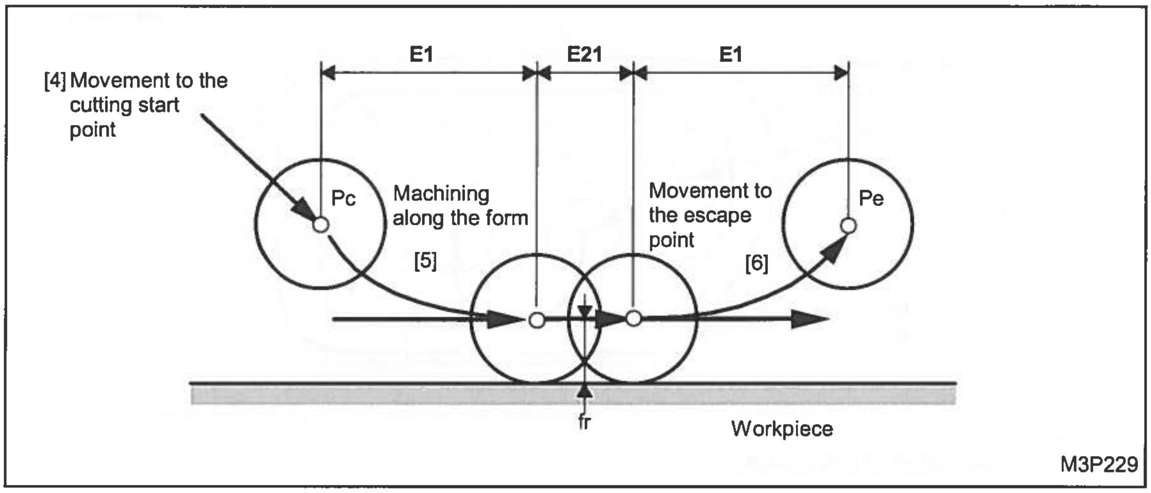
Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Note 1: The feedrate on tool path [3] is dependent upon the **AFD** (axial feed) in the tool sequence.

Note 2: Detail description of tool path near approach point and escape point



The bold codes represent parameter addresses.

fr: An optimum distance is automatically obtained from the data entered in the **PROGRAM** and **TOOL FILE** displays

Note 3: See Subsection 3-5-6, "Precautions in line machining".

3-5-4 Tool sequence data of the line machining unit

For line machining tool sequence data only a tool name is automatically selected once a machining unit has been entered. Other data should be entered by use of menu keys or numeric keys according to a form of the workpiece to be machined or to the procedure for machining.

SNo.	TOOL	NOM-φ	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				
	↑ 1	↑ 2 ↑ 3	↑ 10	↑ 4	↑ 4	↑ 5	↑ 6	↑ 7		↑ 8	↑ 8	↑ 9	↑ 9

◆: Not necessary to be set here.

1. Tool designation: TOOL

The name of a tool can be changed by the use of menu keys.

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL						

In the central linear, right-hand linear, left-hand linear, outside linear and inside linear machining units, either end mill, face mill or ball end mill is selectable. In the right-hand, left-hand, outside and inside chamfering units, only a chamfering cutter is selectable.

2. Nominal diameter of tool: NOM-φ

Approximate diameter of a tool is entered. A nominal diameter is the data to identify by diameter those tools which are of identical type (having an identical name).

3. Tool identification code

A code should be selected out of the menu to identify those tools which are of identical type (having and identical name) and have an identical nominal diameter.

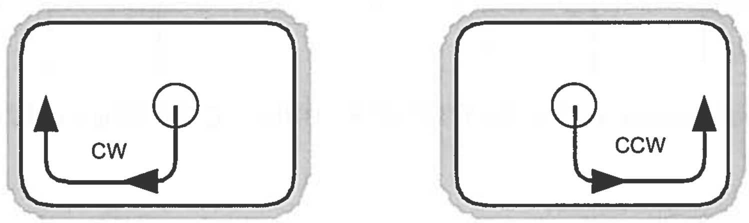
4. Coordinates 1 and 2 of the approach point: APRCH-1, APRCH-2

Enter an X, Y coordinates of the position at which a tool is to cut in axially.
With the **[AUTO SET]** menu item selected, ? is displayed. After the tool path check is performed, ? will automatically change over to the coordinates of a cutting start point. (Refer to tool path by each unit.)

5. Machining method: TYPE

Use menu keys to select the direction in which machining (turning) is performed in the outside and inside linear machining and outside and inside chamfering units.

CW CUT ↻	CCW CUT ↺								



When the **[CW CUT]** menu key is pressed, machining is performed clockwise.

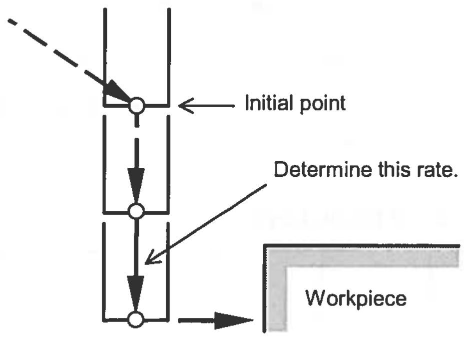
When the **[CCW CUT]** menu key is pressed, machining is performed counterclockwise.

M3P234

6. **Axial feedrate: AFD**

Enter the feedrate in the axial direction. It is also possible, moreover, to select rapid feed (G00) or cutting feed (G01) by the use of menu keys.

CUT G01	RAPID G00								



AFD	Feedrate
G00	Rapid feed
G01	Parameter **E17** may be used to determine: Feed × $\frac{\textbf{E17}}{10}$
Numeric value (α)	Feed × α

M3P235

7. **Axial cutting stroke: DEP-A**

In roughing, a maximum axial cutting stroke in one cycle is entered. With the **[AUTO SET]** menu item selected, a smaller value is entered, either the data **SRV-A** entered in the machining unit or the maximum cutting stroke registered on the **TOOL FILE** display. An actual axial cutting stroke is arithmetically obtained from the data **DEP-A**, **SRV-A** and **FIN-A**, both in the machining unit. (For calculation formula, see Subsection 3-5-6, "Precautions in line machining".)

8. **Cutting conditions (circumferential speed, feed): C-SP, FR**

A spindle speed and a cutting feedrate are entered.

With **[AUTO SET]** menu item selected, optimum cutting conditions are calculated and entered, based on the materials of both workpiece and tool and on the cutting depth. (A circumferential speed is given in meters per minute and a cutting feedrate in millimeters per revolution.)

9. **M-codes: M**

Set the required M-code(s) to be output immediately after mounting the tool onto the spindle in the ATC mode. A maximum of up to two M-codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu. (See 3-19-2 "M-code table".)

10. **Retraction position of the lower turret: #**

For a machine having upper and lower turrets, it is possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.

The following menu is displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: For details refer to Chapter 4 "LOWER-TURRET CONTROL FUNCTIONS".

3-5-5 Shape sequence data of the line machining unit

The data setting articles of shape sequence for the line machining units are the same as those for the face machining units. For the shape sequence data setting, see Input procedure in Subsection 3-6-6.

3-5-6 Precautions in line machining

1. **Tool path during rough-machining with axial removal allowance (SRV-A) > cutting depth-A (DEP-A)**

Cutting is performed at several pass. The tool path is determined by the parameter **E95** which relates with three factors, but not all of these factors may be available for the certain machining unit:

A. Cutting start position along the axis

B. Type of routing through approach points

C. Type of escape along the axis after machining

[Basic tool path]

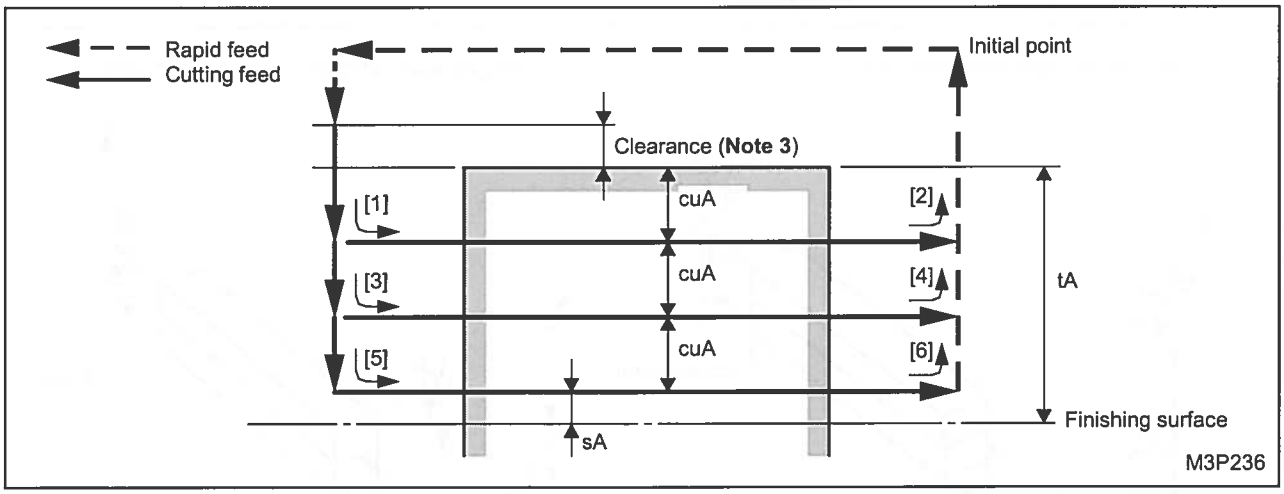


Fig. 3-14 Basic tool path

cuA: Cutting depth in the axial direction per pass

Calculation of cuA:

$$\boxed{cuA = \frac{tA - sA}{n}}$$

$$n = \frac{tA - sA}{cuA}$$

tA: Axial cutting allowance **SRV-A** to be entered in the machining unit

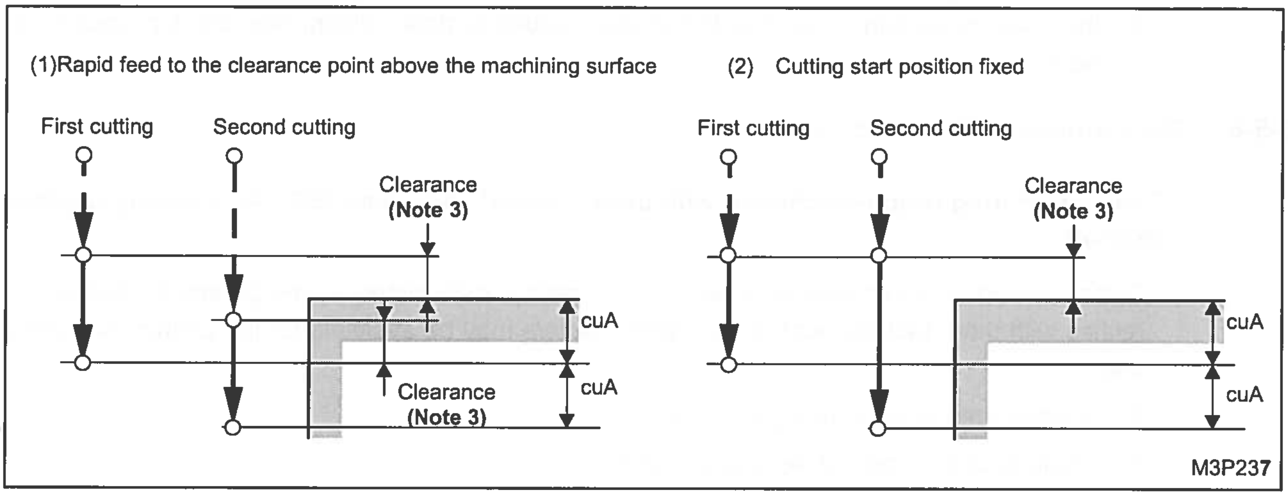
sA: Axial finishing allowance **FIN-A** to be entered in the machining unit

cuA: Axial cutting depth **DEP-A** to be entered in the tool sequence

n: Number of passes in the axial direction (Integer obtained by rounding up the decimal fraction)

A. Cutting start position along the axis direction

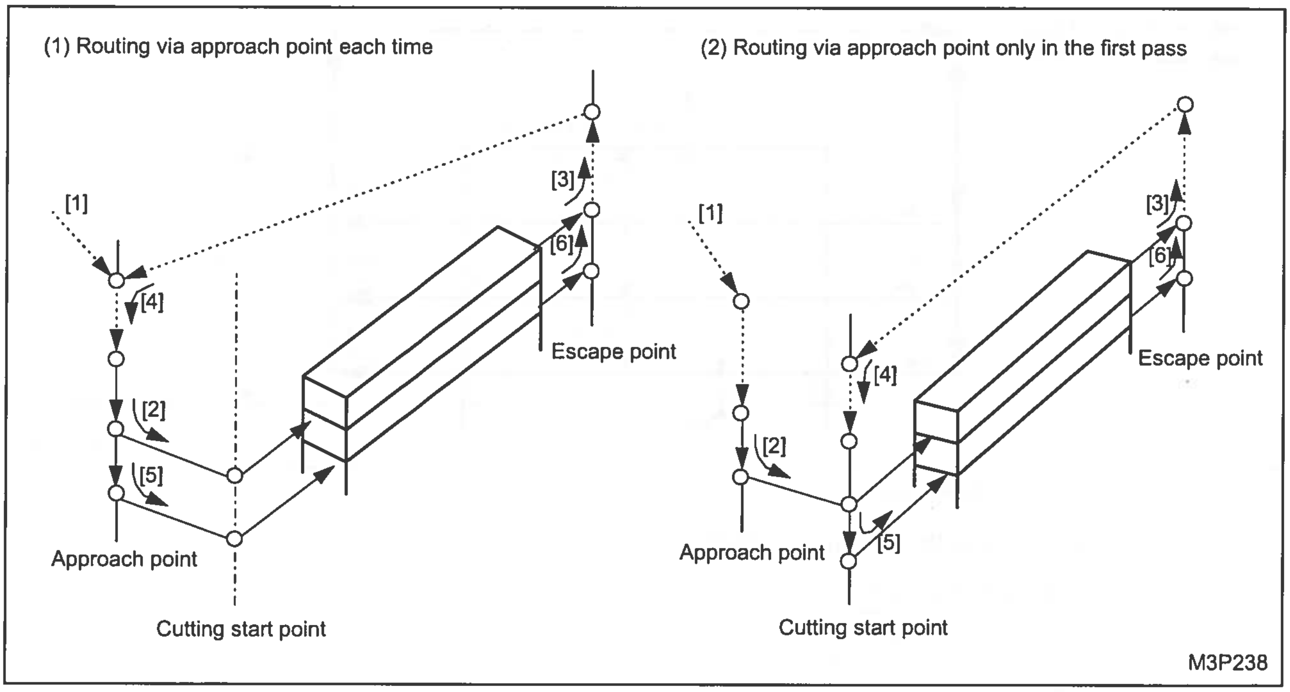
Select one of the following two types:



M3P237

B. Type of routing via approach point

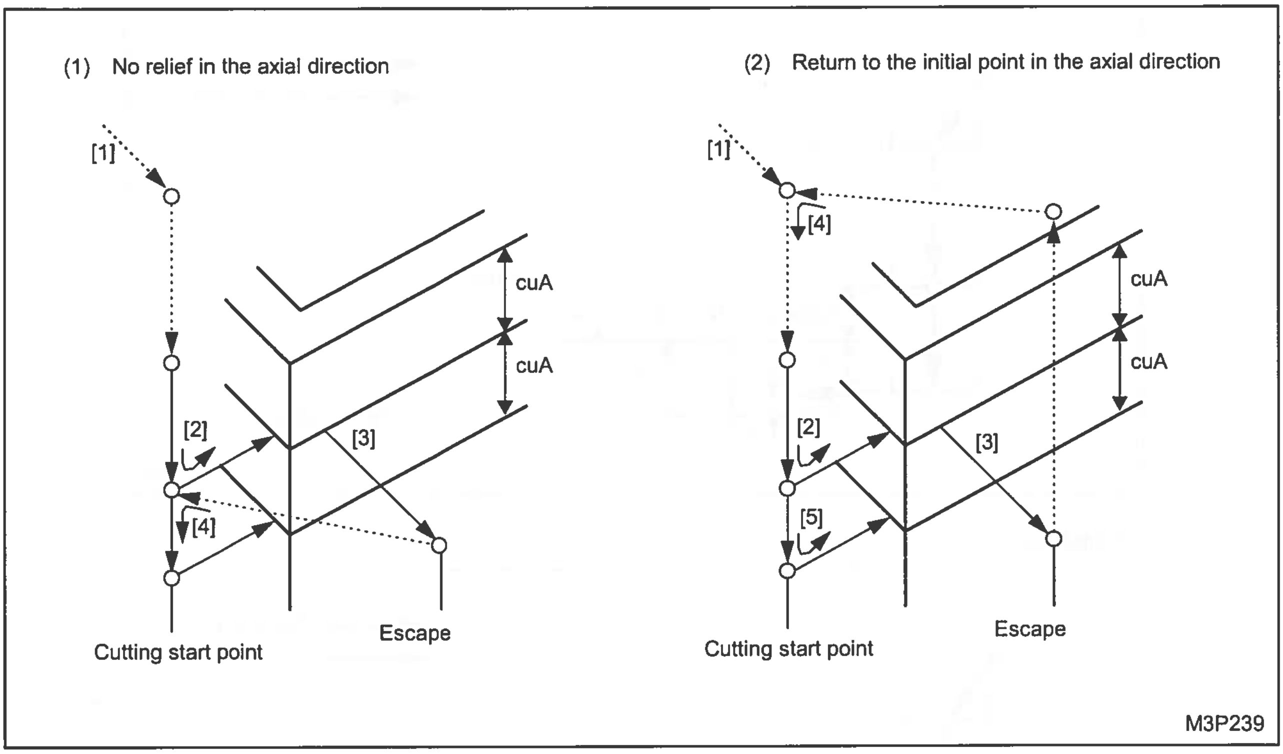
Select one of the following two types:



M3P238

C. Type of escape along the axis after machining

Select one of the following two types:



<u>Tool path setting parameter (Parameter **E95**)</u>

For A: bit 4 = 0: Cutting start position fixed -- (2)

1: Rapid feed to the clearance point above the machining surface -- (1)

* As for pattern (1), the starting position of cutting feed is determined by the setting of parameter **E7** (instead of **E9**) from the second cutting when the following conditions are satisfied:

- Bit 6 of parameter **E95** is set to "1", and
- The unit concerned is **<u>LINE</u> CTR**, **RGT**, **LFT**, **OUT** or **IN**.

For B: bit 2 = 0: Routing via approach points only in the first pass -- (2)

1: Routing via approach points each time -- (1)

For C: bit 3 = 0: Return to the initial point -- (2)

1: No escape along the axis -- (1)

Note 1: Both A and B can be used for all line-machining units, whereas C can only be used for inside linear and outside linear machining units.

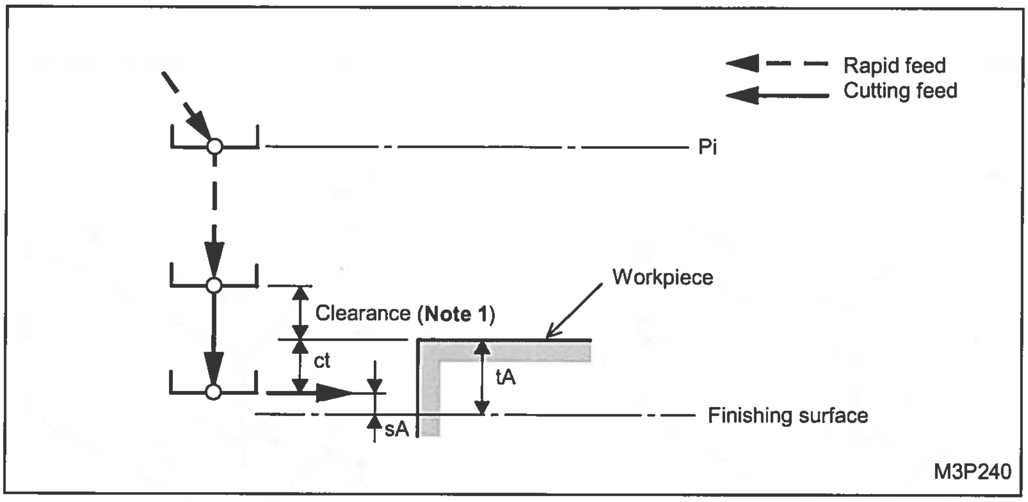
Note 2: The tool path shown at basic tool path above is selected automatically for machining units that are not subject to the selection of the parameter **E95.**

Note 3: The clearance value differs according to the mode specified for the machining unit.

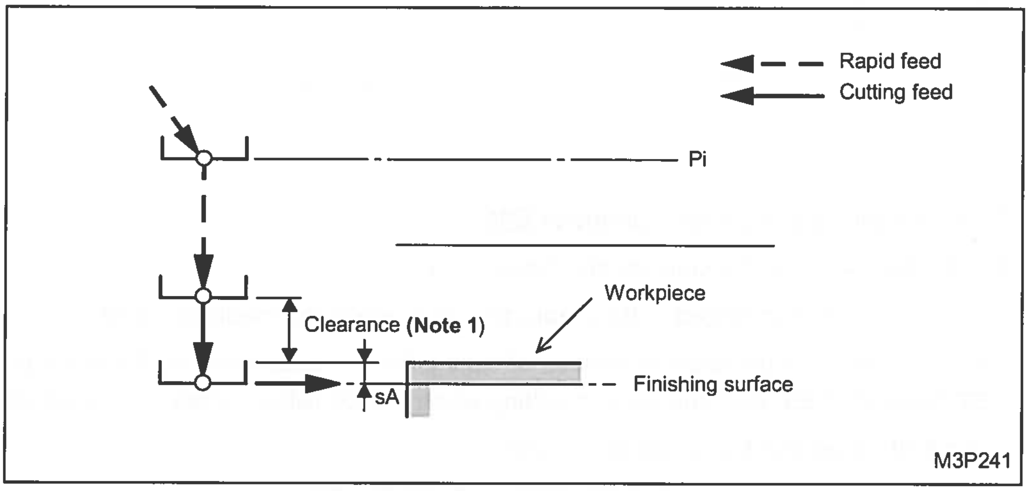
Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

2. **Detail tool path of a cut-in in the axial direction**

- Roughing



- Finishing



Pi: Initial point

ct: Axial cutting depth **DEP-A** to be entered in the tool sequence

tA: Axial cutting allowance **SRV-A** to be entered in a machining unit

sA: Axial finishing allowance **FIN-A** to be entered in a machining unit

Note 1: The clearance value differs according to the mode specified for the machining unit.

Mode	Clearance value
ZC, ZY	$\frac{U3}{2}$
XC, XY, /C, /Y $\boxed{XC}$, $\boxed{XY}$, $\boxed{/C}$, $\boxed{/Y}$	**U5**

The clearance will become equal to parameter **E7** if the following three states occur at the same time:

- Bit 6 of parameter **E95** is set to 1.
- A pre-machining tool is included in that tool sequence.
- The machining unit is either central linear, right-hand linear, left-hand linear, outside linear or inside linear machining.

Note 2: The clearance in radial direction, specified by parameter **U10**, will become equal to parameter **E5** if the following three states occur at the same time:

- Bit 7 of parameter **E95** is set to 1.
- A pre-machining tool is included in that tool sequence.
- The machining unit is either outside linear or inside linear machining.

3. **Other precaution on tool path**

If shape data, tool data or parameter are modified after the automatic determination of coordinates of approach point **APRCH-1** and **APRCH-2** (displayed in yellow), the approach point will not be located on the same cutting start point and the tool path will also be modified.

3-6 Face Machining Units

Face machining unit is used to enter the data relating to the procedures for machining an area and to the form to be machined. Available in this unit are two sequences; one is the tool sequence in which tool-operation-associated data are entered and the other shape sequence in which the data relating to machining dimensions specified on drawing are entered.

3-6-1 Types of face machining units

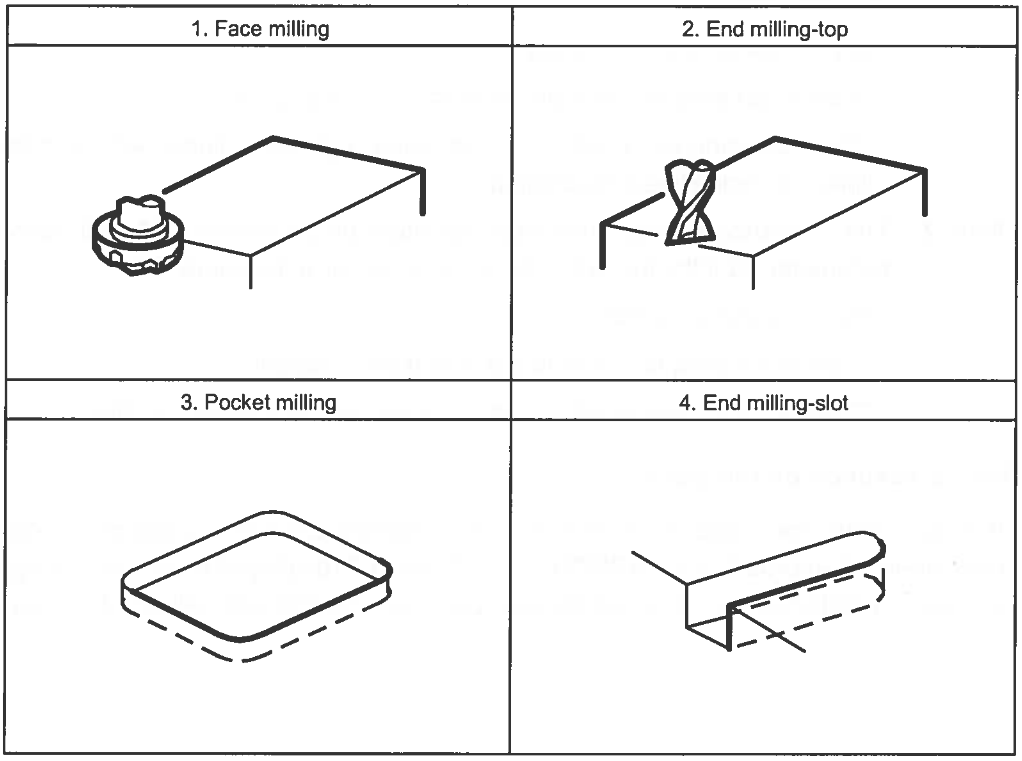


Fig. 3-15 Types of face machining unit

3-6-2 Procedure for selecting face machining unit

(1) Press the menu selector key (key located to the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRM	PLANE AUTO SET	>>>

(2) Pressing on the **[FACE MACH-ING]** menu key displays the following machining unit menu.

FACE MIL	TOP EMIL		POCKET			SLOT			

(3) Press the appropriate menu key of the desired machining unit.

3-6-3 Unit data, automatic tool development and tool path of the face machining unit

1. Face milling unit (FCE MILL)

This unit is selected to machine a workpiece flatly on the surface by the use of a face milling tool.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A		BTM	WAL	FIN-A	FIN-R				
1	FCE MILL	/Y	999.999	-999.999	99.999		9	◆	99.999	◆				
SNo.	TOOL	NOM-φ			APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	FCE MILL							◆						
F2	FCE MILL							◆	◆					

Remark 1: Data in unit represent a maximum input value.

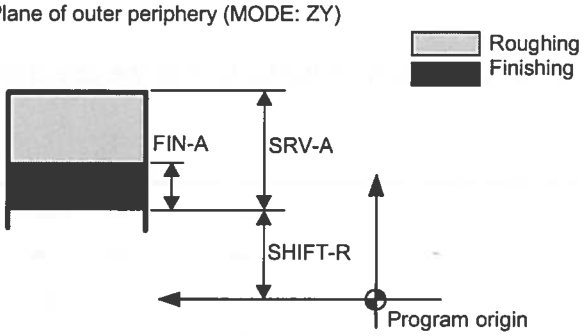
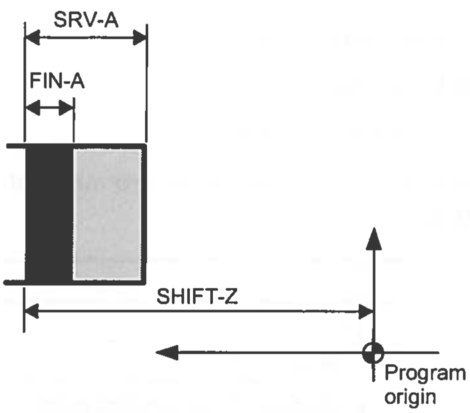
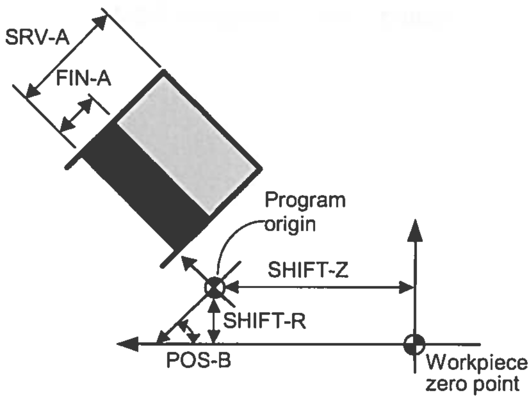
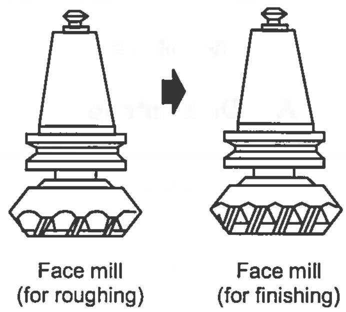
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, face mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A** and on **FIN-A**.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-A = 0
1	F1 (Finishing)	SRV-A ≤ FIN-A
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 3-6-4.

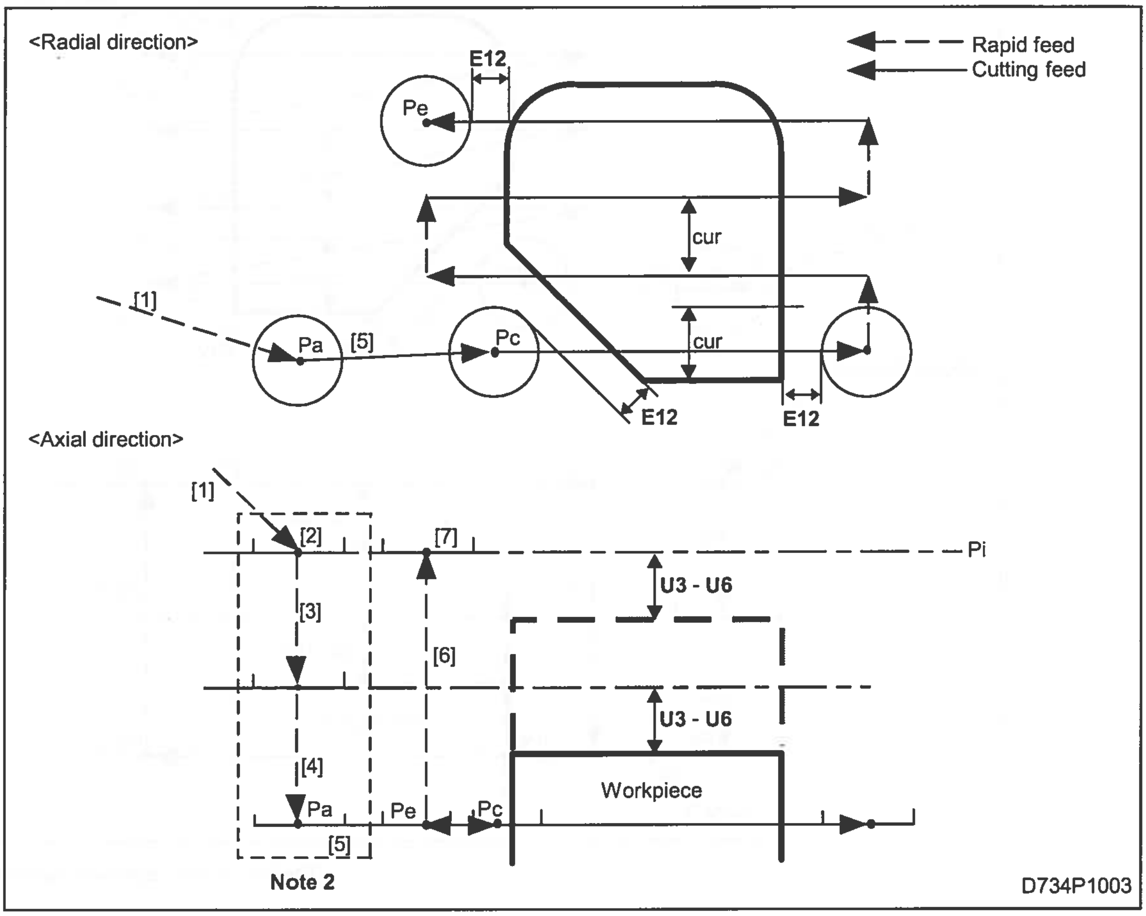
FCE MILL unit	Tool sequence
Set data as follows according to the plane to be machined: a) Plane of outer periphery (MODE: ZY)  b) Edge (MODE: XY, XY)  c) Oblique plane (MODE: /Y, /Y)  D734P1002	 M3P248

BTM: A bottom roughness code is selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a bottom roughness code has been selected.

B. Tool path

When the menu key **[1 BI-DIR]** is selected for the article **TYPE** in the tool sequence



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

cur: Radial cutting depth to be determined by the data **DEP-R** in the tool sequence

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

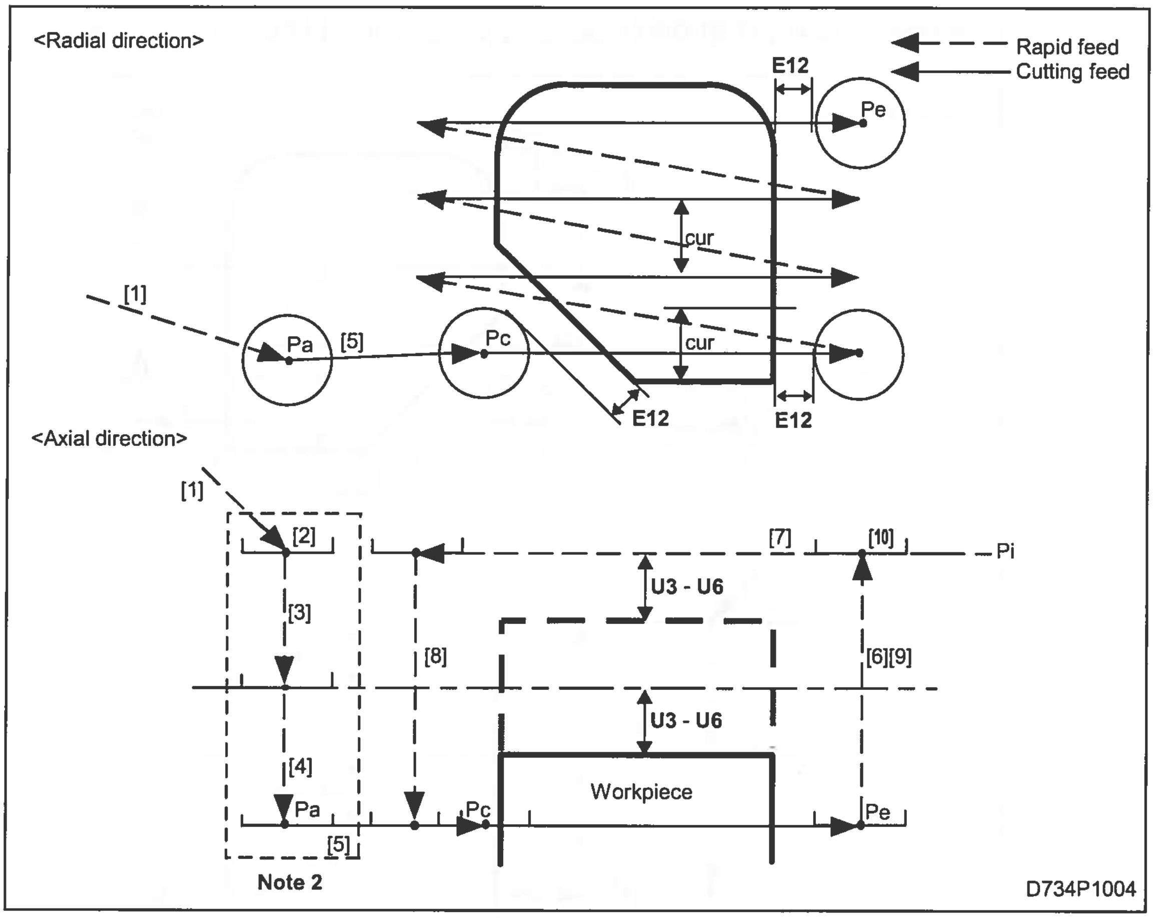
[4] The tool moves at a rapid feedrate to the face to be machined.

[5] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[7] The C-axis is set in the unclamping status at the initial point.

When the menu key **[1 UNI-DIR]** is selected for the article **TYPE** in the tool sequence



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

cur: Radial cutting depth to be determined by the data **DEP-R** in the tool sequence

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves at a rapid feedrate to the face to be machined.

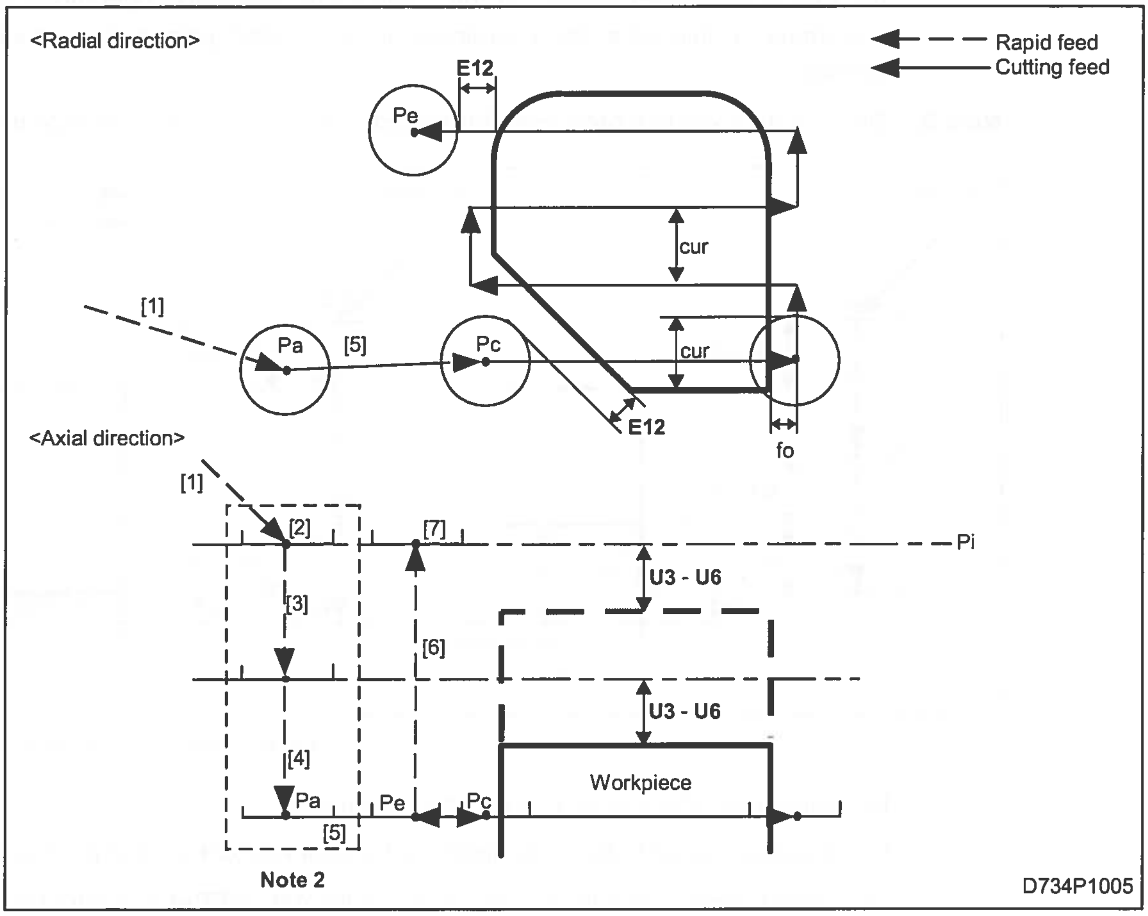
[5] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[6], [7] and [8] Upon completion of machining in one direction, the tool moves at a rapid feedrate to the initial point and to a subsequent cutting start point.

[9] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[10] The C-axis is set in the unclamping status at the initial point.

When the menu key **[1 BI-DIR SHORT]** is selected for the article **TYPE** in the tool sequence



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

cur: Radial cutting depth to be determined by the data **DEP-R** in the tool sequence

fo: Form-offsetting clearance

$$\text{fo} = \text{tool diameter} \times \frac{\textbf{E15}}{10}$$

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] The C-axis set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves at a rapid feedrate to the face to be machined.

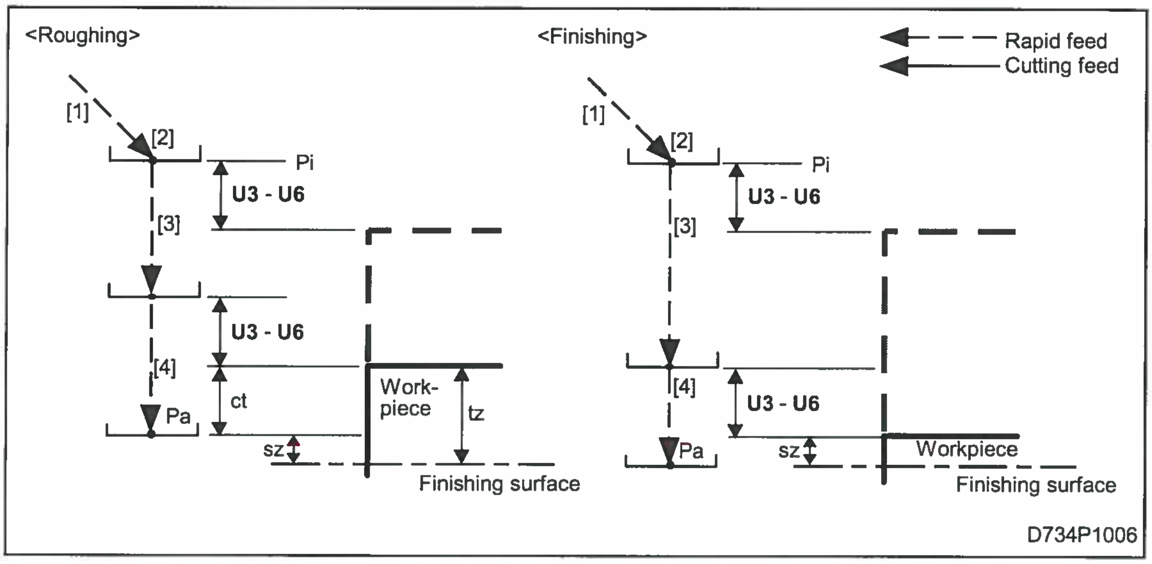
[5] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[7] The C-axis is set in the unclamping status at the initial point.

Note 1: When ? is displayed in the article **APRCH-1**, **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] to [4] are performed. In this case, the coordinate of cutting start point will be entered in these articles.

Note 2: Detail of the axial tool path. (See Subsection 3-6-5, "Precautions in face machining".)



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APPCH-1**, **-2** in the tool sequence

ct: Axial cutting stroke to be determined by the data **DEP-A** in the tool sequence

tz: Axial cutting allowance to be determined by the data **SRV-A** in a machining unit

sz: Axial finishing allowance **FIN-A** in a machining unit

2. End milling-top unit (TOP EMIL)

This unit is selected to machine a workpiece flatly on the machine by the use of an end mill.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A		BTM	WAL	FIN-A	FIN-R				
1	TOP EMIL	/Y	999.999	-999.999	99.999		9	◆	99.999	◆				
SNo.	TOOL	NOM-φ			APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	END MILL													
F2	END MILL								◆					

Remark 1: Data in unit represent a maximum input value.

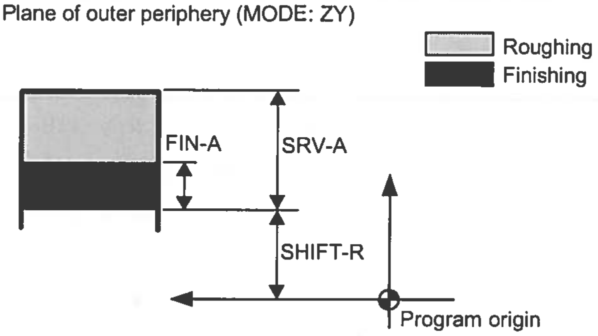
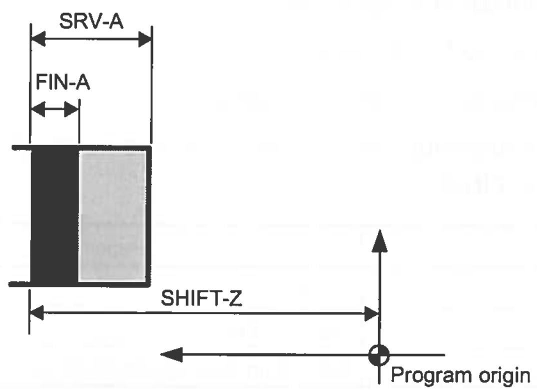
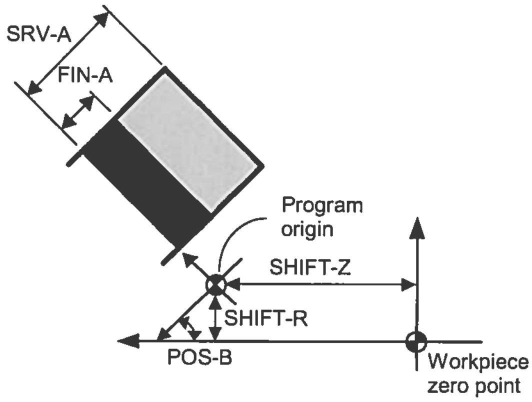
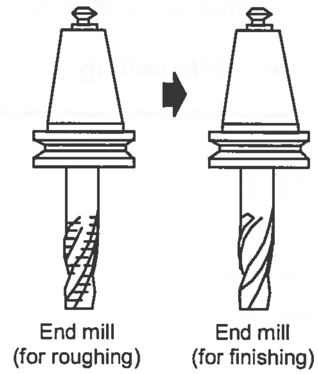
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A** and on **FIN-A**.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-A = 0
1	F1 (Finishing)	SRV-A ≤ FIN-A
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 3-6-4.

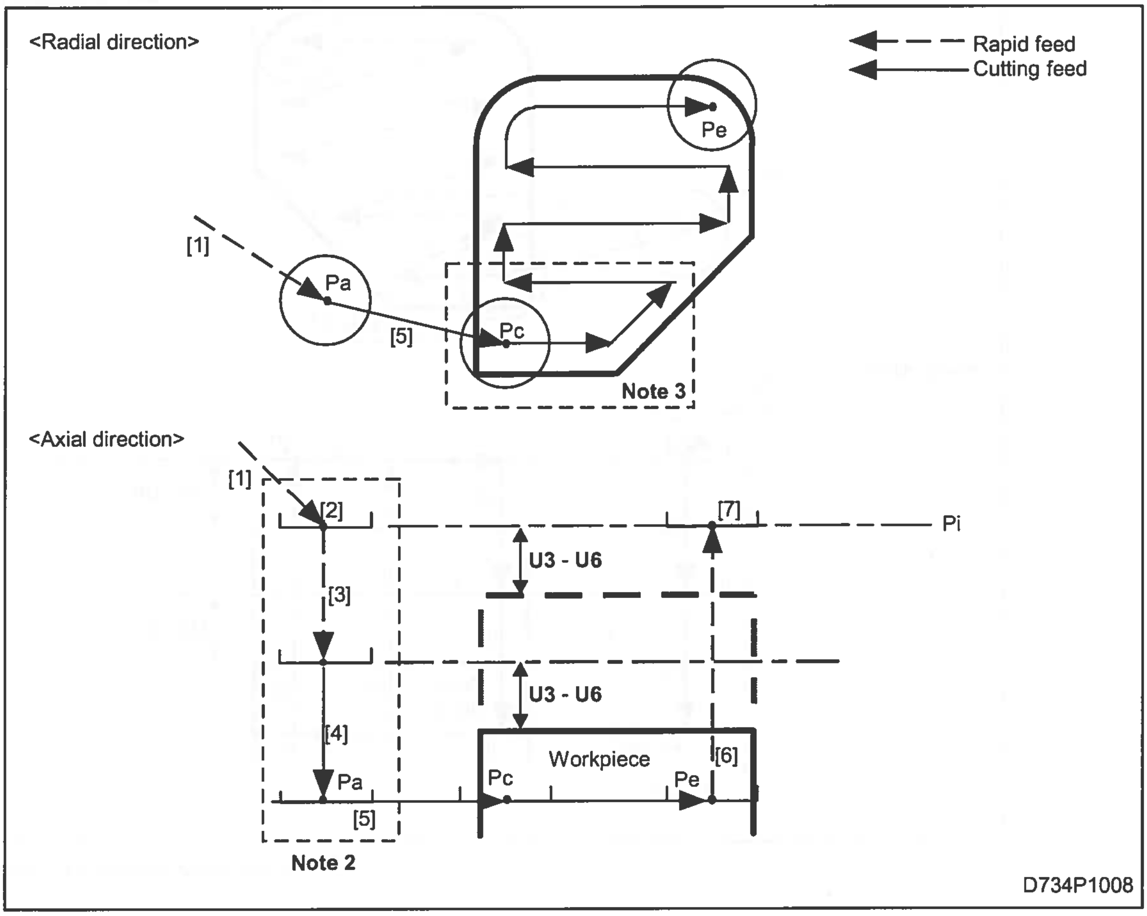
TOP EMIL unit	Tool sequence
Set data as follows according to the plane to be machined. a) Plane of outer periphery (MODE: ZY)  b) Edge (MODE: XY, $\boxed{XY}$)  c) Oblique plane (MODE: /Y, $\boxed{/Y}$)  D734P1007	 M3P255

BTM: A bottom roughness code is selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a bottom roughness code has been selected.

B. Tool path

When the menu key **[1 BI-DIR]** is selected for the article **TYPE** in the tool sequence



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] Set the C-axis in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

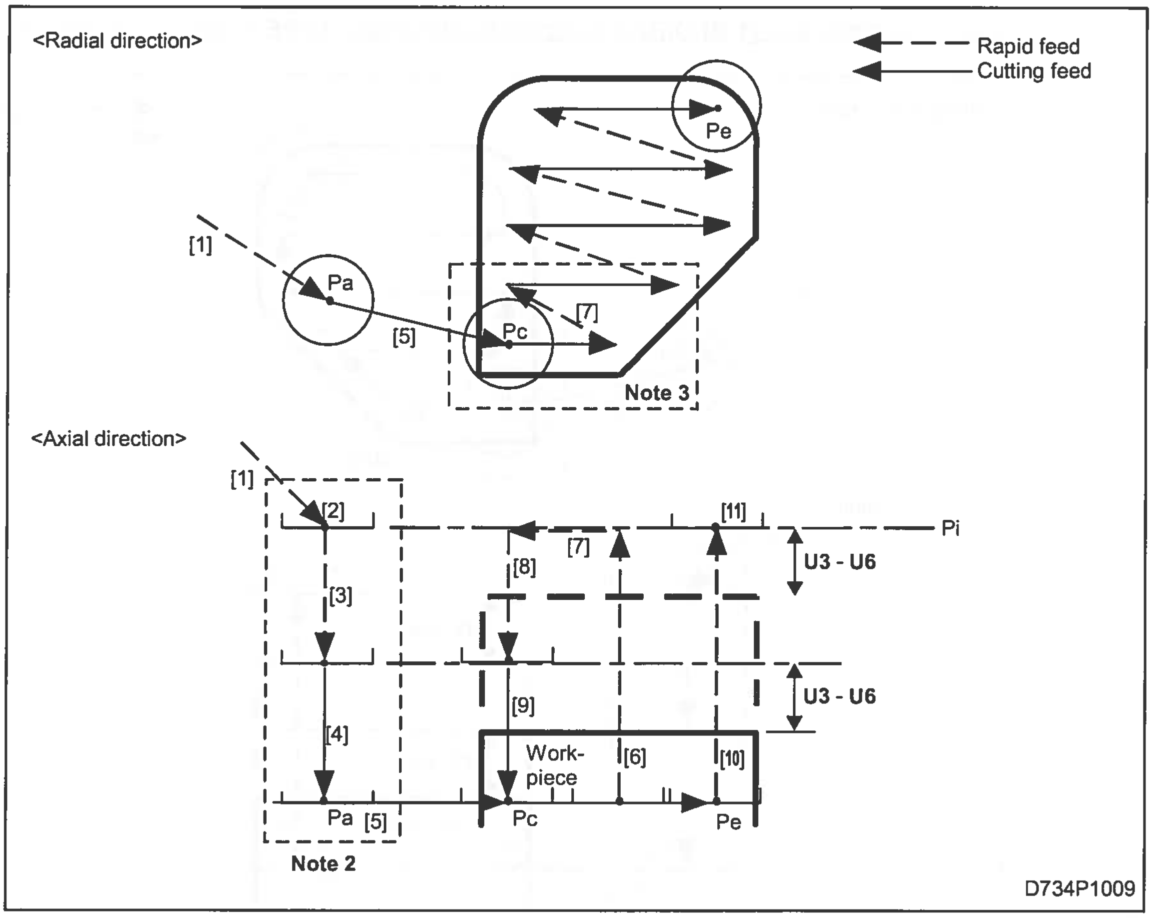
[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[7] The C-axis is set in the unclamping status at the initial point.

When the menu key **[1 UNI-DIR]** is selected for the article **TYPE** in the tool sequence



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon **AFD** in the tool sequence.)

[5] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[6], [7] and [8] Upon completion of machining in one direction, the tool moves at a rapid feedrate to the initial point.
Then, it moves at a rapid feedrate to the subsequent cutting start point specified by the parameter (**U3** to **U6**) above the next cutting start point.

[9] The tool moves at a cutting feedrate to the face to be machined and starts machining.

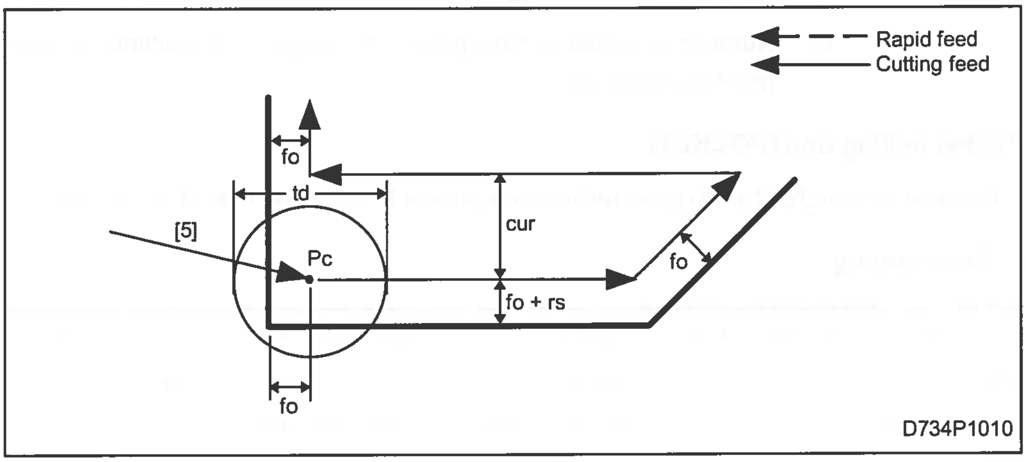
[10] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[11] The C-axis is set in the unclamping status at the initial point.

Note 1: When ? is displayed in the articles **APRCH-1**, **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operations [2] to [4] are performed. In this case, the coordinates of cutting start point will be entered in these articles.

Note 2: See Subsection 6-3-5 "Precautions in face machining".

Note 3: Detail description of tool path



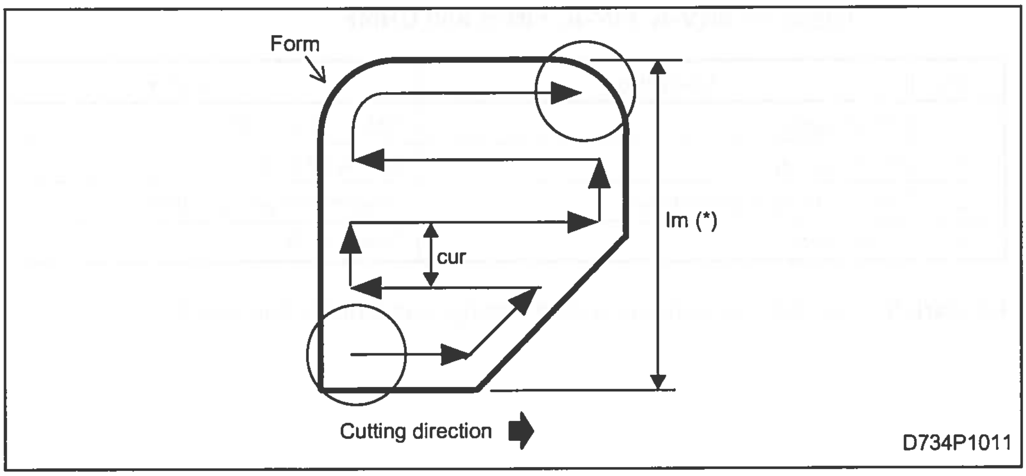
td: Diameter of a tool

fo: Form offset clearance dependent upon both td and parameter **E13**

$$fo = td \times \frac{\mathbf{E13}}{10}$$

rs: Form offset amount rectangular to the cutting direction

$$rs = \frac{td}{20}$$



cur: Radial cutting depth per cycle, obtainable as follows:

$$cur = \frac{lv}{n}$$

$$lv = lm\ (*) - 2 \times (fo + rs)$$

$$n = \frac{lv}{cr}$$

cr: Radial cutting depth (**DEP-R**) to be entered in the tool sequence

n: Number of radial cutting pass (an integer with fractions below the decimal point rounded up)

3. Pocket milling unit (POCKET)

This unit is selected to carry out milling of a pocket form by the use of an end mill.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A		BTM	WAL	FIN-A	FIN-R	INTER-R	CHMF		
1	POCKET	/Y	999.999	-999.999	99.999		9	9	99.999	◆	99.999	99.9		
SNo.	TOOL	NOM-φ			APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	END MILL													
F2	END MILL								◆					
3	CHAMFER								◆	◆				

Remark 1: Data in unit represent a maximum input value.

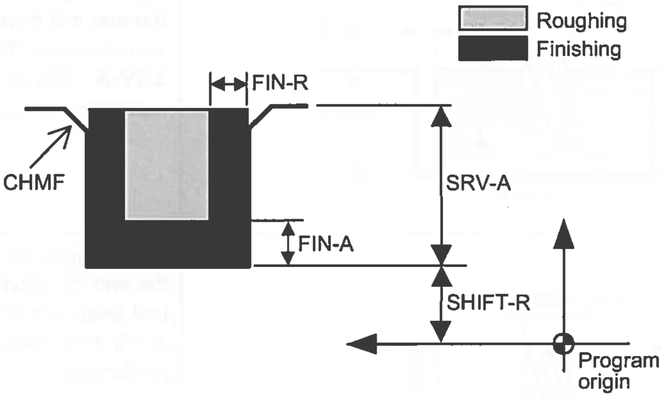
Remark 2: ◆: Data are not necessary to be set here.

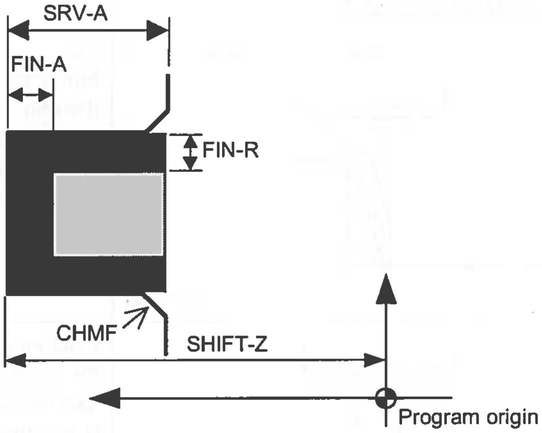
Remark 3: In this unit, end mills and a chamfering cutter are automatically developed.

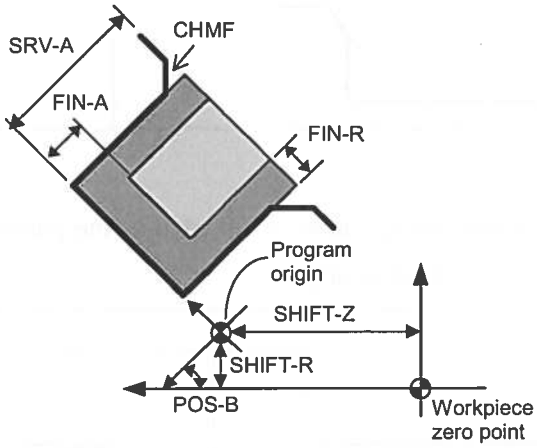
Remark 4: In the tool sequence, a maximum of up to three tools are automatically developed, based on **SRV-A**, **FIN-A**, **FIN-R** and **CHMF**.

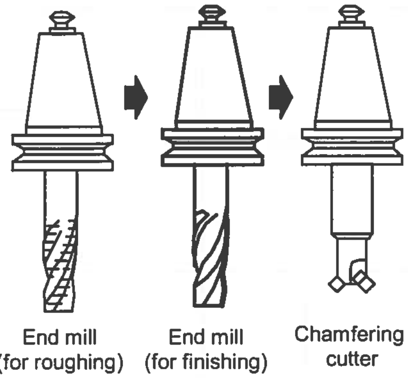
Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-A = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-A ≤ FIN-A
2	R1, F1 (Roughing/Finishing)	Other than those specified above
1	(Chamfering)	CHMF ≠ 0

Remark 5: For the tool sequence data setting, see Subsection 3-6-4.

<table>
<tr><th>POCKET unit</th><th>Tool sequence</th></tr>
<tr><td>
Set data as follows according to the plane to be machined.
a) Plane of outer periphery (MODE: ZY)


b) Edge (MODE: XY, XY)


c) Oblique plane (MODE: /Y, /Y)


D734P1013
</td><td>


M3P255
</td></tr>
</table>

BTM: A bottom roughness code is selected out of the menu.

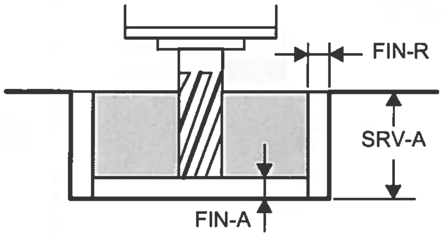
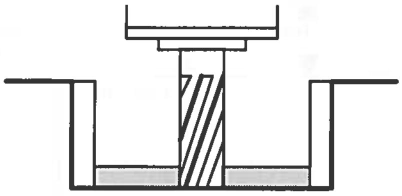
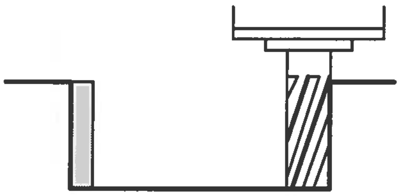
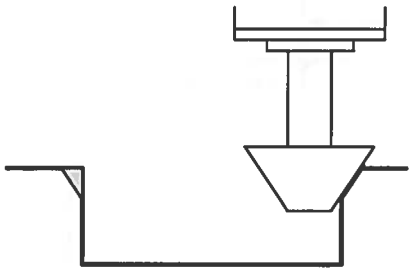
WAL: A wall roughness code is selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a bottom roughness code has been selected.

FIN-R: A radial finishing allowance is automatically established once a wall roughness code has been selected.

B. Machining sequence

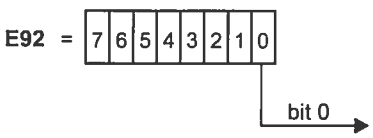
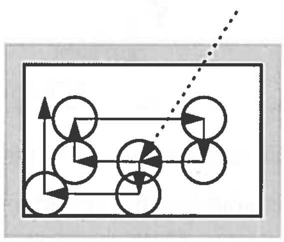
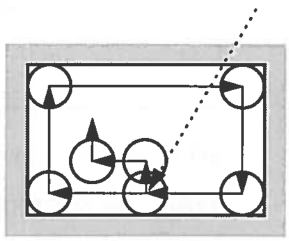
Pocket milling is performed in the following order.

Roughing			Machining is performed with the end mill developed in the tool sequence **R1**. With **SRV-A** = **FIN-A**, this machining is not performed.
Finishing	Bottom		Machining is performed with the end mill developed in the tool sequence **F1**. With **FIN-A** = 0, this machining is not performed.
	Wall	 M3P265	Machining is performed with the end mill developed in the tool sequence **F1**. With **FIN-R** = 0, this machining is not performed.
Chamfering			Machining is performed with the chamfering cutter in the tool sequence. With **CHMF** = 0, this machining is not performed.

C. Machining pattern

For roughing or bottom finishing, a machining pattern is selected by the parameter **E92**.

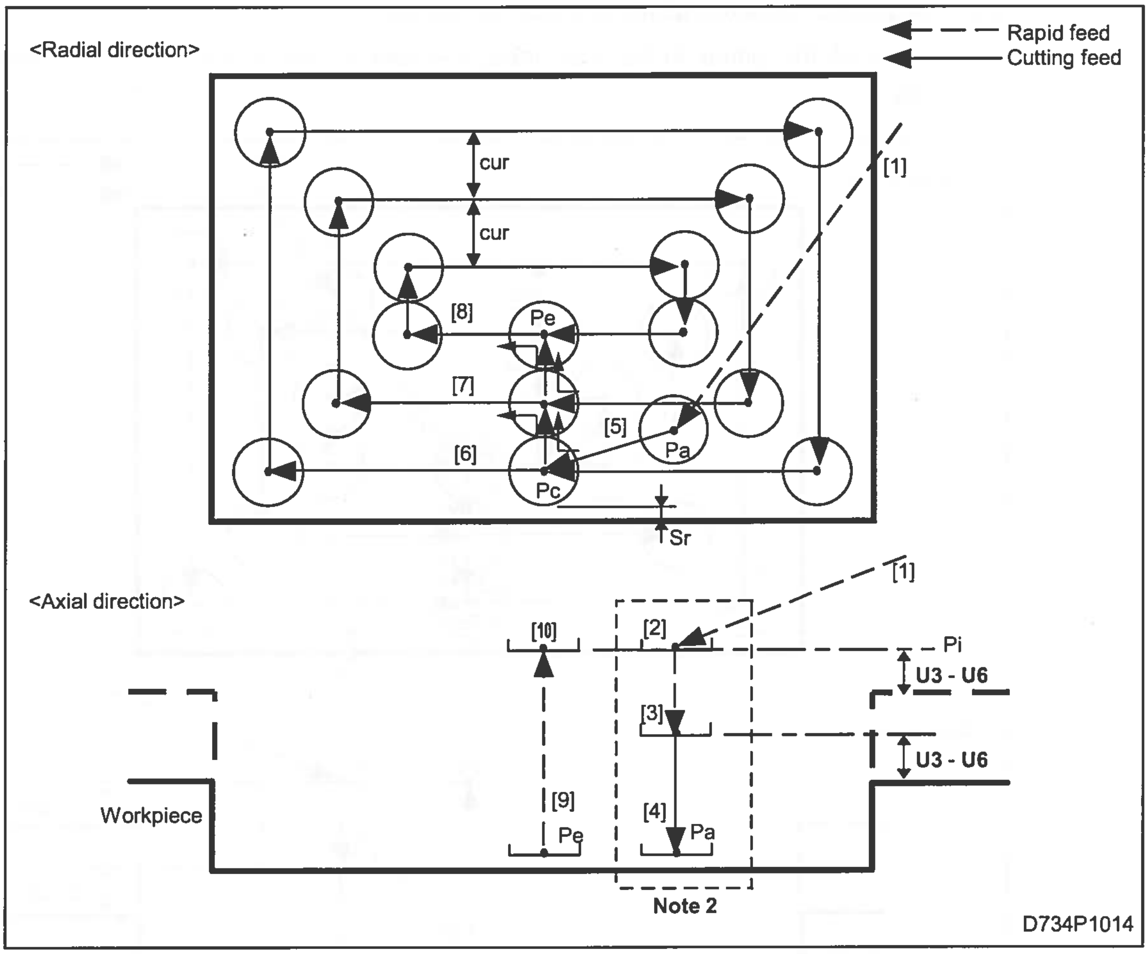
- Either 0 or 1 is entered in the related bit accordingly

	0	1
 M3P266	 Machining is performed from inside to outside.	 Machining is performed from outside to inside.

D. Tool path

Machining from outside (roughing or bottom finishing) **E92** bit 0 = 1

Irrespective of the plane to be machined, the tool is fed as follows in its radial and axial directions.



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point automatically established

cur: Radial cutting depth to be determined by the data **DEP-R** in the tool sequence

sr: Radial finishing allowance to be determined by the data in the machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point.

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The tool moves at a cutting feedrate to the cutting starting point.

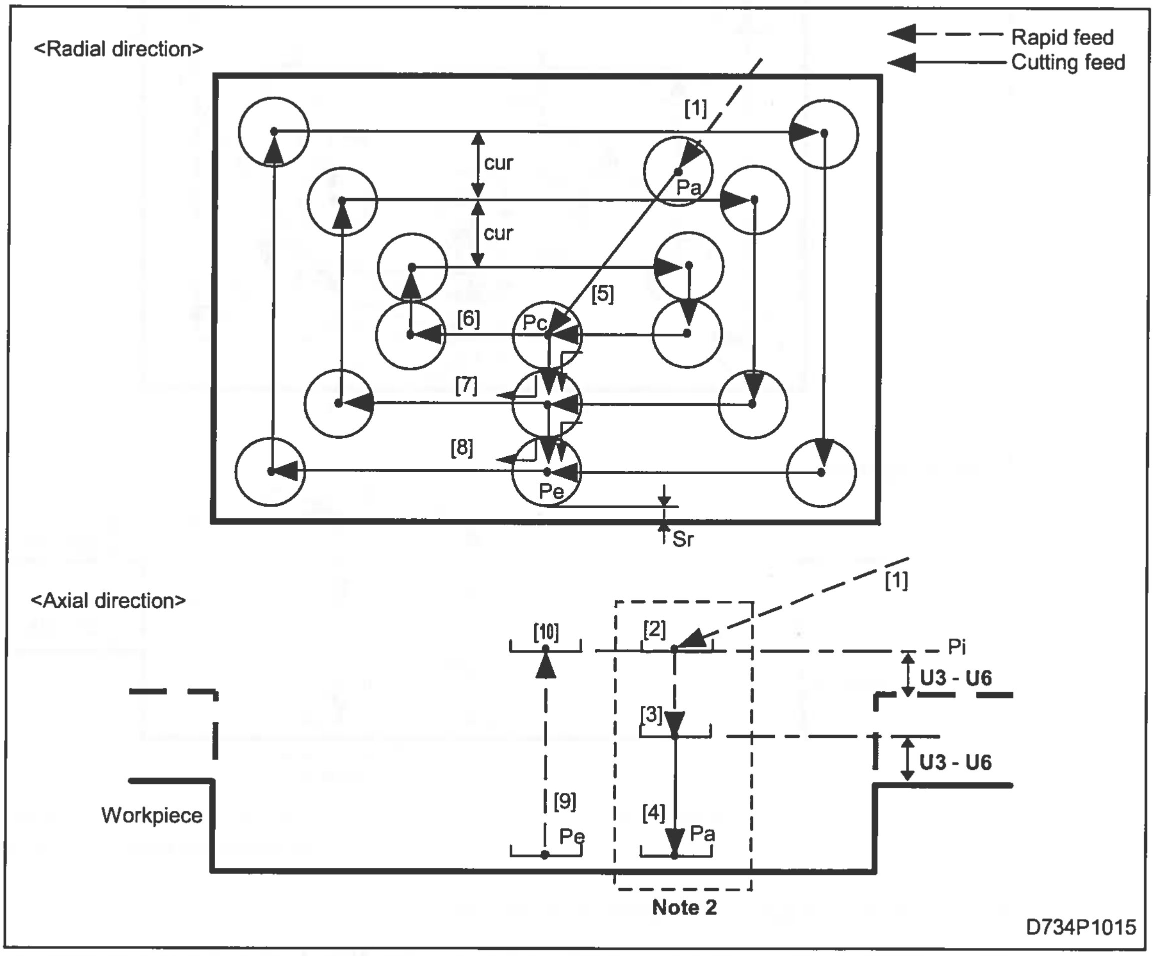
[6], [7] and [8] The tool machines on an around by around basis inwards.

[9] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[10] The C-axis is set in the unclamping status at the initial point.

Machining from inside (roughing or bottom finishing)

Irrespective of the plane to be machined, the tool is fed as follows in its radial and axial directions.



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pa: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point automatically established

cur: Radial cutting depth to be determined by the data **DEP-R** in the tool sequence

sr: Radial finishing allowance to be determined by the data in the machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The tool moves at a cutting feedrate to the cutting starting point.

[6], [7] and [8] The tool machines on an around by around basis outwards.

[9] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[10] The C-axis is set in the unclamping status at the initial point.

Note 1: When ? is displayed in the article **APRCH-1**, **-2** by pressing the menu key **[AUTO SET]**, the tool is positioned directly at the cutting start point and operation [2] to [4] are performed. In this case, the coordinates of cutting start point will be entered in these articles.

Note 2: See Subsection 3-6-5, "Precautions in face machining".

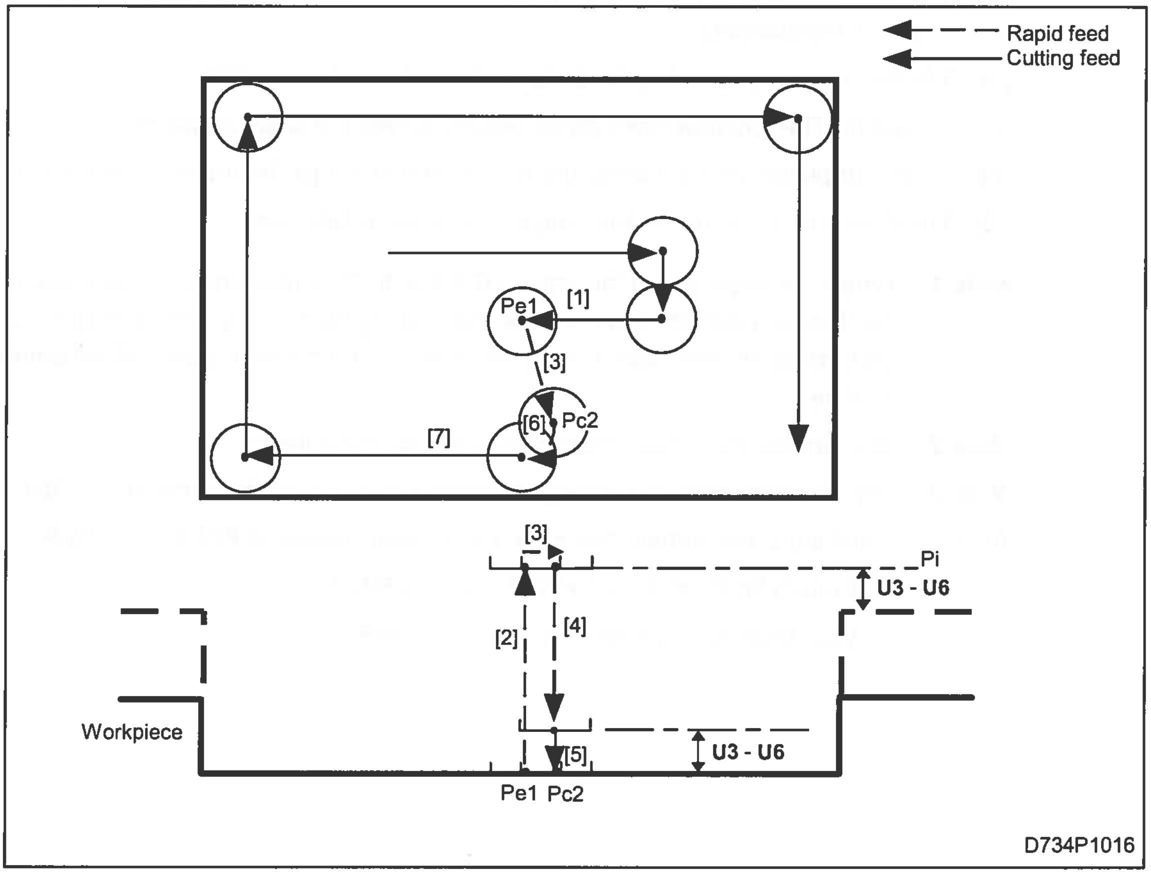
Note 3: The tool path for wall finishing is identical with that for finishing in the **LINE IN** unit.

Note 4: Finishing is performed, based on the entered values of **FIN-A** and **FIN-R**.

- Bottom finishing is performed, with 0 < **FIN-A**.
- Wall finishing is performed, with 0 < **FIN-R**.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data **APRCH-1**, **-2** of the tool sequence will be the approach point in bottom finishing. To transfer from bottom finishing to wall finishing, moreover, the tool moves at a rapid feedrate from the bottom-finishing escape point to the wall-finishing cutting start point as illustrated below.



Pe1: Escape point in bottom finishing

Pc2: Cutting start point in wall finishing

Pi: Initial point (face to be machined + clearance)

4. End milling-slot unit (SLOT)

This unit is selected to carry out slot machining by the use of an end mill.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SLOT-WID	BTM	WAL	FIN-A	FIN-R	PAT.			
1	SLOT	ZY	◆	-999.999	99.999	99.999	9	9	99.999	99.999	1			
SNo.	TOOL	NOM-φ			APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	END MILL									◆				
F2	END MILL								◆	◆				

Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A**, **SLOT-WID**, **FIN-A** and **FIN-R**.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-A = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-A ≤ FIN-A or SLOT-WID ≤ (2 × FIN-R)
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 3-6-4.

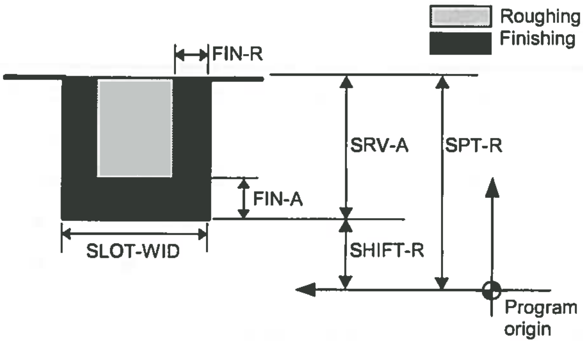
Remark 6: If **ZC** or **XC** is selected for **MODE** (plane to be machined), spreading can not be executed in this unit.

Remark 7: If **ZC** or **XC** is selected for **MODE** (plane to be machined), ◆ is displayed at **FIN-R** (data setting not required).

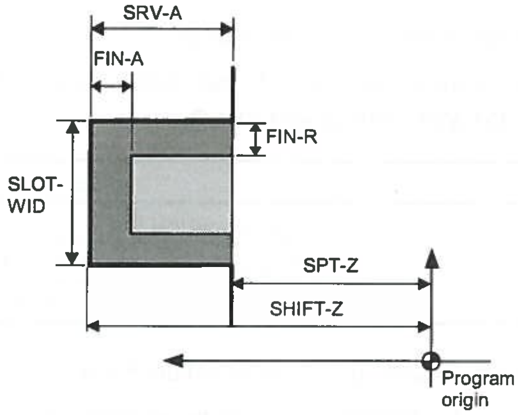
SLOT unit

Set data as follows according to the plane to be machined:

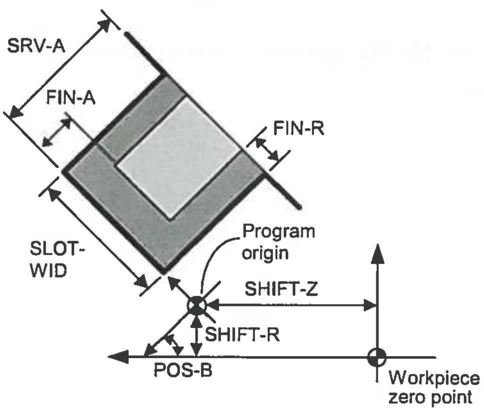
a) Outer periphery (MODE: ZC, ZY)



b) Edge (MODE: XC, XY, XY)

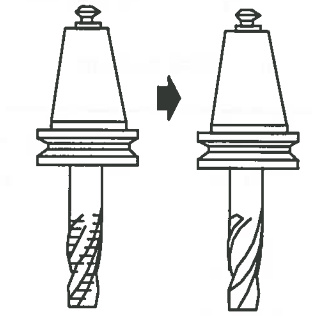


c) Oblique plane (MODE: /Y, /Y)



D734P1012

Tool sequence



M3P255

BTM: A buttom roughness code is selected out of the menu.

WAL: A wall roughness code is selected out of the menu.

FIN-A: An axial finishing allowance is automatically established once a bottom roughness code has been selected.

FIN-R: A radial finishing allowance is automatically established once a wall roughness code has been selected.

Note: Set 0 or 1 in **PAT**. of unit data to select one of the following two machining patterns.

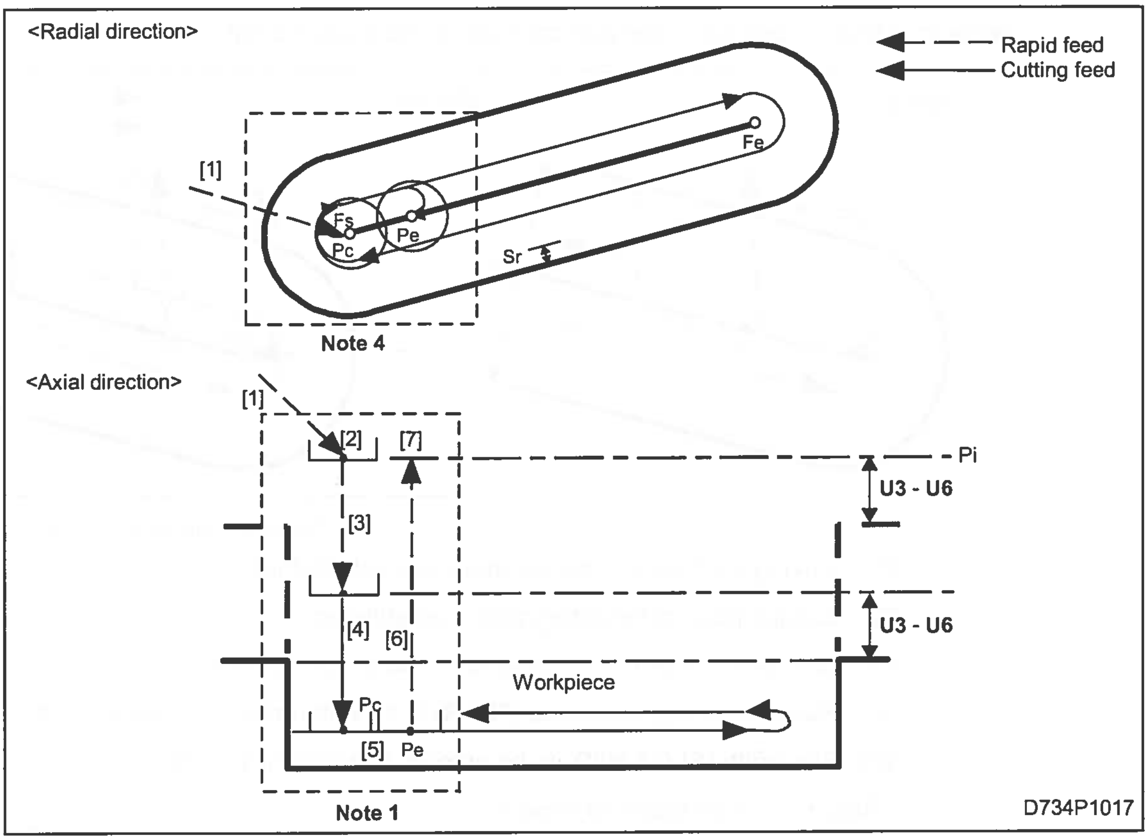
Setting	Description
0	A linear shape can be specified. Set the starting and ending point lines for the shape pattern. The spreading cycle will occur if: **SLOT-WID** > Tool diameter This setting is used if **ZC**, **XC** or **ZY** is selected for **MODE**.
1	Arbitrary shape can be specified. The spreading cycle will occur if:(Tool diameter x 2) ≥ **SLOT WID** > Tool diameter This setting is used if **ZY**, **XY**, **/Y**, **[XY]**, **[/Y]** is selected for **MODE**.

Note: With **MODE** = **ZY** and **PAT**. = 0, the axial finishing allowance (**FIN-A**) is also used for radial finishing (on the wall).

B. Tool path

<u>With tool radius + **FIN-R** < **SLOT-WID**/2 < tool diameter + **FIN-R**</u>

Spreading occurs only during Y-axial machining. Irrespective of the plane to be machined at this time, the tool is fed as follows in its radial and axial directions:



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pc: Cutting start point to be automatically established

Pe: Escape point automatically established

Fs: Start point of form to be entered in the shape sequence

Fe: End point of form to be entered in the shape sequence

sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point (cutting start point).

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The slot is machined at a cutting feedrate.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial the point.

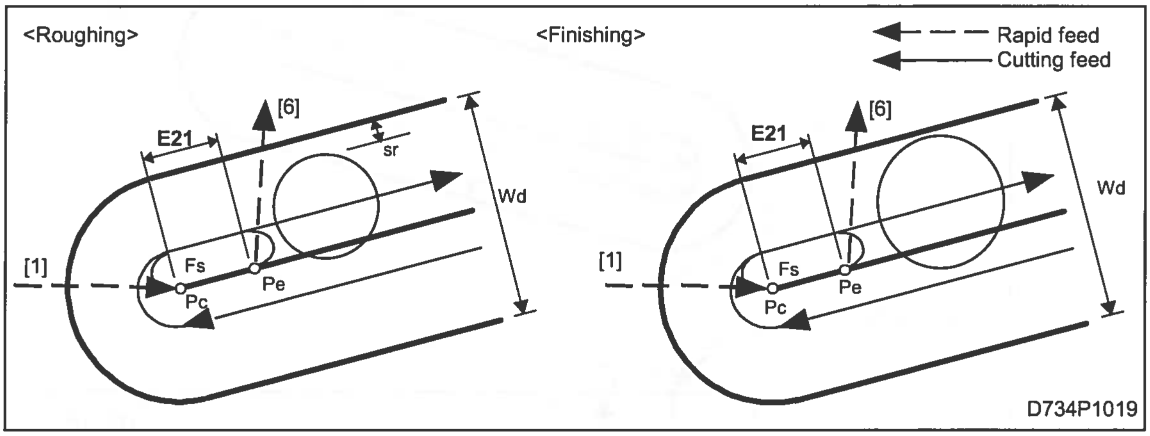
[7] The C-axis is unclamped at the initial point.

Note 1: See Subsection 3-6-5, "Precautions in face machining".

Note 2: The feedrate on tool path [4] is dependent upon the data **AFD** (axial feed) in the tool sequence.

Note 3: Alarm **608 ILLEGAL UNIT DATA** occurs if "Tool radius + **FIN-R** > **SLOT-WID**/2" or if "Tool diameter + **FIN-R** > **SLOT WID**/2".

Note 4: Detail of tool path near approach point and escape point



The bold codes represent parameter addresses.

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Fs: Start point of form to be entered in the shape sequence

sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

Wd: Slot width (**SLOT-WID**) to be entered in a machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point (cutting start point).

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

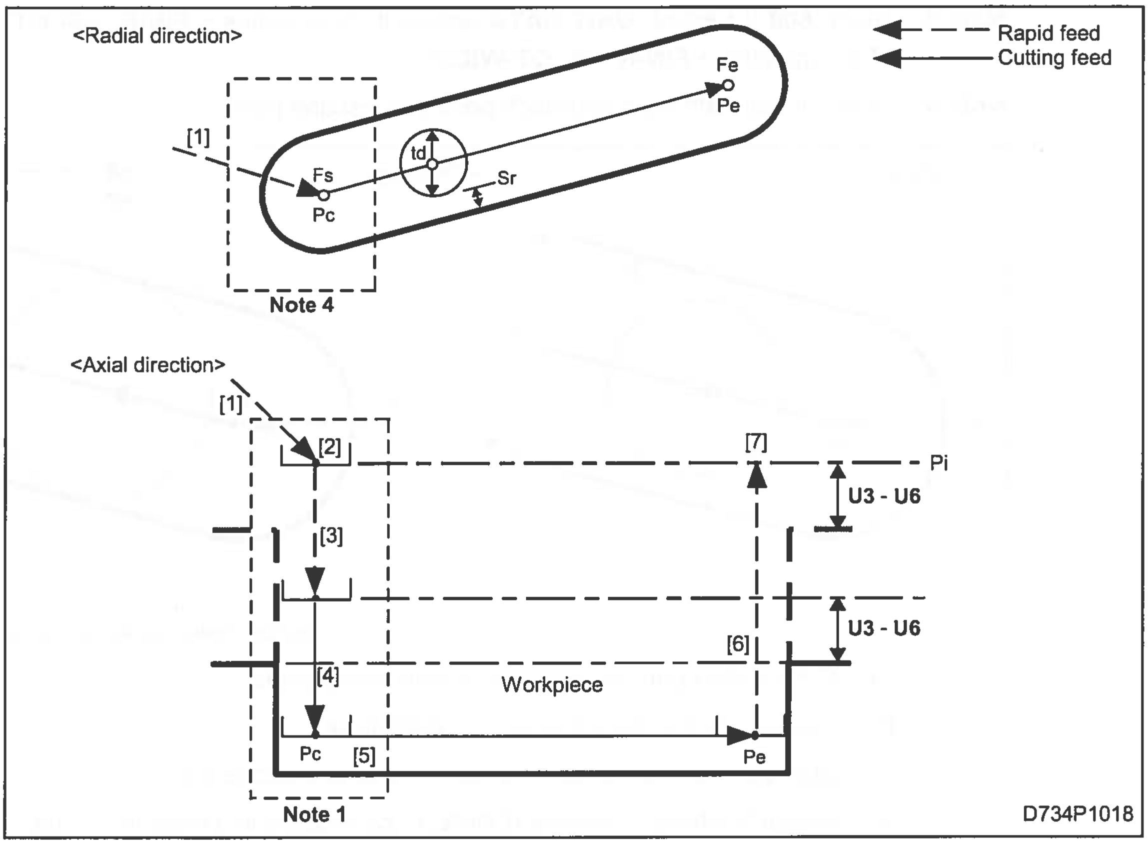
[5] The slot is machined at a cutting feedrate.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[7] The C-axis is unclamped at the initial point.

With tool radius + **FIN-R** = **SLOT-WID/2**

Irrespective of the plane to be machined, the tool is fed as follows in its radial and axial directions.



The bold codes represent parameter addresses.

Pi: Initial point (face to be machined + clearance)

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Fs: Start point of form to be entered in the shape sequence

Fe: End point of form to be entered in the shape sequence

td: Tool diameter to be registered in the **TOOL DATA** display

sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point (cutting start point).

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The slot is machined at a cutting feedrate.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

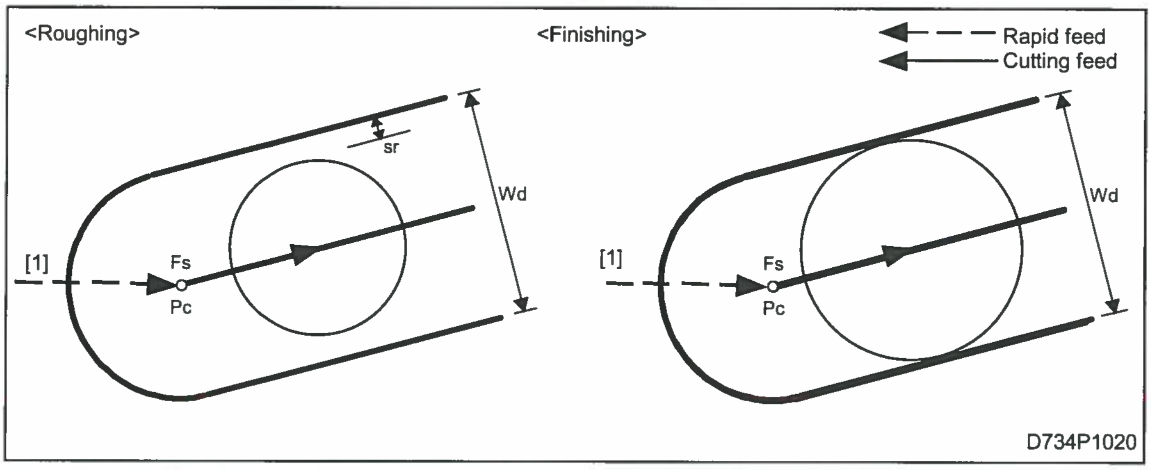
[7] The C-axis is unclamped at the initial point.

Note 1: See Subsection 3-6-5, "Precautions in face machining".

Note 2: The feedrate on tool path [4] is dependent upon the data **AFD** (axial feed) in the tool sequence.

Note 3: Alarm **608 ILLEGAL UNIT DATA** occurs if "Tool radius + **FIN-R** > **SLOT-WID**/2" or if "Tool diameter + **FIN-R** > **SLOT-WID**/2".

Note 4: Detail of tool path near approach point and escape point



The bold codes represent parameter addresses.

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Fs: Start point of form to be entered in the shape sequence

sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

Wd: Slot width (**SLOT-WID**) to be entered in a machining unit

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to the approach point (cutting start point).

[2] The C-axis is set in the clamping status.

[3] The tool moves at a rapid feedrate to the position entered by the parameter (**U3** to **U6**).

[4] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[5] The slot is machined at a cutting feedrate.

[6] Upon completion of machining, the tool moves at a rapid feedrate to the initial point.

[7] The C-axis is unclamped at the initial point.

3-6-4 Tool sequence data of the face machining unit

In the tool sequence a tool name only is automatically selected once a machining unit has been entered. Other data should be entered by use of menu keys or numeric keys according to a form of the workpiece to be machined or to the procedure for machining.

SNo.	TOOL	NOM-ϕ	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL							◆					
	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑
	1	2	3	11	4	4	5	6	7	8	9	9	10	10

◆: Not necessary to be set here.

1. **Tool designation: TOOL**

The name of a tool can be selected by the use of menu keys.

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL						

In the face milling unit, either one of the **[ENDMILL]**, **[FACEMILL]** and **[BALL ENDMILL]** menu items are selectable. In other units, either the **[ENDMILL]** or **[BALL ENDMILL]** menu item is selectable.

2. **Nominal tool diameter: NOM-ϕ**

An approximate tool diameter is entered. A nominal diameter is the data to identify by diameter those tools which are of identical type (having an identical name).

3. **Tool identification code**

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal diameter.

A	B	C	D	E	F	G	H		>>>

4. **Approach point: APRCH-1, APRCH-2**

Enter coordinates of the position at which a tool is to cut in axially.

With **[AUTO SET]** menu key selected, ? is displayed. Check the tool path and ? will automatically change over to the coordinates of a cutting start point. (Refer to tool path by each unit.)

5. Machining method: TYPE

Machining method differs according to the units as follows.

A. Face milling unit

A tool path pattern is selected out of three: "Bidirectional cut", "Unidirectional cut", and "Bidirecional short cut". In each pattern, moreover, it is possible to select whether machining is performed in parallel with the first axis or the second axis.

1 BI-DIR	2 BI-DIR	1 UNI-DIR	2 UNI-DIR	1 BI-DIR SHORT	2 BI-DIR SHORT				

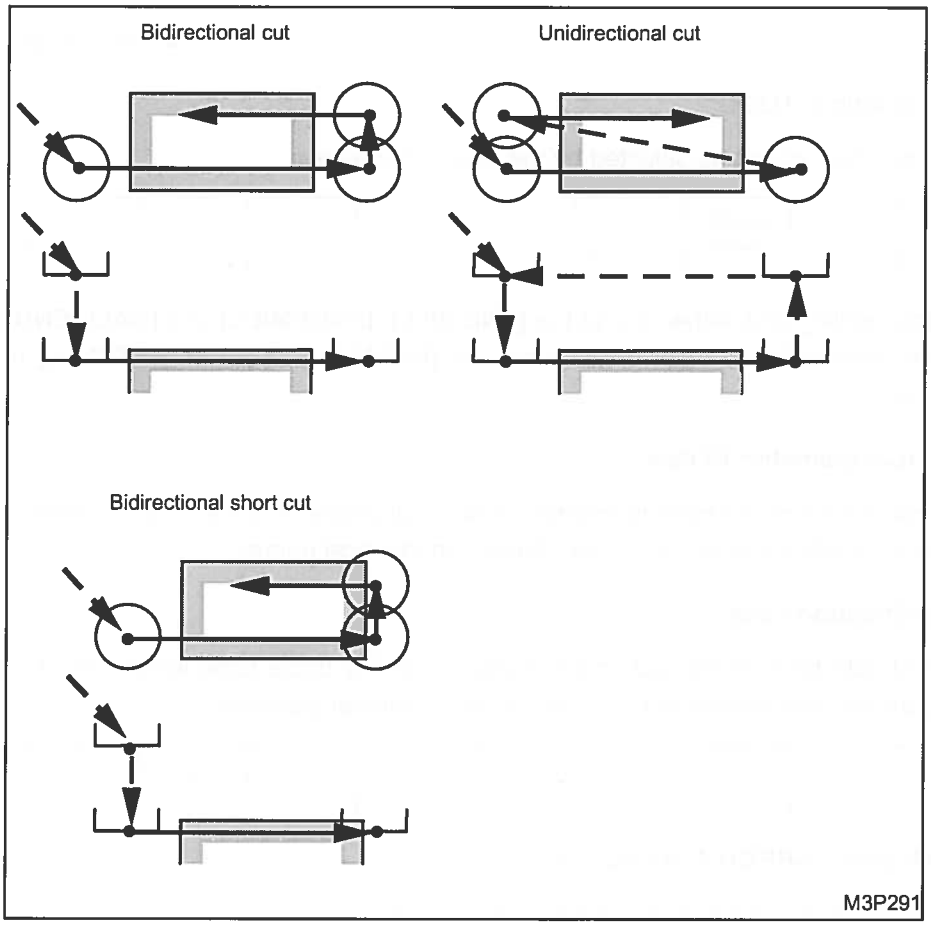
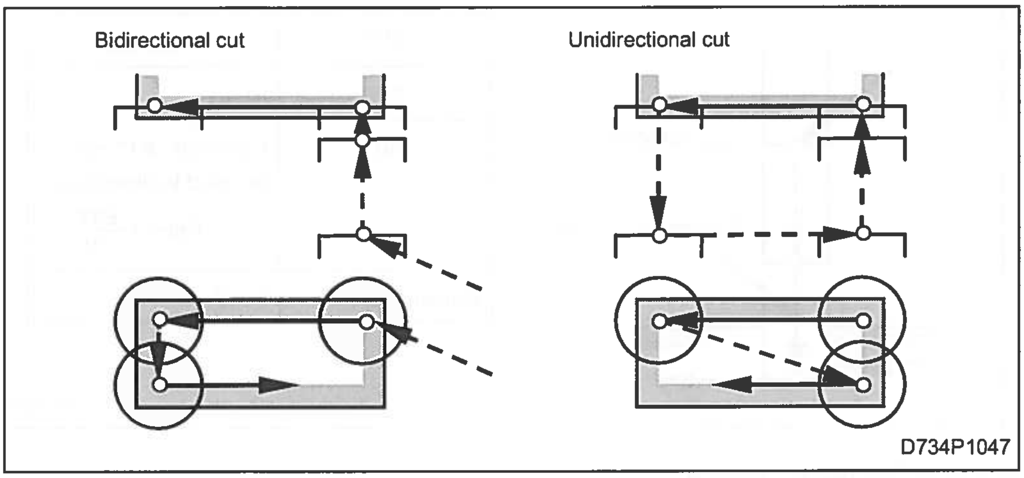


Fig. 3-16 Tool path patterns

B. End milling-top unit

A tool path pattern is selectable out of “Bidirectional cut” or “Unidirectional cut”. In each pattern, moreover, it is possible to select whether machining is performed in parallel with the first axis or with the second axis.

1 BI-DIR	2 BI-DIR	1 UNI-DIR	2 UNI-DIR						

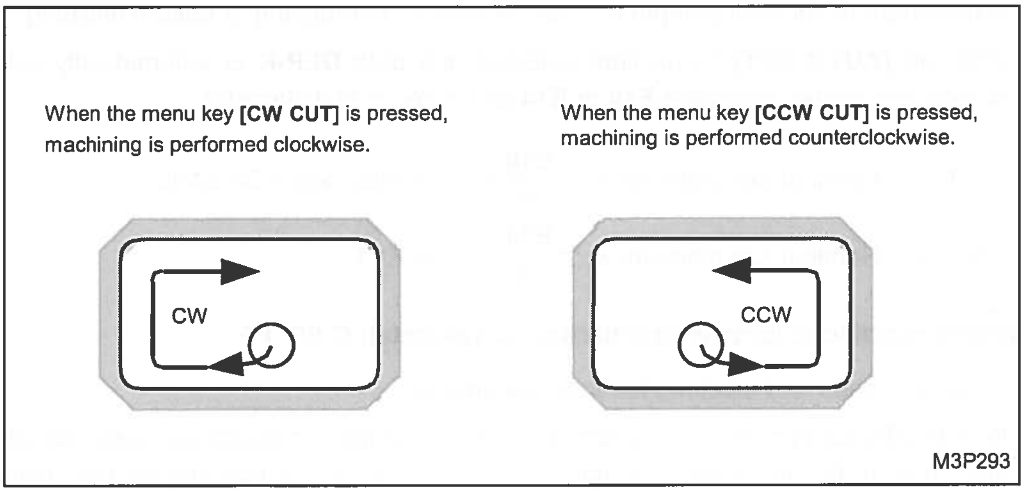


D734P1047

C. Other units

A machining (turning) direction is selected by the use of menu keys.

CW CUT	CCW CUT								

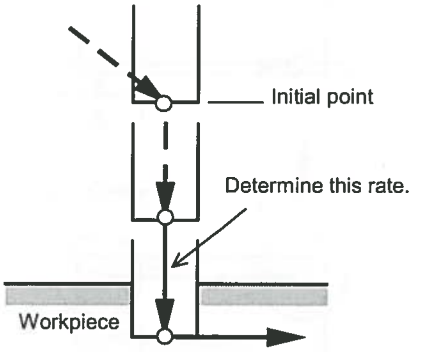


M3P293

6. **Axial feedrate: AFD**

The axial feedrate should be entered in a multiple of feedrate. It is also possible, to select rapid feed (G00) or cutting feed (G01) by the use of menu keys.

CUT G01	RAPID G00								



AFD	Feedrate
G00	Rapid feed
G01	Parameter **E17** may be used to determine: Feed × $\frac{E17}{10}$
Numeric value (α)	Feed × α

M3P294

7. **Axial cutting depth: DEP-A**

In roughing, a maximum axial cutting stroke in one pass is entered. With **[AUTO SET]** menu key pressed, an actual axial cutting depth is arithmetically obtained from the data **DEP-A**, **SRV-A** and **FIN-A** in the machining unit. (For calculation formula, see Subsection 3-6-5, “Precautions in face machining”.)

8. **Radial cutting depth: DEP-R**

A maximum radial cutting depth per pass is entered in roughing or bottom finishing.

With the **[AUTO SET]** menu item selected, the data **DEP-R** is automatically calculated and determined by the parameter **E10** or **E14** and nominal tool diameter.

DEP-R = Nominal tool diameter x $\frac{\mathbf{E10}}{10}$: FCE MILL and TOP EMIL

DEP-R = Nominal tool diameter x $\frac{\mathbf{E14}}{10}$: POCKET

9. **Cutting conditions (circumferential speed, feedrate): C-SP, FR**

A spindle speed and a cutting feedrate are entered.

With **[AUTO SET]** menu item selected, optimum cutting conditions are automatically calculated and entered, based on the materials of both workpiece and tool and on the cutting depth. (A circumferential speed is given in meters per minute and a feedrate in millimeters per revolution.)

10. **M-codes: M**

Enter an M-code which will be output immediately after mounting the tool onto the spindle in the ATC mode. A maximum of up to two codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu. (See 3-19-2 “M-code table”.)

11. Retraction position of the lower turret: #

For a machine having upper and lower turrets, it is possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.

The following menu is displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: For details refer to Chapter 4 "LOWER-TURRET CONTROL FUNCTIONS".

3-6-5 Precautions in face machining

1. Tool path during rough-machining in the case of "axial machining allowance (SRV-A) > axial cutting depth (DEP-A)"

Cutting is performed at several passes. The tool path is determined by the parameter related to the following two factors, but these factors may not be all available in certain machining unit:

A. Cutting start position in the axial direction

B. Type of routing through approach point

[Basic tool path]

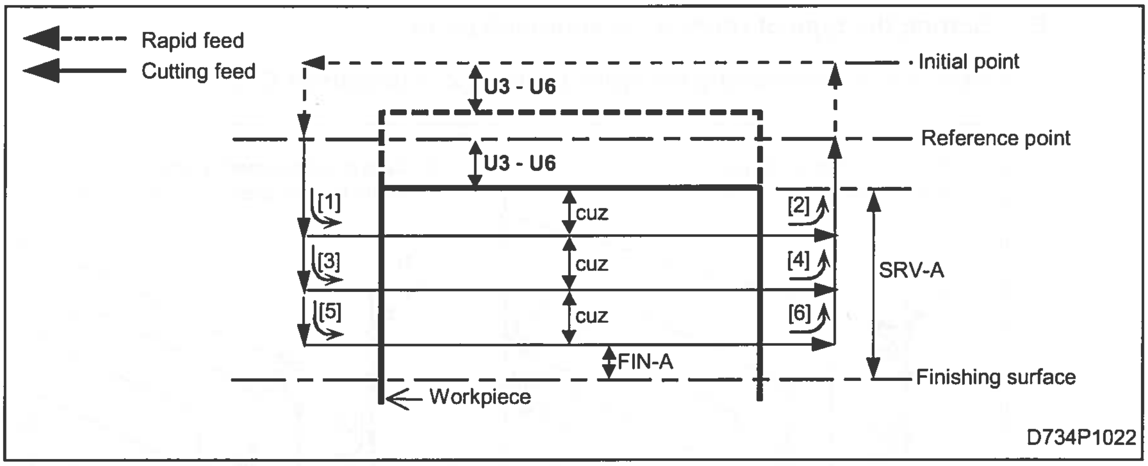


Fig. 3-17 Basic tool path

cuz: Axial cutting depth per pass

Calculation of cuz:

$$cuz = \frac{tz - sz}{n}$$

$$n = \frac{tz - sz}{cz}$$

tz: Axial machining allowance **SRV-A** to be entered in the machining unit

sz: Axial finishing allowance **FIN-A** to be entered in the machining unit

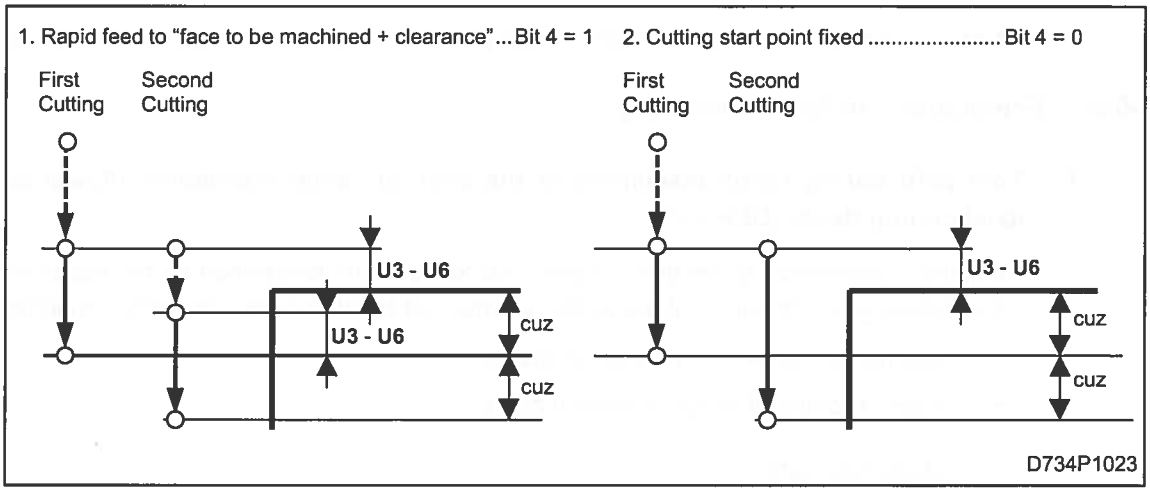
cz: Axial cutting depth **DEP-A** to be entered in the tool sequence

n: Number of passes in the axial direction
(Integer obtained by rounding up the decimal fraction)

A. Setting the cutting start position in the axial direction

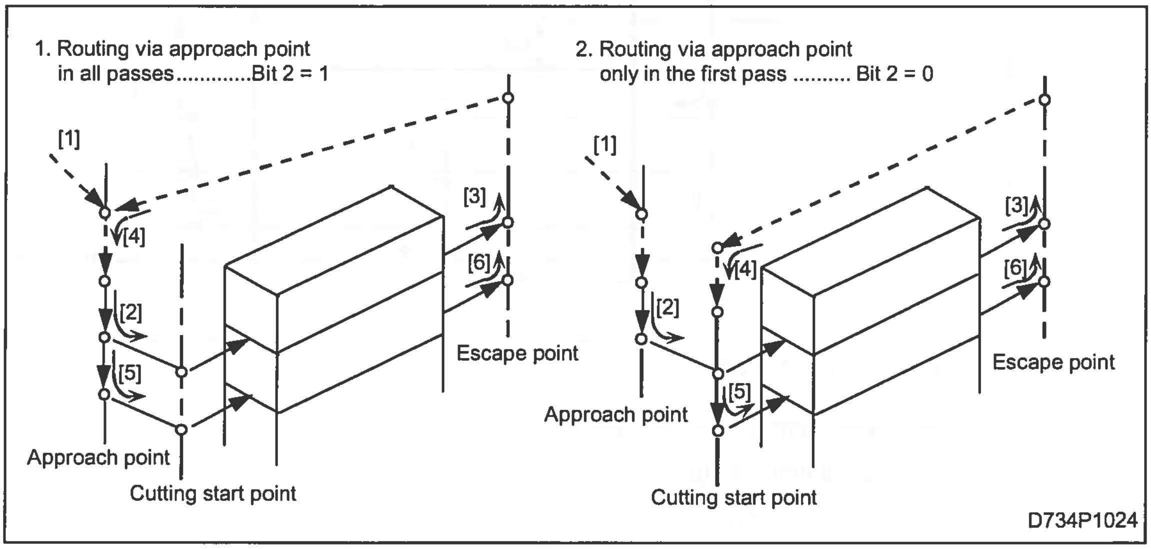
Select the cutting start position using the following parameter:

End milling-top unit (**TOP EMIL**): **E97** bit 4
Pocket milling unit (**POCKET**): **E92** bit 4
End milling-slot unit (**SLOT**): **E96** bit 4



B. Setting the type of routing via approach point

Select one of the following two types using bit 2 of parameter **E96**:

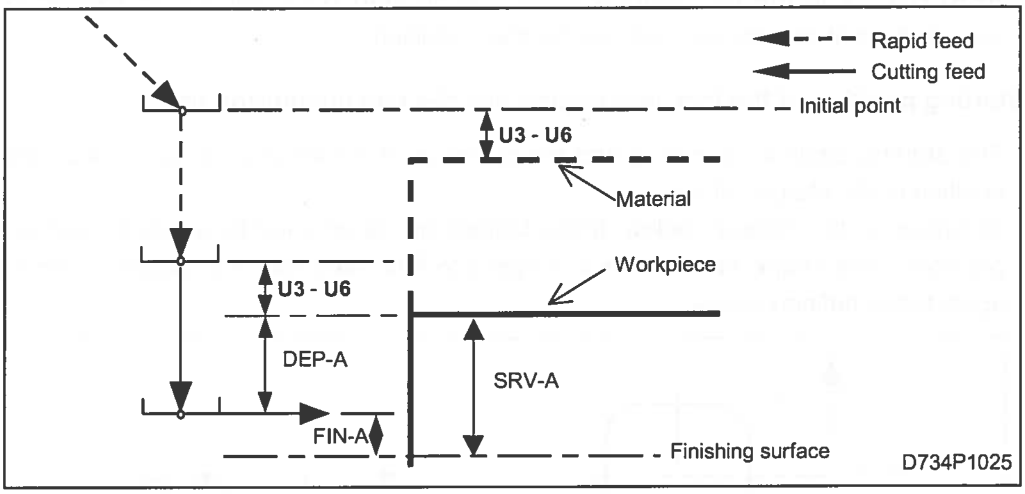


Note 1: B can only be used in the end milling-slot (**SLOT**) unit.

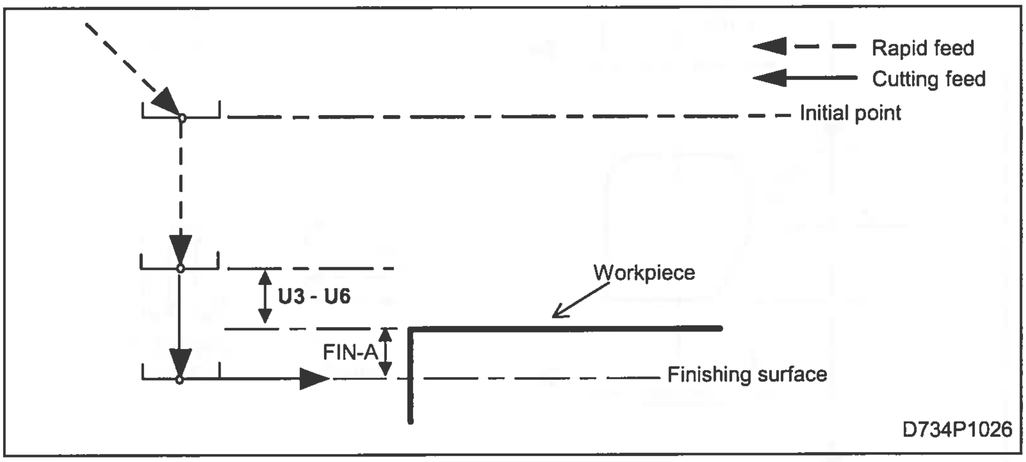
Note 2: The tool path shown at basic tool path above is selected automatically for face machining unit that is not assigned by these parameters.

2. Detail tool path of an axial cutting

- Roughing



- Finishing



The bold codes represent parameter addresses.

Note 1: The starting allowance of axial cutting, specified by parameter **U3** to **U6**, will become equal to parameter **E7** if the following two states occur at the same time:

- The designated parameter for the intended unit is set to 1.

  End milling-top: **E97**, bit 2
  Pocket milling: **E92**, bit 2
  End milling-slot: **E96**, bit 1

- A pre-machining tool is included in that tool sequence.

Note 2: The starting allowance of cutting in radial direction, specified by parameter **U10**, will become equal to parameter **E5** if the following two states occur at the same time:

- The designaed parameter for the intended unit is set to 1.

  Pocket milling: **E92**, bit 2

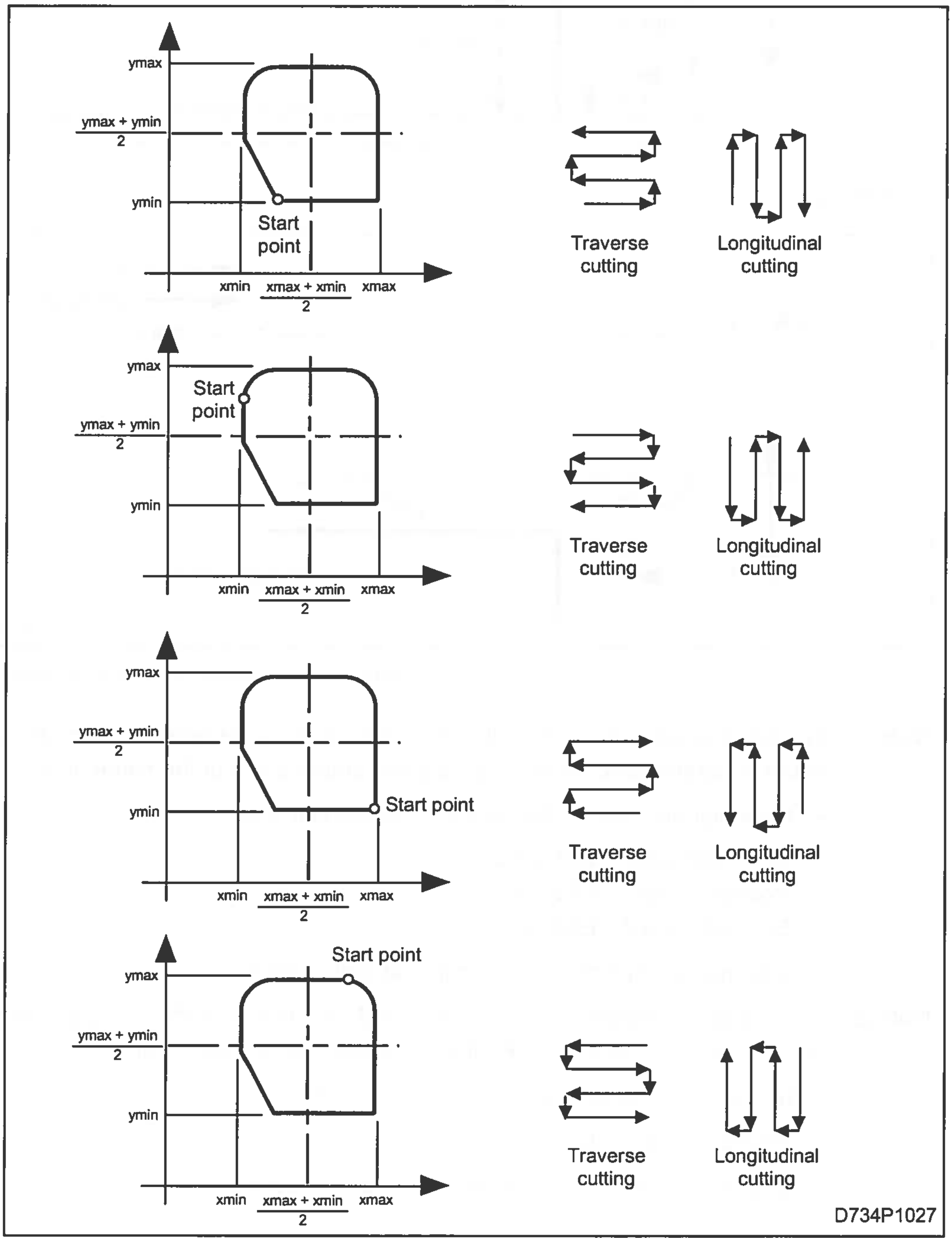
- A pre-machining tool is included in that tool sequence.

3. **Automatic determanation of the approach point**

If shape data, tool data or parameters are modified after the automatic determination of the data **APRCH-1**, **-2** in the tool sequence, the cutting start point will not be located on the same approach point and the tool path will be also modified.

4. **Starting position of the face milling unit and the end milling-top unit**

The starting position of cutting and the direction of machining are determined by the starting position of the shape pattern.
As shown in the diagram below, these factors are determined by in what quadrant the starting position of the shape exists when it is split into four sections. This diagram, however, may not apply to the defined shape.

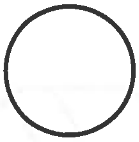
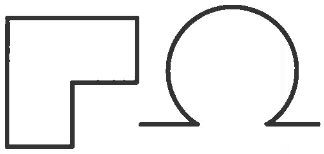


3-6-6 Definitions of forms in line machining and face machining units

After the data in the machining unit and the tool sequence have been entered, enter the data related to the machining form and dimensions in the shape sequence.

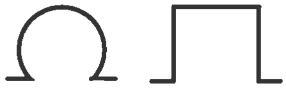
1. Definitions of forms

In line machining and face machining units one of the following three patterns can be selected.

Fixed form		Arbitrary form
SQUARE	CIRCLE	ARBITRY
		 M3P303

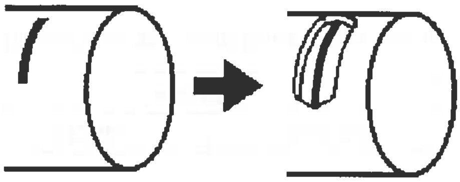
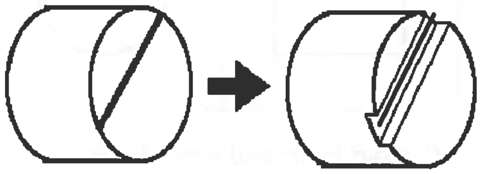
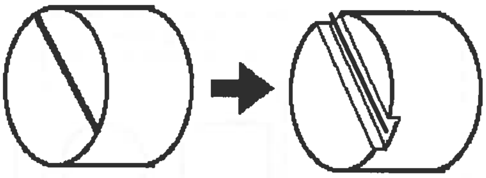
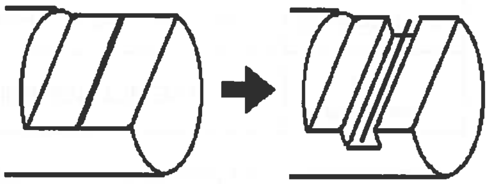
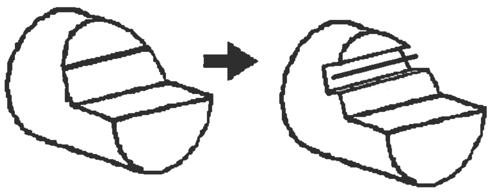
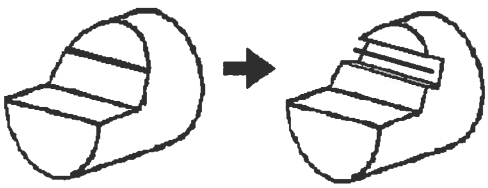
A. Closed form and open form

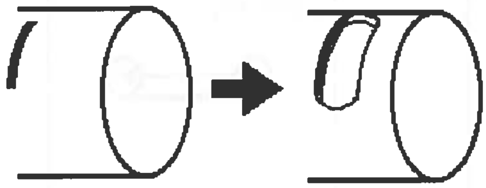
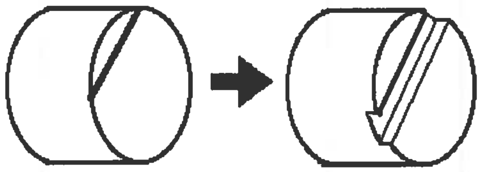
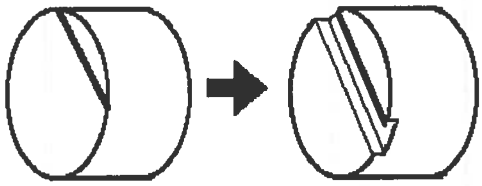
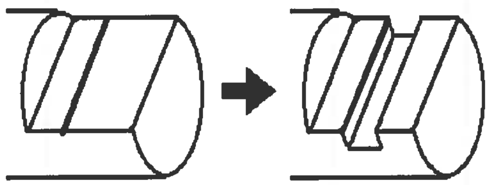
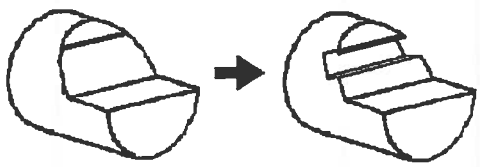
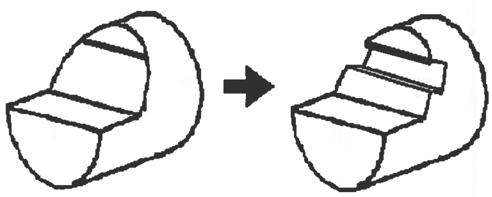
Depending on the machining units, machining form can be divided into the following two types:

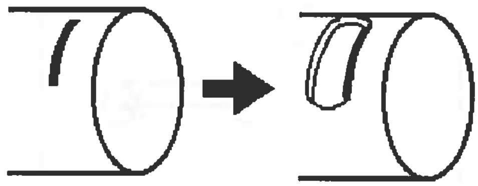
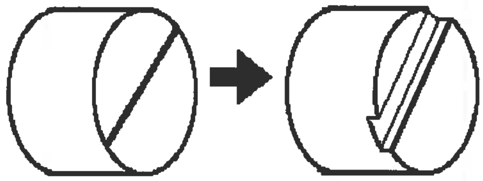
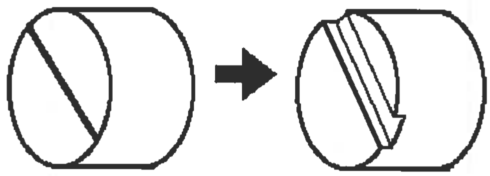
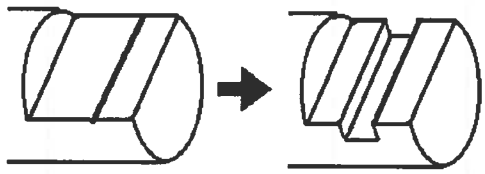
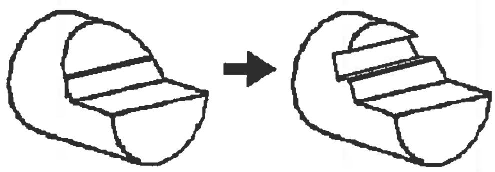
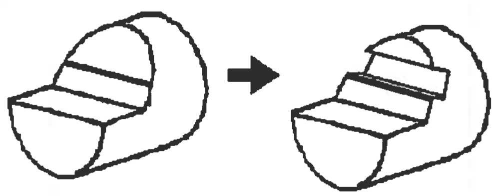
	Closed form		Open form
	Fixed form	Arbitrary form	Arbitrary form
			 M3P304
Line machining	LINE OUT, LINE IN, CHMF OUT, CHMF IN		LINE CTR, LINE RGT, LINE LFT, CHMF RGT, CHMF LFT
Face machining	FCE MILL, TOP EMIL, POCKET		SLOT

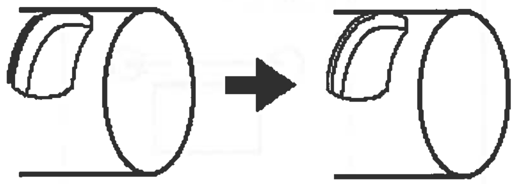
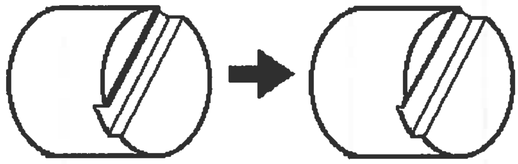
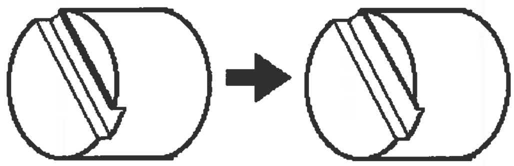
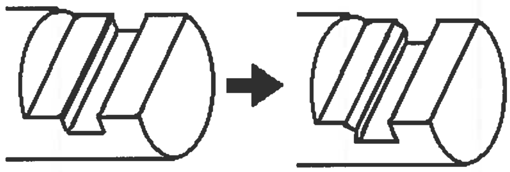
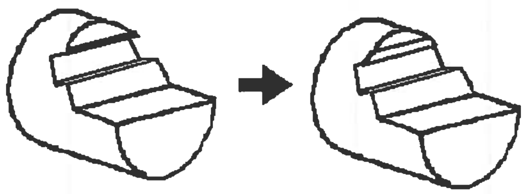
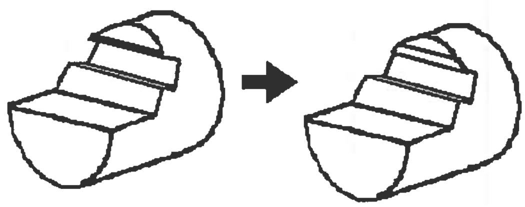
A fixed shape is not provided for the **SLOT** unit.

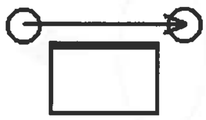
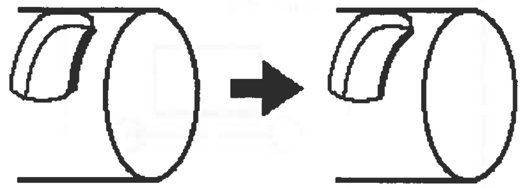
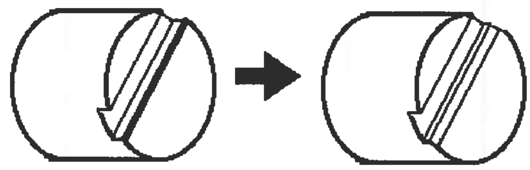
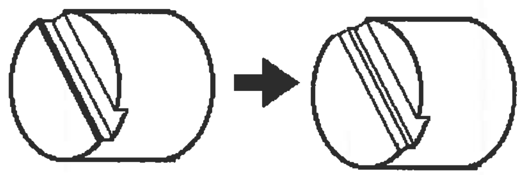
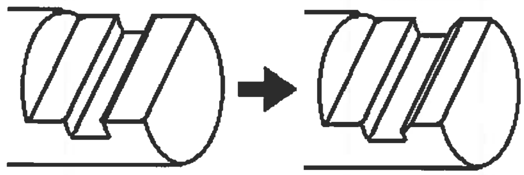
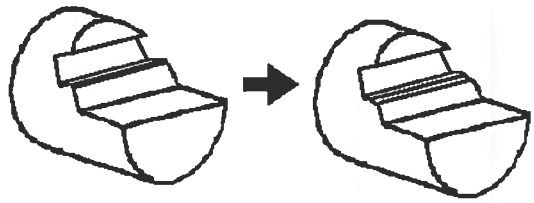
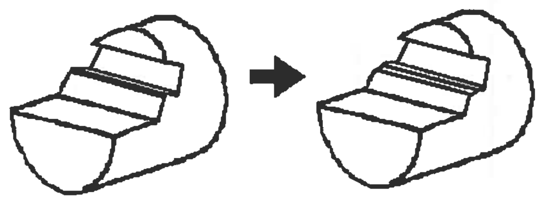
B. Machining units related the open form

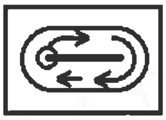
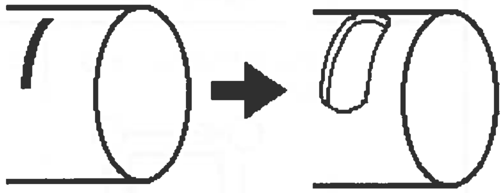
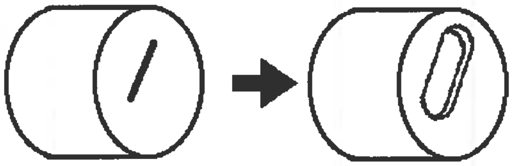
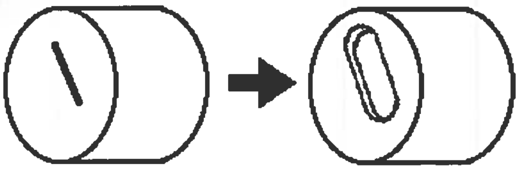
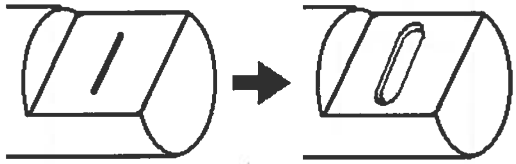
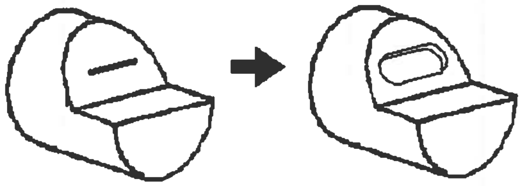
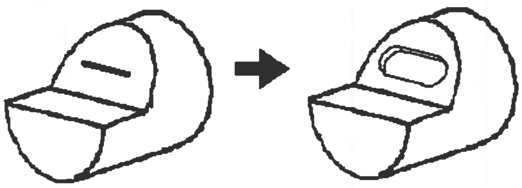
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Central linear machining (LINE CTR) 	[ZC]  [XC], [XY]  [$\boxed{XC}$], [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Right-hand linear machining (LINE RGT) 	[ZC]  [XC], [XY]  [$\boxed{\text{XC}}$], [$\boxed{\text{XY}}$]  [ZY]  [/Y]  [$\boxed{\text{/Y}}$] 

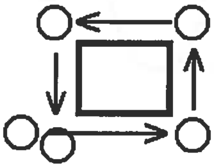
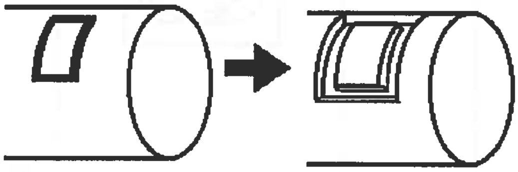
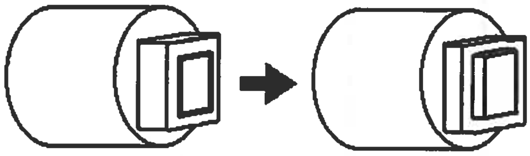
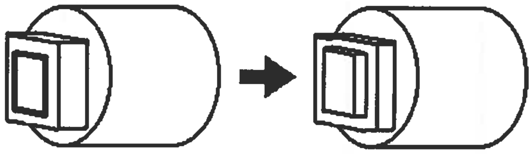
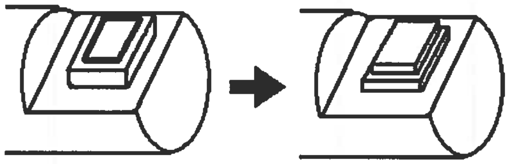
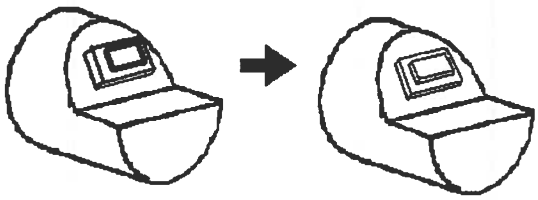
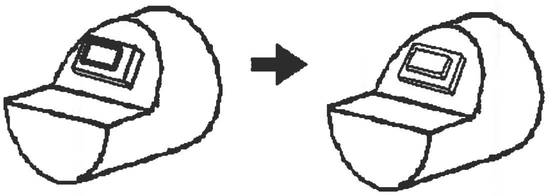
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Left-hand linear machining (LINE LFT) 	[ZC]  [XC], [XY]  [$\boxed{\text{XC}}$], [$\boxed{\text{XY}}$]  [ZY]  [/Y]  [$\boxed{\text{/Y}}$] 

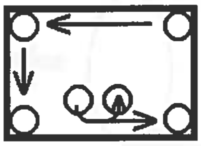
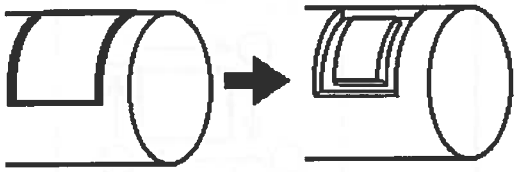
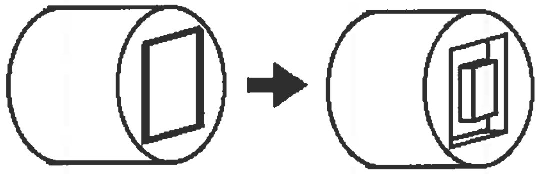
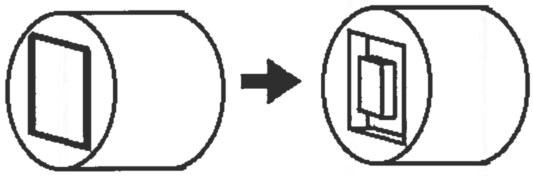
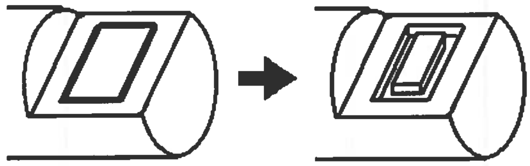
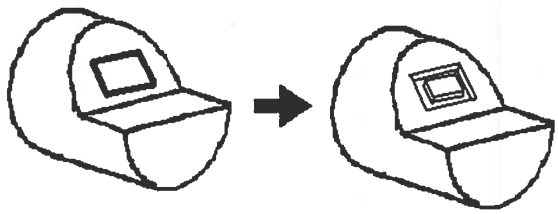
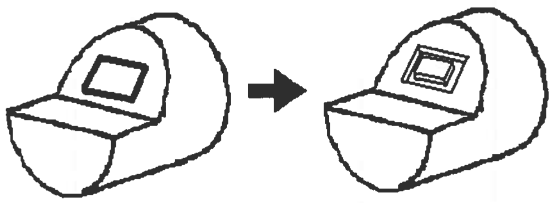
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Right-hand chamfering (CHMF RGT) 	[ZC]  [XC], [XY]  [$\boxed{XC}$], [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

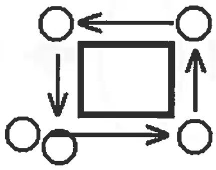
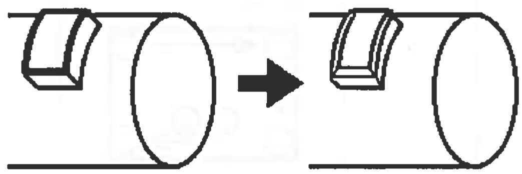
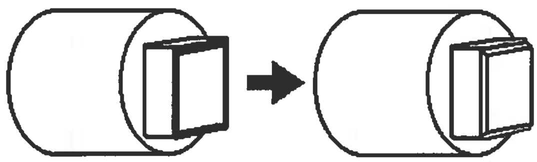
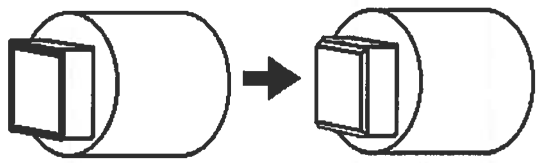
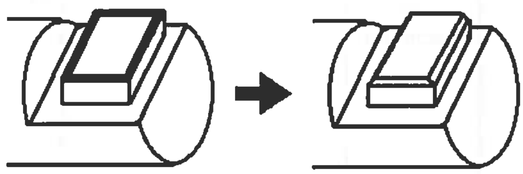
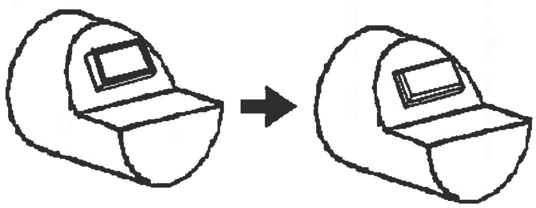
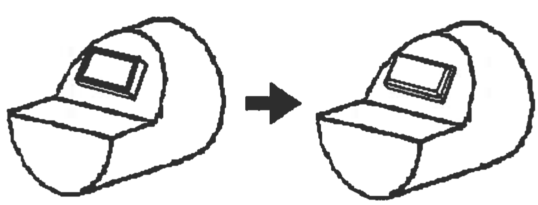
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Left-hand chamfering (CHMF LFT) 	[ZC]  [XC], [XY]  [$\boxed{XC}$], [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

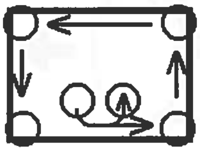
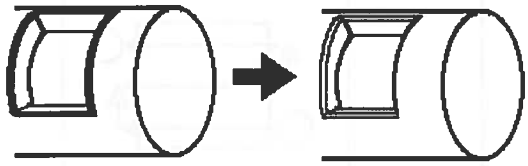
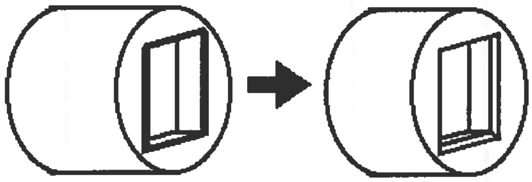
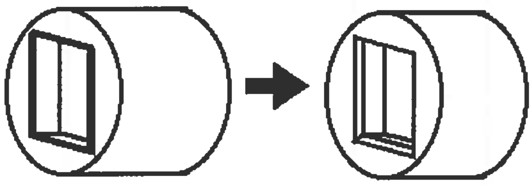
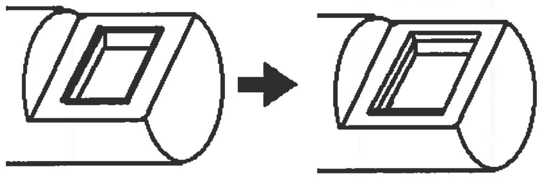
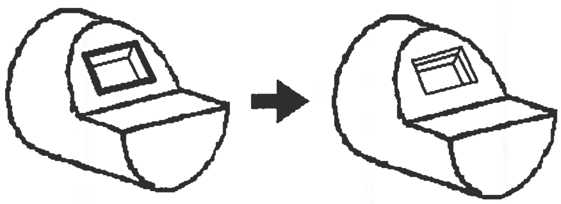
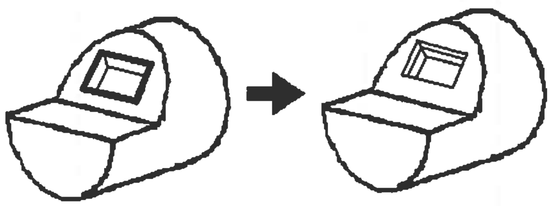
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Face machining	End milling-slot (SLOT) 	[ZC]  [XC], [XY]  [$\boxed{\text{XY}}$]  [ZY]  [/Y]  [$\boxed{\text{/Y}}$] 

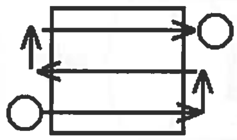
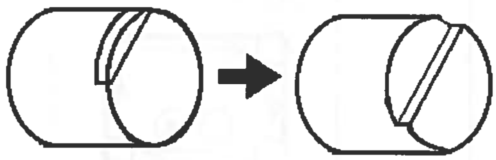
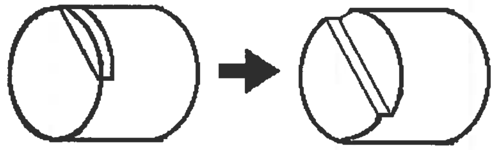
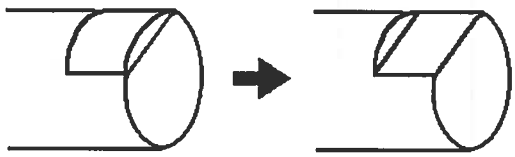
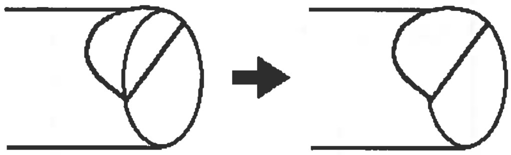
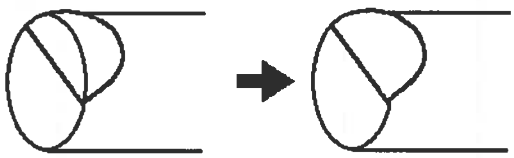
C. Machining units related to the closed form

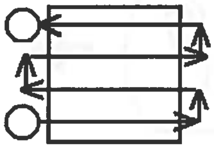
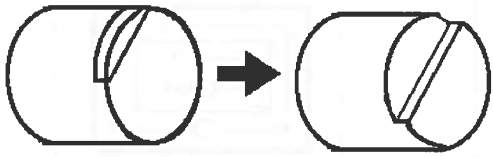
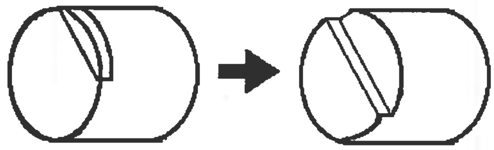
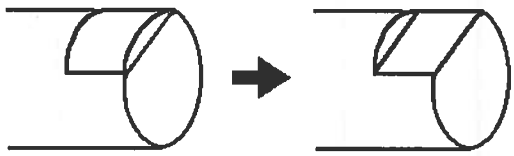
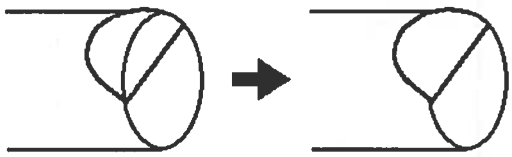
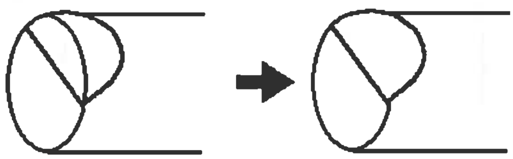
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Outside linear machining (LINE OUT) 	[ZC]  [XC], [XY]  [$\boxed{\text{XC}}$], [$\boxed{\text{XY}}$]  [ZY]  [/Y]  [$\boxed{\text{/Y}}$] 

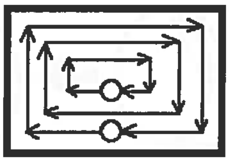
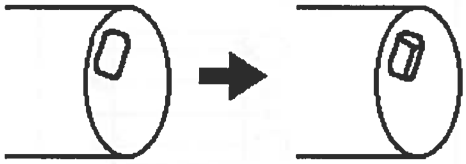
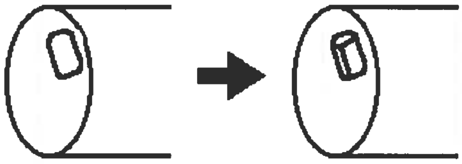
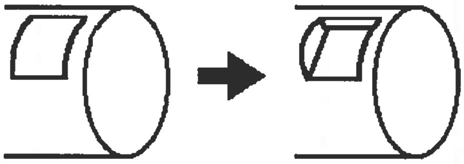
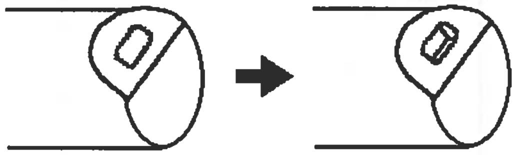
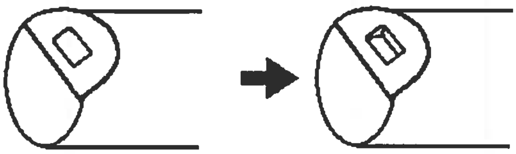
	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Inside linear machining (LINE IN) 	[ZC]  [XC], [XY]  [$\boxed{XC}$], [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Outside chamfering (CHMF OUT) 	[ZC]  [XC], [XY]  [$\boxed{\text{XC}}$], [$\boxed{\text{XY}}$]  [ZY]  [/Y]  [$\boxed{\text{/Y}}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Line machining	Inside chamfering (CHMF IN) 	[ZC]  [XC], [XY]  [$\boxed{XC}$], [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Face machining	Face milling (FCE MILL) 	[XY]  [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Face machining	End milling-top (TOP EMIL) 	[XY]  [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

	Machining unit	Before machining → After machining ([ ] : Mode selected in unit)
Face machining	Pocket milling (POCKET) 	[XY]  [$\boxed{XY}$]  [ZY]  [/Y]  [$\boxed{/Y}$] 

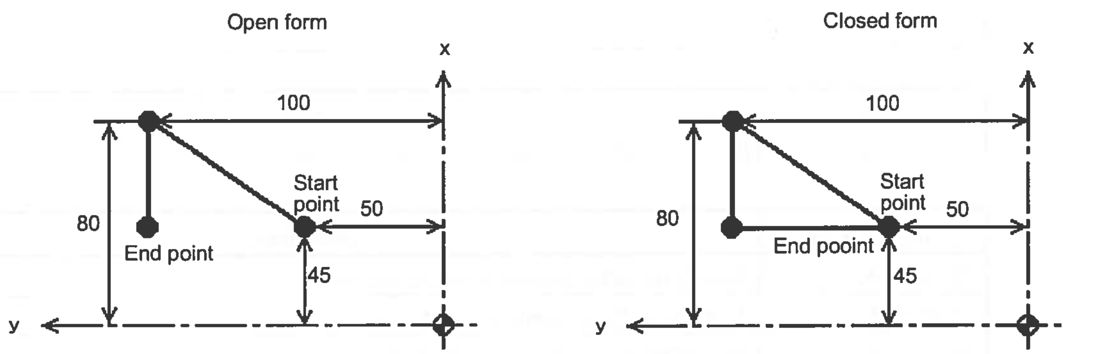
2. Precautions to be taken when defining an arbitrary form

1. For an open form, do not fail to establish coordinate of its start and end points.
2. In an open form, it is impossible to select the corner (C or R) of its start and end points.
3. The meaning of a start point differs between an open form and a closed one.
   - Open form ....The start point a meaning as a point.
   - Closed form......The start point has a meaning as a line from an end point to the start point.

Example: When the mode selected in the machining unit is XY

FIG	SHP	SHIFT-Z	R/x	C/y	RADIUS/th	I	J	P	CNR	RGH
1	LINE		45.	50.						
2	LINE		80.	100.						
3	LINE		45.	100.						

The form in this program is indicated on a is play as follows.



The start point has a meaning as a point.

The start point has a meaning as a line from an end point to the start point.

D734P0035

3. Entry of shape sequence data

- Menu selection

  After setting tool sequence data of the line/face machining unit, the following menu will be displayed.

  Select **[SQUARE]**, **[CIRCLE]** or **[ARBITRY]** from this menu.

SQUARE	CIRCLE		ARBITRY						SHAPE END

If **[ARBITRY]** is selected, the following menu will be displayed.

LINE	CW ARC	CCW ARC	CW SHIFT	CCW SHFT	SHAPE SHIFT	REPEAT END	STARTING POINT		SHAPE END

- The shape data to be set differs according to the **MODE** that was selected for the machining unit. Shape data is explained for each mode below.

A. When the mode selected in the unit is ZC

1. Fixed form

- Square (**SQR**)

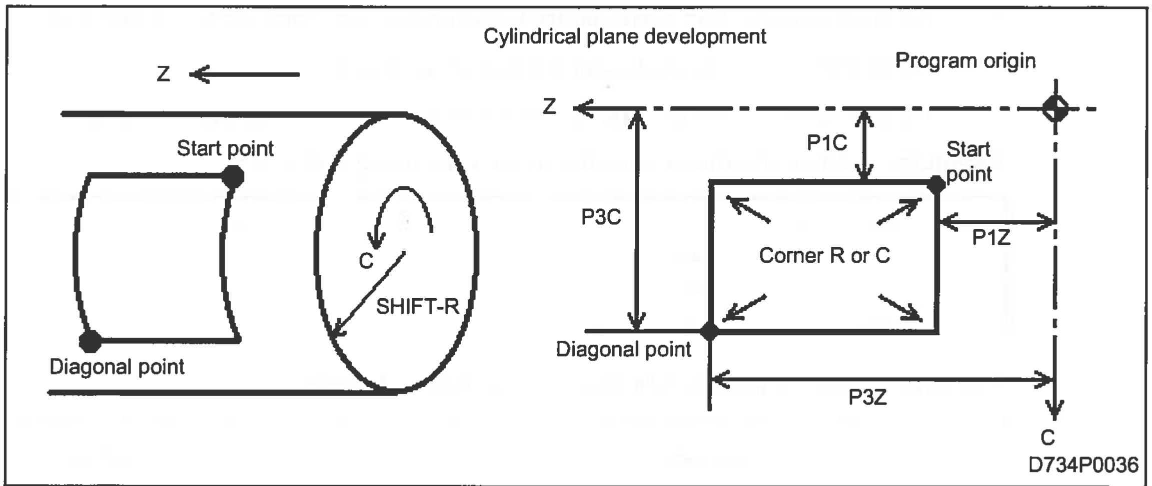


FIG	SHP	SHIFT-R	P1Z/CZ	P1C/CC	P3Z/R	P3C	CNR
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the square shape.
[2] P1Z/CZ	Specify the Z coordinate of a start point.
[3] P1C/CC	Specify the C coordinate of a start point.
[4] P3Z/R	Specify the Z coordinate of diagonal point.
[5] P3C	Specify the C coordinate of diagonal point.
[6] CNR	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

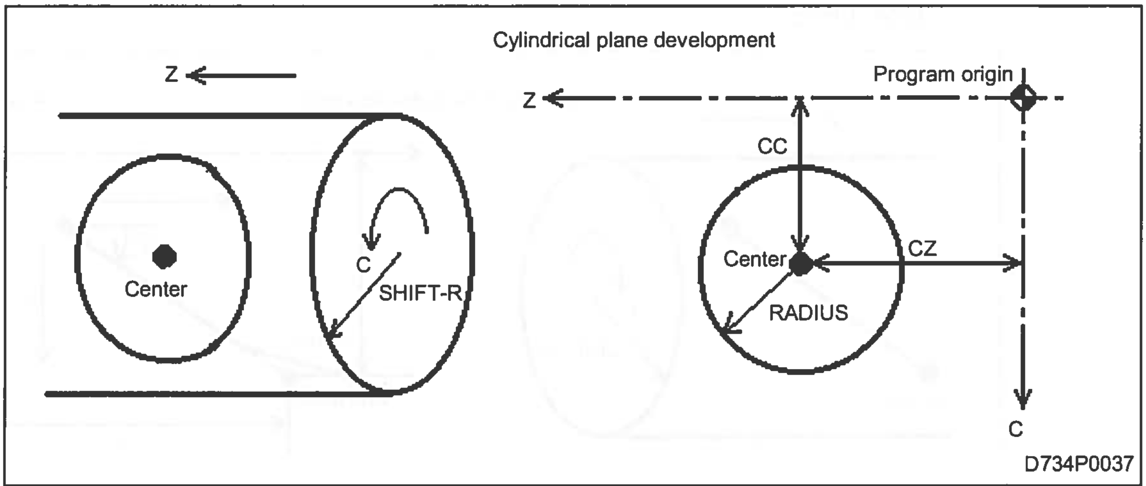


FIG	SHP	SHIFT-R	P1Z/CZ	P1C/CC	P3Z/R	P3C	CNR
2	CIR	[1]	[2]	[3]	[4]	◆	◆

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the circle shape.
[2] P1Z/CZ	Specify the Z coordinate of center.
[3] P1C/CC	Specify the C coordinate of center.
[4] P3Z/R	Specify the radius.

2. Arbitrary shape

- Line (**LINE**)

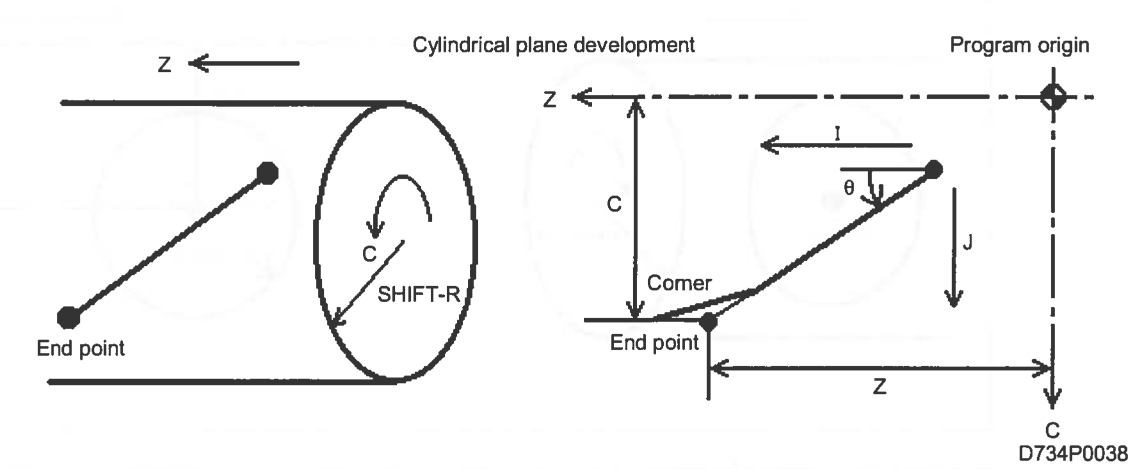


FIG	SHP	SHIFT-R	Z	C	RADIUS/th	I	J	P	CNR	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the linear shape.
[2] Z	Specify the Z coordinate of an end point of linear machining. If it is unknown, select the **[?]** menu key.
[3] C	Specify the C coordinate of an end point of linear machining. If it is unknown, select the **[?]** menu key.
[4] RADIUS/th	Specify the angle th between Z-axis and machining line.
[5] I	Specify the Z-axial vector value.
[6] J	Specify the C-axial vector value.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW, CCW**)

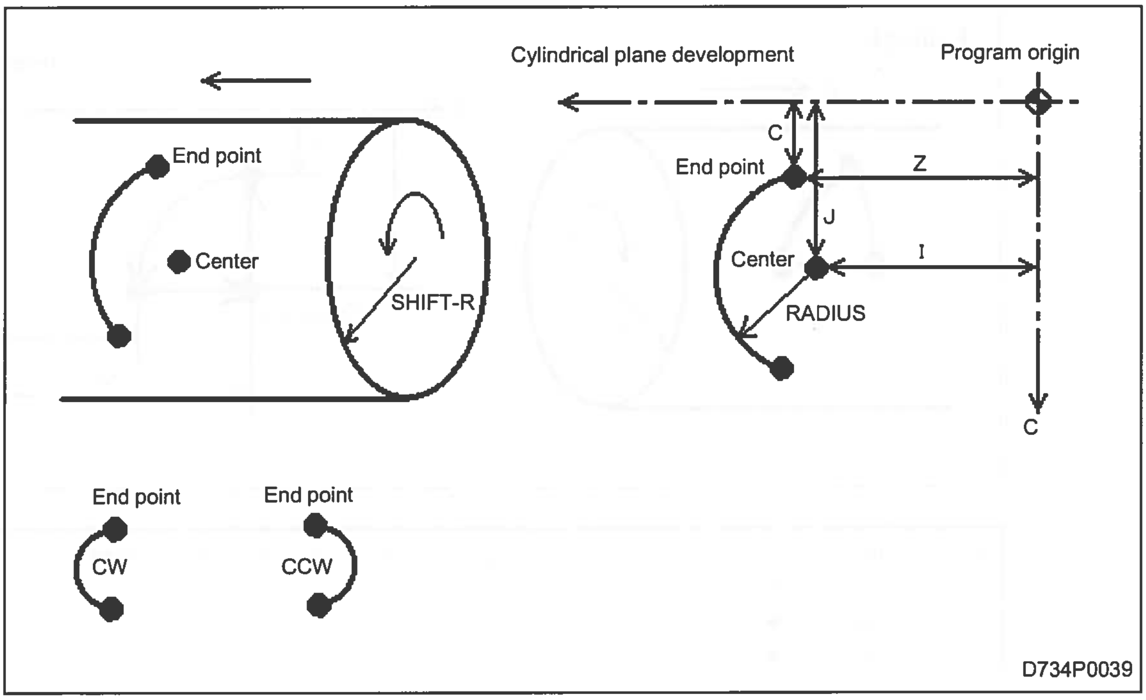


FIG	SHP	SHIFT-R	Z	C	RADIUS/th	I	J	P	CNR	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the arc shape.
[2] Z	Specify the Z coordinate of an end point of the arc. If it is unknown, select the **[?]** menu key.
[3] C	Specify the C coordinate of an end point of the arc. If it is unknown, select the **[?]** menu key.
[4] RADIUS/th	Specify the radius of the arc.
[5] I	Specify the Z coordinate of the center of arc.
[6] J	Specify the C coordinate of the center of arc.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Example of the arbitrary form

Example:

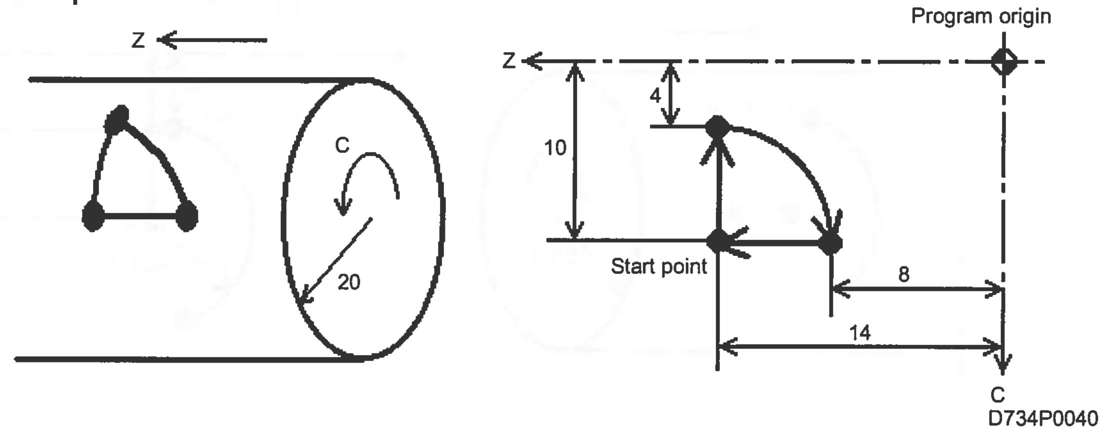
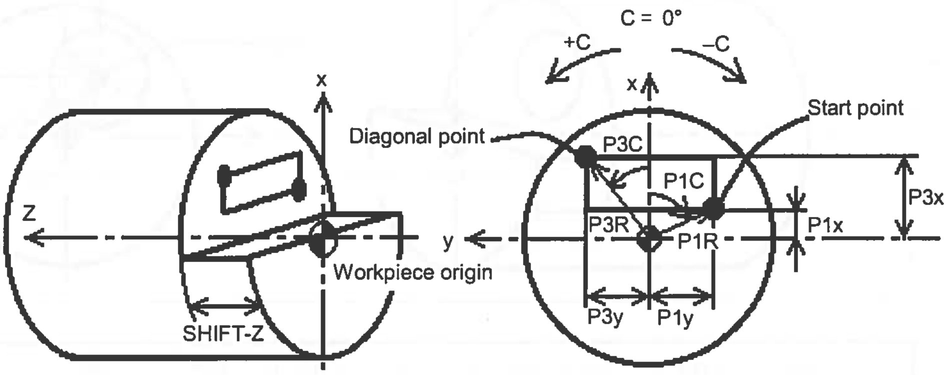


FIG	SHP	SHIFT-R	Z	C	RADIUS/th	I	J	P	CNR	RGH
1	LINE	20.	14.	10.	0.					
2	LINE	◆	14.	4.	90.					
3	CW	◆	8.	10.	6.	14.	10.			

B. When selected mode in the unit is XC, $\boxed{\text{XC}}$, XY or $\boxed{\text{XY}}$

1. Fixed form

   - Square (**SQR**)



D734P0041

FIG	SHP	SHIFT-Z	P1Rx/CRx	P1Cy/CCy	P3Rx/R	P3Cy	CNR
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]

Cursor position	Description
[1] SHIFT-Z	Specify the Z-axial position of the square shape.
[2] P1Rx/CRx [3] P1Cy/CCy	Specify a coordinate of the start point. - To specify the start point in R-C coordinates, specify the radius and the angle as they are. - To specify the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before specifying the data.
[4] P3Rx/R [5] P3Cy	Specify a coordinate of the diagonal point. - To specify the diagonal point in R-C coordinates, specify the radius and the angle as they are. - To specify the diagonal point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before specifying the data.
[6] CNR	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

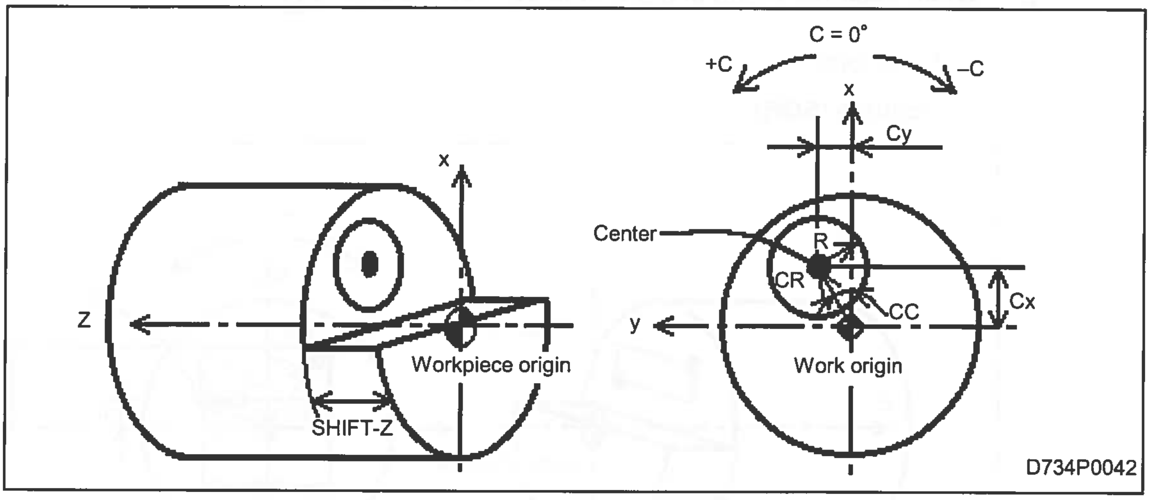


FIG	SHP	SHIFT-Z	P1Rx/CRx	P1Cy/CCy	P3Rx/R	P3Cy	CNR
2	CIR	[1]	[2]	[3]	[4]	◆	◆

Cursor position	Description
[1] SHIFT-Z	Specify the Z-axial position of the circle shape.
[2] P1Rx/CRx [3] P1Cy/CCy	Specify the coordinate of the center. - To specify the center in R-C coordinates, enter the radius and the angle as they are. - To specify the center in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data.
[4] P3Rx/R	Specify the radius.

2. Arbitrary form

- Line (**LINE**)

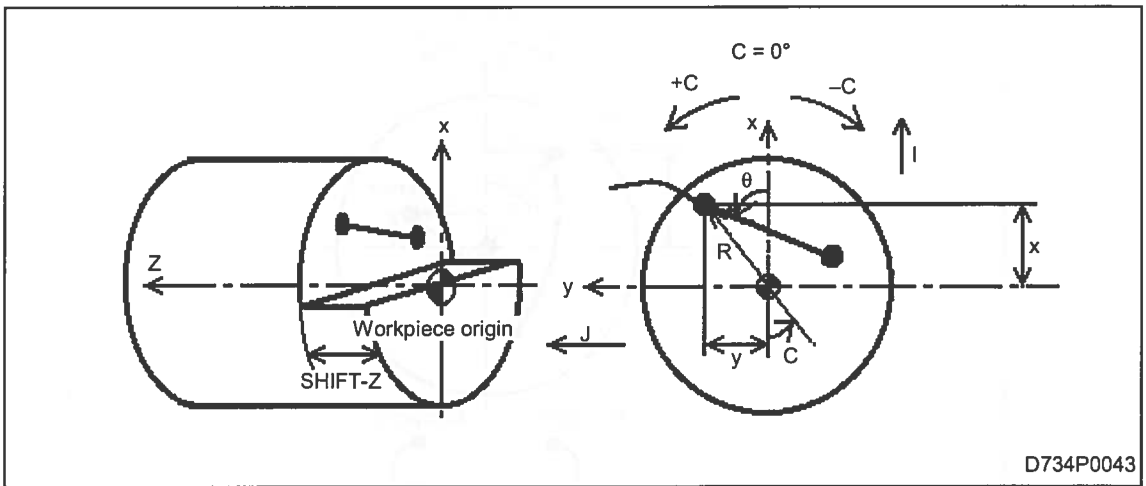


FIG	SHP	SHIFT-Z	R/x	C/y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-Z	Specify the Z-axial position of the line shape.
[2] R/x [3] C/y	Specify the coordinate of the end point of the line machining. - To specify the start point in R-C coordinates, enter the radius and the angle as they are. - To specify the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data.
[4] RADIUS/th	Specify the angle th between x-axis and machining line.
[5] I	Specify the x-axial vector value.
[6] J	Specify the y-axial vector value.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW, CCW**)

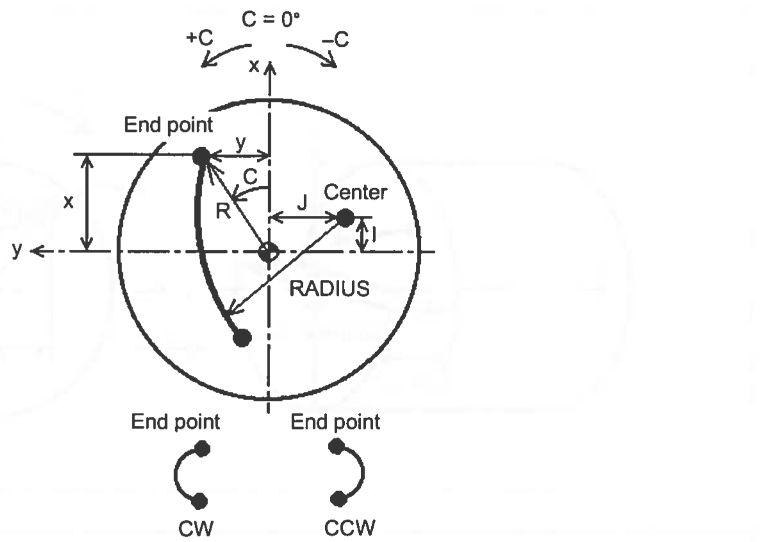
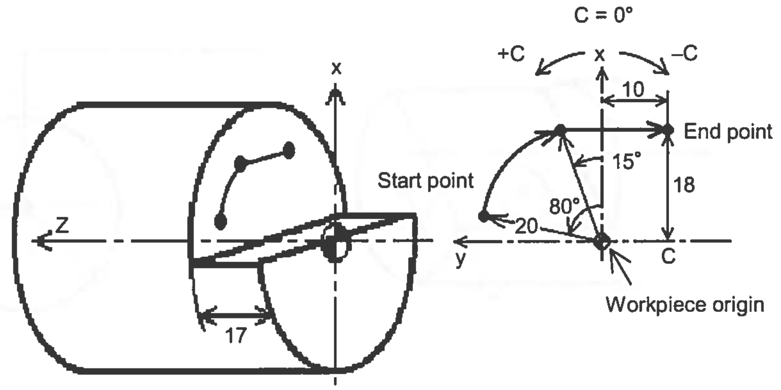


FIG	SHP	SHIFT-Z	R/x	C/y	RADIUS/th	I	J	P	CNR	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-Z	Specify the Z-axial position of the arc shape. See the item "Line (LIN) ".
[2] R/x [3] C/y	Specify the coordinate of an end point of the arc machining. - To specify the end point in R-C coordinates, enter the radius and the angle as they are. - To specify the end point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display mode before entering data. If it is unknown, select the **[?]** menu key.
[4] RADIUS/th	Specify the radius of the arc.
[5] I	Specify the x coordinate of the center of arc.
[6] J	Specify the y coordinate of the center of arc.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Example of the arbitrary form

Example:



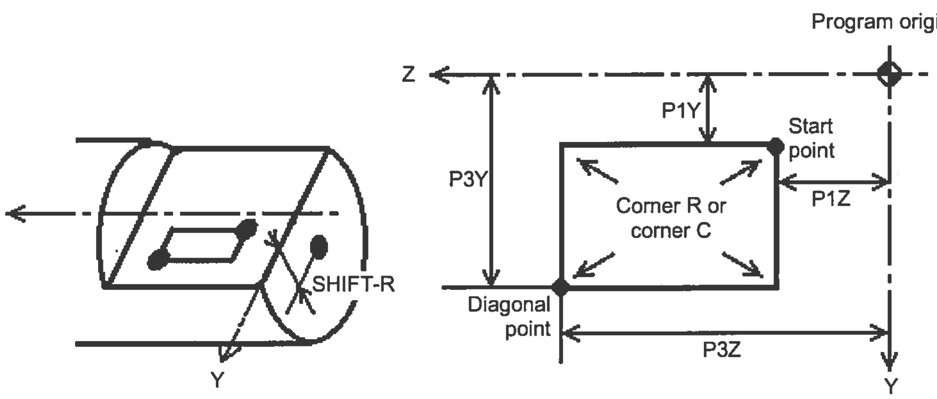
D734P0045

FIG	SHP	SHIFT-Z	R/x	C/y	RADIUS/th	I	J	P	CNR	RGH	
1	LINE	17.	20.	80.							← R-C coordinate system
2	CW	◆	20.	15.	20.	0.	0.				← R-C coordinate system
3	LINE	◆	18.	-10.	90.						← x-y coordinate system

C. When selected mode in the unit is ZY

1. Fixed form

- Square (**SQR**)



D734P0046

FIG	SHP	SHIFT-R	P1Z/CZ	P1Y/CY	P3Z/R	P3Y	CNR
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the ZY plane.
[2] P1Z/CZ	Specify the Z coordinate of a start point.
[3] P1Y/CY	Specify the Y coordinate of a start point.
[4] P3Z/R	Specify the Z coordinate of diagonal point.
[5] P3Y	Specify the Y coordinate of diagonal point.
[6] CNR	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

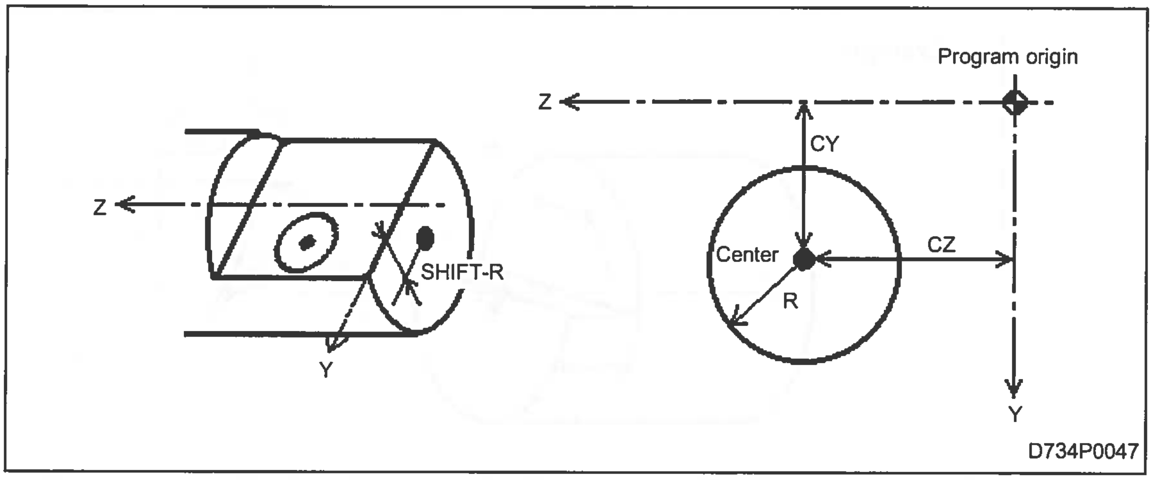


FIG	SHP	SHIFT-R	P1Z/CZ	P1Y/CY	P3Y/R	P3Y	CNR
2	CIR	[1]	[2]	[3]	[4]	◆	◆

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the ZY plane.
[2] P1Z/CZ	Specify the Z coordinate of center.
[3] P1Y/CY	Specify the Y coordinate of center.
[4] P3Z/R	Specify the radius.

2. Arbitrary form

- Line (**LINE**)

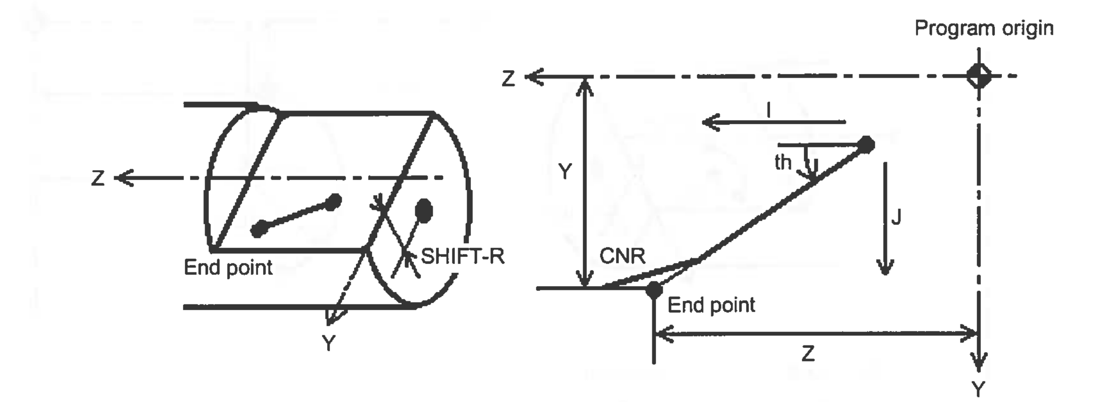


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the ZY plane.
[2] Z	Specify the coordinate of an end point of linear machining. If it is unknown, select the **[?]** menu key.
[3] Y	Specify the Y coordinate of an end point of linear machining. If it is unknown, select the **[?]** menu key.
[4] RADIUS/th	Specify the angle th between Z-axis and machining line.
[5] I	Specify the Z-axial vector value.
[6] J	Specify the Y-axial vector value.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW, CCW**)

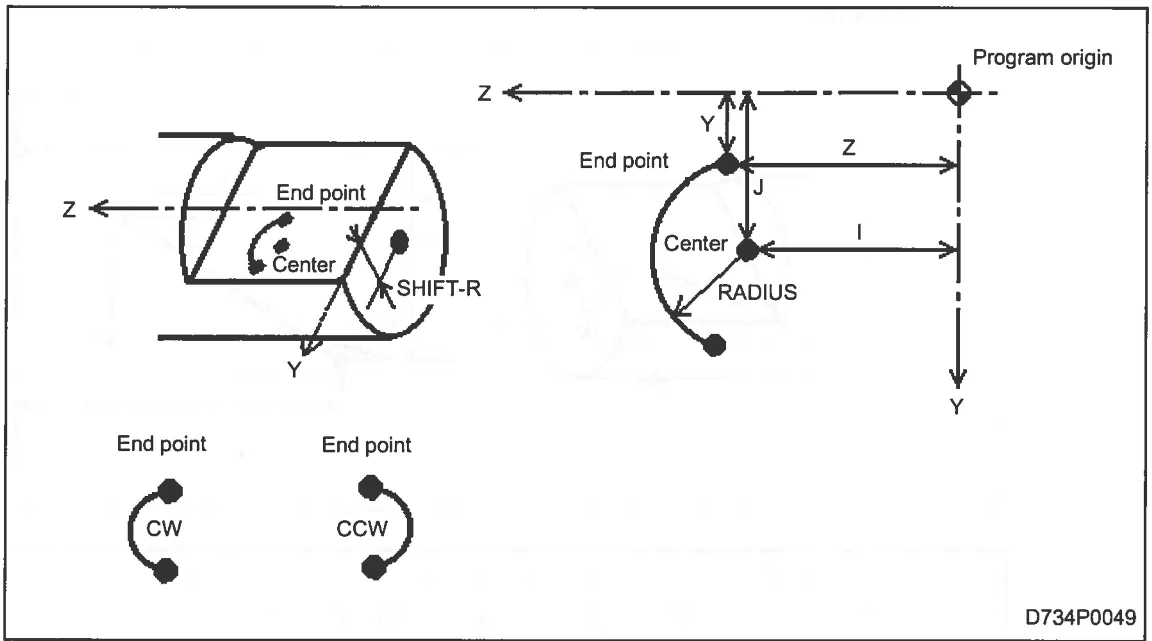
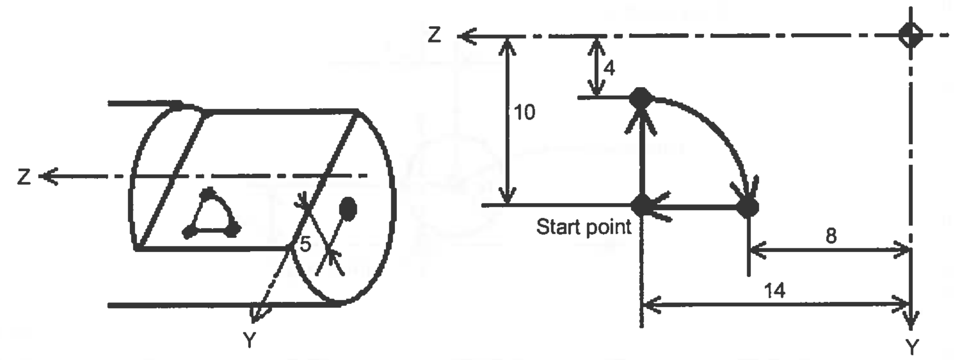


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

Cursor position	Description
[1] SHIFT-R	Specify the radial position of the ZY plane.
[2] Z	Specify the Z coordinate of an end point of the arc. If it is unknown, select the **[?]** menu key.
[3] Y	Specify the Y coordinate of an end point of the arc. If it is unknown, select the **[?]** menu key.
[4] RADIUS/th	Specify the radius of the arc.
[5] I	Specify the Z coordinate of the center of arc.
[6] J	Specify the Y coordinate of the center of arc.
[7] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Example of the arbitrary form

Example:



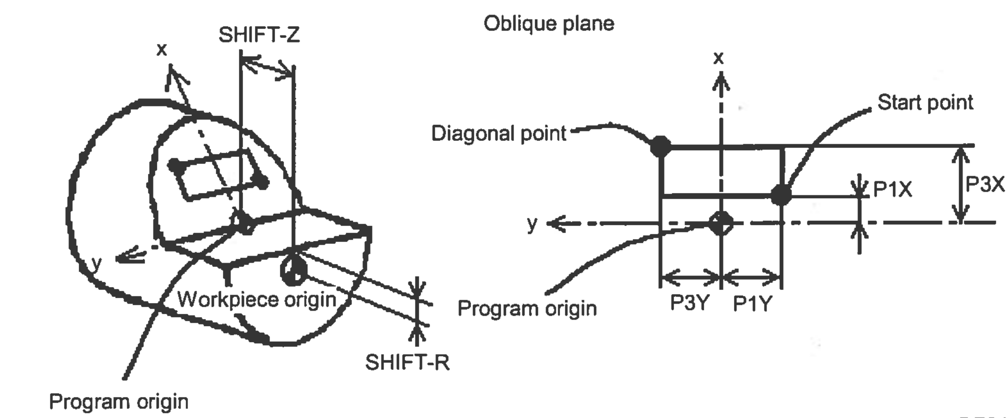
D734P0050

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	5.	14.	10.	0.					
2	LINE	◆	14.	4.	90.					
3	CW	◆	8.	10.	6.					

D. When selected mode in the unit is /Y or $\boxed{/Y}$

1. Fixed form

- Square (**SQR**)

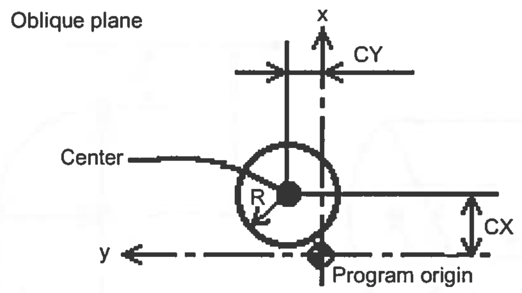


D734P0051

FIG	SHP	SHIFT-Z	SHIFT-R	P1X/CX	P1Y/CY	P3X/R	P3Y	CNR
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the amount of shift from the workpiece origin of the x-y plane.
[3] P1X/CX [4] P1Y/CY	Specify the coordinate of the start point.
[5] P3X/R [6] P3Y	Specify the coordinate of diagonal point.
[7] CNR	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)



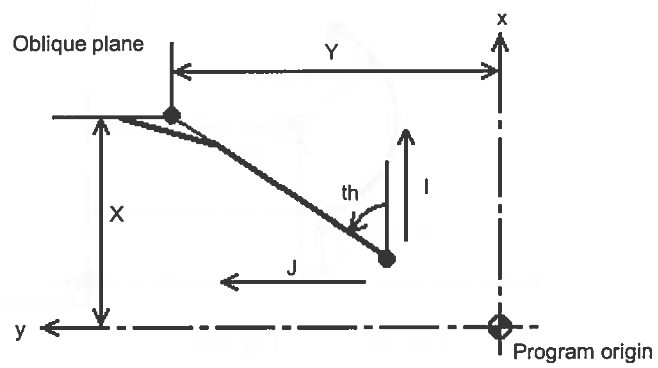
D734P0052

FIG	SHP	SHIFT-Z	SHIFT-R	P1X/CX	P1Y/CY	P3X/R	P3Y	CNR
2	CIR	[1]	[2]	[3]	[4]	[5]	◆	◆

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the amount of shift from the work origin of the x-y plane. See the item "Square (SQR)".
[3] P1X/CX [4] P1Y/CY	Specify the coordinate of the center.
[5] P3X/R	Specify the radius.

2. Arbitrary form

- Line (**LINE**)



734P0053

FIG	SHP	SHIFT-Z	SHIFT-R	X	Y	RADIUS/th	I	J	P	CNR	RGH
999	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the amount of shift from the work origin of the x-y plane. See the item of the square of the fixed form.
[3] X [4] Y	Specify the coordinate of an end point of linear machining.
[5] RADIUS/th	Specify the angle th between x-axis and machining line.
[6] I	Specify the x-axial vector value.
[7] J	Specify the y-axial vector value.
[8] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[9] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[10] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW, CCW**)

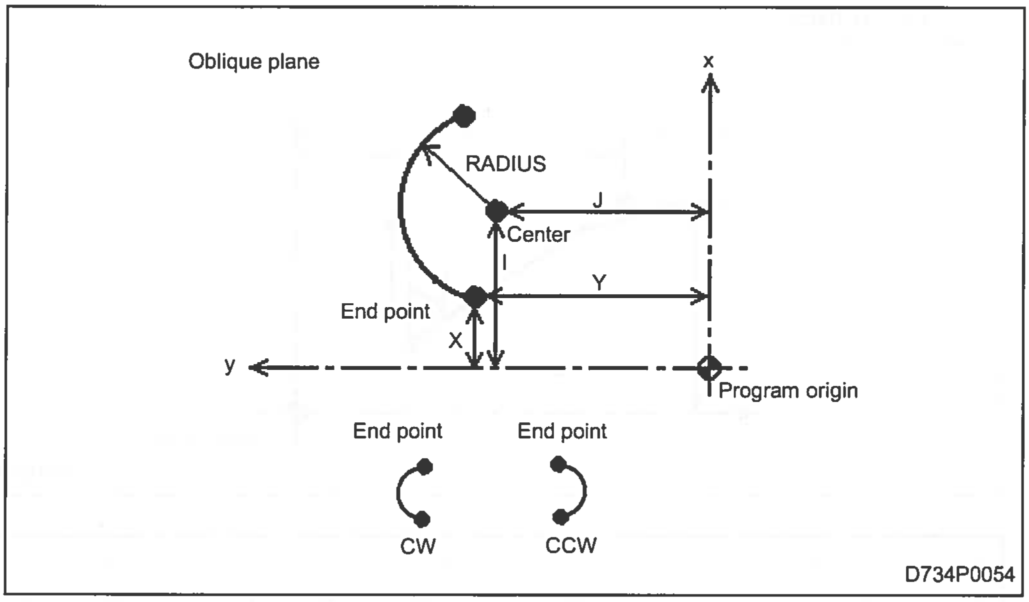


FIG	SHP	SHIFT-Z	SHIFT-R	X	Y	RADIUS/th	I	J	P	CNR	RGH
999	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the amount of shift from the workpiece origin of the x-y plane. See the item of the square of the fixed form.
[3] X [4] Y	Specify the coordinate of an end point of arc machining.
[5] RADIUS/th	Specify the radius of the arc.
[6] I	Specify the x coordinate of the center of the arc.
[7] J	Specify the y coordinate of the center of the arc.
[8] P	Select from the menu the position of the point crossing the next shape. **Note:** See the section of the Automatic Cutting-Conditions Setting Function for further details.
[9] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[10] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Example of the arbitrary shape

Example:

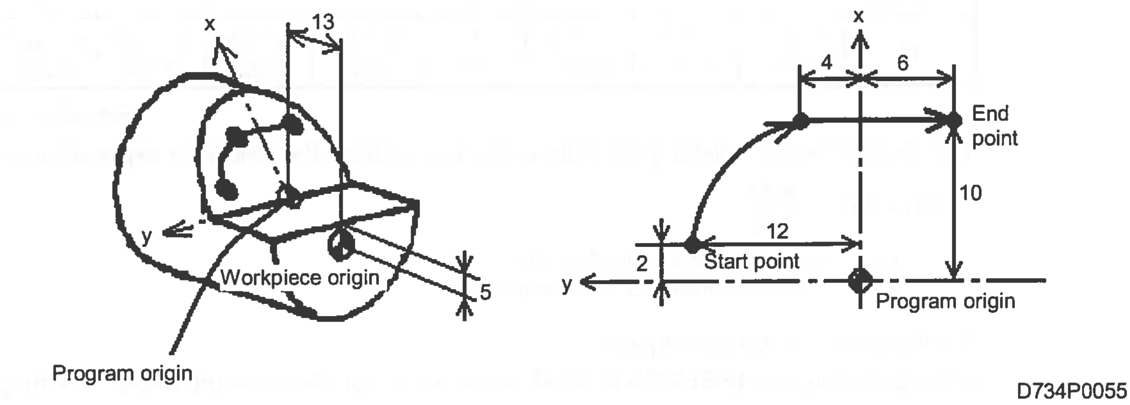


FIG	SHP	SHIFT-Z	SHIFT-R	X	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	13.	5.	2.	12.						
2	CW	◆	◆	10.	4.	8.	2.	4.			
3	LINE	◆	◆	10.	-6.	90.					

Remark1: Roughness

Set a finishing feedrate appropriate for particular surface roughness.
For setting a finishing feedrate, two methods are available: selection of a surface roughness code (for this case, the NC unit automatically calculates the appropriate feedrate for the selected surface roughness code), and direct setting of any desired feedrate.
The following menu will be displayed when the cursor is placed at this item:

ROUGHNES	FEEDRATE / rev								

- If a surface roughness code is to be selected:
  The code can be entered either by setting the desired code number directly with numerical keys or using the following procedure:

1) First, press the **[ROUGHNES]** menu key.
   The following menu will be displayed:

∇ 1	∇ 2	∇ 3	∇ 4	∇∇ 5	∇∇ 6	∇∇ 7	∇∇∇ 8	∇∇∇ 9	

2) Next, from the above menu, select one of the surface roughness codes indicated on the machining drawing.

The finishing feedrate in radial direction is calculated from the following expression automatically.
If the diameter of the tool to be used is taken as D, one can have:

In case of $D <$ **K32**............ $Ff_1 = \mathbf{U19} \times \frac{D}{\mathbf{K32}} \times Kf \times Z$

In case of $D \geq$ **K32**............ $Ff_1 = \mathbf{U19} \times Kf \times Z$

K32 : Parameter used to set a reference diameter for the feedrate of finishing during milling
Ff_1 : Radial-direction finishing feedrate
U19 : Parameter used to set a feedrate for reference degree of surface roughness (∇∇4)
Kf : Feed factor
Z : Number of teeth of the tool

Each surface roughness code and feed factor are correlated as follows:

Surface roughness	∇ 1	∇ 2	∇∇∇ 3	∇∇∇ 4	∇∇∇ 5	∇∇∇ 6	∇∇∇ 7	∇∇∇∇ 8	∇∇∇∇ 9
Kf	$K_0/0.8^3$ (0.977)	$K_0/0.8^2$ (0.781)	$K_0/0.8$ (0.625)	K_0 (0.5)	$K_0 \times 0.8$ (0.4)	$K_0 \times 0.8^2$ (0.32)	$K_0 \times 0.8^3$ (0.256)	$K_0 \times 0.8^4$ (0.205)	$K_0 \times 0.8^5$ (0.164)

Reference value: $K_0 = 0.5$

The axial-direction finishing feedrate is calculated from the following expression automatically.

$$Ff_2 = \mathbf{Ff_1} \times \frac{\mathbf{K32}}{100}$$

Ff_2 : Axial-direction finishing feedrate
K23 : Factor to set an axial direction feedrate

- If a feedrate is to be directly set:
After pressing the **[FEEDRATE/rev]** menu key, set the desired value (finishing feedrate in radial-direction).
The axial-direction finishing feedrate is calculated from the above expression automatically.

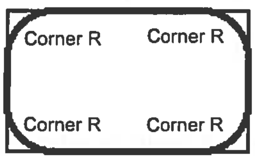
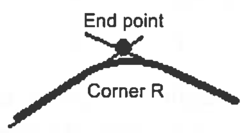
CAUTION

- During sequences having no data set for this item, finishing is done at the feedrate that was set for tool sequence data item **FR**.
- This item can be set for **LINE CTR**, **LINE RGT**, **LINE LFT**, **LINE OUT** and **LINE IN** units.

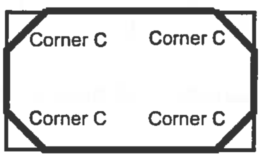
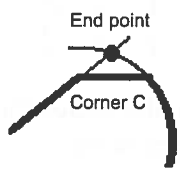
Remark 2: Corner

Set the machining pattern for the corner.

- R machining (rounding) : Set data as it is.

[Fixed form] corner R of the square	[Arbitrary form] corner R of the end point
	

- C chamfering: Set data after pressing the **[CORNER CHAMFER]** menu key.

[Fixed form] corner C of the square	[Arbitrary form] corner C of the end point
	

- Pressing the **[CORNER CHAMFER]** menu key changes the menu to reverse display mode and then setting data returns the menu to the original display mode.

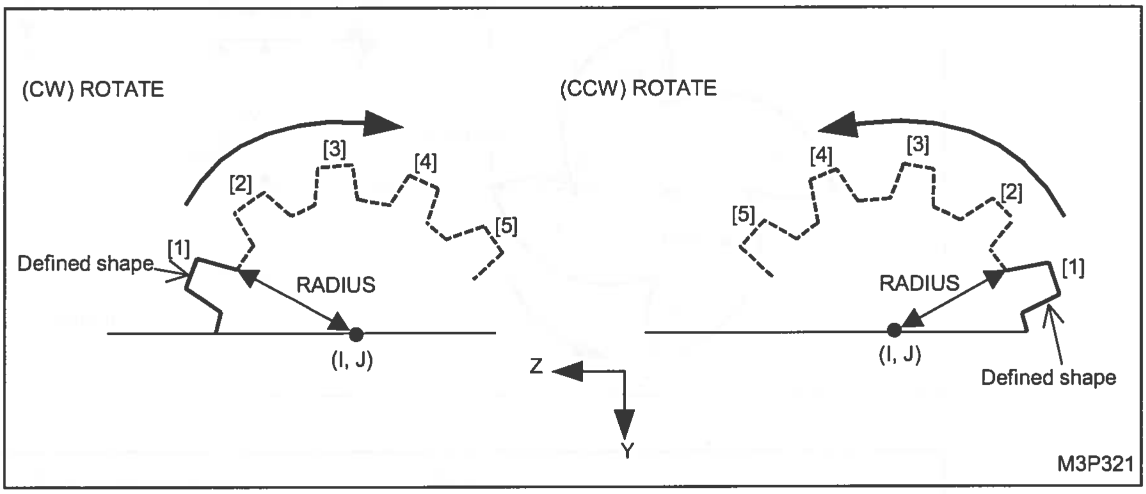
4. Shape rotation and shifting

The defined shape can be rotated or shifted.

The method of rotating or shifting a shape during the "ZY" mode of the machining unit is explained as an example below.

(The shape can be rotated or shifted similarly in other modes.)

A. Shape rotation (CW and CCW)



1. Menu selection

   Press the **[SHAPE ROTATE]** and **[CW SHIFT]** or **[CCW SHIFT]** menu keys in this order.

2. Data setting in shape sequence **CW/CCW-SH** (see figure above)

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	CW-SH (CCW-SH)	[1]	◆	◆	[2]	[3]	[4]	◆	[5]	◆
	Use LINE, CW or CCW ARC to enter a defined shape.									
999	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆

◆: Data are not necessary to be set here.

Cursor position	Description
[1] SHIFT-R	Specify the radius position of the ZY plane. See the square shape of the ZY mode for further details.
[2] RADIUS/th	Specify the radius to rotate a defined shape. If it is unknown, select the **[?]** menu key .
[3] I	Specify the Z coordinate of the center to rotate a defined shape. If it is unknown, select the **[?]** menu key.
[4] J	Specify the Y coordinate of the center to rotate a defined shape. If it is unknown, select the **[?]** menu key.
[5] CNR	Specify the number of defined shape repetitions (p).

3. **[REPEAT END]** menu function

   Press the **[REPEAT END]** menu key and a shape sequence of **CW/CCW-SH** will be brought to the end.

   Example: CW-SH

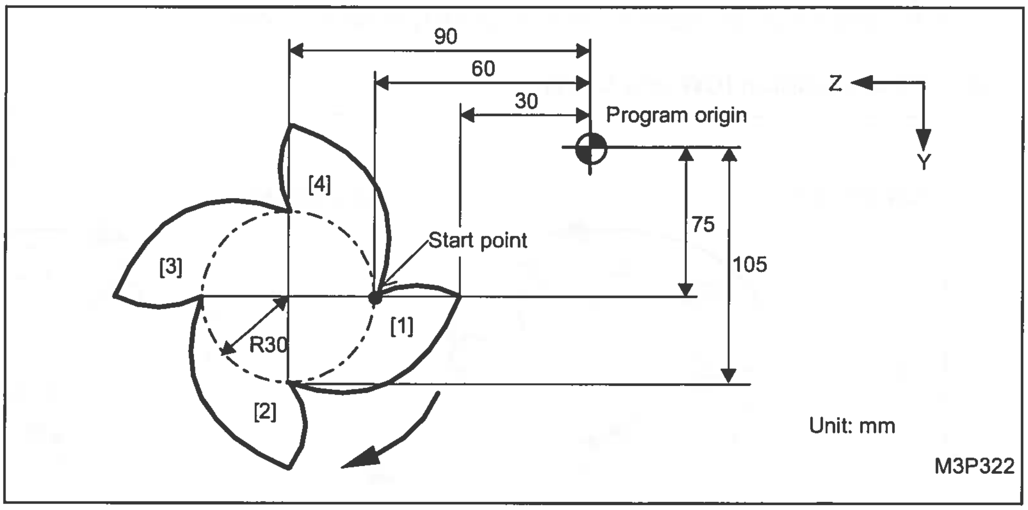
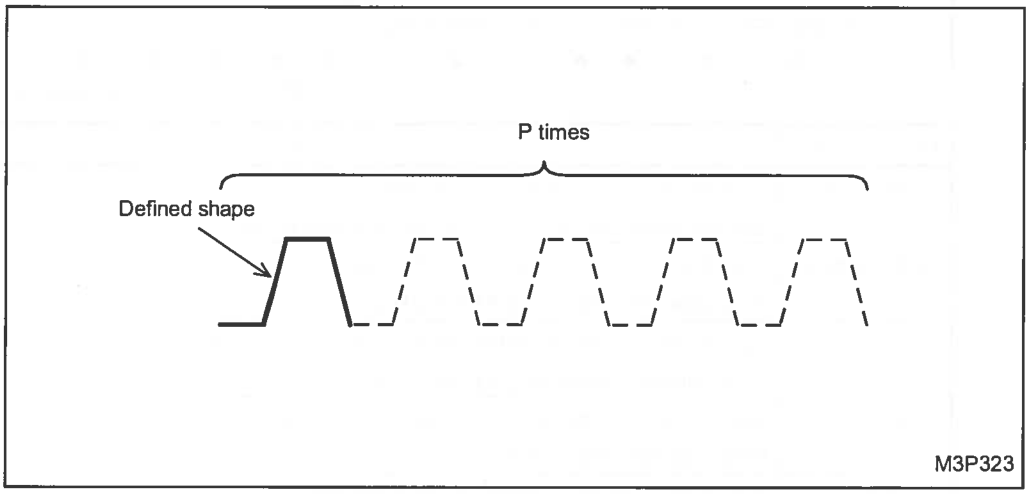


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	CW-SH	10.	◆	◆	30.	90.	75.	◆	4	◆
2	LINE	◆	60.	75.						
3	CW	◆	30.	75.	100.					
4	CW	◆	90.	105.	50.					
5	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆

B. Shape shifting

The end point will be regarded as the next start point.



1. Menu selection
   Press the **[SHAPE SHIFT]** menu key.

2. Data setting in shape sequence **FIG-SH** (see figure above)

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
6	FIG-SH	[1]	◆	◆	◆	◆	◆	◆	[2]	◆
Use LINE, CW ARC or CCW ARC to enter a defined shape.										
999	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆

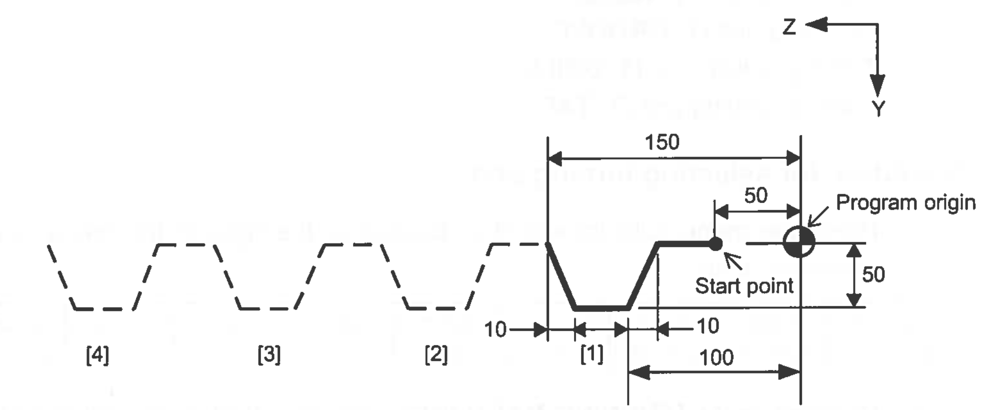
◆ : Data are not necessary to be set here.

Cursor position	Description
[1] SHIFT-R	Specify the radius position of the ZY plane. See the square shape of the ZY mode for further details.
[2] CNR	Specify the number of repetitions for a defined shape.

3. **[REPEAT END]** menu function

Press the menu key **[REPEAT END]** and a shape sequence of **FIG-SH** will be brought to the end.

Example:



M3P324

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	FIG-SH	10.	◆	◆	◆	◆	◆	◆	4	◆
2	LINE	◆	50.	0.						
3	LINE	◆	90.	0.						
4	LINE	◆	100.	50.						
5	LINE	◆	140.	50.						
6	LINE	◆	150.	0.						
7	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆

3-7 Turning Units

The turning units are intended to specify data on the machining method to be used for turning, and data on the shape of the section to be machined.
Specify the coordinates of the shape in the axial direction of the machine coordinate system, regardless of the spindle head angle.
Each turning unit includes the following two sequences:

- Tool sequence ....Enter the tool operation data to be used in the turning unit.
- Shape sequence ......Enter data on the machining dimensions shown in the drawing.

3-7-1 Types of turning units

Eight types of turning units are provided.

- Bar-materials machining unit (**BAR**)
- Copy-machining unit (**CPY**)
- Corner-machining unit (**CORNER**)
- Facing unit (**FACING**)
- Threading unit (**THREAD**)
- Grooving unit (**T. GROOVE**)
- Turning drilling unit (**T. DRILL**)
- Turning tapping unit (T. **TAP**)

3-7-2 Procedure for selecting turning unit

(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Pressing on the **[TURNING MACH-ING]** menu key displays the following unit menu.

BAR	CPY	CORNER	FACING	THREAD	T.GROOVE	T.DRILL	T.TAP		FINISH ALLOW

(3) Press the menu key corresponding to the desired machining unit.

<u>Notes on the menu option **[FINISH ALLOW]**:</u>

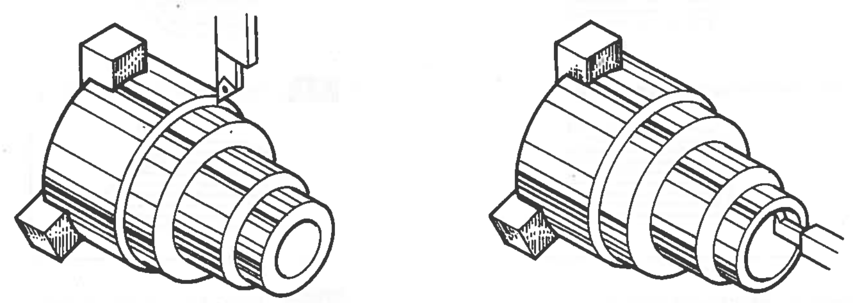
When the **[FINISH ALLOW]** menu key is selected from the turning unit selection menu and then a turning unit is created with the **[FINISH ALLOW]** menu item highlighted as follows, if another turning unit already exists in front of that created unit, the operator can automatically set the same values as the finishing allowances specified in the existing turning unit.

BAR	CPY	CORNER	FACING	THREAD	T.GROOVE	T.DRILL	T.TAP		FINISH ALLOW

- For **FIN-X** and **FIN-Z** each, independent finishing allowance values are automatically determined from the values that have been specified in the existing turning unit.
- When no finishing allowances are specified in the previous turning unit, older turning units will be referred to in sequence and if the settings of finishing allowances are not detected in any turning units down to the beginning of the program, finishing allowances will not be auto-set.
- The highlighted status of the **[FINISH ALLOW]** menu option is maintained, even after power has been turned off.

3-8 Bar-Materials Machining Unit (BAR)

Select the bar-materials machining unit to lathe the outer peripheries, inner peripheries, front faces, or back faces of round-bar-materials using general-purpose cutting tools.



M210-00429

Press the **[BAR]** menu key to select this unit.

3-8-1 Setting unit data

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
*	BAR	[1]	[2]	[3]	[4]	[5]

[1] PART

The following menu will be displayed when the cursor is placed at this item.

OUT	OUT	IN	IN	FACE	FACE	BACK	BACK		

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

OUT : Outer periphery (Cutting from the front face)
.........................Outside-diameter open type

OUT : Outer periphery (Cutting from the middle of the outer periphery)
.........................Outside-diameter middle type

IN : Inner periphery (Cutting from the front face)
.........................Inside-diameter open type

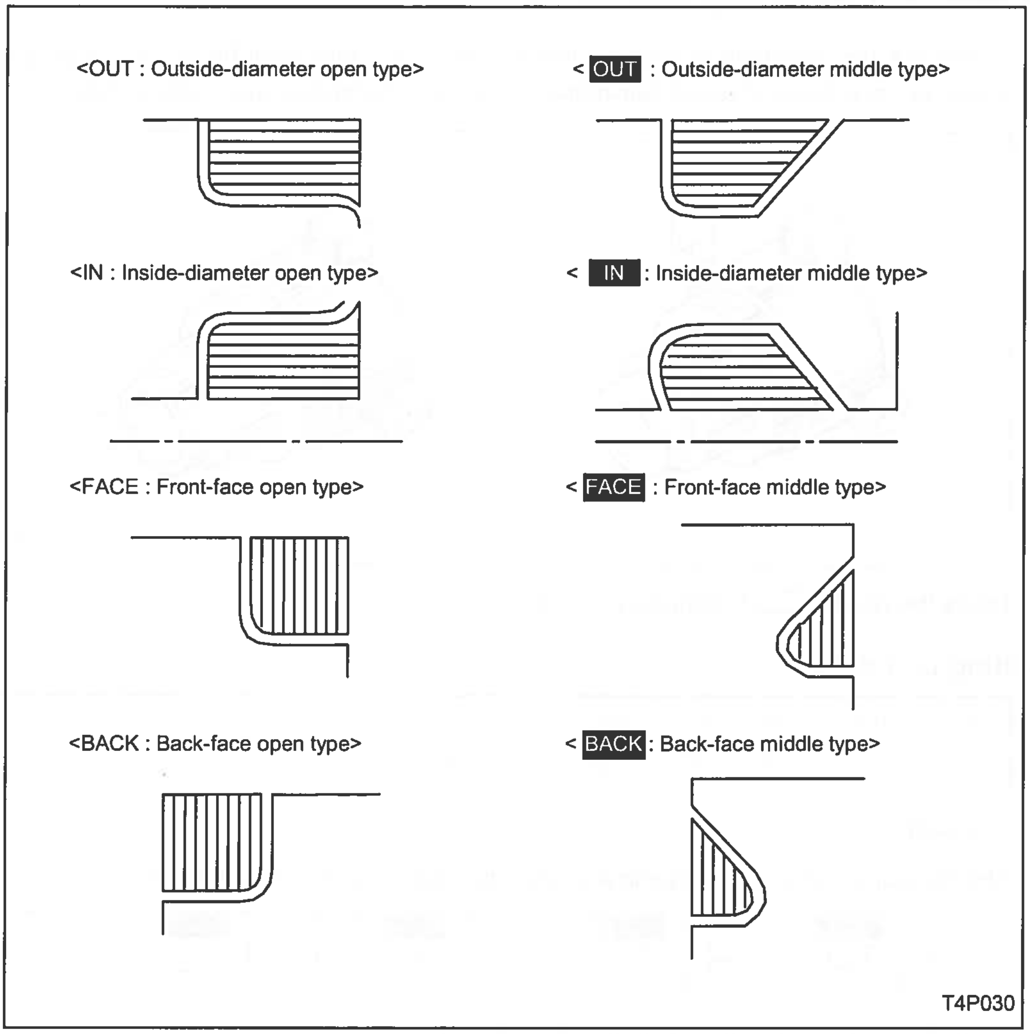
IN : Inner periphery (Cutting from the middle of the inner periphery)
.........................Inside-diameter middle type

FACE : Front face (Cutting from the outer or inner periphery)
.........................Front-face open type

FACE : Front face (Cutting from the middle of the front face)
.........................Front-face middle type

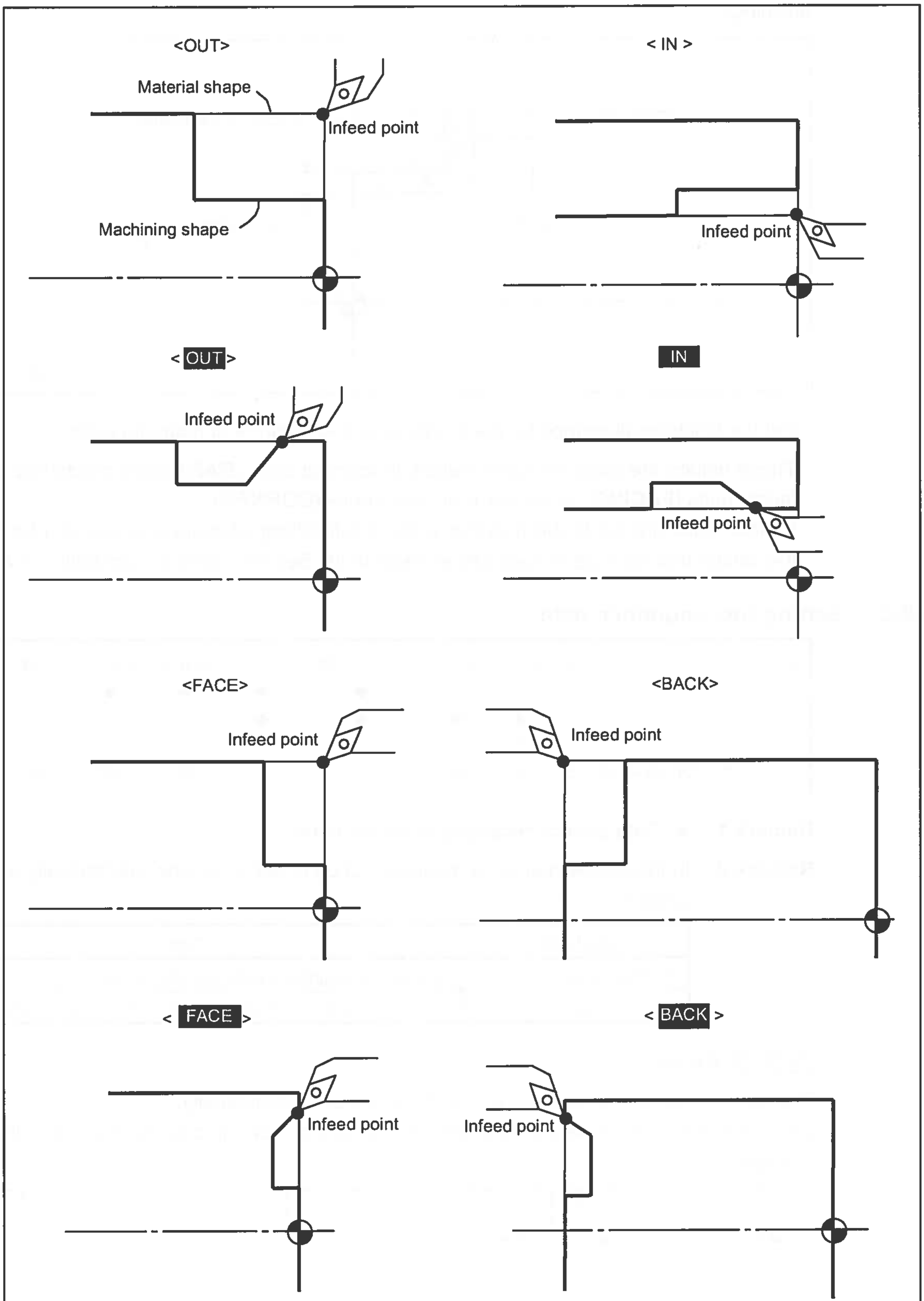
BACK : Back face (Cutting from the outer or inner periphery)
.........................Back-face open type

BACK : Back face (Cutting from the middle of the back face)
.........................Back-face middle type


<OUT : Outside-diameter open type>
< OUT : Outside-diameter middle type>
<IN : Inside-diameter open type>
< IN : Inside-diameter middle type>
<FACE : Front-face open type>
< FACE : Front-face middle type>
<BACK : Back-face open type>
< BACK : Back-face middle type>
T4P030

[2] CPT-X, [3] CPT-Z

Set the X- and Z-coordinates of the desired infeed point.

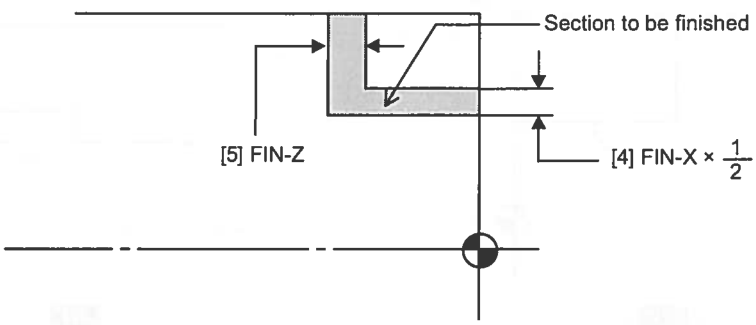


T4P036

- The infeed point refers to the cutting start point of the tool tip.
  Data thus set and sequence data automatically determine the actual cutting area.

[4] FIN-X, [5] FIN-Z

Set the finishing allowances for the X-axis and Z-axis directions (removal allowances during finishing).



T4P018

- Set the finishing allowance for the X-axis direction in terms of diameter data.
- These values are used for bar-materials machining units (**BAR**), copy-machining units (**CPY**), facing units (**FACING**), or corner-machining units (**CORNER**).
  If these units are set in the previous units, the finishing allowance values can be copied from the values that have been specified in these units. See the notes in Subsection 3-7-2.

3-8-2 Setting tool sequence data

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1						◆	◆	◆	◆				
F2				◆	◆	◆	◆						
	↑ ↑	↑ ↑ ↑	↑	↑	↑			↑	↑	↑	↑	↑	↑
	[1] [2]	[3] [4] [5]	[6]	[7]	[8]			[9]	[10]	[11]	[12]	[13]	[13]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	One tool for roughing is automatically selected.
F2 (Finishing)	Finishing allowance > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a bar-materials machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
OUT		OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
IN		IN INNER DIAMETER, IN INNER (BAK)
FACE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
FACE		EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)
BACK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys. **NOM.** is a data item that identifies tools of the same type. The tools of the same type that match in item **NOM.** and "Tool identification code" registered on the **TOOL DATA** display are used during actual machining.

Enter numeric data in item **NOM.** for the purpose of identifying tools. Although the numeric data can be either the "Nose R", "Nose angle", and/or any other characteristic factor of the tools, the data must be the same as that of the desired tools registered on the **TOOL DATA** display.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Simultaneous machining No., balanced cutting, or retraction position of the lower turret)

For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify either the simultaneous machining number or balanced cutting.
It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed. When specifying the simultaneous machining number, enter the number directly from the keyboard, not using the menu:

	BALANCE FEED 2								TURRET 2 ESCAPE

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

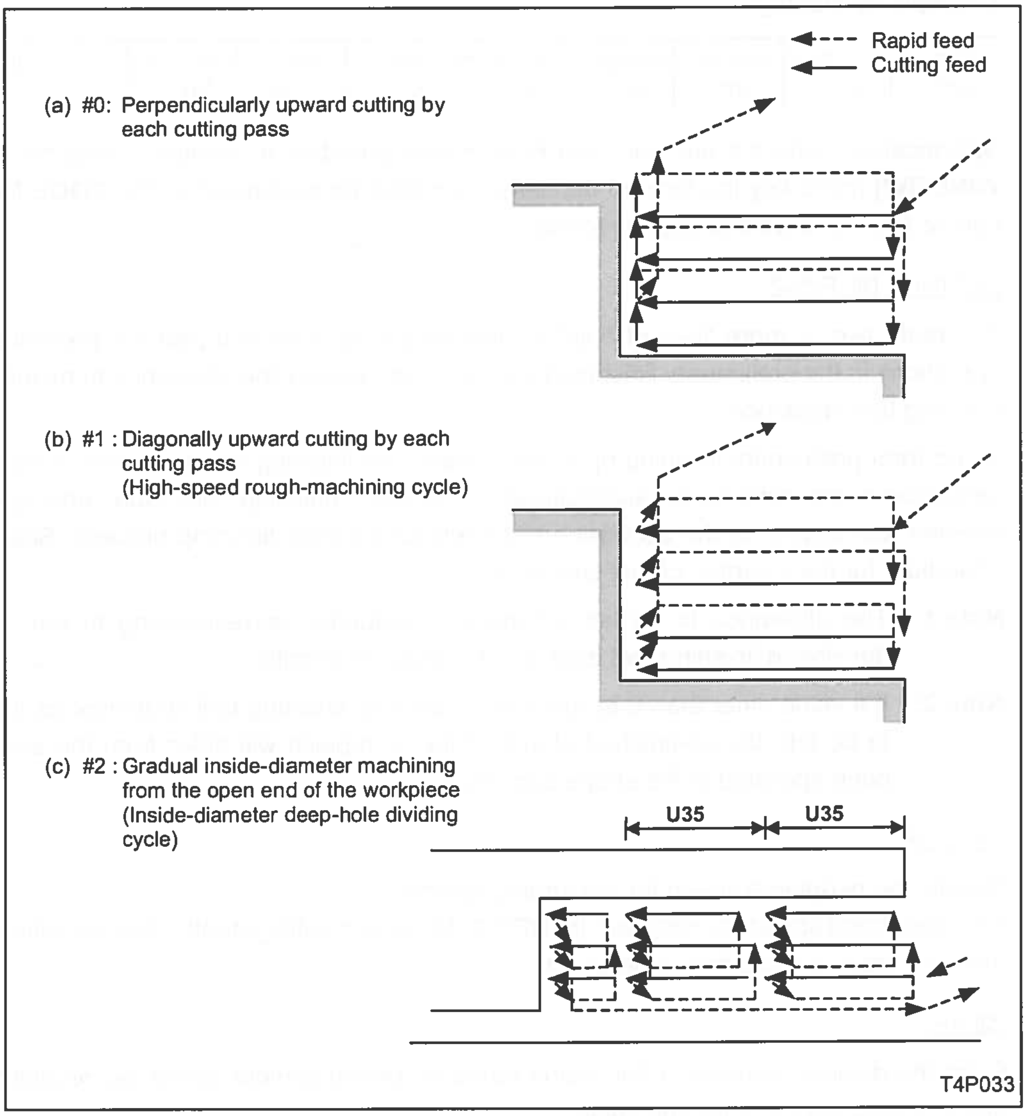
[7] PAT. (Machining pattern)

The following menu will be displayed when the cursor is placed at this item.

#0	#1	#2							

(a) (b) (c)

Select the machining pattern from (a) through (c) above. The data of the displayed menu denote the following machining patterns:



#2 machining can be used only when **IN** is selected for item [1] of the unit.
The #2 cycle, however, cannot be selected to turn inside diameter for a shape of which the size increases with the depth.
Chips may clog the hole bottom during conventional inside-diameter machining of deep holes. No such problems occur with this machining pattern (#2) since gradual cutting from the open end of a workpiece ensures highly efficient, automatic removal of chips.
Use parameter **U35** to specify the depth of cut per pass.

[8] DEP-1 (Maximum cutting depth)

Specify the maximum cutting depth per roughing pass. The maximum cutting depth in the X-axial direction is to be specified in terms of radius.

For automatic setting of items [8] **DEP-1**, [11] **C-SP**, and [12] **FR**, select the corresponding tool material from the menu.
The tool materials that have been specified in the cutting conditions item (workpiece materials/tool materials) are listed in the menu.
To register new tool materials, refer to "CUTTING CONDITION Display" of the Operating Manual.

Example of display:

CARBIDEL AUTO	UNINTRPT AUTO	COATINGL AUTO	CERMET L AUTO	CERAMICL AUTO	CBN L AUTO	HSS D AUTO	CARBIDED AUTO	>>>	TOOL DAT WINDOW

Specification using the numeric data keys is also possible. In addition, using the **[TOOL DAT WINDOW]** menu key the tools of the same type that are registered on the **TOOL DATA** display can be listed in a window display format.

[9] FIN-X, [10] FIN-Z

To create two or more lines of finishing tool sequence data and perform preliminary finishing operations in the preliminary finishing tool sequence, specify the allowance to be left for the next finishing tool sequence.

To perform preliminary finishing operations, insert the finishing tool sequence in front of the tool sequence corresponding to automatically developed finishing tool data, and specify in the inserted tool sequence the allowance to be left for the next finishing process. See Section 6-3 "Insertion" for the insertion of tool sequences.

Note 1: The allowance to be left for the tool sequence corresponding to the automatically developed finishing tool data is set to 0 automatically.

Note 2: If a value other than 0 is specified in the final finishing tool sequence as the allowance to be left, the as-finished shape of the workpiece will differ from the shape that has been specified in the shape sequence.

[11] C-SP

Specify the peripheral speed for the turning spindle.
This peripheral speed, as with item [8] **DEP-1** (Maximum cutting depth), can be selected from the menu or entered using the numeric data keys.

[12] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution. Use the numeric data keys to enter the value.
For the roughing tool sequence, this feedrate, as with items [8] **DEP-1** (Maximum cutting depth) and [11] **C-SP**, can be selected from the menu or entered using the numeric data keys.

[13] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).

Select the desired code from the menu (the codes displayed vary from model to model) or enter the desired code using the numeric data keys.

3-8-3 Setting shape sequence data

FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] SHP

The following menu will be displayed when the cursor is placed at this item.

LIN	TPR				CENTER				SHAPE END
(a)	(b)	(c)	(d)		(e)				(f)

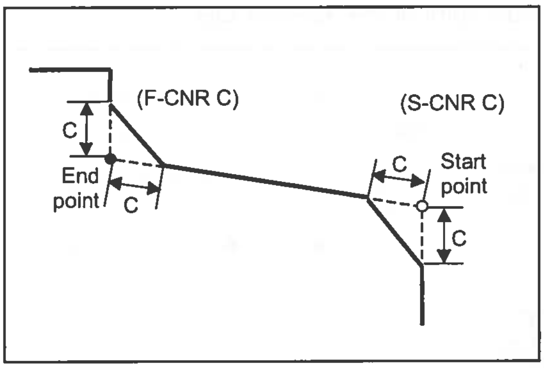
Select the type of machining shape pattern from the above four types (a) to (d).

Menu function	Description
(a)	Select to specify the straight line parallel to the center line of the workpiece.
(b)	Select to specify a straight line not parallel to the center line of the workpiece (namely, a taper).
(c)	Select to specify a bulged arc.
(d)	Select to specify a recessed arc.
(e)	Select to use the automatic crossing-point calculation function on the bulged (convex) arc or recessed (concave) arc drawn on the previous sequence line.
(f)	Select to proceed to the next unit after entering all shape data.

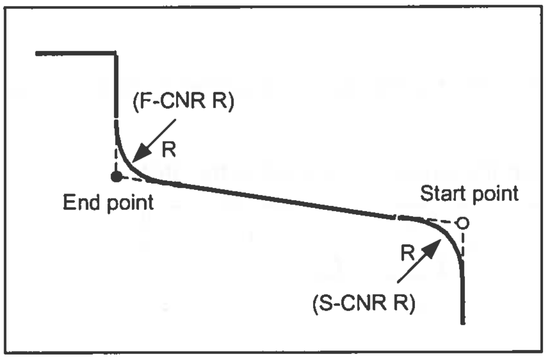
[2] S-CNR

Set data for this item when C-chamfering (cornering) or R-chamfering (rounding) is to be done at the start point of the shape.

- If C-chamfering is to be done:
  Set the amount of chamfering (C in the diagram).

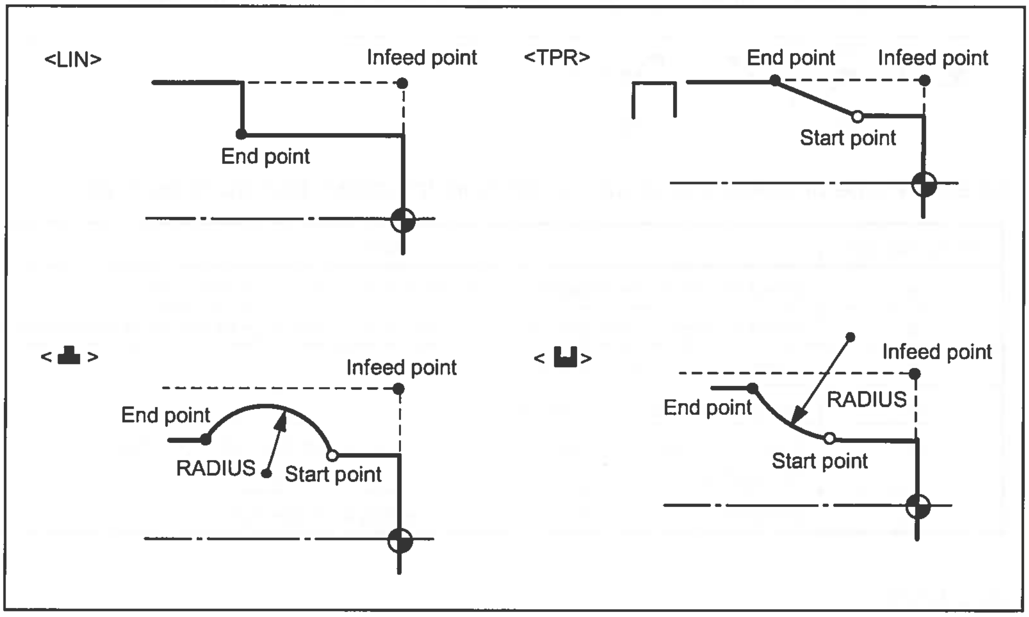


- If R-chamfering is to be done:
  After pressing the **[CORNER R]** menu key, set the radius of rounding (R in the diagram).

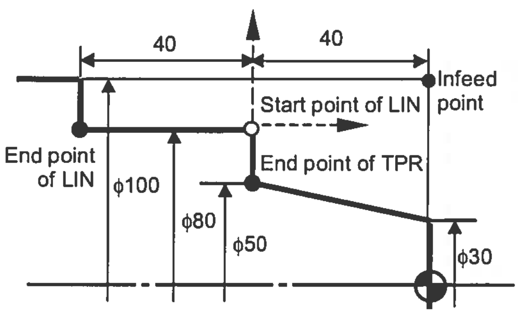


[3] SPT-X, [4] SPT-Z, [5] FPT-X, [6] FPT-Z

- Set the coordinates of the start and end points of the shape you selected in item [1] above. The terms “start” and “end” refer to the infeed point.
- In **[CENTER]** was selected in item [1], set the central coordinates of the arc.
  If the crossing point cannot be found, press the **[INTER PT]** or **[CONT PT]** menu key.
  See “Automatic Crossing-Point Calculation Function” for further details.



- If the selected shape type is **LIN**, the coordinates of the start point need not be set.
  The NC unit will auto-set those coordinates. A horizontal line will be drawn from the end point of **LIN** towards the infeed point, and the crossing point of this line and the line that is perpendicularly drawn from the end point of the preceding **FIG** (or from the infeed point for an **LIN** as the first **FIG**) will be set as the start point of the relevant **LIN**.



UNIT	PART	CPT-X	CPT-Z
BAR	OUT	100.	0.

FIG	SHP	SPT-X	SPT-Z	FPT-X	FPT-Z
	TPR	30.	0.	50.	40.
	LIN	◆	◆	80.	80.

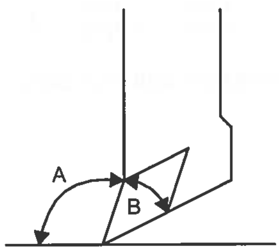
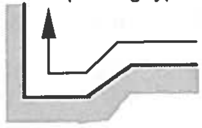
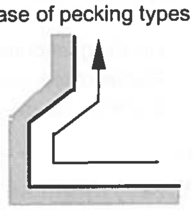
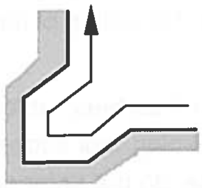
[7] F-CNR/$

Set data for this item when C-chamfering, R-chamfering, or pecking is to be done at the end point of the shape.

The following menu will be displayed when the cursor is placed at this item:

CORNER R	$1	$2	$3	$4	$5	$6			
	(a)	(b)	(c)	(d)	(e)	(f)			

- The setting procedure for **F-CNR** is the same as for **S-CNR** (refer to the description and diagram for item [2] **S-CNR**).
- If pecking is to be done, select a pecking type from (a) through (f) above.
  Pecking types $4, $5 and $6 are the same as $1, $2 and $3, respectively. Use parameters **K33** to **K40** to select the dimensions of pecking. (See the Parameter List for further details.)
- Pecking can be done only if the following conditions are satisfied:
  1) During finishing.
  2) The selected shape and the next shape are linear and orthogonal.
  3) The cutting-edge angle and tip angle of the tool to be used satisfy the conditions listed in the table below.

<table>
<tr><td rowspan="3">A: Tool blade angle
B: Tool tip angle
T4P047</td><td>In the case of pecking types $1, $4:
</td><td>A ≥ 93°
B ≤ 57°
A + B ≤ 150°</td></tr>
<tr><td>In the case of pecking types $2, $5:
</td><td>A ≥ 120°
B ≤ 57°
A + B ≤ 177°</td></tr>
<tr><td>In the case of pecking types $3, $6:
</td><td>A ≥ 120°
B ≤ 30°
A + B ≤ 150°</td></tr>
</table>

[8] RADIUS/th

- If you selected or for item **SHP** [1] above, set the radius of the desired circle (see the diagram shown previously for items [3] to [6]).
- If you selected T**PR** for item [1] above and typed the question mark "?" for one of the four items from [3] to [6], set a taper angle.
  See "Function of automatically calculating a point of intersection" for details.
- Data setting is not required in any other cases (◆ mark will be displayed for this item).

[9] RGH

Set a finishing feedrate appropriate for particular surface roughness.

For setting a finishing feedrate, two methods are available: selection of a surface roughness code (for this case, the NC unit automatically calculates the appropriate feedrate for the selected surface roughness code), and direct setting of any desired feedrate.
The following menu will be displayed when the cursor is placed at this item:

ROUGHNES	FEEDRATE /rev								

- If a surface roughness code is to be selected:
  The code can be entered either by setting the desired code number directly with numerical keys or using the following procedure:
  1) First, press the **[ROUGHNES]** menu key. The following menu will be displayed:

▼ 1	▼ 2	▼▼ 3	▼▼ 4	▼▼▼ 5	▼▼▼ 6	▼▼▼ 7	▼▼▼▼ 8	▼▼▼▼ 9	

  2) Next, from the above menu, select one of the surface roughness codes indicated on the machining drawing. The above codes of the displayed menu denote the following levels of surface roughness:

▼ 1	▼ 2	▼▼ 3	▼▼ 4	▼▼▼ 5	▼▼▼ 6	▼▼▼ 7	▼▼▼▼ 8	▼▼▼▼ 9	
↓	↓	↓	↓	↓	↓	↓	↓	↓	
100	50	25	12.5	6.3	3.2	1.6	0.8	0.4	Surface roughness (μm)
(100-S ▽	50-S ▽	25-S ▽▽	12-S ▽▽	6-S ▽▽▽	3-S ▽▽▽	1.5-S ▽▽▽	0.8-S ▽▽▽▽	0.4-S ▽▽▽▽)	finishing symbols

  The finishing feedrate is calculated from the following expression automatically:

$$F = \sqrt{\frac{8R\mu}{1000}}$$

F : Finishing feedrate (mm/rev)
R : Radius of tool nose (mm)
μ : Surface roughness (μm)

- If a feedrate is to be directly set:
  After pressing the **[FEEDRATE/rev]** menu key, set the desired value.

Note 1: The feedrate that has been specified in this item is incorporated only during the finishing process, and the setting of item [12] **FR** in the tool sequence is used during the roughing process.

Note 2: The feedrate data that has been entered in this item takes priority over the setting of item [12] **FR** in the finishing tool sequence. If the feedrate is to be changed with each finishing tool sequence, do not enter data in this item.

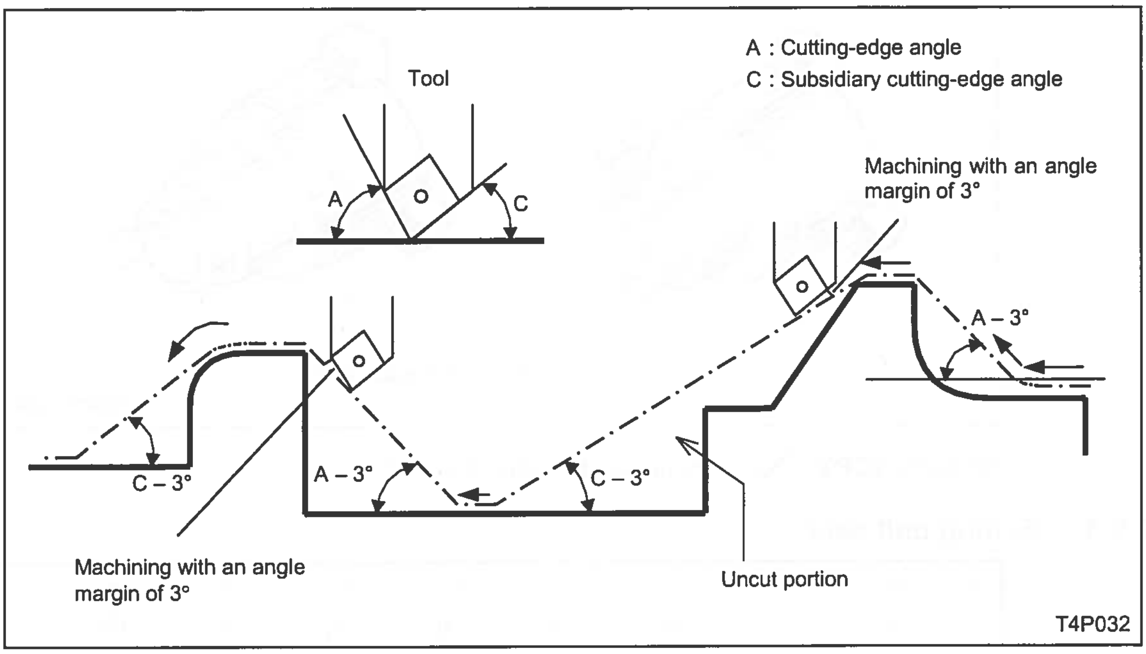
Note 3: If no data is entered in this item, the settings of feedrates in item [12] **FR** of each tool sequence will be used for each machining operation.

Note 4: The same value will automatically be set here if the preceding sequence has a set data of **RGH**.

<Precautions for BAR unit>

Some parts may remain uncut because of the tool shape.

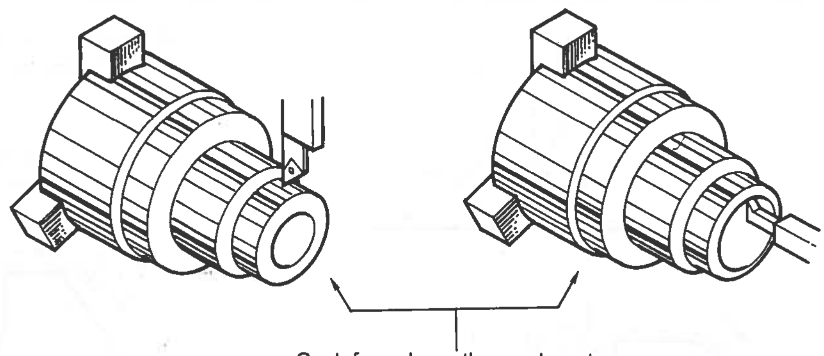
For a tool having a cutting-edge angle of A and a subsidiary cutting-edge angle of C, uncut portions occur at an angle of A – 3° in the machining direction and an angle of C – 3° in an opposite direction because machining will occur generally with an angle margin of 3°.



* The description given above also applies to **CPY** unit.

3-9 Copy-Machining Unit (CPY)

Select the copy-machining unit when cast, forged, or other such parts are to be cut along their profiles.



Cast, forged, or other such parts

NM210-00430

Press the **[CPY]** menu key to select this unit.

3-9-1 Setting unit data

UNo.	UNIT	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	FIN-X	FIN-Z
*	CPY	[1]	[2]	[3]	[4]	[5]	[6]	[7]

<u>[1] PART</u>

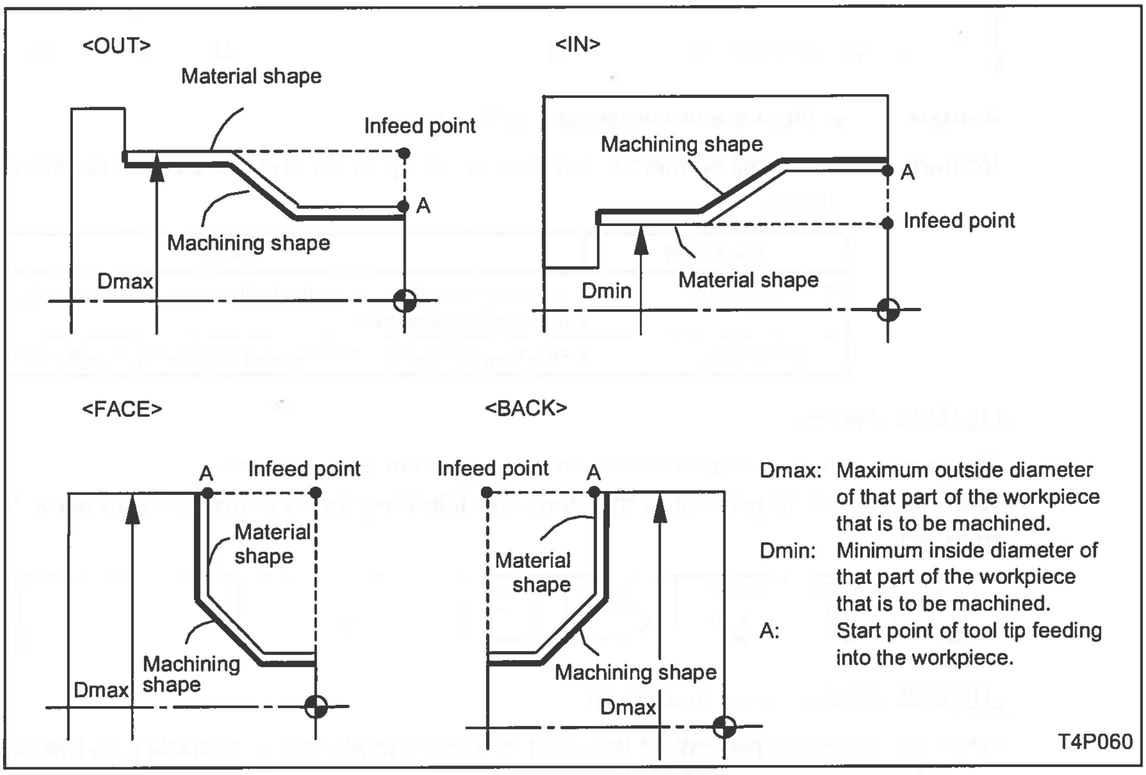
The following menu will be displayed when the cursor is placed at this item.

OUT	OUT	IN	IN	FACE	FACE	BACK	BACK		

From the menu, select the section to be machined. The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (**BAR**).

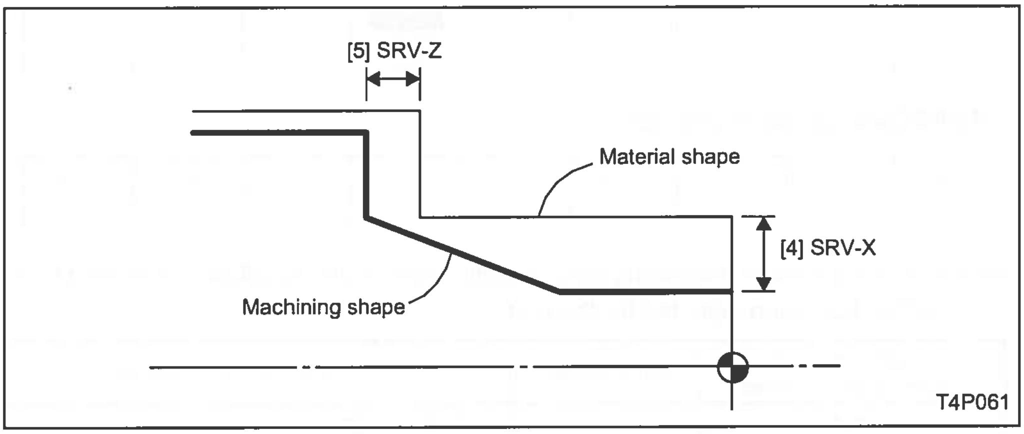
[2] CPT-X, [3] CPT-Z

Set the X- and Z-coordinates of the desired infeed point.
In general, the infeed point means the start point of tool tip feeding into a workpiece. For copy-machining units, however, the points shown in the diagrams below are infeed points.



[4] SRV-X, [5] SRV-Z

Set the maximum removal allowances in the X-axis and Z-axis directions (removal allowances for the section that is to be cut most deeply).
The removal allowance in the X-axis direction must be set with the radius value (half the workpiece thickness).



[6] FIN-X, [7] FIN-Z

Set the finishing allowances for the X-axis and Z-axis directions (removal allowances during finishing). The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (**BAR**).

3-9-2 Setting tool sequence data

SNo.	TOOL		NOM.			#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1							◆		◆	◆	◆	◆				
F2							◆	◆	◆	◆						
	↑	↑	↑	↑	↑	↑		↑			↑	↑	↑	↑	↑	↑
	[1]	[2]	[3]	[4]	[5]	[6]		[7]			[8]	[9]	[10]	[11]	[12]	[12]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	Removal allowance > Finishing allowance: One tool for roughing is automatically selected.
F2 (Finishing)	Finishing allowance > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[1] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a copy-machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
OUT		OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
IN		IN INNER DIAMETER, IN INNER (BAK)
FACE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
FACE		EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)
BACK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant items for **BAR** unit.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Simultaneous machining No., balanced cutting, or retraction position of the lower turret)

For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify either the simultaneous machining number or balanced cutting.
It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed. When specifying the simultaneous machining number, enter the number directly from the keyboard, not using the menu:

	BALANCE FEED 2								TURRET 2 ESCAPE

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] DEP-1 (Maximum cutting depth)

Specify the maximum cutting depth per roughing pass. The maximum cutting depth in the X-axial direction is to be specified in terms of radius.
See the description of the relevant item for **BAR** unit.

[8] FIN-X, [9] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for **BAR** unit.

[10] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for **BAR** unit.

[11] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for **BAR** unit.

[12] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change). See the description of the relevant item for **BAR** unit.

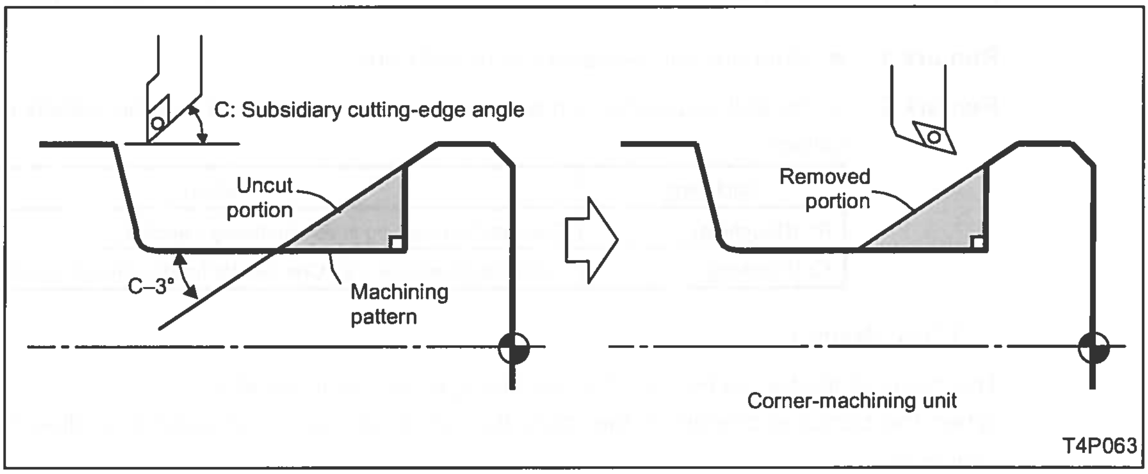
3-9-3 Setting shape sequence data

FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

The shape sequence data for the copy-machining unit is the same as that for the bar-materials machining unit. See the description of the relevant item in Section 3-8 "Bar-Materials Machining Unit (BAR)".

3-10 Corner-Machining Unit (CORNER)

Part of the corners of a workpiece may remain uncut because of the particular tool shape for the bar-materials machining unit (**BAR**) or the copy-machining unit (**CPY**). Select the corner-machining unit (**CORNER**) when uncut portions are to be removed to make all corners right-angled.



Press the **[CORNER** ⊿ **]** menu key to select this unit.

3-10-1 Setting unit data

UNo.	UNIT	PART	FIN-X	FIN-Z
*	CORNER	[1]	[2]	[3]

[1] PART

The following menu will be displayed when the cursor is placed at this item.

OUT	IN	FACE	BACK						

From the menu, select the section to be machined.

Sections to be machined that correspond to the data of the displayed menu are as follows.

OUT : Uncut portion on outer periphery
IN : Uncut portion on inner periphery
FACE : Uncut portion on front face
BACK : Uncut portion on back face

[2] FIN-X, [3] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for **BAR** unit.

3-10-2 Setting tool sequence data

```
SNo.  TOOL   NOM.   #  PAT. DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP  FR  M  M
 R1                                ◆          ◆      ◆      ◆
 F2                      ◆    ◆    ◆          ◆
      ↑    ↑    ↑ ↑ ↑  ↑    ↑    ↑                  ↑      ↑      ↑     ↑   ↑  ↑
     [1]  [2]  [3][4][5][6] [7]  [8]               [9]    [10]   [11]  [12][13][13]
```

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	One tool for roughing is automatically selected.
F2 (Finishing)	Finishing allowance > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a corner-machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FACE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for **BAR** unit.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Simultaneous machining No., balanced cutting, or retraction position of the lower turret)

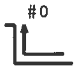
For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify either the simultaneous machining number or balanced cutting.
It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed. When specifying the simultaneous machining number, enter the number directly from the keyboard, not using the menu:

	BALANCE FEED 2								TURRET 2 ESCAPE

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] # (Machining pattern)

The following menu will be displayed when the cursor is placed at this item.

#0 	#1 								
(a)	(b)								

Select a rough-machining pattern from (a) or (b) above.
See the description of the relevant item for **BAR** unit.

[8] DEP-1 (Maximum cutting depth)

Specify the maximum cutting depth per roughing pass. The maximum cutting depth in the X-axial direction is to be specified in terms of radius.
See the description of the relevant item for **BAR** unit.

[9] FIN-X, [10] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for **BAR** unit.

[11] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for **BAR** unit.

[12] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for **BAR** unit.

[13] M

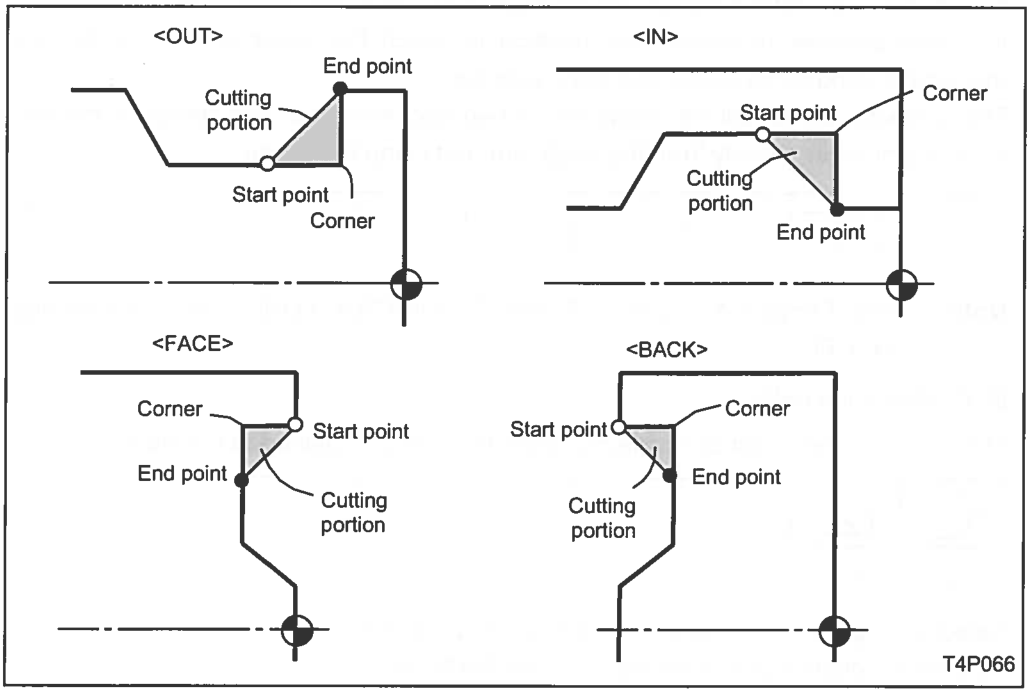
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

3-10-3 Setting shape sequence data

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]

[1] SPT-X, [2] SPT-Z, [3] FPT-X, [4] FPT-Z

Set the coordinates of the desired start and end points of cornering.
The position of the start point and the end point are shown below.



[5] F-CNR/$

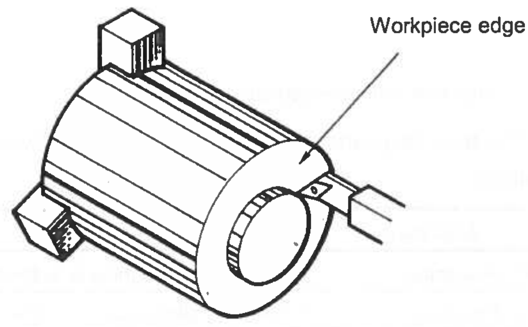
This item applies to the corners shown in the figure below, not the ending-point section. Enter data in this item to perform C-chamfering, R-chamfering, or polish-necking operations.
For data setting method, see the relevant items for **BAR** unit.

[6] RGH

Set the appropriate, finish-machining feedrate for particular finishing surface roughness.
For data setting method, see the relevant items for **BAR** unit.

3-11 Facing Unit (FACING)

Select the facing unit (**FACING**) when chipping off any protrusions of the workpiece edges (front face or back face).



NM210-00431

Press the **[FACING]** menu key to select this unit.

3-11-1 Setting unit data

UNo.	UNIT	PART	FIN-Z
*	FACING	[1]	[2]

[1] PART

The following menu will be displayed when the cursor is placed at this item.

FACE	BACK								

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FACE : Right edge of the workpiece
BACK : Left edge of the workpiece

Note: The **[BACK]** menu item may not be selectable for special machine specifications.

[2] FIN-Z

Set the finishing allowances for the Z-axis directions (removal allowances during finishing). See the description of the relevant item for **BAR** unit.

3-11-2 Setting tool sequence data

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1				◆		◆	◆	◆	◆				
F2				◆	◆	◆	◆	◆					
	↑ ↑	↑ ↑ ↑	↑		↑				↑	↑	↑	↑	↑
	[1] [2]	[3] [4] [5]	[6]		[7]				[8]	[9]	[10]	[11]	[11]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	One tool for roughing is automatically selected.
F2 (Finishing)	Finishing allowance > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a facing unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
FACE	GENERAL	OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for **BAR** unit.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Simultaneous machining No. or retraction position of the lower turret)

For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify the simultaneous machining number. It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] DEP-1 (Maximum cutting depth)

Specify the maximum cutting depth per roughing pass. The maximum cutting depth in the X-axial direction is to be specified in terms of radius.
See the description of the relevant item for **BAR** unit.

[8] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for **BAR** unit.

[9] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for **BAR** unit.

[10] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for **BAR** unit.

[11] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

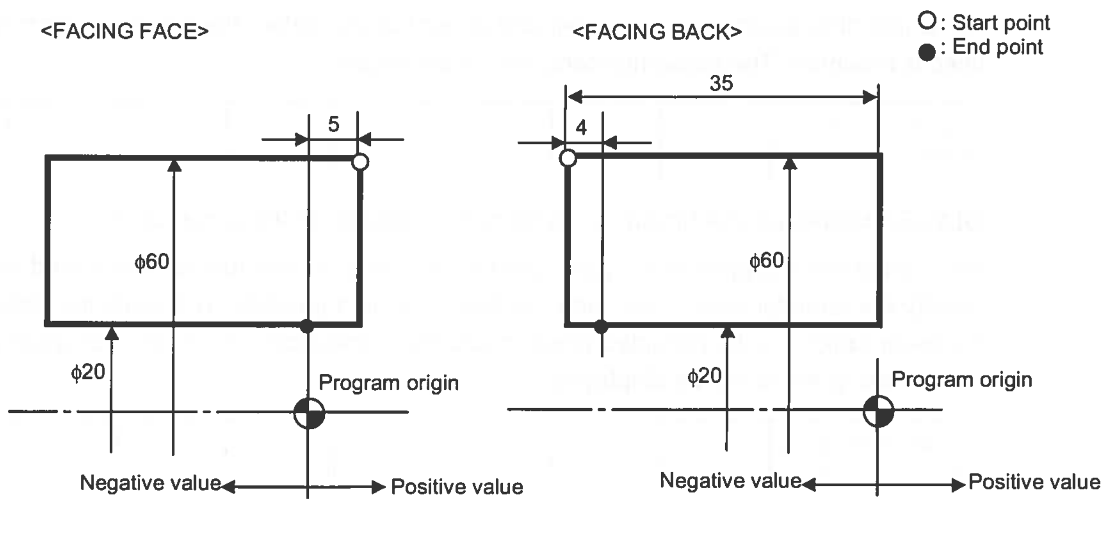
3-11-3 Setting shape sequence data

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	RGH
1	[1]	[2]	[3]	[4]	[5]

[1] SPT-X, [2] SPT-Z, [3] FPT-X, [4] FPT-Z

Set the coordinates of the machining start point and end point.

For the facing unit, set as a plus value the Z-coordinates of all points located to the right of the program zero-point, or set as a minus value the Z-coordinates of all points located to the left of the program origin.



SPT-X	SPT-Z	FPT-X	FPT-Z
60.	5.	20.	0.

SPT-X	SPT-Z	FPT-X	FPT-Z
60.	-35.	20.	-31.

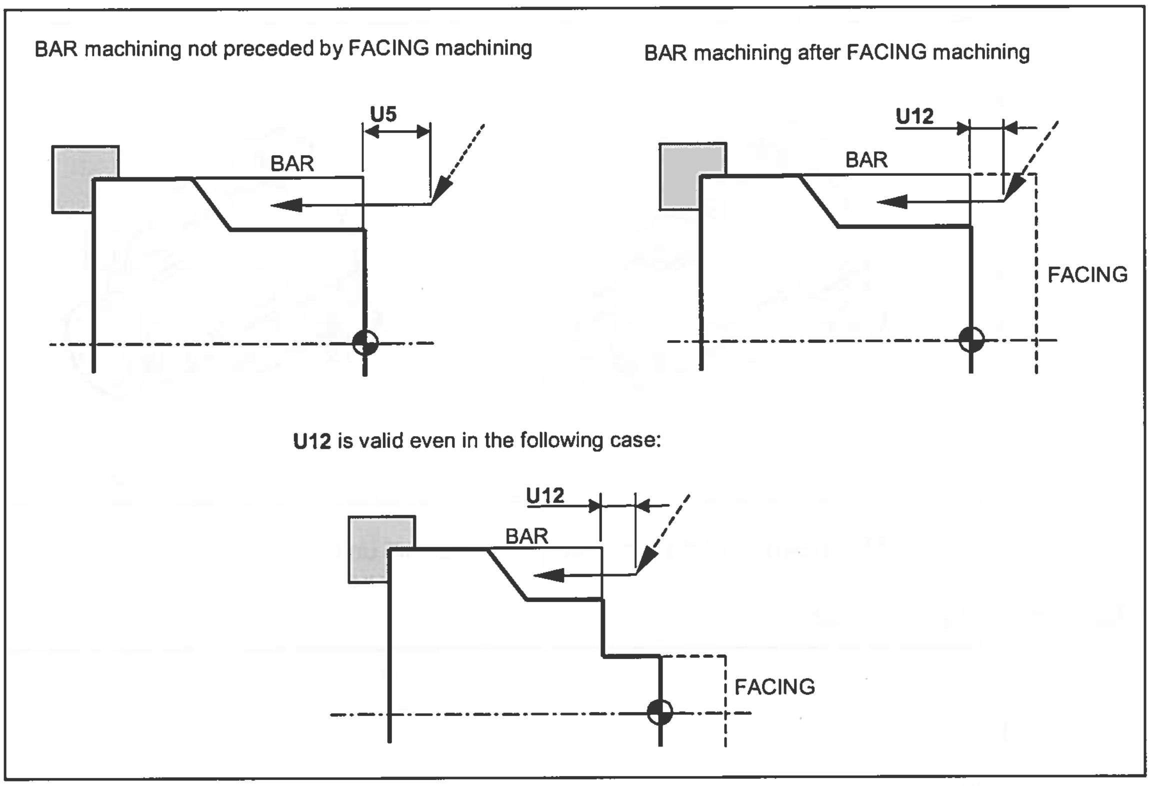
[5] RGH

Set the appropriate finish-machining feedrate for particular finishing surface roughness.

This setting can be done by selecting a surface roughness code or by directly setting any desired feedrate.

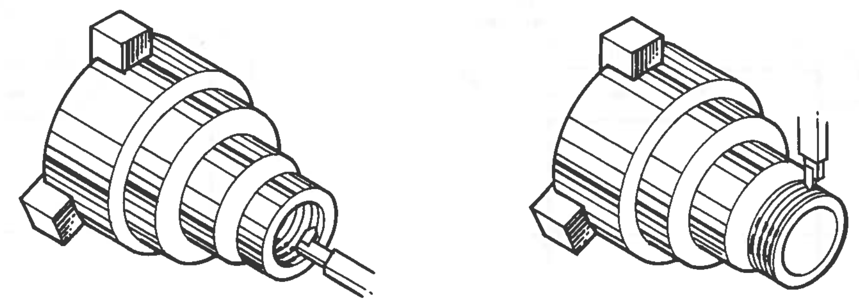
See related items of “Bar-materials machining unit (**BAR**)”.

Note: The tool path for **BAR** and **CPY** units is calculated in general with the end-face clearance set in parameter **U5**.
For a **BAR** or **CPY** unit preceded by a **FACING** unit, in particular, the value of parameter **U12** is used as the end-face clearance.



3-12 Threading Unit (THREAD)

Select the threading unit to thread the outer peripheries, inner peripheries or front faces or back faces of a workpiece.



NM210-00432

Press the **[THREAD** -\/\/- **]** menu key to select this unit.

3-12-1 Setting unit data

UNo.	UNIT	PART	CHAMF	LEAD	ANG	MULTI	HGT
*	THREAD	[1]	[2]	[3]	[4]	[5]	[6]

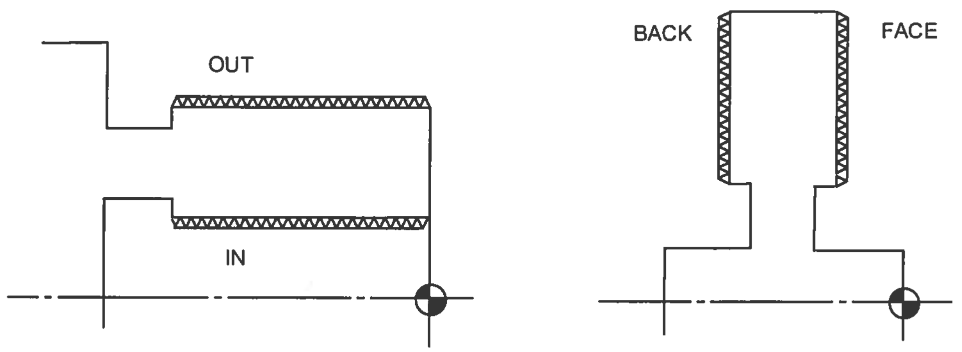
[1] PART

The following menu will be displayed when the cursor is placed at this item.

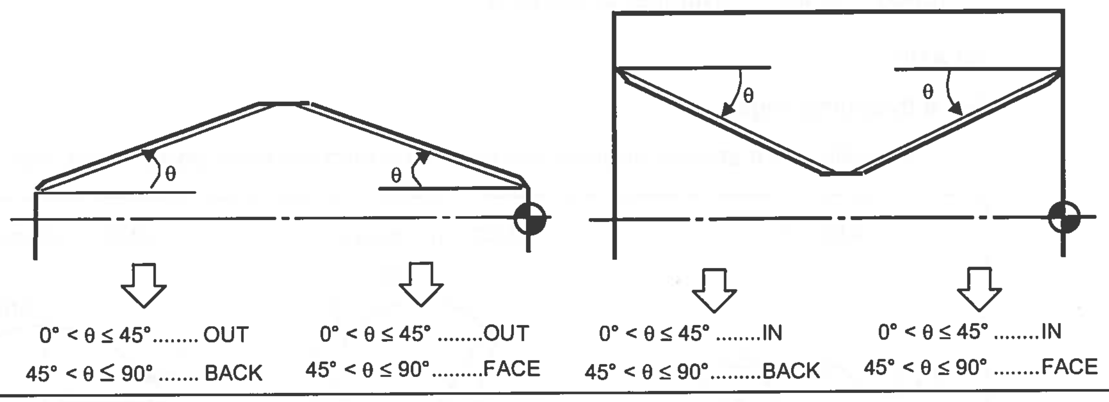
OUT	IN	FACE	BACK						

From the menu, select the section to be machined. Sections to be machined that correspond to the data of the displayed menu are as follows.

OUT : Outer periphery (male thread)
IN : Inner periphery (female thread)
FACE : Right edge of the workpiece (front face)
BACK : Left edge of the workpiece (back face)



- For taper threading, select the appropriate machining section as follows according to the desired taper angle:

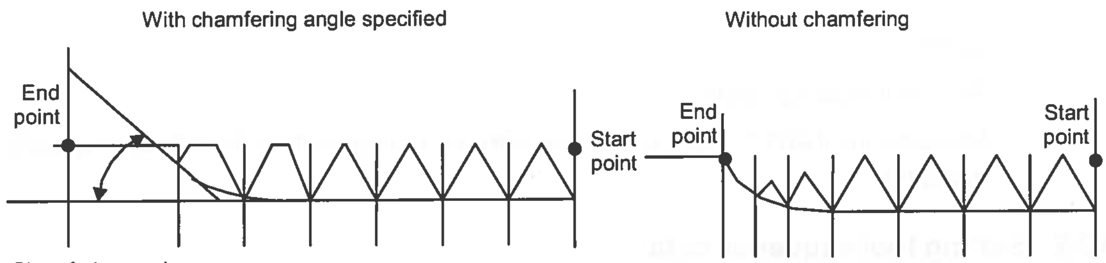


[2] CHAMF

Set a chamfering angle for the section you want to thread.

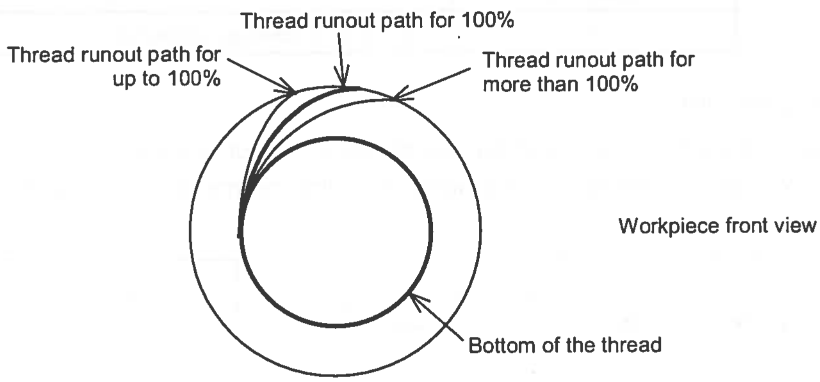
- Set 0 if chamfering is not required.
- Set 1 for a chamfering angle of 45 degrees.
- Set 2 for a chamfering angle of 60 degrees.

Designate chamfering to maintain the lead up to the ending point of threading.



Use the parameter **K19** to specify the chamfering amount.

Note: For the machine specifications with a threading start position automatic correction option, when the spindle override ratio is updated during the threading process, if no chamfering is required, the thread runout path will change. The thread runout speed will increase for a spindle override ratio up to 100%, or decrease for a spindle override ratio greater than 100%.



Examples of changes in the thread runout path by spindle override ratio updating

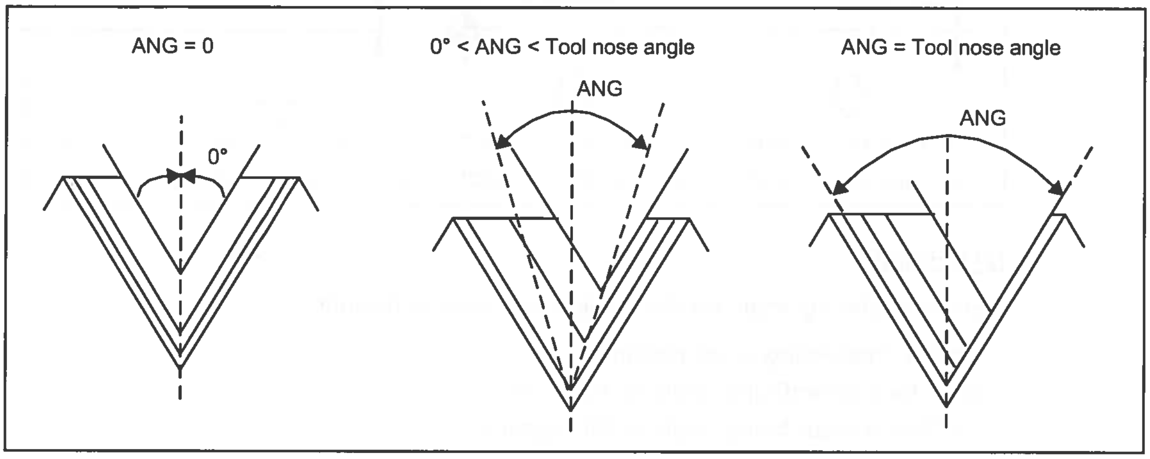
[3] LEAD

Set the threading lead given by the following expression:
(Lead) = (Pitch) × (Number of threads)

[4] ANG

Set a threading angle.

- Usually, set a several degrees smaller value than the nose angle of the tool.



[5] MULTI

Set the desired number of threads.

[6] HGT

Set the threading height.

Pressing the **[AUTO SET]** menu key with the cursor at item [6] will automatically set data into items [6].

3-12-2 Setting tool sequence data

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
1							◆	◆	◆		◆		
	↑ ↑	↑ ↑ ↑	↑	↑	↑	↑				↑		↑	↑
	[1] [2]	[3] [4] [5]	[6]	[7]	[8]	[9]				[10]		[11]	[11]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a threading unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	THREAD	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FACE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for **BAR** unit.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Retraction position of the lower turret)

For a machine equipped with upper and lower turrets, specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.

The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] PAT. (Machining pattern)

The following menu will be displayed when the cursor is placed at this item:

#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA	#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA				
(a)	(b)								

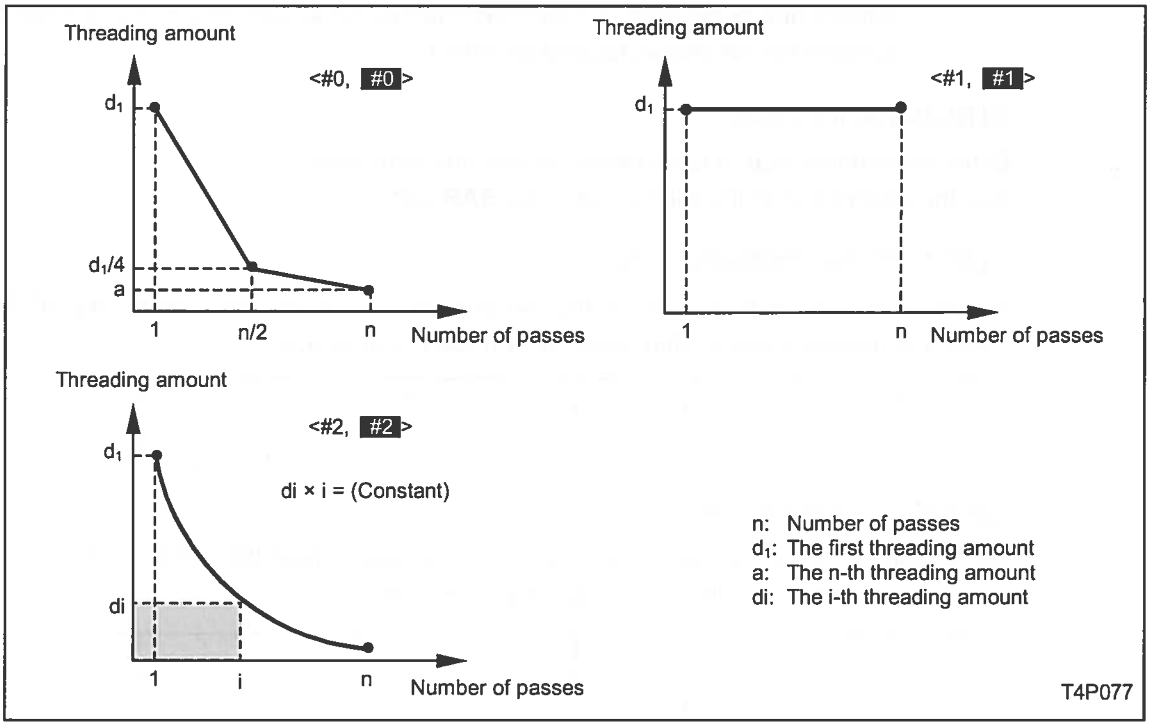
From the menu, select the threading pattern to be machined.

Patterns to be machined that correspond to the data of the displayed menu are as follows.

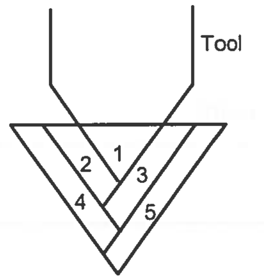
#0, #0: Standard patterns
The threading amount gradually decreases as the number of passes increases.

#1, #1: Constant-threading pattern
The threading amount always stays constant, irrespective of the number of passes.

#2, #2: Area-constant pattern
The threading amount is inversely proportional to the number of passes.



Note: If you select #0, #1 or #2 zigzag threading (alternate threading with the left and right cutting edges) will occur unless you set a value of 30 or less in item [4], ANG in the unit data.



1 to 5: Threading order

[8] DEP-1 (First cutting depth)

Enter the first cutting depth during the threading pass. For X-axial cutting, enter this value in terms of radius. The above value can likewise be auto-set by pressing the menu key **[AUTO SET]**.

[9] DEP-2/NUM. (Number of cutting passes)

Enter the number of cutting passes (how often the threading pass is to be repeated).

Note: Specify at least three cutting passes.

[10] C-SP

Specify the peripheral speed for the tool in terms of turning spindle.
See the description of the relevant item for **BAR** unit.

[11] M

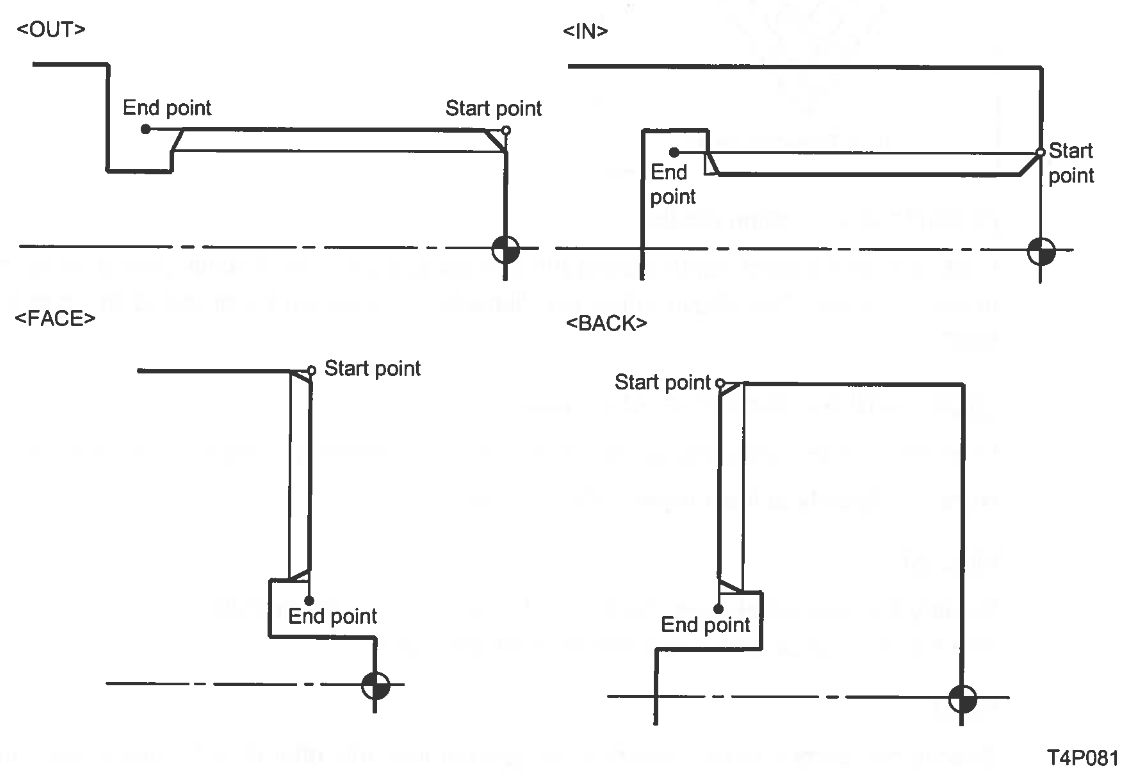
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

3-12-3 Setting sequence data

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z
1	[1]	[2]	[3]	[4]

[1] SPT-X, [2] SPT-Z, [3] FPT-X, [4] FPT-Z

Set the coordinates of the machining start point and end point.



- For usual threading, set the nominal diameter of the thread as the X-coordinate.
- Incompletely threaded portions occur near the end point of threading. Therefore, if grooves are present at the position of the end point as shown in the diagrams above, set the end point at a position slightly deeper than the section to be threaded.
- Even if the spindle override value is changed using the threading start position automatic correction option, the acceleration distance for threading will be the distance existing when the spindle override value is 100%.
  Since the use of a spindle override value exceeding 100% may result in an incomplete thread due to the insufficiency of the acceleration distance, specify a spindle override value not exceeding 100%.
  However, do not set the override value to 0%. Otherwise, operation will stop during threading.

Note 1: The continuous threading pattern shown in the diagram below can be generated by setting multiple lines of sequence data.
In that case, the coordinates of the second and subsequent start points do not need be set (items [1] and [2] will be marked with ◆).

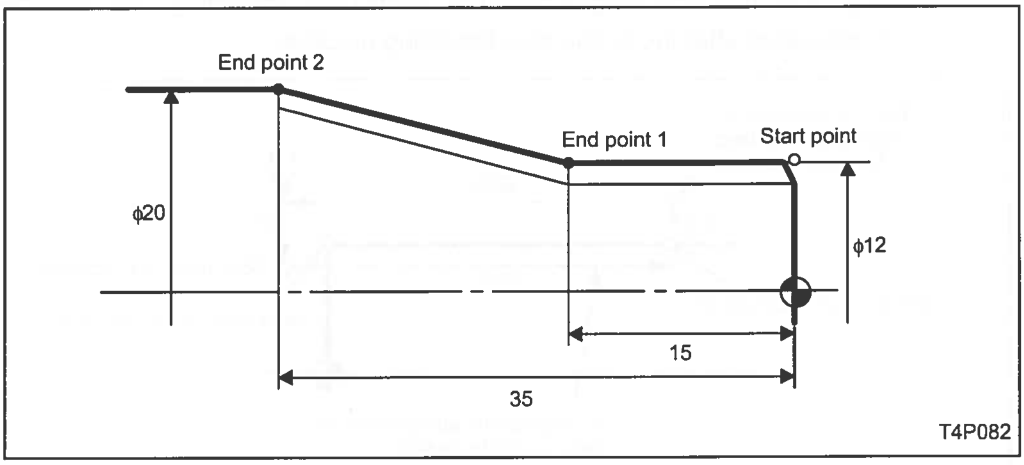
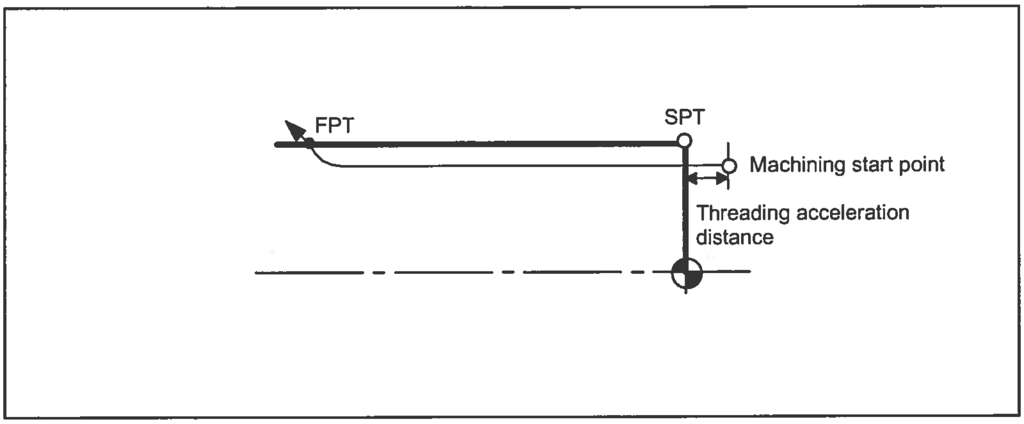


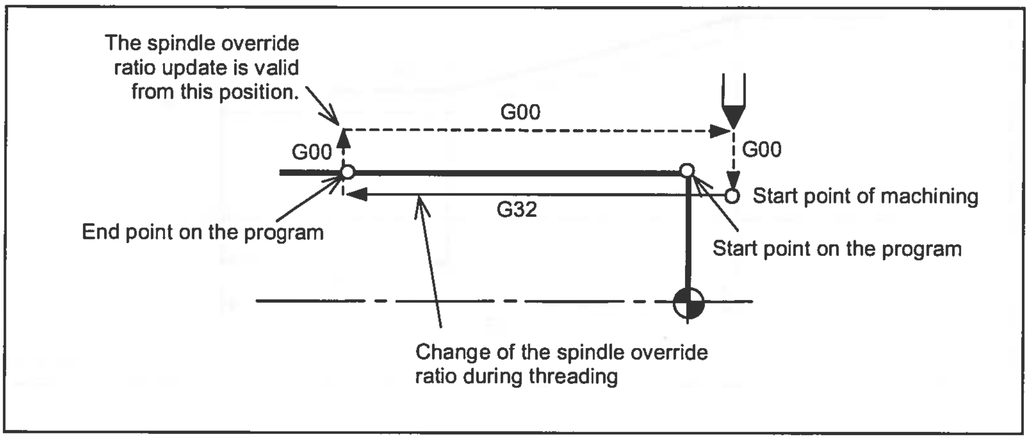
FIG	SPT-X	SPT-Z	FPT-X	FPT-Z
1	12.	0.	12.	15.
2	◆	◆	20.	35.

Note 2: Threading action begins at a position that is away from the start point specified in the program by the acceleration distance. Before carrying out a threading operation, therefore, check for possible interference with the tailstock or the workpiece during threading.



Note 3: The actuation of feed-hold function during a pass of threading will not interrupt the machine operation until the chamfering at the end point of threading has been completed.

Note 4: For the machine specifications with a threading start position automatic correction option, percentage (%) display can be changed by pressing the spindle/milling spindle override key during the threading process. The spindle speed, however, does not change. The specified percentage value is incorporated into the actual spindle speed, only after the threading block. For continuous threading, the specified value is incorporated after the continuous threading process.

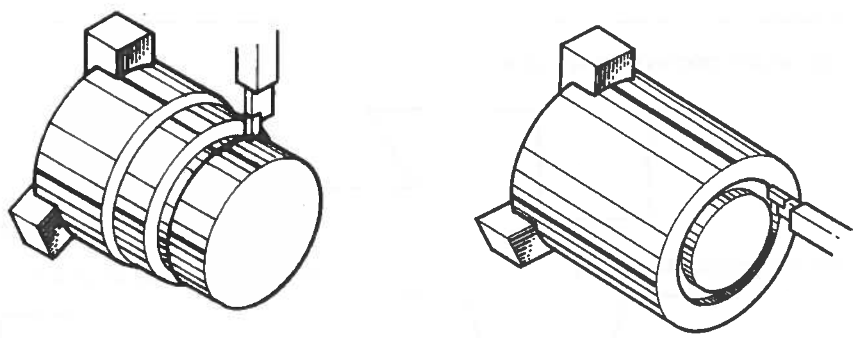


Note 5: Threading start position automatic correction and re-threading functions (both optional) are only valid for a longitudinal threading by cutting feed on the Z-axis: **THREAD OUT** or **IN**. (Invalid for **THREAD FACE** and **BACK**)

Note 6: Re-threading function (optional) is only valid for constant lead threading.

3-13 Grooving Unit (T. GROOVE)

Select the grooving unit to groove the outer peripheries, inner peripheries, front faces or back faces or to cut off workpiece.



NM210-00433

Press the **[T. GROOVE]** menu key to select this unit.

3-13-1 Setting unit data

UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FINISH
*	T.GROOVE	[1]	[2]	[3]	[4]	[5]	[6]

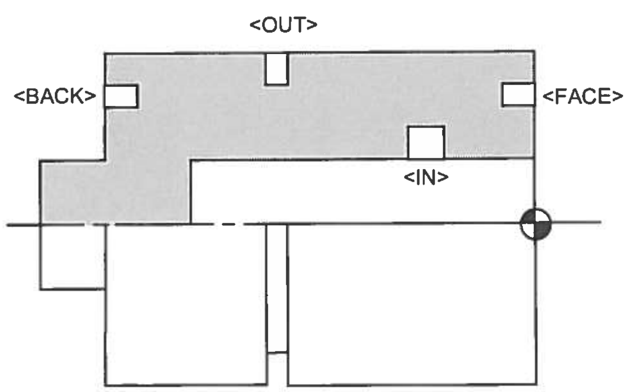
[1] PART

The following menu will be displayed when the cursor is placed at this item.

OUT	IN	FACE	BACK						

From the menu, select the section to be machined. Sections to be machined that correspond to the data of the displayed menu are as follows.

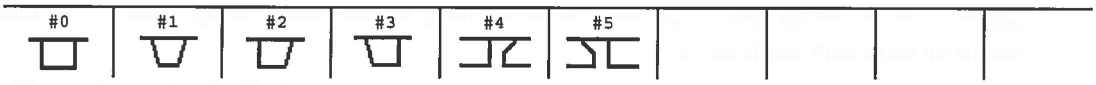
OUT : Outer periphery
IN : Inner periphery
FACE : Right edge of the workpiece (front face)
BACK : Left edge of the workpiece (back face)



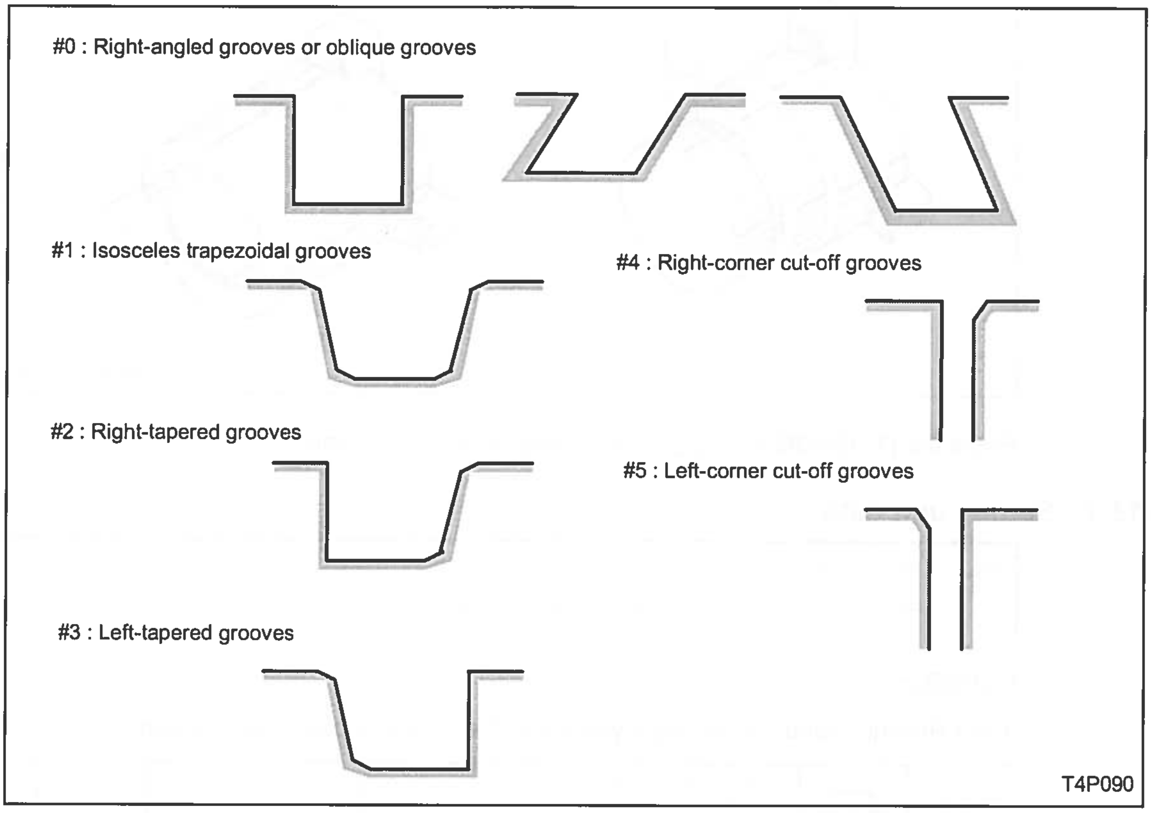
T4P089

[2] PAT.

The following menu will be displayed when the cursor is placed at this item.



Select a grooving pattern from the menu. The data of the displayed menu denote the following grooving patterns:



Note 1: Grooving patterns #4 and #5 (both, cutting-off) are available only when outside diameter (**OUT**) is selected for item [1] **PART**.

Note 2: For grooving patterns #4 and #5, the feed reduction count can be changed using parameter **U56**.

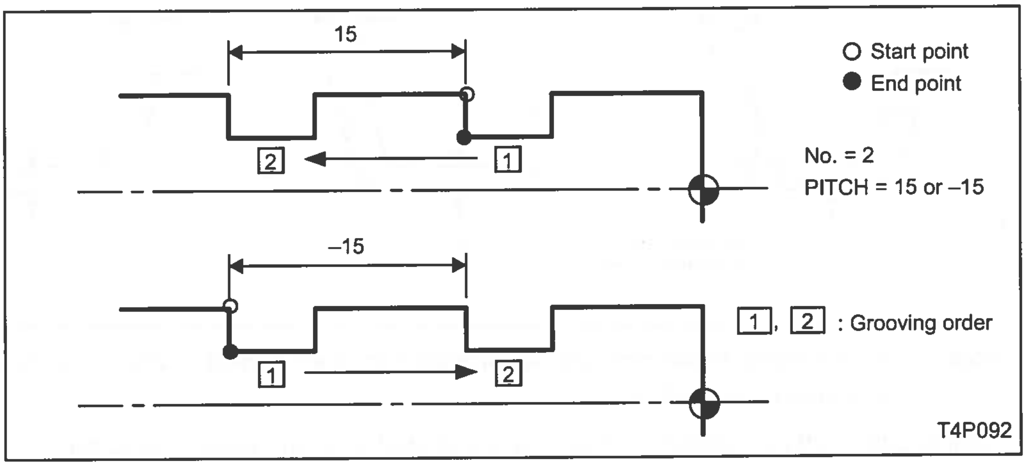
[3] No.

Set the number of grooves when multiple grooves of the same shape are to be machined at fixed spacings.

[4] PITCH

Set a pitch when multiple grooves of the same shape are to be made at fixed spacings.

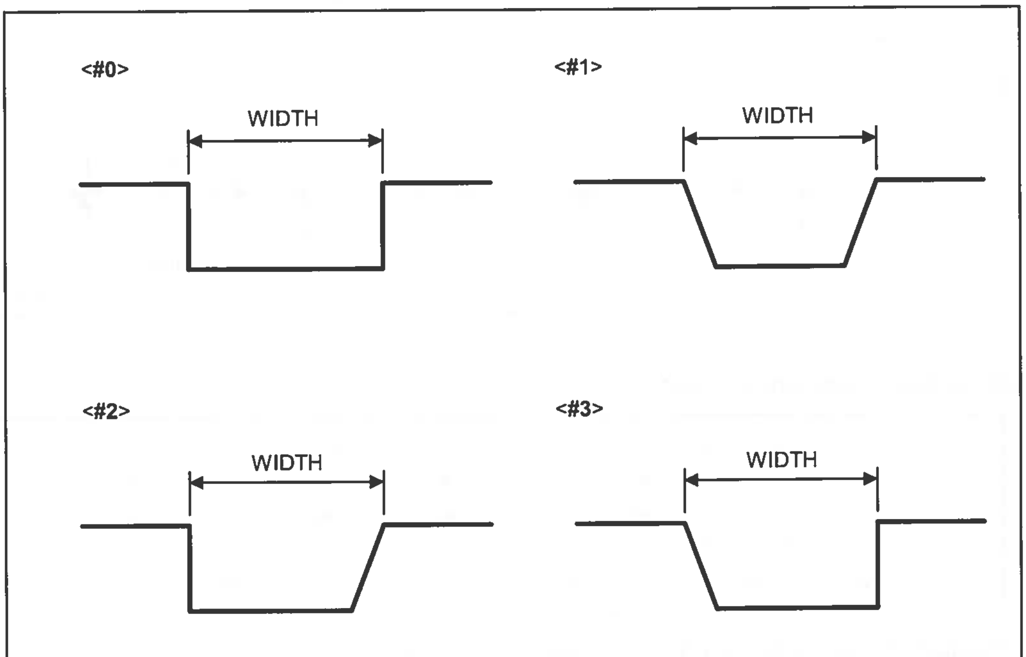
The pitch can be set as either a plus value or a minus value. Setting the pitch as a plus value causes sequential grooving in a forward direction. Setting the pitch with a minus sign causes sequential grooving in a reverse direction.



T4P092

[5] WIDTH

Set a grooving width.

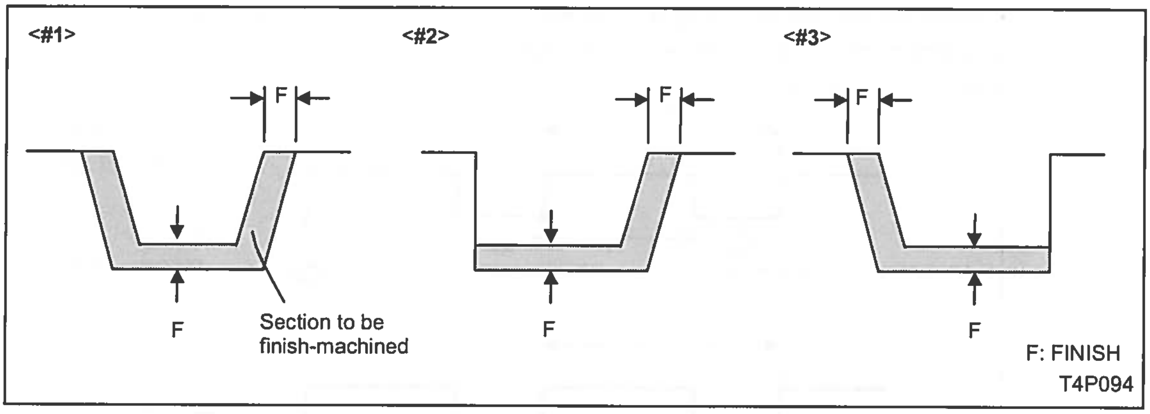


T4P093

- If you have selected grooving pattern #4 or #5, a cutting-off tool tip width is considered as a grooving width.

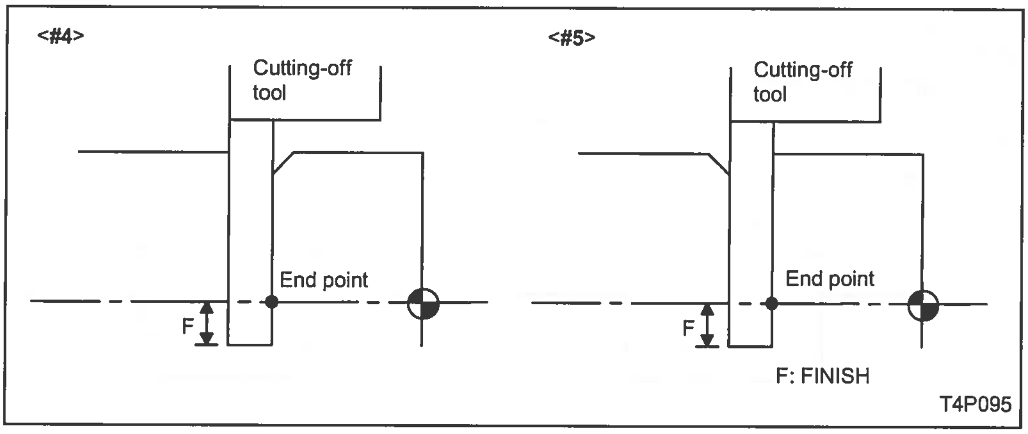
[6] FINISH

- No data can be set if you have selected grooving pattern #0.
- Set a finish-machining removal allowance if you have selected grooving pattern #1, #2 or #3.



Note: No finishing allowance will be provided to right-angled walls if you have selected grooving pattern #2 or #3.

- Set a cutting-off tool overshoot if you have selected grooving pattern #4 or #5.



3-13-2 Setting tool sequence data

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1				◆		◆	◆	◆	◆				
F2				◆		◆	◆		◆				
	↑ ↑ [1] [2]	↑ ↑ ↑ [3] [4] [5]	↑ [6]		↑ [7]			↑ [10]		↑ [8]	↑ [9]	↑ [11]	↑ [11]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	Grooving patterns #1 to #3: One tool for roughing is automatically selected.
F2 (Finishing)	One tool for machining is automatically selected.

Remark 3: For grooving patterns #1 to #3, **DEP-1** in the finishing tool sequence is displayed with a ◆ mark to indicate that a data cannot be set here.

Remark 4: For grooving patterns #0, #4, and #5, **FIN-X** is displayed with a ◆ mark to indicate that a data cannot be be set here.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a grooving unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GROOVE	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FACE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BACK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for **BAR** unit.

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Simultaneous machining No. or retraction position of the lower turret)

For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify the simultaneous machining number. It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] DEP-1 (Max. depth of cut), [8] C-SP, [9] FR

- For each grooving pattern, set data into these items as follows:

Pattern	Sequence	**DEP-1** (Max. depth of cut)	**C-SP**	**FR**
#0	F (Finishing)	Max. depth of cut per pass (Designate in radius for **OUT** or **IN**)	Peripheral speed during grooving	Feedrate during grooving (Shape seq. data **RGH** ineffective)
#1, #2 or #3	R (Roughing)		Peripheral speed during roughing	Feedrate during roughing
	F(Finishing)	—	Peripheral speed during finishing	Feedrate during finishing
#4 or #5 (Parameter **U56** = 0, 1)	F (Finishing)	Max. depth of cut per pass (Designate in radius; without pecking if 0 is set)	Peripheral speed during grooving (limited by the rotational speed specified by the parameter **U54**)	Feedrate during grooving (Shape seq. data **RGH** effective for cutting-off area specified by the parameter **K8**)
#4 or #5 (Parameter **U56** ≥ 2)	F (Finishing)		Number of revolutions during grooving (*)	Starting feedrate for grooving (**)

* The monitor will display "S500" if "500" is entered in an attempt to set a number of revolutions of 500 min^{-1}.
The section from the starting point of machining before the cutting-off area (specified by the parameter **K8**) is machined at the rotational speed designated here. In the cutting-off area the machining is performed at the rotational speed set by the parameter **U54**.

** The feedrate is reduced in several steps (set by the parameter **U56**) to the value of shape sequence data RGH.

[10] FIN-X

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for **BAR** unit.

[11] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

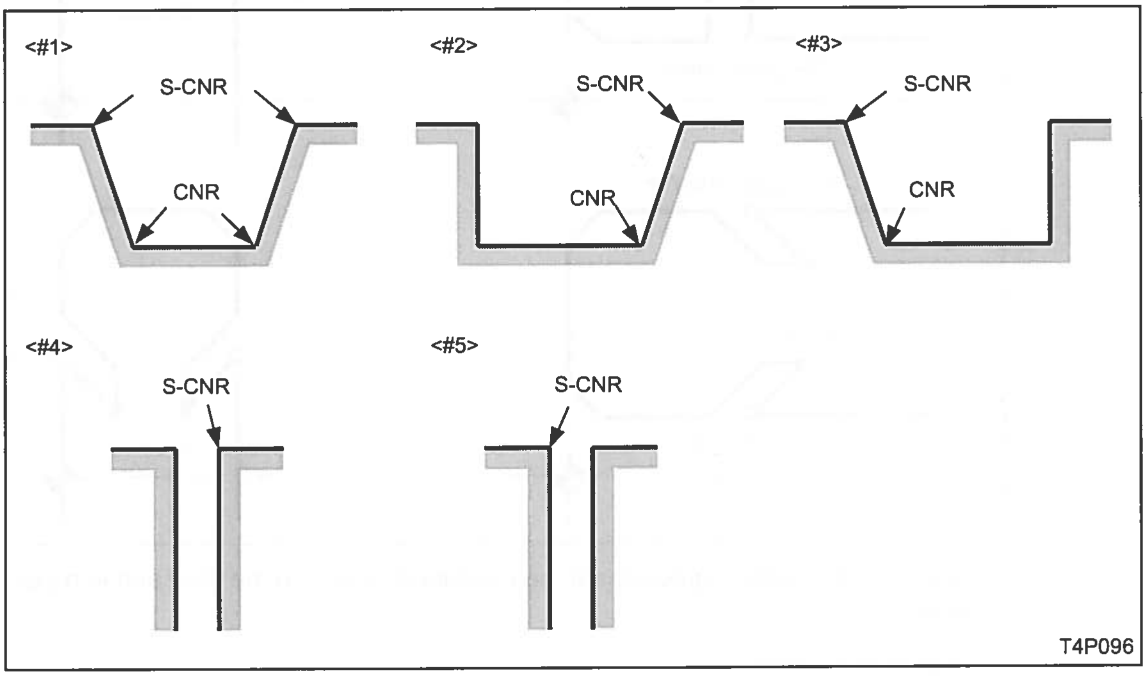
3-13-3 Setting shape sequence data

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	CNR	ANG	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]

[1] S-CNR

Set the chamfering amount if C-chamfering is required.
For R-chamfering, set a rounding radius after pressing the **[CORNER R]** menu key.

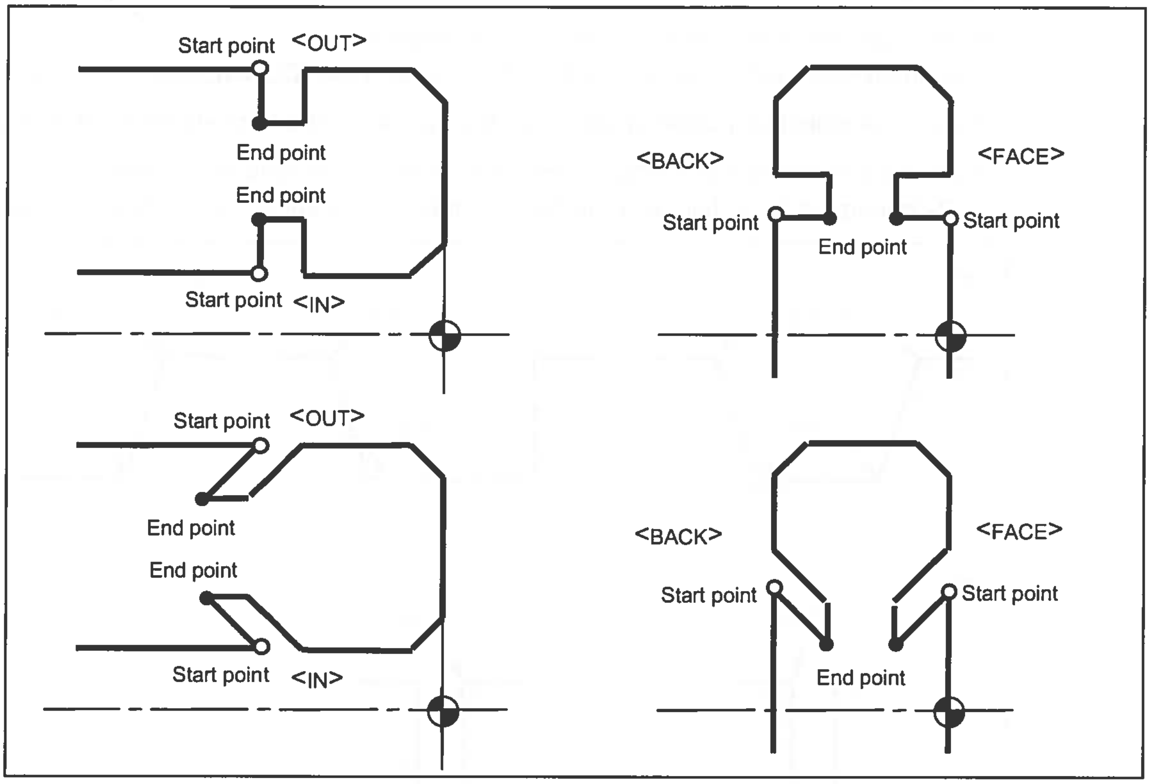
- If you have selected grooving pattern #0, the data specified here will become invalid.
- If you have selected a grooving pattern other than #0, set data in this item when C-chamfering or R-chamfering (rounding) is to be done on the corners shown in the diagrams below.



[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z

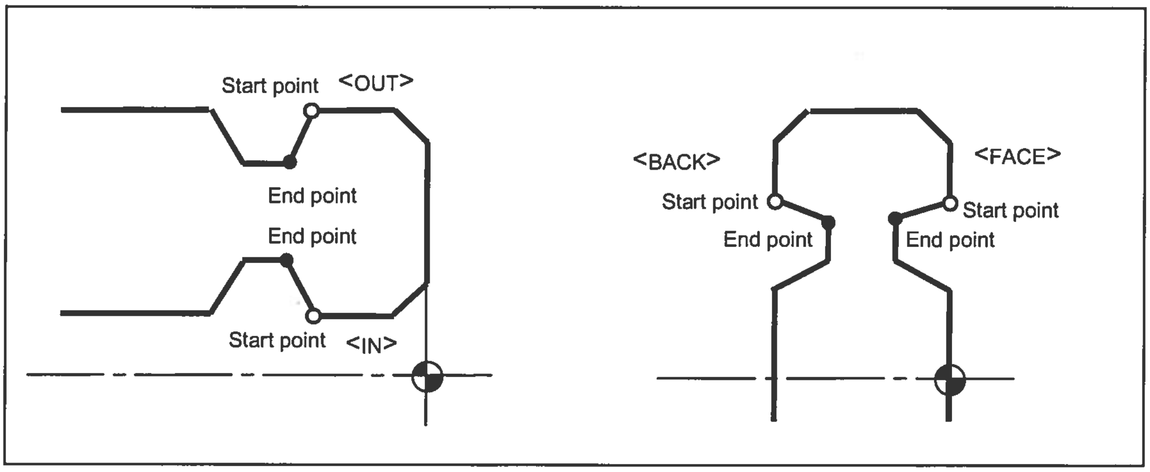
Set the coordinates of the start point and end point of grooving.
Set the position of the start point and the end point as follows according to the selected grooving pattern.

- For grooving pattern #0:

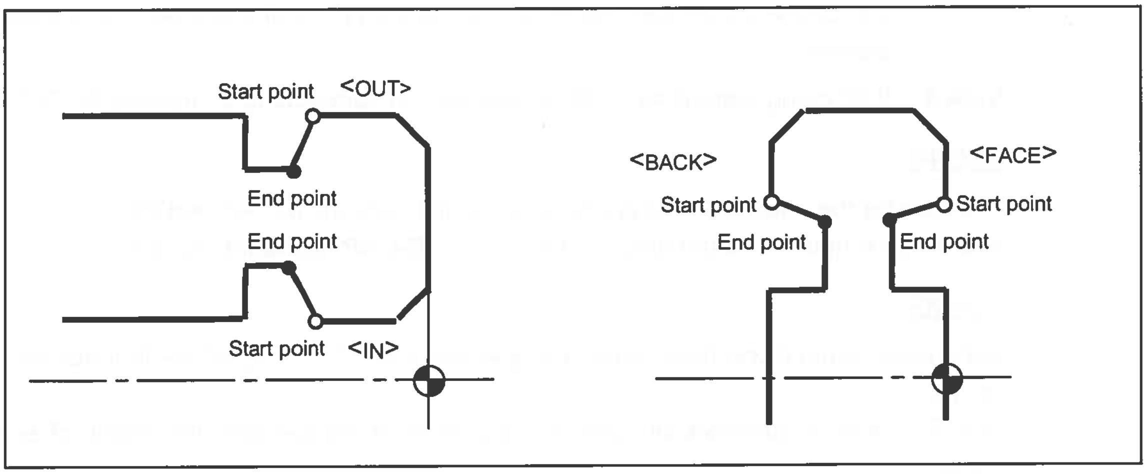


For pattern #0, oblique grooves can be machined by setting the start and end points as shown above.

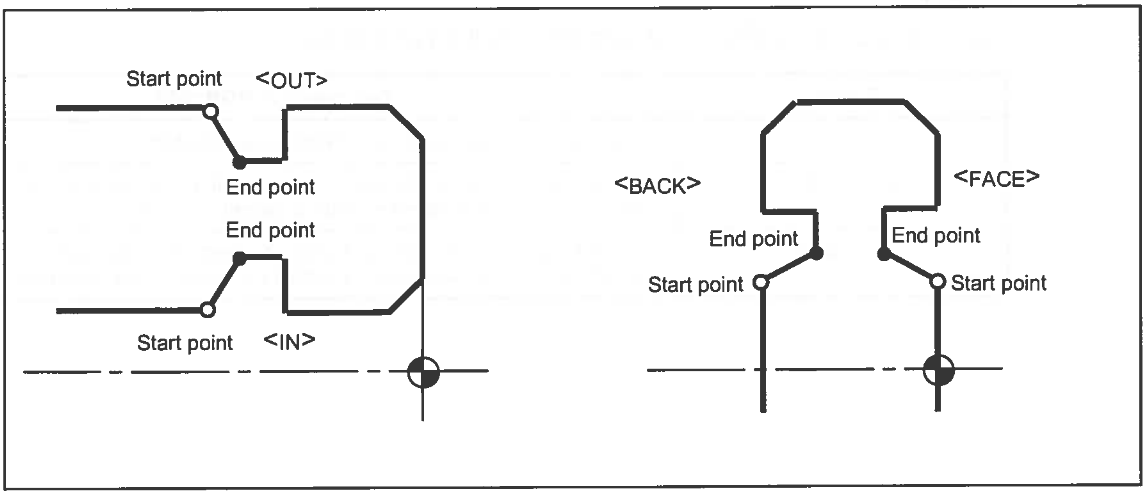
- For grooving pattern #1



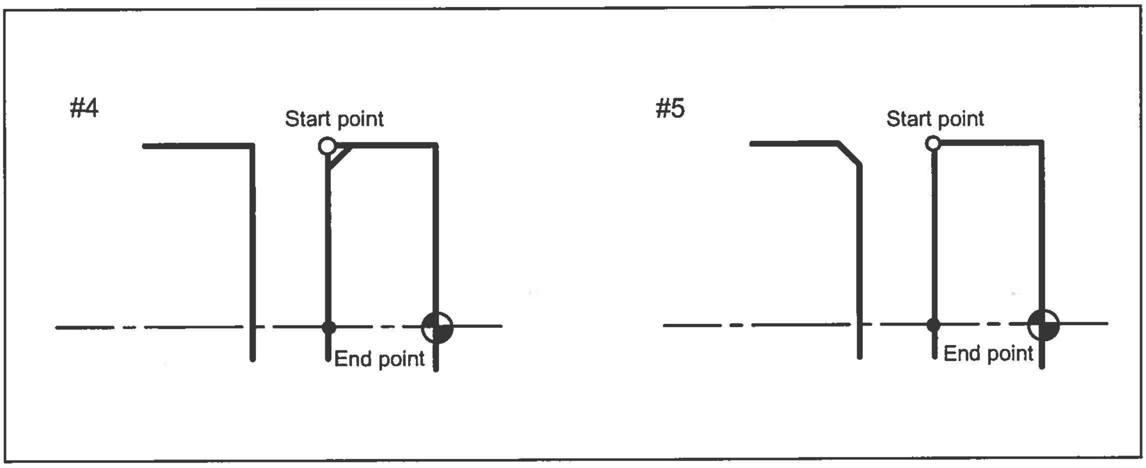
- For grooving pattern #2:



- For grooving pattern #3:



- For grooving pattern #4 or #5:



Note 1: For grooving pattern **#0, #1**, **#2** or **#3**, the machining drawing may have an indicated taper angle but not have a clearly indicated position for the start point or the end point. In such cases, temporarily type the question mark "?" in all unclear items by pressing the menu key **[?]**. You will be able to set data automatically at a later time using the automatic calculation function of the crossing-point.
See "Automatic Crossing-Point Calculation Function" for further details.

Note 2: If multiple grooves of the same shape are to be machined (according to the setting of unit data **No.**), set the coordinates of the start point and end point of grooving of the first groove.

Note 3: If grooving pattern #4 or #5 is selected, no data setting is required for **FPT-Z**.

[6] CNR

The data for the ending corner is only effective for patterns #1, #2, and #3.
See the description and the diagram for item [1], **S-CNR**, for further details.

[7] ANG

Set a taper angle if you have typed the question mark "?" in one of the four items from [2] to [5] above.
See "Function of automatically calculating a point of intersection" for details of setting a taper angle.

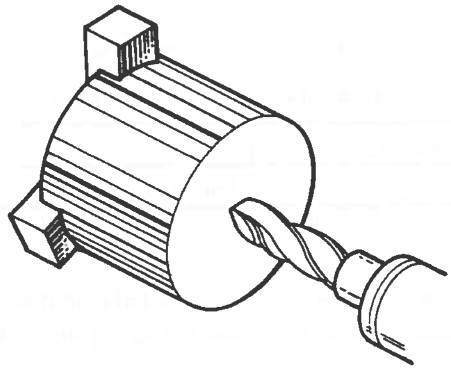
[8] RGH

For each grooving pattern, set data into this item as follows:

Pattern	Description of **RGH** data
#0	Invalid. (Set the feedrate in item **FR** of sequence data.)
#1, #2, #3	Set the feedrate during finishing. (Finishing will also be executed at the **FR** feed rate of tool sequence data if no data is designated here.)
#4 and #5	Set the feedrate for the cutting-off area. (Cutting-off will be executed at the half of the **FR** feed rate of tool sequence data if no data is designated here.)

3-14 Turning Drilling Unit (T. DRILL)

Select the turning drilling unit when preholes are to be drilled in the middle of a workpiece using a turning drill.



NM210-00434

Press the **[T. DRILL**] menu key to select this unit.

3-14-1 Setting unit data

UNo.	UNIT	PART	DIA
*	T.DRILL	[1]	[2]

[1] PART

The following menu will be displayed when the cursor is placed at this item.

FACE	BACK								

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FACE : Right edge of the workpiece
BACK : Left edge of the workpiece

Note: **PART** may not be specified for special machine specifications.

[2] DIA

Set the diameter of the hole to be drilled (nominal diameter of the turning drill).

3-14-2 Setting tool sequence data

SNo.	TOOL		NOM.			#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
1											◆	◆				
	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑			↑	↑	↑	↑
	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]			[11]	[12]	[13]	[13]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a turning drilling unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
FACE	T. DRILL	EDG EDGE
BACK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal diameter)

Set the diameter of the hole to be drilled (nominal diameter of the turning drill).

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal diameter.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Retraction position of the lower turret)

For a machine equipped with upper and lower turrets, specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] PAT. (Machining pattern)

The following menu will be displayed when the cursor is placed at this item.

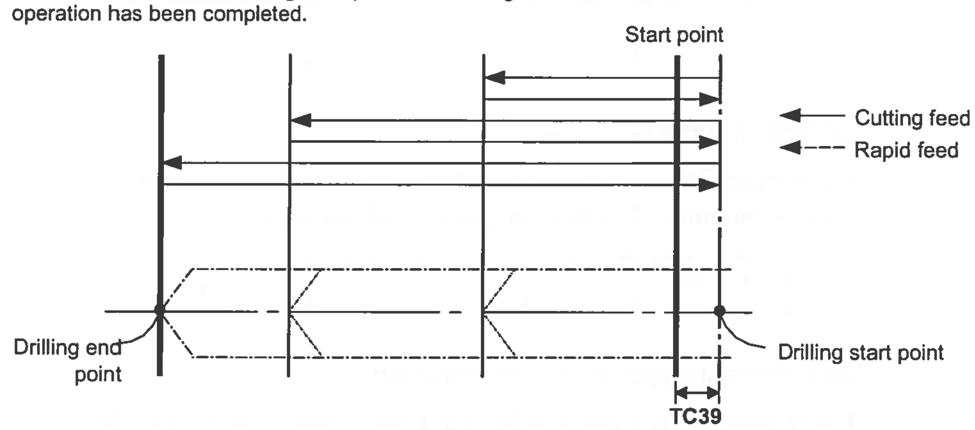
#0 DRILLING BOTTOMED	#1 PECKING BOTTOMED	#2 PECKING BOTTOMED	#3 REAMER BOTTOMED	#4 PECKING BOTTOMED					>>>

#0 DRILLING THROUGH	#1 PECKING THROUGH	#2 PECKING THROUGH	#3 REAMER THROUGH	#4 PECKING THROUGH					>>>

From the menu, select the turning drilling pattern.

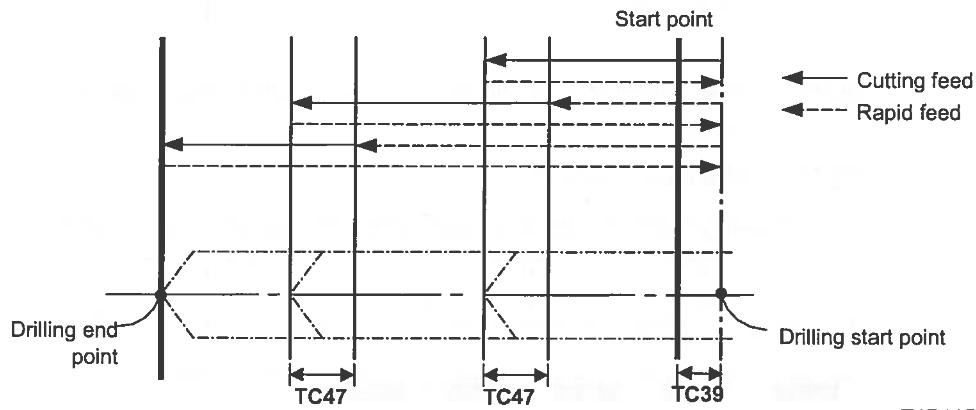
The data of the displayed menu represent following drill patterns.

#0, #0 : Conventional type of drilling cycle
The drill returns to the drilling start point at a cutting feedrate after each infeed operation has been completed.



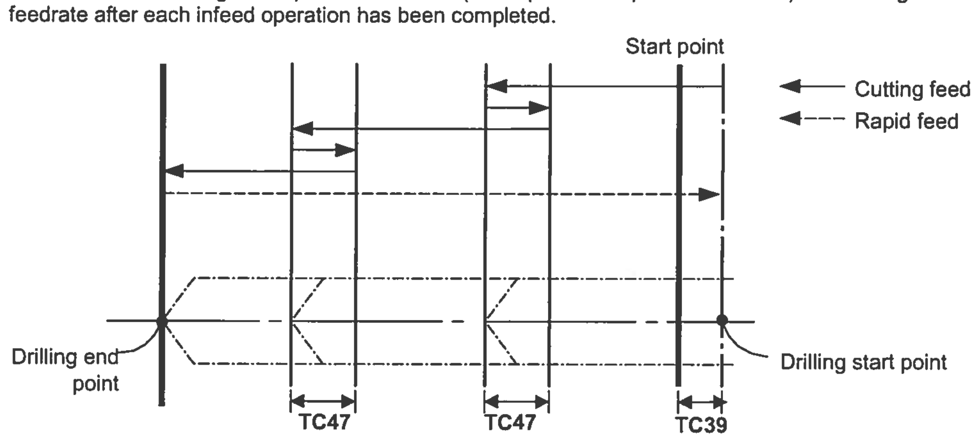
T4P114

#1, #1 : Deep-hole drilling cycle
The drill returns to the drilling start point at a rapid feedrate after each infeed operation has been completed.



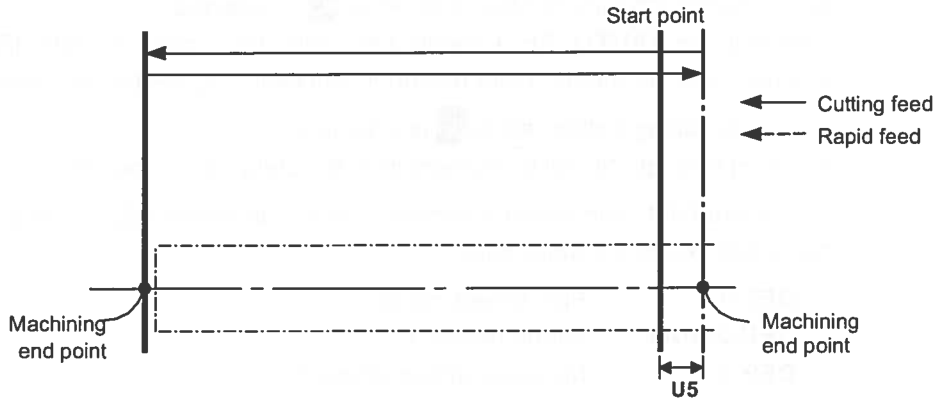
T4P115

#2, #2 : High-speed deep-hole drilling cycle
The drill returns through the specified distance (data specified in parameter **TC47**) at a cutting feedrate after each infeed operation has been completed.



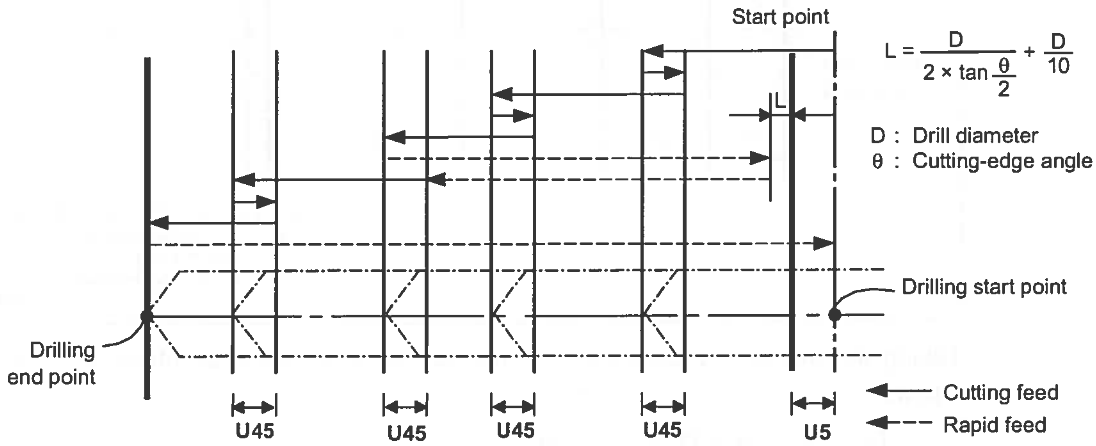
T4P116

#3, #3 : Reaming cycle
Drilled holes are finished with a reamer.



T4P117

#4, #4 : Very-deep-hole drilling cycle
Each time the normal pecking has been executed the parameter-designated number of times, the tool returns at a rapid feedrate to a position near the drilling start point.



T4P118

Note 1: Select either #0, #1, #2, #3 or #4 to drill stop-holes. Select either #0, #1, #2, #3 or #4 to drill through-holes.

Note 2: For patterns #0 to #4, the tool dwells at the bottom of the hole while the spindle rotates in accordance with the parameter-designated value. For patterns #4 and #4 the tool dwells for the same while after it has returned to a position near the drilling start point.

Note 3: With patterns #4 and #4, the rapid feedrate during the cycle can be reduced to the value designated in parameter **K76**.

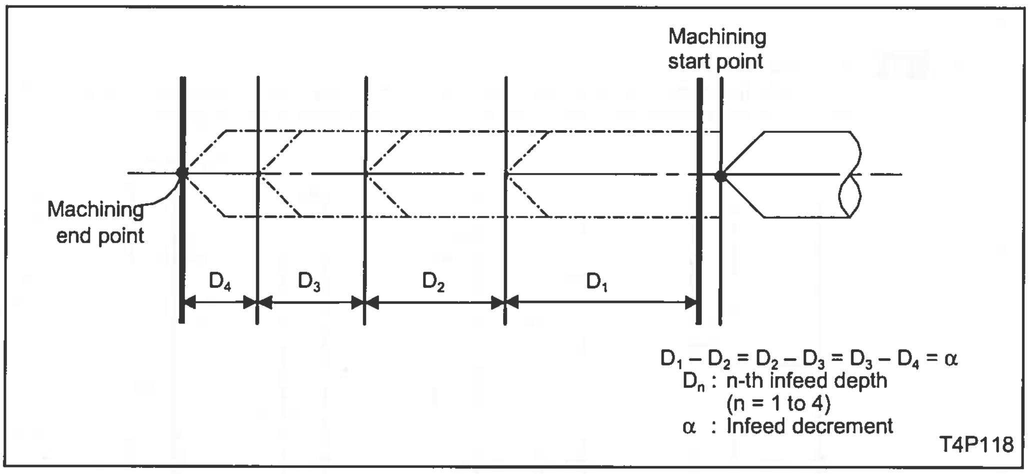
[8] DEP-1, [9] DEP-2/NUM., [10] DEP-3

You can automatically set data in these items.
When machining pattern other than #3 or #3 is selected.
Pressing the **[AUTO SET]** menu key with the cursor at item [7] and setting a tool will automatically set the data that has been calculated by the NC unit into items [8] through [10].

When machining pattern #3 or #3 is selected.
Items [8] through [10] will be marked with ◆. (Data cannot be set.)

Any desired data can be set in these items, and automatically set data can be changed. The data items denote the following data:

DEP-1 : First infeed depth
DEP-2/NUM. : Infeed decrement
DEP-3 : Minimum infeed amount



Taking the infeed decrement as α, one can calculate the n-th infeed amount, D_n ($n \geq 2$), as follows:

$$D_n = D_{n-1} - \alpha = D_1 - \alpha (n - 1)$$

However, if $D_{n-1} - \alpha \leq D_{min}$ (D_{min}: minimum infeed amount), then

$$D_n (= D_{n+1} = D_{n+2} = \cdots\cdots) = D_{min}$$

[11] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for **BAR** unit.

[12] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for **BAR** unit.

[13] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

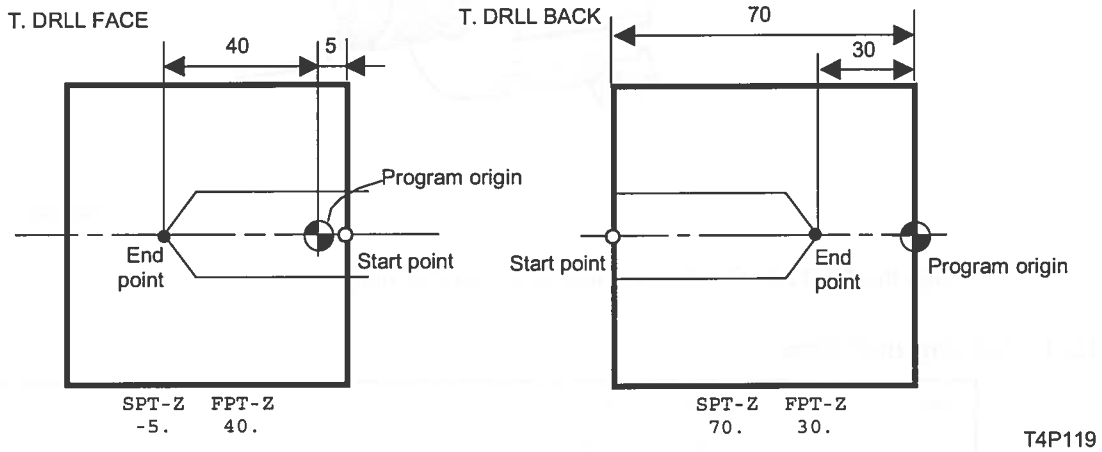
3-14-3 Setting shape sequence data

FIG	SPT-Z	FPT-Z
1	[1]	[2]

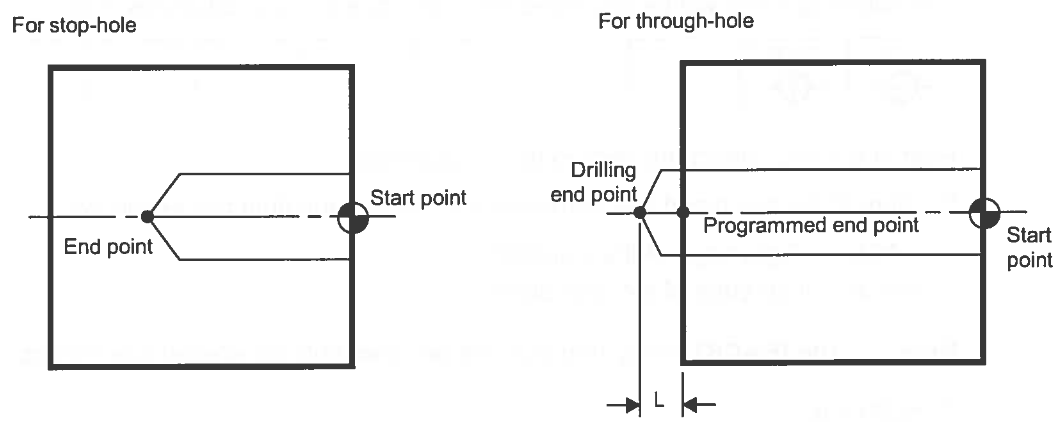
[1] SPT-Z, [2] FPT-Z

Set the coordinates of the start point and end point of the intended drilling pattern.

- The start point and end point for drilling stop-holes, for example, are positioned as shown below.



- The position of the end point differs from stop-hole to through-hole as follows:



The drilling feed will be stopped when the tool tip has reached the programmed end point.

Program the through-hole depth as the end point. Automatic drilling will then be performed up to the position (of tool tip) deeper than the programmed end point by the distance L given by the following expression:

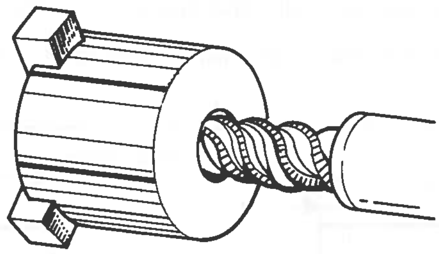
$$L = \frac{D}{2 \times \tan\frac{\theta}{2}} + \frac{D}{10} \quad (0 < \theta < 180°)$$

$$L = \frac{D}{10} \quad (\theta = 180°)$$

D : Tool diameter
θ : Cutting-edge

3-15 Turning Tapping Unit (T. TAP)

Select the turning tapping unit when holes in the middle of a workpiece are to be tapped using a turning tap.



NM210-00435

Press the **[T. TAP]** menu key to select this unit.

3-15-1 Setting unit data

UNo.	UNIT	PART	NOM-DIA	PITCH
*	T.TAP	[1]	[2]	[3]

[1] PART

The following menu will be displayed when the cursor is placed at this item.

FACE	BACK								

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FACE : Right edge of the workpiece
BACK : Left edge of the workpiece

Note: The **[BACK]** menu item may not be selectable for special machine specifications.

[2] NOM-DIA

The following menu will be displayed when the cursor is placed at this item.

METRIC THRD(M)	UNFY THRD(UN)	PIPE THRD(PT)	PIPE THRD(PF)	PIPE THRD(PS)					OTHER
(a)	(b)	(c)	(d)	(e)					(f)

From (a) through (f) above, select the type of threads to be tapped. Then, set the nominal diameter of the threads.

The data of the displayed menu denote the following types of threads:

(a) [METRIC THRD(M)] : Metric threads

(b) [UNFY THRD(UN)] : Unified threads

(c) [PIPE THRD(PT)] : Tapered pipe threads (PT)

(d) [PIPE THRD(PF)] : Parallel pipe threads (PF)

(e) [PIPE THRD(PS)] : Tapered pipe threads (PS)

(f) [OTHER] : Other thread types

- If you select metric threads, the message **NOMINAL SIZE OF TAP?** will be displayed. In that case, set the nominal diameter of the threads to be tapped.

  Example: To tap M8 metric threads:

  Press the keys [METRIC THRD(M)], [8], and [⇨] in this order.

- If you select unified thread types, the message **TAP NOMINAL SIZE <MENU → INPUT>?** will be displayed and then the menu will change over to:

No.	H(1/2) HALF	Q(1/4) QUARTER	E(1/8) EIGHTH	S(1/16) SIXTENTH		NOM-ϕ SELECT			

  Using the menu, set the nominal diameter of the threads to be tapped.

  Example 1: To tap unified thread of the 3/4-16UN size:

  Press the keys [UNFY THRD(UN)], [Q(1/4) QUARTER], [3], [−], [1], [6], and [⇨] in this order.

  Example 2: To tap unified thread of the 1-1/8-7UN size:

  Press the keys [UNFY THRD(UN)], [E(1/8) EIGHTH], [9], [−], [7], and [⇨] in this order.

  Example 3: To tap unified thread of the No. 1-16UN size:

  Press the keys [UNFY THRD(UN)], [No.], [1], [−], [1], [6], and [⇨] in this order.

- Also, a press of the **[NOM-ϕ SELECT]** menu key displays the following nominal tap diameter window to allow the desired nominal thread diameter to be entered by selecting it using the cursor keys.

UNIFY THREAD

No. 1 - 64 UN	3/4 - 10 UN
No. 2 - 56 UN	7/8 - 9 UN
No. 3 - 48 UN	1 - 8 UN
No. 4 - 40 UN	1[1/8]- 7 UN
No. 5 - 40 UN	1[1/4]- 7 UN
No. 6 - 32 UN	1[3/8]- 6 UN
No. 8 - 32 UN	1[1/2]- 6 UN
No. 10 - 24 UN	1[3/4]- 5 UN
No. 12 - 24 UN	2[1/2]- 4 UN
1/4 - 20 UN	2[3/4]- 4 UN
5/16- 18 UN	3 - 4 UN
3/8 - 16 UN	3[1/4]- 4 UN
7/16- 14 UN	3[1/2]- 4 UN
1/2 - 13 UN	3[3/4]- 4 UN
9/16- 12 UN	
5/8 - 11 UN	

- If you select pipe thread types, the message **TAP NOMINAL SIZE <MENU → INPUT>?** will be displayed and then the menu will change over to:

H(1/2) HALF	Q(1/4) QUARTER	E(1/8) EIGHTH	S(1/16) SIXTENTH		NOM-ϕ SELECT		

Using the menu, set the nominal diameter of the threads to be tapped.

Example 1: To tap pipe thread of the PT3/8 size:

Press the keys [PIPE THRD(PT)], [E(1/8) EIGHTH], [3], and [⇔] in this order.

Example 2: To tap pipe thread of the PF1/4 size:

Press the keys [PIPE THRD(PF)], [Q(1/4) QUARTER], [1], and [⇔] in this order.

Example 3: To tap pipe thread of the PS1/8 size:

Press the keys [PIPE THRD(PS)], [E(1/8) EIGHTH], [1], and [⇔] in this order.

- Also, a press of the **[NOM-ϕ SELECT]** menu key displays the following nominal tap diameter window to allow the desired nominal thread diameter to be entered by selecting it using the cursor keys.

[Tapered pipe thread (PT)]

PIPE THREAD PT	
PT 1/16	PT 1
PT 1/8	PT 1[1/8]
PT 1/4	PT 1[1/4]
PT 3/8	PT 1[1/2]
PT 1/2	PT 2
PT 5/8	
PT 3/4	
PT 7/8	

[Tapered pipe thread (PF)]

PIPE THREAD PF	
PF 1/8	PF 1[1/8]
PF 1/4	PF 1[1/4]
PF 3/8	PF 1[1/2]
PF 1/2	PF 2
PF 5/8	
PF 3/4	
PF 7/8	
PF 1	

[Tapered pipe thread (PS)]

PIPE THREAD PS	
PS 1/8	PS 1[1/8]
PS 1/4	PS 1[1/4]
PS 3/8	PS 1[1/2]
PS 1/2	PS 2
PS 5/8	
PS 3/4	
PS 7/8	
PS 1	

[3] PITCH

Set the pitch of the threads to be tapped (pitch of turning tapping tool to be used).
When the appropriate nominal tool diameter is entered in item [2] **NOM-DIA**, data will be auto-set, except for special threads. Any data, however, can also be entered instead.

3-15-2 Setting tool sequence data

SNo.	TOOL		NOM.			#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
1							◆	◆	◆	◆	◆	◆		◆		
	↑	↑	↑	↑	↑	↑							↑		↑	↑
	[1]	[2]	[3]	[4]	[5]	[6]							[7]		[8]	[8]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.
When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item [1] **TOOL** (Name) is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T-DRILL**, or **T-TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a turning-tapping unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (Section to be machined)	TOOL (Name)	TOOL (Section to be machined)
FACE	T. TAP	EDG EDGE
BACK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal diameter)

Set the diameter of the hole to be tapped (nominal diameter of the turning tap).

[4] NOM. (Tool identification code)

A code should be selected out of the menu to identify those tools which are of identical type (having an identical name) and have an identical nominal size.

A	B	C	D	E	F	G	H		>>>

[5] NOM. (Turret selection)

For a machine equipped with upper and lower turrets, select the turret in which the tool to be used is mounted. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[6] # (Retraction position of the lower turret)

For a machine equipped with upper and lower turrets, specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [6].

[7] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for **BAR** unit.

[8] M

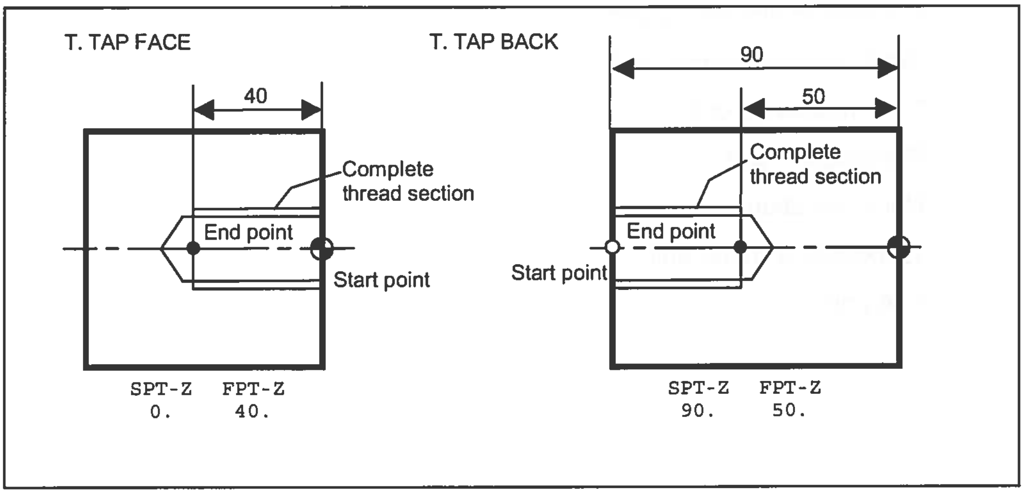
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for **BAR** unit.

3-15-3 Setting shape sequence data

FIG	SPT-Z	FPT-Z
1	[1]	[2]

[1] SPT-Z, [2] FPT-Z

Set the coordinates of the start point and end point of the thread shape.



3-16 Other Units

Ten units are provided moreover:

- Turning manual program machining unit
- Milling manual program machining unit
- M-code unit
- Coordinate measuring unit
- Workpiece measuring unit
- Tool measuring unit
- Subprogram unit
- Head selection unit
- Workpiece transfer unit
- End unit

3-17 Turning Manual Program Machining Unit (MANUAL. P)

The manual program machining unit complements the turning units described so far (**BAR**, **CPY**, **CORNER**, **FACING**, **THREAD**, **T. GROOVE**, **T. DRILL** and **T. TAP**).

These turning units have respective tool paths automatically generated according to the unit data and sequence data you have set, whereas the manual program machining unit requires user setting of its tool path.

Select this unit if a machining type or machine action that cannot be programmed in usual machining units is required, or if it is likely to be more convenient to directly set a tool path.

Press the **[MANUAL PROGRAM]** menu key to select this unit.

3-17-1 Setting unit data

UNo.	UNIT	CHANGE-PT	GEAR	TOOL	#
*	MANUAL.P	[1]	[2]	[3]	[4]

[1] CHANGE-PT

Specify whether you want to return the tool to a predetermined tool change position before executing the turning manual program machining unit.

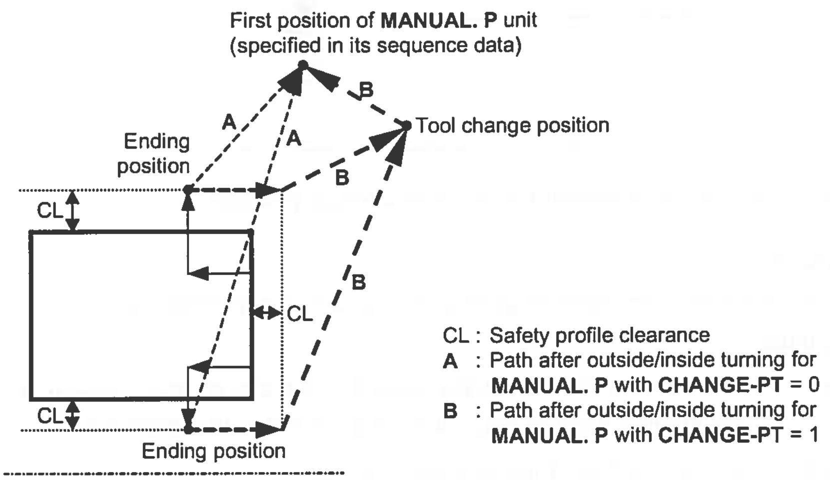
- Set 1 if tool return is required.
- Set 0 if tool return is not required.

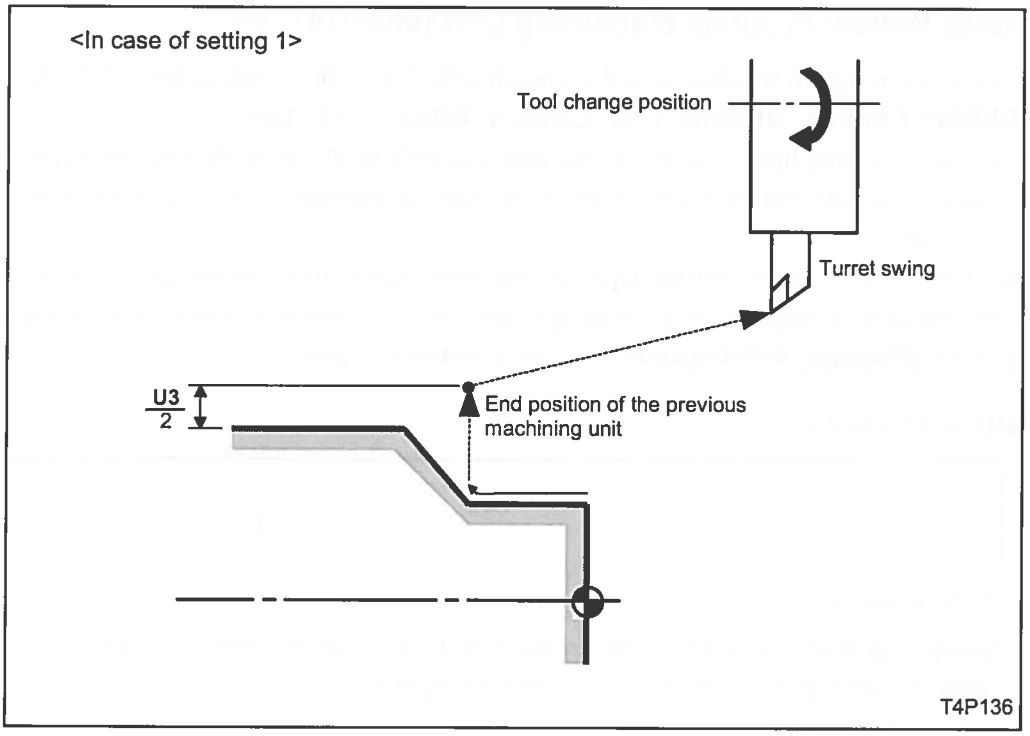
Setting 1 in this item returns a turret to predetermined tool change position, that is, the position previously specified using parameter **P17** (**P68** for the lower turret), after the previous unit ends, and then the tool to be used for the turning manual program machining unit is indexed at that position.

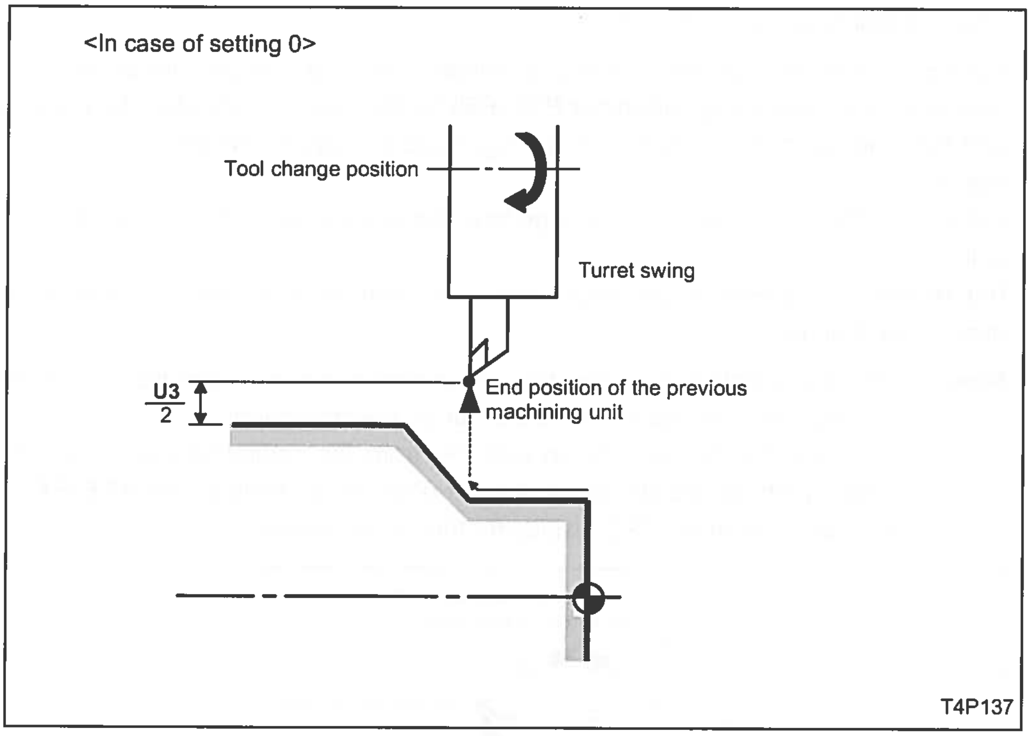
Setting 0 in this item causes tool change to occur at the end position of the previous machining unit.

Before setting 0, therefore, you must make sure that the tool does not contact the workpiece during tool change.

Note: The figure below illustrates the tool motion in the transition from an outside or inside turning unit to the turing manual program machining unit.
To avoid the interference on path "A" from the ending position of an inside turning, select path "B" (return to toll change position) by setting **CHANGE-PT** = 1 or set an escape route in the **TPC** display for the inside turning.







Note: See the Parameter List for tool change position.

[2] GEAR

Set the number of the spindle gear to be used. Gear shifting will not occur if no data is set in this data field.

Note 1: A gear is not automatically selected for the turnng manual program machining unit. Gear shifting will not occur if no data is set in this data field.

Note 2: Set 1 in this item if the spindle has stepless, automatic transmission capabilities.

[3] TOOL

Select a tool to be used. See the relevant items in "Bar-Materials Machining Unit (BAR)" for tool selection method. For manual program machining unit, R tool/F tool distinction is not drawn. The NC regards the tool as R tool.
If a tool is not designated here, the currently valid tool will be used as it is.

[4] # (Simultaneous machining No. or retraction position of the lower turret)

For a machine equipped with upper and lower turrets, to use the tools mounted in both turrets, specify the simultaneous machining number. It is also possible to specify the position to which the lower turret is to be retracted when machining workpieces using only the upper turret.
The following menu will be displayed:

TURRET 2 POS.1	TURRET 2 POS.2								

Note: See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for details of item [4]

3-17-2 Setting sequence data

UNo.	UNIT	CHANGE-PT				GEAR	TOOL	#	
*	MANUAL.P	***				***	***	***	
SEQ	G	DATA-1	DATA-2	DATA-3	RADIUS/VARIABLE	RPM	FEED	M	OFS
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] G

The following menu will be displayed when the cursor is placed at this item.

G00	G01	G02	G03	G04	G32	G34		SHAPE END	
(a)	(b)	(c)	(d)	(e)	(f)	(g)			

From (a) through (g) above, select G-code to be used. The data of the displayed menu denote the following function.

G00 : Rapid feed positioning
G01 : Linear interpolation
G02 : Clockwise arc interpolation
G03 : Counterclockwise arc interpolation
G04 : Dwell
G32 : Fixed-pitch threading
G34 : Variable-pitch threading

Note: The codes G00, G01, G02, G03, G32 and G34 are modal information — information that remains valid until you have set any other G-code. The code G04 is unmodal information — information that becomes valid only for the sequence data you set.

[2] DATA-1, [3] DATA-2, [4] DATA-3

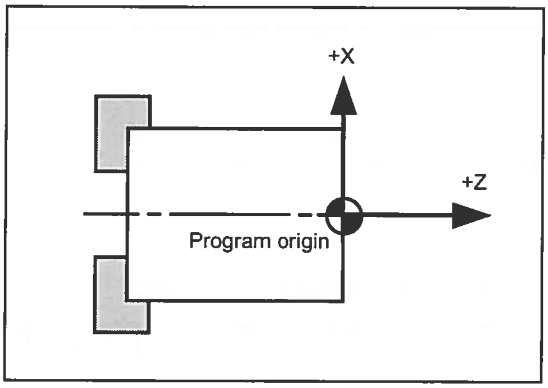
The following menu will be displayed when the cursor is placed at either one of these three items:

X	Z						W SUB		

- If the G-code you have selected for item [1] above is other than G04:
  Select an axis from the menu shown above, and then specify the position to which the movement is to be performed on the axis.
- If the G-code you have selected for item [1] above is G04:
  After pressing the **[X]** menu key, set the desired dwell time in seconds.
- Select the **[W SUB]** menu item to designate the Z-axis of the secondary spindle.
  This menu item is displayed only when the secondary spindle is equipped.

For the turning manual program machining unit, use the following coordinate system to specify the position to which the cutting edge of the tool is to be moved.

<X- and Z-axis>



Either absolute programming or incremental programming must be used to specify the moving position.

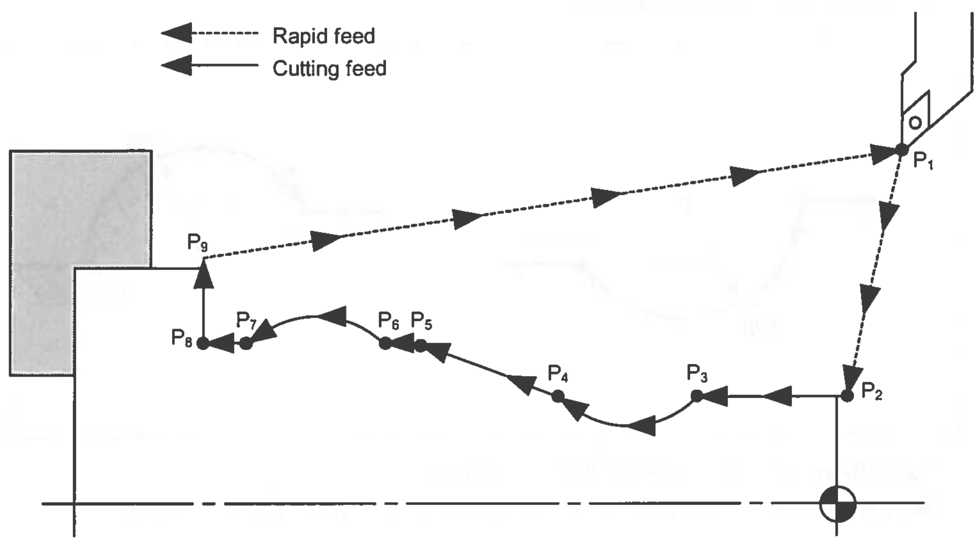
(a) Absolute programming
Directly set the coordinates of the intended moving position in the coordinate system shown in the diagram above.

(b) Incremental programming
Set an increment or decrement in distance from one moving position to another.
Selection of an axis displays the following menu:

INCRMENT INPUT									

To use this programming method, first press the **[INCRMENT INPUT]** menu key to highlight its display and then set data.

Example: To move the cutting edge of the tool in the following order ($P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow \cdots \rightarrow P_9 \rightarrow P_1$):



P_1 (300, 50) : Tool change position
P_2 (70, 5) : Approach point
P_3 (70, –35) : End point of a line
P_4 (70, –70) : End point of a clockwise arc
P_5 (80, –95) : End point of a taper line
P_6 (80, –105) : End point of a line
P_7 (80, –135) : End point of a counterclockwise arc
P_8 (80, –145) : End point of a line
P_9 (130, –145) : End point of a line (Return point)

T4P140

For absolute programming, set the following data:

UNo.	UNIT	CHANGE-PT		
*	MANUAL.P	1		
SEQ	G	DATA-1	DATA-2	DATA-3
1	00	X 70.	Z 5.	
2	01	X 70.	Z -35.	
3	02	X 70.	Z -70.	
4	01	X 80.	Z -95.	
5	01 (*1)	X 80.	Z -105.	
6	03	X 80.	Z -135.	
7	01	X 80.	Z -145.	
8	01 (*1)	X 130.	Z -145.	

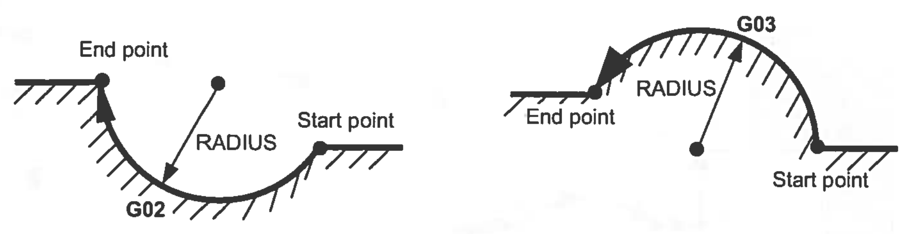
For incremental programming, set the following data:

UNo.	UNIT	CHANGE-PT		
*	MANUAL.P	1		
SEQ	G	DATA-1	DATA-2	DATA-3
1	00	X 70.	Z 5.	
2	01	X 0. <	Z -40. <	Incremental programming (*2)
3	02	X 0. <	Z -35. <	
4	01	X 10. <	Z -25. <	
5	01 (*1)	X 0. <	Z -10. <	
6	03	X 0. <	Z -30. <	
7	01	X 0. <	Z -10. <	
8	01 (*1)	X 50. <	Z -0. <	

*1: Since G01 is modal information, corresponding data can be omitted.
*2: Setting data using the incremental programming method displays the data with a < marking on the right side.

[5] RADIUS/VARIABLE

- If you have selected whether G02 or G03 for item [1] above:
  Set a radius for the desired arc.



T4P141

- If you have selected G34 for item [1] above:
  Set the desired increment or decrement in thread pitch per revolution.
- If you have selected other G-code: No data can be set. (A ◆ mark will be displayed.)

[6] RPM

Set a rotational speed (rpm) or peripheral speed for the spindle.

1) To set a rotational speed:
   Set the desired rpm value directly. An **S** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM     FEED
      ◆          S2000
                   ↑
```
Setting data

2) To set a peripheral speed:
   After highlighting **[SURF SPD V]** by pressing its menu key, set the desired value in m/min or ft/min units.
   A **V** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM     FEED
      ◆          V130
                   ↑
```
Setting data

Note: The rotational speed or peripheral speed that you have set remains valid until new such value is set.

[7] FEED

If G01, G02, G03, G32 or G34 is selected for item [1] above, set the desired feedrates for the X-axis and Z-axis in either feedrate per revolution (synchronous feed) or feedrate per minute (asynchronous feed).

1) Setting in feedrate per revolution:
   Directly set the feedrate per spindle revolution (mm/rev or inch/rev).
   A **REV** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM     FEED
      ◆          V130
      ◆                 REV 0.35
                              ↑
```
Setting data

2) Setting in feedrate per minute:
   After highlighting **[FEEDRATE/min]** by pressing its menu key, set the desired value in mm/min or inch/min units.
   A **MIN** marking will then be displayed to the left of that value.

```
RADIUS/VARIABLE   RPM     FEED
      ◆          V130   MIN 100
      ◆                      ↑
```
Setting data

Note 1: During sequence of G00, the axis movement will be carried out at the rapid feedrate preset in the specifications. The feedrate you set in this item is valid only for sequences corresponding to your selection of either G01, G02, G03, G32 or G34.

Note 2: Feedrate preset remains valid until new data is set.

Note 3: If you have selected either G32 or G34 for item [1] above, set a lead for the threads.
(Lead) = (Pitch) × (Number of threads)

[8] M

When an M-code (auxiliary function code) is to be used, set the number of that M-code.
The M-code you set in this item becomes valid only for the next and subsequent sequences.

Note: If a feedrate is to be set in mm/rev or inch/rev units for the next sequence, you must first select either M03 (forward rotation of the spindle) or M04 (reverse rotation of the spindle), unless the rotational direction of the spindle with respect to the tool to be used has been specified on the **TOOL DATA** display.

[9] OFS

If tool offsetting is to be done using either one of the offset amounts registered on the **TOOL OFFSET** display, set the offset number corresponding to the desired offset amount.
During tool offsetting, the offset data specified in this item [9] will be added to the **WEAR COMP.** data on **TOOL DATA** display.

Note: Nose R offsetting cannot be performed for manual program machining.

3-18 Milling Manual Program Machining Unit (M-MANUAL)

The milling manual program machining unit (**M-MANUAL**) complements the point, line and face machining units described so far.
These point, line and face machining units have respective tool paths automatically generated according to the unit data and sequence data you have set, whereas the milling manual program machining unit requires user setting of its tool path.
If a machining type or machine action that cannot be programmed for usual point, line and face machining units is required, select this unit when it is likely to be more convenient to directly set a tool path.
Press the **[MILLING MANUAL P]** menu key to select this unit.

3-18-1 Setting unit data

UNo.	UNIT	CHANGE-PT	GEAR	TOOL
*	M-MANUAL	[1]	[2]	[3]

[1] CHANGE-PT, [2] GEAR, [3] TOOL

The meaning of each item is the same as for turning manual program machining unit.
Refer to the relevant items in “3-17 Turning Manual Program Machining Unit (MANUAL. P)” for data-setting method and other details.

3-18-2 Setting sequence data

UNo.	UNIT	CHANGE-PT					GEAR	TOOL	NOM-φ
*	M-MANUAL	***					***	***	***
SEQ	G	DATA-1	DATA-2	DATA-3	RADIUS	RPM	FEED	M	OFS
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] G

The following menu will be displayed when the cursor is placed at this item.

G00	G01	G02	G03	G04				SHAPE END	
(a)	(b)	(c)	(d)	(e)					

From (a) through (e) above, select G-code to be used.
The data of the displayed menu denote the following function.

G00 : Rapid feed positioning
G01 : Linear interpolation
G02 : Clockwise arc interpolation
G03 : Counterclockwise arc interpolation
G04 : Dwell

Note: The codes G00, G01, G02 and G03 are modal information — information that remains valid until you have set any other G-code.
The code G04 is unmodal information — information that becomes valid only for the sequence data you set.

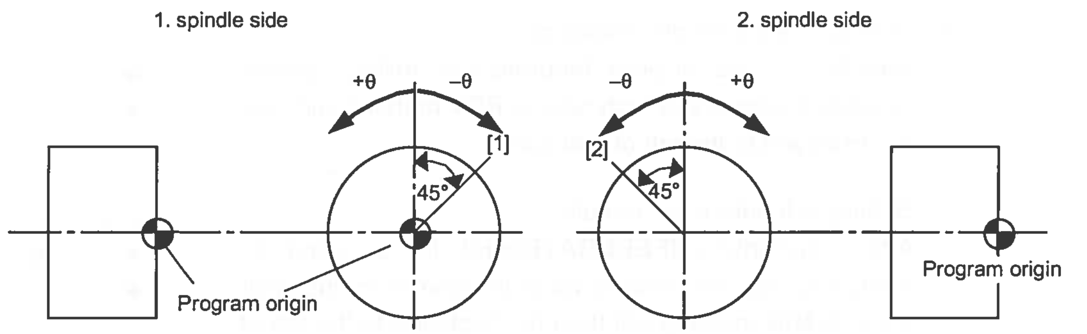
[2] DATA-1, [3] DATA-2, [4] DATA-3

The following menu will be displayed when the cursor is placed at either one of these three items:

X	Z	C		Y		B	W SUB		

- If the G-code you have selected for item [1] above is other than G04:
  Select an axis from the menu shown above, and then specify the position to which the movement is to be performed on the axis.
- If the G-code you have selected for item [1] above is G04:
  After pressing **[X]**, set the desired dwell time in seconds.

- Select the **[W SUB]** menu item to designate the Z-axis of the secondary spindle.
  This menu item is displayed only when the secondary spindle is equipped.
- For the milling manual program machining unit you must program a C-axis position as shown in the diagram below (refer to the relevant items in "Turning Manual Program Machining Unit (MANUAL. P)" for details of positioning on the X-, Z- and B-axis).



Data setting for positioning to [1]

```
UNo.   UNIT      TYPE   HEAD
 *     HEAD      SIN     1
UNo.    UNIT     ......
 *    M-MANUAL
SEQ     G       DATA-1
 1      0      C  -45.
```

Data setting for positioning to [2]

```
UNo.   UNIT      TYPE   HEAD
 *     HEAD      SIN     2
UNo.    UNIT     ......
 *    M-MANUAL
SEQ     G       DATA-1
 1      0      C  -45.
```

T4P264

Either absolute programming or incremental programming must be used to specify the moving position.
See the Section "3-17 Turning Manual Program Machining Unit (MANUAL. P)" for further details of this programming method.

[5] RADIUS

- If you have selected either G02 or G03 for item [1] above:
  Set a radius for the desired arc.
- If you have selected other G-code:
  No data can be set. (A ◆ mark will be displayed.)

[6] RPM

Set a rotational speed (rpm) or peripheral speed for the milling spindle.

1) To set a rotational speed:
Set the desired rpm value directly.
An **S** marking will then be displayed to the left of that value.

```
RADIUS    RPM     FEED
  ◆      S1200
           ↓
       Setting data
```

2) To set a peripheral speed:
After highlighting **[SURF SPD V]** by pressing its menu key, set the desired value in m/min or ft/min units.
A **V** marking will then be displayed to the left of that value.

```
RADIUS    RPM     FEED
  ◆      V100
           ↓
       Setting data
```

Note: The rotational speed or peripheral speed that you have set remains valid until new such value is set.

[7] FEED

Set the desired feedrates for the axis to be moved in either feedrate per revolution (synchronous feed) or feedrate per minute (asynchronous feed).

1) Setting in feedrate per revolution:
Directly set the desired feedrate per milling spindle revolution (mm/rev or inch/rev). A **REV** marking will then be displayed to the left of that value.

```
RADIUS    RPM     FEED
  ◆      V130
  ◆              REV 0.35
                       ↓
                  Setting data
```

2) Setting in feedrate per minute:
After highlighting **[FEEDRATE/min]** by pressing its menu key, set the desired value in mm/min or inch/min units. A **MIN** marking will then be displayed to the left of that value.

```
RADIUS    RPM     FEED
  ◆      V130
  ◆              MIN 100
                      ↓
                  Setting data
```

Note 1: During sequence for which you selected G00, the axis moves at the rapid feedrate preset in the specifications, not the feedrate you set in this item. The feedrate you set in this item becomes valid only for sequences corresponding to your selection of either G01, G02, or G03.

Note 2: Feedrate once set remains valid until new data is set.

[8] M

When an M-code (auxiliary function code) is to be used, set the number of that M-code.
The M-code you set in this item becomes valid only for the next and subsequent sequences.

Note: If a feedrate is to be set in mm/rev or inch/rev units for the next sequence, you must first select either M203 (forward rotation of the milling spindle) or M204 (reverse rotation of the milling spindle), unless the rotational direction of the milling spindle with respect to the tool to be used has been specified on the **TOOL DATA** display.

[9] OFS

If tool offsetting is to be done using either one of the offset amounts registered on the **TOOL OFFSET** display, set the offset number corresponding to the desired offset amount.
During tool offsetting, the offset data specified in this item [9] will be added to the **WEAR COMP.** data on **TOOL DATA** display.

3-19 M-Code Unit (M-CODE)

Select the M-code unit when M-codes (miscellaneous function codes) are to be set.
Up to a maximum of 12 M-codes can be set for one M-code unit.
Press **[M CODE]** menu key for this unit.

3-19-1 Setting unit data (M-code)

UNo.	UNIT	#	#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12
*	M-CODE	[1]	[2]	[2]	[2]	[2]	[2]	[2]	[2]	[2]	[2]	[2]	[2]	[2]

[1] # (Turret selection)

For a machine equipped with upper and lower turrets, select the turret for which the M-code is to be issued. The following menu will be displayed:

SET TURRET1	SET TURRET2								

[2] #1 to #12

Set the desired M-code number in each of the 12 items.
The M-codes you have set are executed in the following order:

#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12
***	***	***	***	***	***	***	***	***	***	***	***

1. (Synchronous) → 2. (Synchronous) → 3. (Synchronous)

If not all of the intended M-codes are to be executed at the same time, therefore, divide them into three groups (#1 through #4, #5 through #8, and #9 through #12) and then set those M-codes separately.

Note 1: For the list of M-codes provided in the NC system, see the Operating Manual.
The M-codes M02 (Program End), M98 (Subprogram Call), or other dedicated M-codes for EIA/ISO program cannot be selected.

Note 2: For a machine with an optional second miscellaneous function, second miscellaneous function codes can be issued with #4, #8, and #12. In such a case, make the **[OTHER CODE]** menu function valid and enter code numbers.

3-19-2 M-code table

O: Standard
Δ: Optional
—: Special order

M-codes	Description	Specification	Remarks
M00	Program stop	O	
M01	Optional stop	O	
M02	Program end	O	
M03	Spindle normal rotation	O	
M04	Spindle reverse rotation	O	
M05	Spindle stop	O	
M06	Chuck unclamp (holding workpiece)	Δ	
M07	Chuck clamp (releasing workpiece)	Δ	
M08	Coolant ON	O	
M09	Coolant OFF	O	
M10	Tailbody connection release/Independent movement of tailstock spindle	O	
M11	Tailbody connection/Simultaneous movement of tailstock spindle	O	
M12	Milling spindle mode cancel (Turning mode selection)	O	
M13	Milling tool normal rotation	O	
M14	Milling tool reverse rotation	O	
M15	Milling tool stop	O	
M16	Spindle orientation 0° (for AJC)	—	
M17	Spindle orientation 120° (for AJC)	—	
M18	Spindle orientation 240° (for AJC)	—	
M19	Spindle orientation position (robot work insertion)	Δ	
M20	Robot service call	Δ	
M21	Robot service call	Δ	
M22	Robot service call	Δ	
M23	Robot service call	Δ	
M24	Robot service call	Δ	
M25	Robot service call	Δ	
M26	Robot service call	Δ	
M27	Robot service call	Δ	
M28	Robot service call	Δ	
M29	Robot service call	Δ	
M30	Reset & tape rewind	O	
M31	Tailstock spindle & tailbody advance	O	
M32	Tailstock spindle & tailbody retract	O	
M33	Low chuck pressure	—	
M34	High chuck pressure	—	
M35			
M36			
M37	Spindle gear change low speed	O	
M38			
M39			
M40			
M41	Spindle gear change low speed	O	
M42			

M-codes	Description	Specification	Remarks
M43			
M44			
M45	Air coolant blow ON (coolant stop: M09)	Δ	
M46			
M47			
M48	Parts catcher advance	Δ	
M49	Parts catcher retract	Δ	
M50			
M51	Error detect OFF	O	
M52	Error detect ON	O	
M53	Chamfering OFF	O	
M54	Chamfering ON	O	
M55	Piece count	—	
M56	Front door open	Δ	
M57	Front door close	Δ	
M58	Chuck air blast	Δ	
M59	Program end preparation announce	—	
M60	C-axis unclamp	O	
M61			
M62			
M63			
M64			
M65			
M66	C-axis clamp	O	
M67	C-axis brake (G01 machining)	O	
M68	Cycle bar feeder call 1	—	
M69	Cycle bar feeder call 2	—	
M70	Ejector drill coolant ON	—	
M71	Ejector drill coolant OFF	—	
M72	Inside chuck clamp	—	
M73	Outside chuck clamp	—	
M74	Steady rest - turret connection mode release	—	
M75	Steady rest - turret connection mode start	—	
M76	Spindle jaw return to AJC unit (AJC)	—	
M77	Mounting AJC unit jaw to spindle	—	
M78	Steady rest auto unclamp limit switch, valid	—	
M79	Steady rest auto unclamp limit switch, invalid	—	
M80			
M81	Workpiece measurement start	—	
M82	Workpiece measurement end	—	
M83	Tool measurement start	Δ	
M84	Tool measurement end	Δ	
M85	Mist coolant ON (OFF: M09)	—	
M86	Steady rest 1 unclamp	—	
M87	Steady rest 1 clamp	—	
M88	Steady rest 2 unclamp	—	
M89	Steady rest 2 clamp	—	

M-codes	Description	Specification	Remarks
M90			
M91			
M92			
M93			
M94			
M95			
M96	User macro interruption, valid	—	
M97	User macro interruption, invalid	—	
M98	EIA → EIA Subprogram call	O	
M99	EIA → EIA Return to main EIA program	O	
M100			
↓			
M130			
M131	Chuck pressure selection 1 (minimum pressure)	—	
M132	Chuck pressure selection 2	—	
M133	Chuck pressure selection 3/Steady rest clamp high pressure	—	
M134	Chuck pressure selection 4/Steady rest clamp low pressure	—	
M135	Chuck pressure selection 5	—	
M136	Chuck pressure selection 6	—	
M137	Chuck pressure selection 7	—	
M138	Chuck pressure selection 8	—	
M139	Chuck pressure selection 9	—	
M140	Chuck pressure selection 10	—	
M141			
M142			
M143			
M144			
M145			
M146			
M147			
M148			
M149			
M150	Workpiece unload demand to robot	—	
M151	Milling spindle through air blow ON	O	
M152	Milling spindle through air blow OFF	O	
M153	Milling spindle through coolant ON	O	
M154	Milling spindle through coolant OFF	O	
M155	Chuck coolant ON (composed movement)	Δ	
M156	Chuck air blow ON (composed movement)	Δ	
M157	Spindle through coolant blow ON (OFF: M159)	Δ	
M158	Spindle through air blow ON (OFF: M159)	Δ	
M159	M157, M158 OFF	Δ	
M160	Shower coolant ON/chuck stopper extend	Δ	
M161	Shower coolant OFF/chuck stopper retract	Δ	
M162	Workpiece rechucking (after delivering robot work)	Δ	
M163	Tailstock spindle thrust low pressure	Δ	
M164	Tailstock spindle thrust high pressure	Δ	

M-codes	Description	Specification	Remarks
M165			
M166			
M167	Turret coolant ON	—	
M168	Turret coolant OFF	—	
M169	Magnum coolant ON	—	
M170		—	
M171		—	
M172	Index 0°(KOUYOU)	—	
M173	Index 90°(KOUYOU)	—	
M174	Index 180°(KOUYOU)	—	
M175	Index 270°(KOUYOU)	—	
M176	Index 270°(KOUYOU)	—	
M177	Index 270°(KOUYOU)	—	
M178	Index 270°(KOUYOU)	—	
M179	Index 270°(KOUYOU)	—	
M180			
M181	Spindle jaw No. 1 selection (AJC)	—	
M182	Spindle jaw No. 2 selection (AJC)	—	
M183	Spindle jaw No. 3 selection (AJC)	—	
M184	Spindle jaw No. 4 selection (AJC)	—	
M185	Spindle jaw No. 5 selection (AJC)	—	
M186	Spindle jaw No. 6 selection (AJC)	—	
M187	Spindle jaw No. 7 selection (AJC)	—	
M188	Spindle jaw No. 8 selection (AJC)	—	
M189	Spindle jaw No. 9 selection (AJC)	—	
M190	Spindle jaw No. 10 selection (AJC)	—	
M191	Spindle jaw No. 11 selection (AJC)	—	
M192	Spindle jaw No. 12 selection (AJC)	—	
M193	Spindle jaw No. 13 selection (AJC)	—	
M194	Spindle jaw No. 14 selection (AJC)	—	
M195	Spindle jaw No. 15 selection (AJC)	—	
M196			
M197			
M198	EIA → MAZATROL program call	Δ	
M199	EIA → MAZATROL program end	Δ	
M200	Milling point machining start	O	
M201	Milling line machining start	O	
M202	Turning mode	O	
M203	Milling tool normal rotation	O	
M204	Milling tool reverse rotation	O	
M205	Milling tool stop	O	
M206			
M207	M208, M209 mode cancel	O	
M208	ATC prohibition mode during finishing, coolant ON	O	
M209	ATC prohibition mode during finishing, coolant OFF	O	
M210	C-axis clamp (for milling)	O	
M211	C-axis brake (for milling)	O	

M-codes	Description	Specification	Remarks
M212	C-axis unclamp (for milling)	○	
M213	C-axis brake only (brake by M211)	○	
M214	C-axis unclamp only (unclamp by M212)	○	
M215	M213, M214 and M216 mode cancel	○	
M216	C-axis unclamp neglect mode (cancel: M215)	○	
M217			
M218			
M219	Milling tool orient	○	
M220			
↓			
M229			
M230	Grinding mode ON	—	
M231	Grinding speed setting	—	
M232			
M233	Steady rest 2 clamp low pressure selection	—	
M234	Steady rest 2 clamp high pressure selection	—	
M235			
M236	C-axis servo gain normal	○	
M237	C-axis servo gain low	○	
M238	C-axis servo gain middle	○	
M239	C-axis servo gain high	○	
M240			
M241			
M242			
M243			
M244			
M245			
M246			
M247			
M248	Spindle speed check (cutting start interlock)	○	
M249	Turret selection preparation	○	
M250	Turret/B-axis unclamp	○	
M251	B-axis clamp	○	
M252	Milling spindle unclamp	○	
M253	Milling spindle clamp	○	
M254	Turret/B-axis unclamp	○	
M255			
M256			
M257			
M258	Tool air blow	○	
M259			
M260	Polygon mode ON	○	
M261	Polygon mode OFF	○	
M262			
↓			
M273			
M274	Steady rest coolant ON	—	

M-codes	Description	Specification	Remarks
M275	Steady rest coolant OFF	—	
M276	Steady rest 1 up	—	
M277	Steady rest 1 down	—	
M278			
↓			
M329			
M330	Stopper 1 in the spindle	—	
M331	Stopper 2 in the spindle	—	
M332	Chip conveyor start	—	
M333	Chip conveyor stop	—	
M334			
↓			
M347			
M348	Yt-axis selection	○	
M349	Y-axis selection	○	
M350			
M351	M352 cancel	○	
M352	Spindle speed arrival signal check cancel	○	
M353			
↓			
M369			
M370	Axis load detection, invalid	Δ	
M371	Axis load detection, valid	Δ	
M372	Axis load detection, temporarily invalid	Δ	
M373	Axis load detection, re-start	Δ	
M374	NC feed holding on overload detection	Δ	
M375	NC feed holding & spindle stop on overload detection	Δ	
M376	Overload detection level % setting	Δ	
M377	Overload detection time setting (unit: 0.1 sec)	Δ	
M378	Overload detection peak %/detection frequency setting	Δ	
M379	Overload detection table No. registration	Δ	
M380			
↓			
M389			
M390	M391 cancel	Δ	
M391	Spindle mis-chucking cancel (M3/M4 completed with chuck open)	Δ	
M392			
M393	Spindle face driver valid	—	
M394			
M395			
M396			
M397			
M398	Turret coolant ON	—	
M399			
↓			
M589			
M590	T code execution during axis movement	—	

M-codes	Description	Specification	Remarks
M591			
M592			
M593			
M594			
M595			
M596	M-code execution during axis movement	—	
M597			
M598			
M599			
M600	Tool setting for turret	○	
M601	Magazine tool rotation	○	
M602	Stand-by tool setting	○	
M603			
↓			
M999			

Note: The table above does not include the M-codes for some of the optional functions or equipments.
For detailed description of the optional functions and equipments etc. refer to the Operating Manual of the machine.

3-20 Head Selection Unit (HEAD)

With the process segmentation unit, select a spindle (No. 1 or No. 2) you are going to operate, before programming various machining units.

Once an operation side (pattern) has been selected, it will remain valid until a different operation pattern is selected with another head selection unit.

Press the **[SEPARATE PROCESS]** menu key to select this unit.

3-20-1 Setting unit data

UNo.	UNIT	TYPE	HEAD	SPDL	TURRET
*	HEAD	[1]	[2]	[3]	[4]

[1] TYPE

From the following menu, select an operation pattern for each spindle.

SINGLE	SYNCH.								

SINGLE : Only the spindle you are going to set in the following item [2] will operate.

SYNCH. : The spindle specified in item [2] will rotate (master action) and the other spindle will perform synchronous rotation (slave action; same direction and speed).

[2] HEAD

Select the spindle to be operated.

- Set 1 or 2 to operate the No. 1 or the No. 2 spindle, respectively.

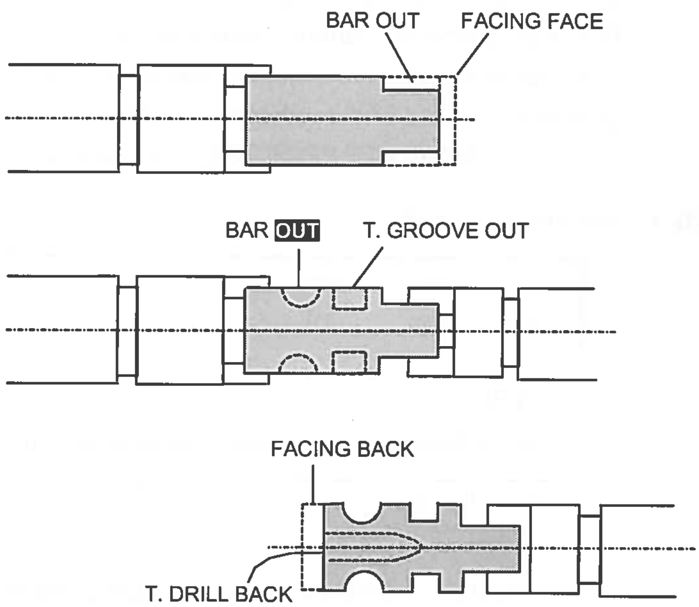
Note: Enter 1 for **SYNC** set in item [1].

[4] TURRET

If the pattern that was selected in item [1] above is **SYNC**, select the turret to be operated, namely, the turret for which the peripheral speed (revolutions) of the turning spindle is to be programmed. See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for further details.

Example: **SIN** and **SYNC** machining

```
      UNo.
       0
      UNo.   UNIT        TYPE     HEAD
       1     HEAD         SIN       1
    ( UNo.   UNIT       PART   ~
    (  2     FACING     FACE
  A (
    ( UNo.   UNIT       PART   ~
    (  3     BAR         OUT
      UNo.   UNIT              ~  HEAD
       4     TRANSFER   BAR         2
      UNo.   UNIT        TYPE     HEAD
       5     HEAD        SYNC       1
    ( UNo.   UNIT       PART   ~
    (  6     T.GROOVE   OUT
  B (
    ( UNo.   UNIT       PART   ~
    (  7     BAR        OUT
      UNo.   UNIT              ~  HEAD
       8     TRANSFER   BAR         1
      UNo.   UNIT        TYPE     HEAD
       9     HEAD        SIN        2
    ( UNo.   UNIT       PART   ~
    (  10    FACING     BACK
  C (
    ( UNo.   UNIT       PART   ~
    (  11    T.DRILL    BACK
      UNo.   UNIT         ~
       12    END
```



A: Machining on the No. 1 spindle side
B: Machining in synchronization of the No. 1 and No. 2 spindles
C: Machining on the No. 2 spindle side

3-21 Workpiece Transfer Unit (TRANSFER)

Select the workpiece transfer unit to change the chucking position of the workpiece, to deliver it from one spindle to the other or to move the No. 2 spindle on the B-axis.
Press the **[TRANSFER WORKPICE]** menu key to select this unit.

3-21-1 Setting unit data

UNo.	UNIT	SETUP-No.	HEAD	SPDL	PUSH	CHUCK
*	TRNSFER [1]	[2]	[3]	[4]	[5]	[6]

[1] UNIT

From the following menu, select a transfer pattern.

CHK CHUCK WORK	BAR BAR LOOP	MOV S-SPDL MOVE							

CHUCK: To deliver a chuck work from No. 1 to No. 2 spindle or vice versa.
BAR: To rechuck a bar work.
MOVE: To move the No. 2 spindle on the Z-axis.

[2] SETUP-No.

Enter the registration number under which the data of delivering position, Z-offset etc. are to be registered on the **TRANSFER** display.

[3] HEAD

- If you have selected **CHUCK** for data item [1] above:
  The following menu will be displayed when the cursor is placed at this item.

HEAD 1 →HEAD 2	HEAD 2 →HEAD 1								

  Select **[HEAD 1 → HEAD 2]** to deliver a workpiece from the No. 1 to the No. 2 spindle.
  Select **[HEAD 2 → HEAD 1]** to deliver a workpiece from the No. 2 to the No. 1 spindle.

- If you have selected **BAR** for data item [1] above:
  The following menu will be displayed when the cursor is placed at this item.

HEAD 1	HEAD 2								

  Select **[HEAD 1]** to rechuck a workpiece on the No. 1 spindle side.
  Select **[HEAD 2]** to rechuck a workpiece on the No. 2 spindle side.

- If you have selected **MOVE** for data item [1] above:
  No data setting is required.

[4] SPDL

From the following menu, specify a spindle action (status) during the unit.

0 SPINDLE STOP	1 FORWARD	2 REVERSE	3 ORIENT	4 C-AXIS POSITION	5 KEEP				

0 SPINDLE STOP: The spindle does not rotate.

1 FORWARD: The spindle rotates forward at the speed set previously in parameter **U27**.

2 REVERSE: The spindle rotates backward at the speed set previously in parameter **U27**.

3 ORIENT: The spindle is orientated.

4 C-AXIS POSITION: The No. 1 spindle undergoes C-axis positioning. The No. 2 spindle undergoes orientation or C-axis positioning, depending on the machine specifications.

5 KEEP: The spindle status specfied in the previous unit is held.

Note: Spindle status pattern 5 (spindle status hold) can be selected when bit 5 in parameter **P107** is “1”.

[5] PUSH

Only when **CHUCK** is selected for data item [1] above, select whether the workpiece is to be pressed by the No. 2 spindle when workpiece delivery is performed.
Set 0 if pressing is to be done, or set 1 if pressing is not to be done.

[6] CHUCK

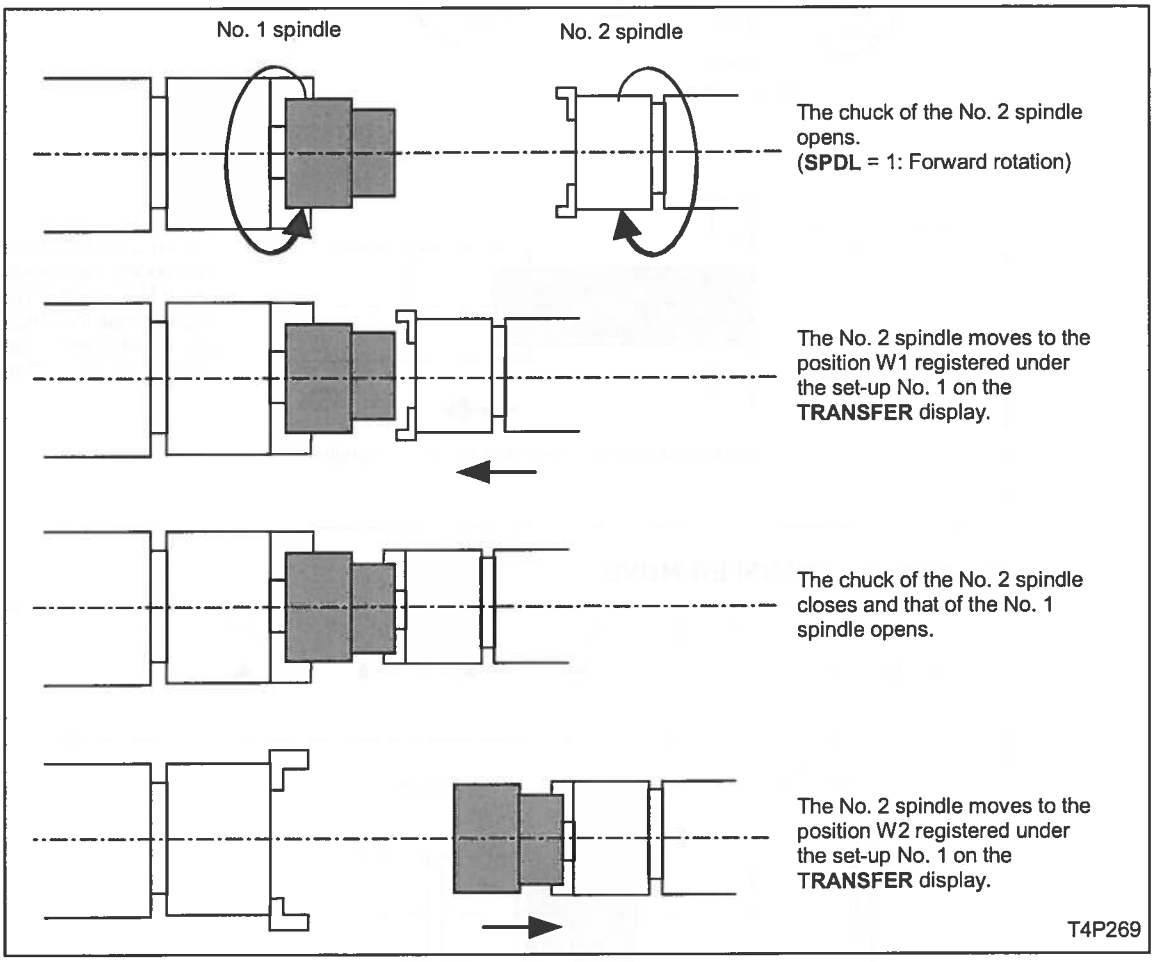
Only when **BAR** is selected for data item [1] above, specify whether the chuck of the spindle indicated in item [3] is to be left open or to be closed after movement of the No. 2 spindle.

Set 0 to leave the chuck open

Set 1 to close the chuck.

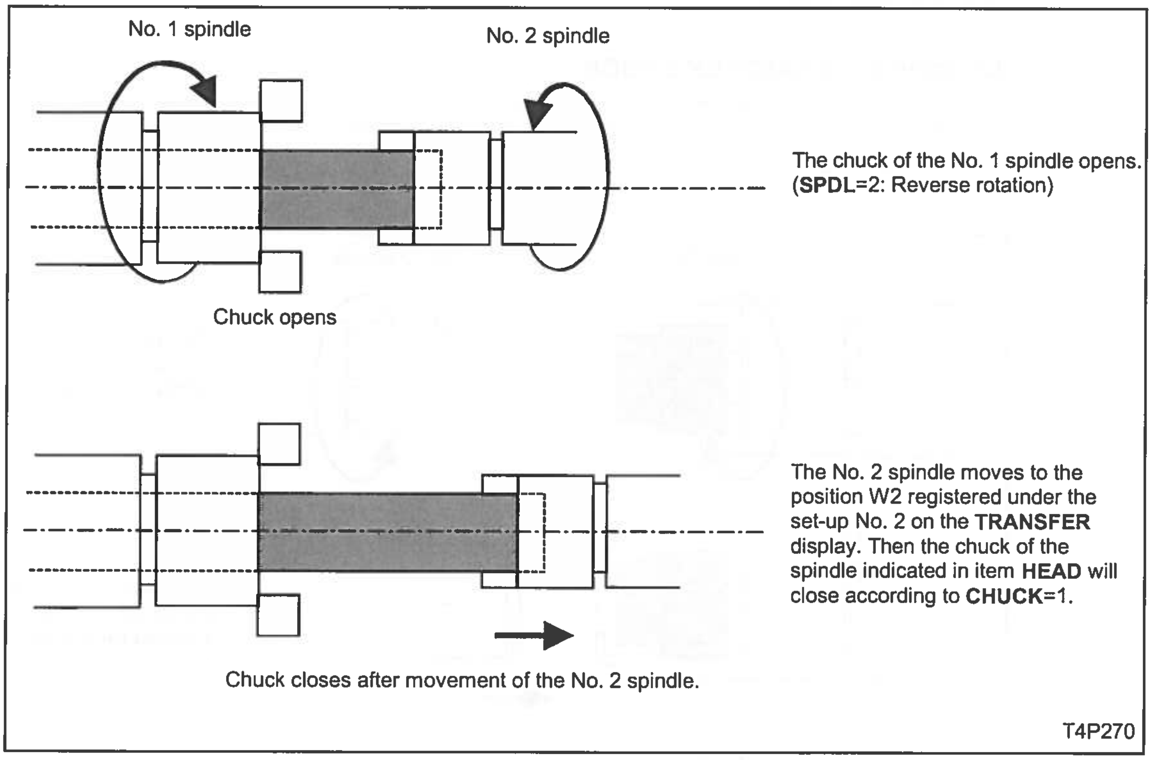
Example 1: TRANSFER CHUCK

UNIT		SETUP-No.	HEAD	SPDL	PUSH	CHUCK
TRANSFER	CHUCK	1	1→2	1	1	◆



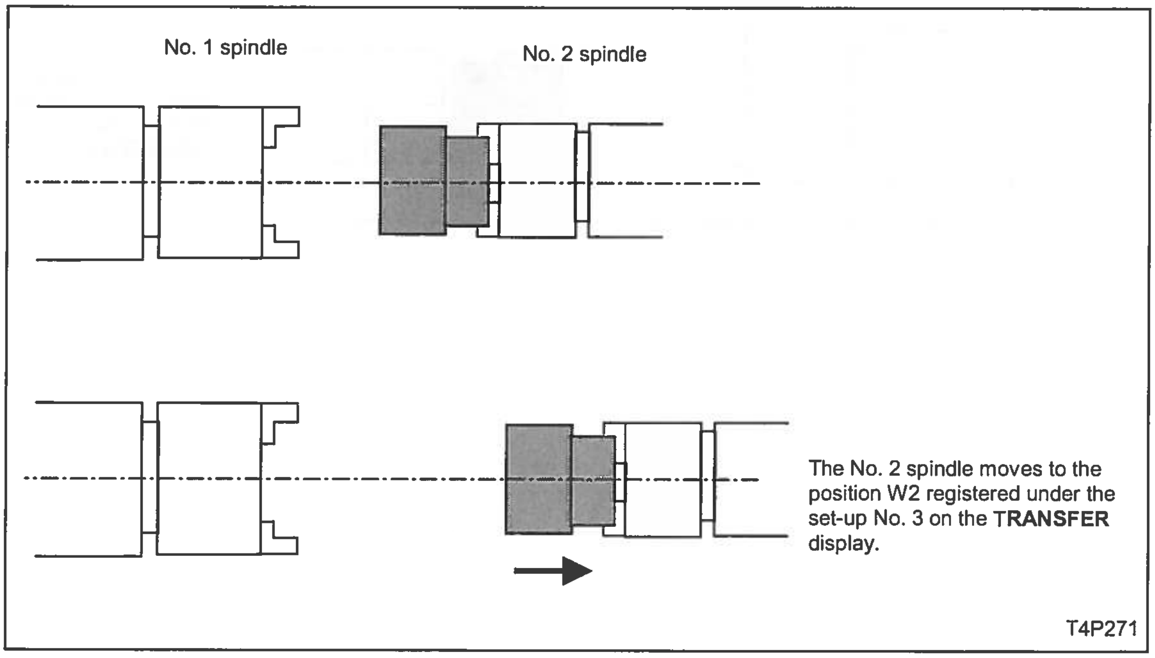
Example 2: TRANSFER BAR

UNIT		SETUP-No.	HEAD	SPDL	PUSH	CHUCK
TRANSFER	BAR	2	1	2	◆	1



Example 3: TRANSFER MOVE

UNIT		SETUP-No.	HEAD	SPDL	PUSH	CHUCK
TRANSFER	MOVE	3	◆	◆	◆	◆



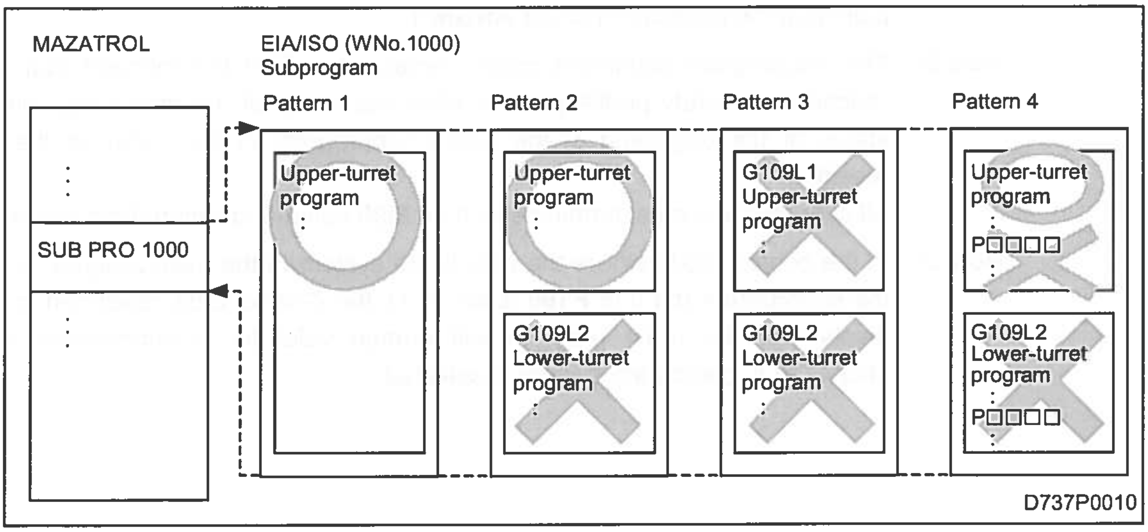
3-22 Subprogram Unit (SUB PRO)

Select the subprogram unit to call up an EIA/ISO or a macro program as a subprogram from the MAZATROL program.
This unit is only available with the optional function for EIA/ISO or macro programming.
Press the **[SUB PROGRAM]** menu key to select this unit.

Note: For the twin-turret specifications, when G109 or a queuing P-code is executed, machining with the EIA/ISO subprogram that includes upper-turret/lower-turret commands will come to an alarm stop. See the figure below.

The cases that the alarm occurs

1. When the G109 command is present in the subprogram
2. When the queuing P-code is present in the subprogram



Pattern 1: The alarm will not occur if the EIA/ISO subprogram does not include any G109 code and the queuing P-code and applies only to the upper turret.

Pattern 2: If the G109L1 command is not specified in the program of the upper turret, the machine will come to an alarm stop when the G109L2 command in the program of the lower turret is executed.

Pattern 3: If the G109L1 command is specified in the program of the upper turret, when this command is executed, the machine will come to an alarm stop.

Pattern 4: If a queuing P-code is specified in the upper turret's program without preceding the G109L1 command, the machine will come to an alarm stop when the P-code is executed.

3-22-1 Setting unit data

UNo.	UNIT	WK.No.	NUM.
*	SUB PRO	[1]	[2]

[1] WK. No.

Enter the work number of the desired subprogram.

[2] NUM.

Enter the number of repetitions of the subprogram.
If no data is entered here, the subprogram will be executed one time.

3-22-2 Setting sequence data

UNo.	UNIT	WK.No.	NUM.			
*	SUB PRO	***	***			
SEQ	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
1	[1] [2]	[1] [2]	[1] [2]	[1] [2]	[1] [2]	[1] [2]

[1] [2] ARGM

First enter the address [1] and then input the data [2] of an argument in pairs.

- To set a macro variable as the argument data, press the **[MACRO INPUT]** menu key before entering the number of the macro variable. In this case, a symbol "#" is displayed before the numerical data.

Note 1: If no argument is required, press the **[SEQUENCE END]** menu key with the cursor placed at the position [1] under **ARGM 1**.

Note 2: The subprogram command mode comes into effect the moment that the tool has reached the safety profile position after machining of the preceding unit. The modal status at the beginning of the called subprogram is the same as the status after resetting.
All subprograms must terminate with an M99 command (return from subprogram).

Note 3: In the control mode where the coordinate system of the main program is conveyed to the subprogram (bit 5 of **P108** is set to 1), the Z-offset data registered in the **SET UP** display for the main program will remain valid for a subprogam in which the MAZATROL coordinate system is selected.

3-23 End Unit (END)

Select the end unit after the entire program data required for machining has been set.
For this unit, set data about the machine action to occur at the end of machining and about the program execution mode. Such data is referred to as end data.
You must set this unit on the last line of a program.
Press **[END]** menu key for this unit.

3-23-1 Setting unit data

UNo.	UNIT	COUNTER	RETURN	WK.No.	CONT.	NUM.	SHIFT
*	END	[1]	[2]	[3]	[4]	[5]	[6]

[1] COUNTER

Specify whether you want the NC unit to count the number of machined workpieces (number of program loops).

- Set 1 if counting is desired.
- Set 0 if counting is not desired.

If you set 1, the number of machined workpieces will be displayed at **COUNTER** of the **POSITION** display.

Note: Counting does not occur if no data has been set in this item.

[2] RETURN

Specify the position to which the tool is to be returned after machining.

- Set 0 to return the tool to the required tool change position.
- Set 1 to return the tool to the machine home position.
- Set 2 to return the tool to the required fixed point.

Note 1: The tool change position is specified in parameter **P17** (for a machine having upper and lower turrets, the tool change position corresponding to the lower turret is specified in parameter **P68**), and the fixed position is specified in parameter **A5**. (See separate Parameter List for further details.)

Note 2: For program loops, the tool return position at the end of machining cycles other than the last one differs according to the particular data setting in bit 7 of parameter **P1**. (See separate Parameter List for further details.)

Note 3: If no data has been set in this item, the NC will interpret that 0 has been set.

Note 4: For a machine having upper and lower turrets, operation by the specification of returning is as shown in the table below. Operation may vary from turret to turret.

	Setting of RETURN		
	0/Blank	1	2
Operation of turret not specified in the program when **P113** bit 4 = 0	No operation		
Others	No operation	Movement to home position after machining	Movement to fixed position after machining

P113 bit 4: Specifies whether a non-specified turret is to be moved during program execution using only the upper or lower turret (1: moved, 0: not moved). See the Parameter List for further details.

[3] WK. No.

If the starting part of a different program is to be called up after machining, set the work number of that program.

Note: If no data has been set in this item, the starting part of the current program will be called up automatically after machining.

[4] CONT.

Specify whether you want to carry out the machining operation repeatedly in succession.

- Set 1 to execute the current program perpetually or to execute a different program following completion of the current program.
- Set 0 to execute the current program once or to repeatedly execute the current program the number of times that is to be specified in item [5] below.

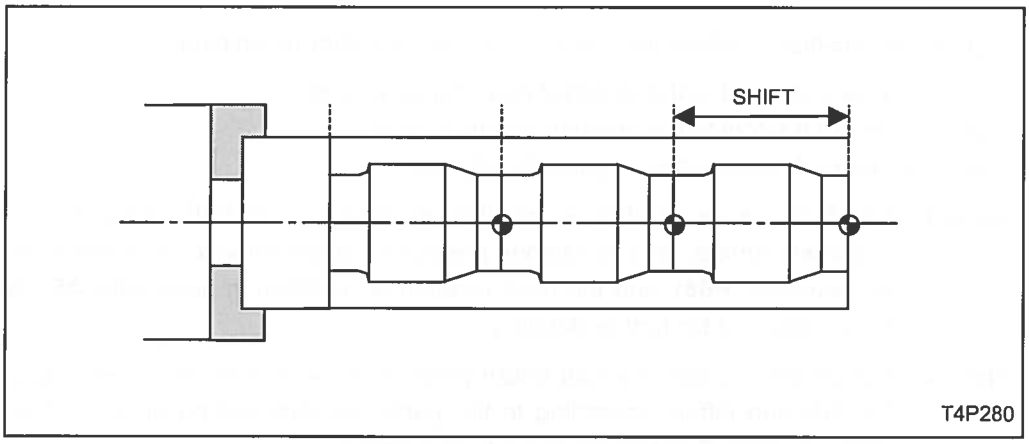
Note: If no data has been set in this item, the NC will interpret that 0 has been set.

[5] NUM.

If the current program is to be executed repeatedly, set the desired number of times of execution.

[6] SHIFT

Shifting the origin of the current program and repeatedly executing it enable multiple parts of the same shape, or a single part of identical recurring shape patterns as shown in diagram below, to be made from one workpiece.
For such machining, set the desired shift amount of the program origin in this item.



Note 1: Data set in this item becomes valid only if the number of times of program execution has been set in item [5] above.

Note 2: If no data has been set in this item, the shift amount will be regarded as 0.

Note 3: Data must not be set in this item if a measurement unit is to be executed. Setting data other than 0 may cause contact between the touch sensor and the workpiece.

Note 4: As for repetitive machining on a single workpiece, the following condition must be satisfied:

LENGTH	>	NUM. × SHIFT	+	WORK FACE
(Common data)		(END unit)		(Common data)

Examples of program execution mode

The program execution mode is determined by the data that has been set in items [3] through [6] above.

If the following data has been set for the end unit of the program of work number A:

	WK. No.	CONT.	NUM.	SHIFT
Example 1	Blank	0 or blank	0 or blank	0 or blank
Example 2	B	0 or blank	0 or blank	0 or blank
Example 3	Blank	1	0 or blank	0 or blank
Example 4	B	1	0 or blank	0 or blank
Example 5	Blank	0 or blank	N	0 or blank
Example 6	B	0 or blank	N	0 or blank
Example 7	Blank	1	N	0 or blank
Example 8	B	1	N	0 or blank
Example 9	Blank	0 or blank	N	s
Example 10	B	0 or blank	N	s
Example 11	Blank	1	N	s
Example 12	B	1	N	s

1: The program of work number **A** is executed only once and the machine stops. At that time, the starting part of the same program is called up automatically.

2: The program of work number **A** is executed only once and the machine stops. At that time, the starting part of the program of work number **B** is called up automatically.

3: The program of work number **A** is executed repeatedly.

4: The program of work number **A** is executed only once, and following this, the program of work number **B** is executed.

5: The program of work number **A** is executed an **N** number of times and the machine stops. At that time, the starting part of the same program is called up automatically.

6: The program of work number **A** is executed an **N** number of times and the machine stops. At that time, the starting part of the program of work number **B** is called up automatically.

7: The program of work number **A** is executed repeatedly.
(Data set in "NUM." becomes invalid.)

8: The program of work number **A** is executed an **N** number of times, and following this, the program of work number **B** is executed.

9: The program of work number **A** is repeatedly executed an **N** number of times while having its origin shifted through the distance **s** and the machine stops. At that time, the starting part of the same program is called up automatically.

10: The program of work number **A** is repeatedly executed an **N** number of times while having its origin shifted through the distance **s** and the machine stops. At that time, the starting part of the program of work number **B** is called up automatically.

11: The program of work number **A** is repeatedly executed an **N** number of times while having its origin shifted through the distance **s**. Then the shift amount is canceled and the same operation is executed repeatedly.

12: The program of work number **A** is repeatedly executed an **N** number of times while having its origin shifted through the distance **s** and following this, the program of work number **B** is executed.

3-24 Simultaneous Machining Unit (SIMULTAN)

For a machine equipped with upper and lower turrets, select the simultaneous machining unit when performing turning operations using both turrets at the same time. Press the **[SIMUL]** menu key to select the simultaneous machining unit.

See Chapter 4, "LOWER-TURRET CONTROL FUNCTIONS", for further details of this unit.

3-24-1 Setting unit data

UNo.	UNIT	SIMUL.No.	RPM
*	SIMULTAN	[1]	[2]

[1] SIMUL No.

Specify the group number for the simultaneous machining using both turrets.

[2] RPM

Enter the turning spindle speed of the simultaneous machining group specified in [1] above.

3-25 Coordinate Measuring Unit (WPC MSR)

The coordinate measuring unit measures coordinates using a touch sensor during automatic operation and automatically establishes the workpiece coordinate system.

The position of the reference face is measured and the Z-offset value is automatically set. Or the center of a projection or a groove is measured and the C-offset value is automatically set.

3-25-1 Procedure for calling up the WPC MSR unit

(1) Press the menu selector key (key located at the right of the menu keys). The following menu will be displayed.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Press the **[>>>]** menu key. The following menu will be displayed.

	M CODE	SUB PROGRAM	SEPARATE PROCESS	TRANSFER WORKPICE	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	SIMUL.	>>>

(3) Press the **[WPC MSR]** menu key.

3-25-2 Setting unit data

1. **Setting WPC MSR unit data**

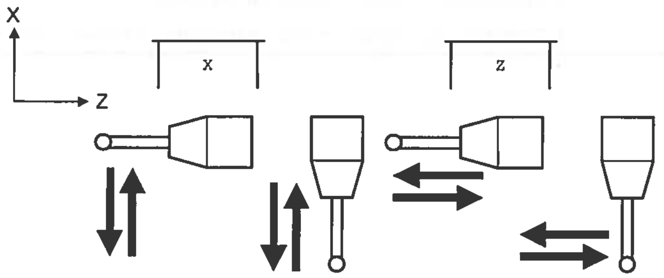
UNo.	UNIT	TOOL	NOM-ϕ
	WPC MSR	TOL SENS OUT	45.A

Cursor position	Description
NOM-ϕ	Specify the nominal diameter of the feeler. Enter the approximate radius of the point of feeler by means of numeric keys.

3-25-3 Setting sequence data

1. Setting WPC MSR sequence data

SNo.	PTN	X	Y	Z	C	DIR.	R	D/L	K	DIR.
1										

<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>PTN</td><td>Select the type of measurement from the following menu.
[Z FACE] [C FACE] [C GRV] [C STP]
For details, refer to “Type of measurement”.</td></tr>
<tr><td>X,Y,Z,C</td><td>Specify the measurment starting position by means of numeric keys.</td></tr>
<tr><td>DIR.</td><td>Select a measuring approach direction from the menu.
[CW] [CCW]
Specify the relative direction for the measuring sensor with respect to the workpiece (namely, the direction in which the tool is to be moved during tool path checking).</td></tr>
<tr><td>R</td><td>Specify the surface coordinates to be measured by means of numeric keys.</td></tr>
<tr><td>D/L</td><td>Specify the width of the groove, the width of the projection, etc.</td></tr>
<tr><td>K</td><td>Specify the returning distance from the stop point after the skip measurement with the measurement approach speed has been completed.</td></tr>
<tr><td>DIR.</td><td>Select from the menu the approach/escape direction to be applied to the measurement.
[X] [Z]
Selection of the [X] menu item specifies the approach/escape in X-axial direction during the measurement.


D736P0107
The tool direction is determined according to the setting on the TOOL DATA display.</td></tr>
</table>

Note: The coordinate measuring unit operates in the original programmed coordinate system until all measuring patterns contained in the unit have been executed to completion. Measured data becomes valid for the next unit onward. Under normal operating conditions, set this unit at the beginning of the program.

3-25-4 Type of measurement

Select the type of measurement for the offset of the coordinates system. The four types of measurement available are described in the following descriptions 1 to 4:

Measurement of the reference surface .... **Z-FACE, C-FACE**
Measurement of groove center .... **C-GRV**
Measurement of center of projection width ... **C-STP**

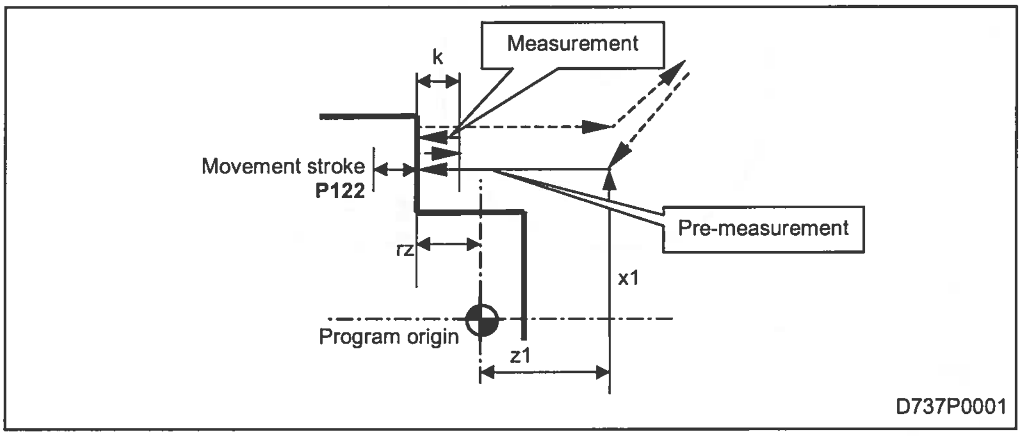
1. Z-FACE

[Function]

The Z-offset value can be adjusted by entering the distance from the workpiece origin to reference surface Z.

SNo.	PTN	X	Y	Z	C	DIR.	R	D/L	K	DIR.
1	Z-FACE	x1	◆	z1	◆	◆	rz	◆	k	X

◆: Not necessary to be set here.



If 0 is entered in "k" (the amount of measurement return), only pre-measurement at the measurement approach speed will occur and measurement at the measuring speed will not occur.

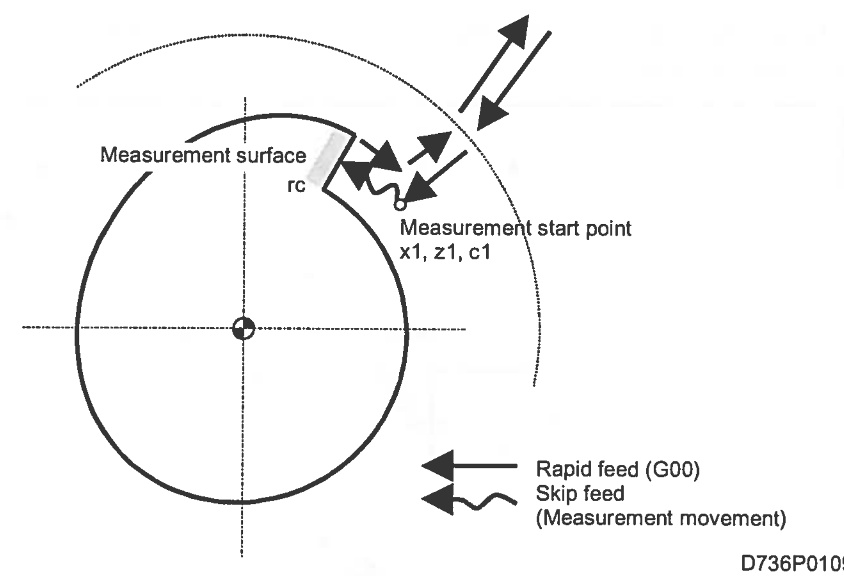
2. C-FACE

[Function]

The C-offset value can be adjusted by entering the distance from the workpiece origin to reference surface C.

SNo.	PTN	X	Y	Z	C	DIR.	R	D/L	K	DIR.
1	C-FACE	x1	◆	z1	c1	◆	rc	◆	◆	X

◆: Not necessary to be set here.



D736P0109

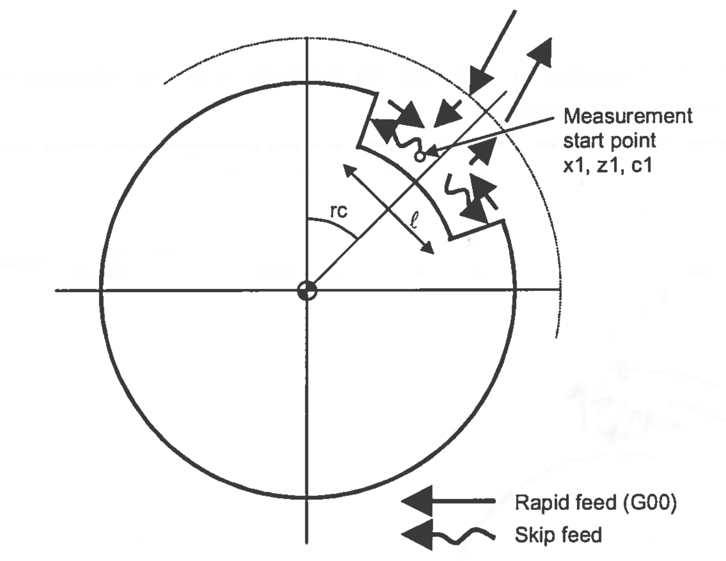
3. C-GRV

[Function]

The C-offset value can be adjusted by entering the distance from the workpiece origin to the groove center and the groove width.

SNo.	PTN	X	Y	Z	C	DIR.	R	D/L	K	DIR.
1	C-GRV	x1	◆	z1	c1	CCW	rc	ℓ	k	Z

◆: Not necessary to be set here.



x1, z1, c1:
Coordinates of the measurement start point based on the workpiece origin

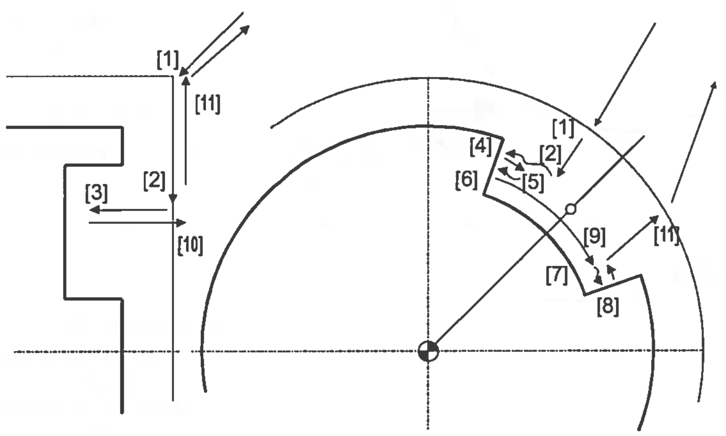
rc: Distance from the origin to the center of the groove on C (Dimension shown on drawing)

ℓ: Width of groove

k: Feed distance at skip speed

D736P0110

[Measurement movement]



The above sequence is an example of tool approach or escape in the Z-axial direction. For tool approach or escape in the X-axial direction, the moving sequence of the X-axis and Z-axis in steps [2], [3], [10], and [11] reverses.

[1] Movement to the clearance point
[2] Movement to measurement start point x1, c1
[3] Movement to measurement start point z1 at the measuring approach speed
[4] Movement at the measuring approach speed in the measuring direction (pre-measurement)
[5] Movement only through measuring return stroke "k" from the skipping position
[6] The first measurement at the skip feed
[7] Movement to the measuring position at the opposite side
[8] The second measurement at the skip feed
[9] Movement only through measuring return stroke "k" from the skipping position
[10] Movement to the edge clearance point
[11] Movement to the outside diameter clearance point

D736P0111

Note 1: If the sensor operates during movement at the measuring approach speed in steps [2] and [3], these steps will be repeated again (this is referred to as the retry function). The retry function is described later in this manual.

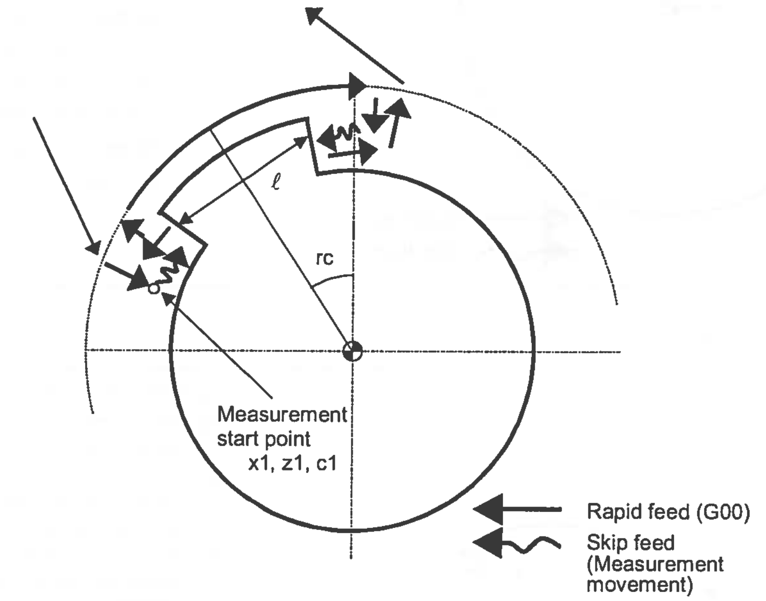
Note 2: If 0 is entered for the amount of measurement return, only pre-measurement at the measuring approach speed for one side will occur and both-side measurement at the measuring speed will not occur.

4. C-STP

The C-offset value can be adjusted by entering the distance from the workpiece origin to the projection center and the projection width.

SNo.	PTN	X	Y	Z	C	DIR.	R	D/L	K	DIR.
1	C-STP	x1	◆	z1	c1	CW	rc	ℓ	k	X

◆: Not necessary to be set here.



x1, z1, c1: Coordinates of the measurement start point based on the workpiece origin

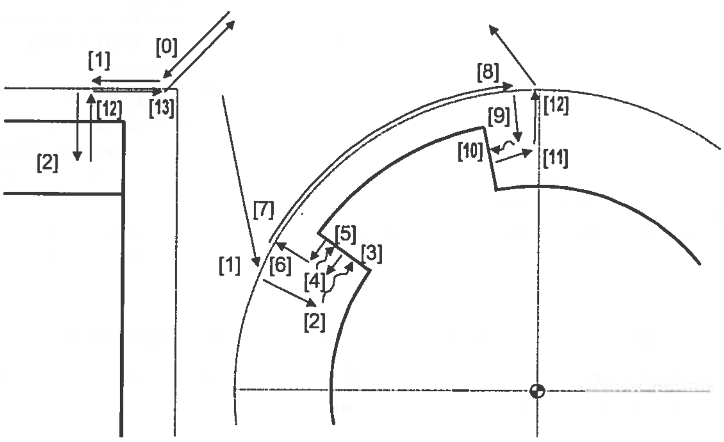
rc: Distance from the origin to the center of the projection on C (Dimension shown on drawing)

ℓ: Width of projection

k: Feed distance at skip speed

D736P0112

[Measurement movement]



The above sequence is an example of tool approach or escape in the X-axial direction. For tool approach or escape in the Z-axial direction, the moving sequence of the X-axis and Z-axis in steps [1], [2], and [12] reverses.

- [0] Movement to the clearance point
- [1] Movement to measurement start point z1, c1
- [2] Movement to measurement start point x1 at the measuring approach speed
- [3] Movement at the measuring approach speed in the measuring direction (pre-measurement)
- [4] Movement only through measuring return stroke "k" from the skipping position
- [5] The first measurement at the skip feed
- [6] Movement only through measuring return stroke "k" from the skipping position
- [7] Movement to the outside diameter clearance point
- [8] [9] Movement to the measurement starting position at the opposite side
- [10] The second measurement at the skip feed
- [11] Movement only through measuring return stroke "k" from the skipping position
- [12] Movement to the outside diameter clearance point
- [13] Movement to the edge clearance position

D736P0136'

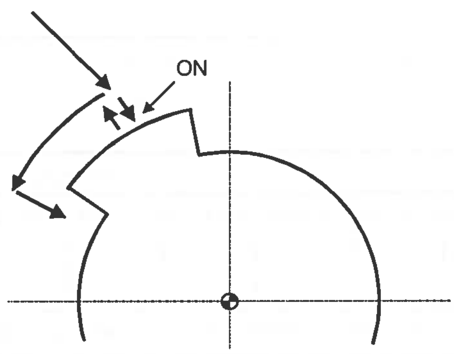
Note 1: If the sensor operates during movement at the measuring approach speed in step [2], this step will be repeated again (this is referred to as the retry function). The retry function is described below.

Note 2: If 0 is entered for the amount of measurement return, only pre-measurement at the measuring approach speed for one side will occur and both-side measurement at the measuring speed will not occur.

[Measurement retry]

After temporarily returning to the outside diameter clearance point, the feeler shifts in a circumferential direction through the distance corresponding to [Measuring width × Parameter **U61** (Measurement retry width) / 100] and then returns to measurement.
The retry function is executed the number of times specified in parameter **P20** (measurement retry count). If the sensor operates in spite of this count being exceeded, an alarm will result.

Note: During the retry function for **C-STP** measurement, an actual retry count may be less than the setting of **P20** (specified retry count).
Since the retry count is limited to such a value that does not cause the total shifting width by retry to exceed a projection width of 100%, the system operates as follows:
If **P20** × **U61** < 100, measurement is repeated as often as the setting of **P20**.
If **P20** × **U61** ≥ 100, measurement is repeated as often as the number of times obtained by rounding any fractions of 100/**U61**.



D736P0139

3-26 Workpiece Measuring Unit (WORK MES)

The workpiece measuring unit uses a touch sensor during the automatic operation to measure the dimensions of the workpiece. Also, measurement results are used to correct the tool wear and tool diameter offset data.

3-26-1 Procedure for selecting workpiece measuring unit

(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Press the **[>>>]** menu key. The following menu will be displayed.

	M CODE	SUB PROGRAM	SEPARATE PROCESS	TRANSFER WORKPICE	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	SIMUL.	>>>

(3) Press the **[WORKPICE MEASURE]** menu key.

3-26-2 Setting the unit data

1. Setting WORK MES unit data

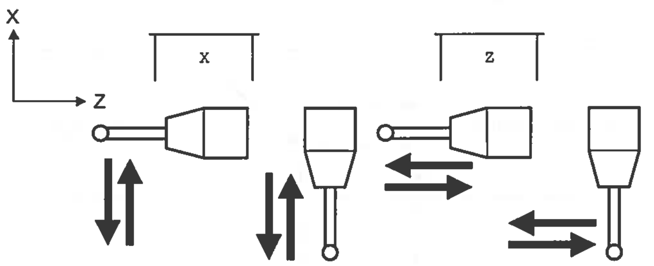
UNo.	UNIT	OFS	OFS-TOOL	COMP.DATA	SNS-TOOL	OUTPUT
	WORK MES				TOL SENS	

Cursor position	Description
OFS	Select from the menu whether the measurement results are to be used to correct tool data. [YES] [NO]
OFS-TOOL	Specify by its tool name, nominal diameter (nominal size), identification code, and turret number, the tool for which the measurement results are to be incorporated. If **NO** has been specified in **OFS**, a ◆ mark is displayed to indicate that no data can be entered.
COMP.DATA	When the offset tool is either an end-milling tool, a face-milling tool, a ball end-milling tool, a special tool, or a tap, select the measurement results incorporating destination from the menu. [DIAMETER] [LENGTH] If a tool other than those mentioned above has been selected as the offset tool, a ◆ mark is displayed to indicate that no data can be entered. If **NO** has been specified in **OFS**, a ◆ mark is displayed to indicate that no data can be entered.
SNS-TOOL	Enter the nominal diameter, machining part and identification code of the touch sensor.
OUTPUT	Select whether the measurement results are to be sent to external equipment. 0: No output 1: Output to a text file on the HDD 2: Output to a serial printer via an RS-232C interface **Note:** Specify output items in parameter **P112**.

3-26-3 Setting the sequence data

1. Setting WORK MES sequence data

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1											

<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>PTN</td><td>Select a workpiece measuring pattern from the menu.
A press of the [>>>] menu key displays menus in the order of [1]→[2] →[3] →[1].
<table>
<tr><td>OUTER X DIA</td><td>OUTER Y DIA</td><td>INNER X DIA</td><td>INTER Y DIA</td><td></td><td>X GRV</td><td>Y GRV</td><td>Z GRV</td><td></td><td>>>></td><td>[1]</td></tr>
<tr><td>X WIDTH</td><td>Y WIDTH</td><td>Z WIDTH</td><td>+X STEP</td><td>-X STEP</td><td>+Y STEP</td><td>-Y STEP</td><td>+Z STEP</td><td>-Z STEP</td><td>>>></td><td>[2]</td></tr>
<tr><td>INNER GRV</td><td>INNER WIDTH</td><td>EXT M.TOOL</td><td>EXT T.TOOL</td><td></td><td></td><td></td><td></td><td></td><td>>>></td><td>[3]</td></tr>
</table>
When [OUTER X DIA] or [INNER X DIA] is set here, select the single-side measurement or both-side measurement next.</td></tr>
<tr><td>SPT-X
SPT-Y
SPT-Z</td><td>Specify the starting position of measurement.
Setup data items differ according to the selected measuring pattern.</td></tr>
<tr><td>FPT-X
FPT-Y
FPT-Z</td><td>Specify the ending position of measurement.
Setup data items differ according to the selected measuring pattern.</td></tr>
<tr><td>T LIM+</td><td>Set the upper-limit value of the tolerance.</td></tr>
<tr><td>T LIM-</td><td>Set the lower-limit value of the tolerance.</td></tr>
<tr><td>BASE</td><td>Set the reference position for measurement.
0: The starting position is defined as reference.
1: The ending position is defined as reference.</td></tr>
<tr><td>DIR.</td><td>Select from the menu the approach/escape direction to be applied to the measurement.
<table>
<tr><td>X</td><td>Z</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
</table>
Selection of the [X] menu item specifies the approach/escape in X-axial direction during the measurement.


D736P0107
The approach/escape directions that can be used for each measuring pattern are shown in the table below.
<table>
<tr><th>Pattern</th><th>Approach dir.</th><th>Pattern</th><th>Approach dir.</th><th>Pattern</th><th>Approach dir.</th></tr>
<tr><td>X WIDTH</td><td>Z</td><td>+X STEP</td><td>Z</td><td>INNER X</td><td>Z</td></tr>
<tr><td>Y WIDTH</td><td>X, Z</td><td>–X STEP</td><td>X, Z</td><td>INNER Y</td><td>Z</td></tr>
<tr><td>Z WIDTH</td><td>X</td><td>+Y STEP</td><td>X, Z</td><td>OUTER X</td><td>Z</td></tr>
<tr><td>X GRV</td><td>Z</td><td>–Y STEP</td><td>X, Z</td><td>OUTER Y</td><td>X, Z</td></tr>
<tr><td>Y GRV</td><td>X, Z</td><td>+Z STEP</td><td>X, Z</td><td>IN WIDTH</td><td>Z</td></tr>
<tr><td>Z GRV</td><td>X</td><td>–Z STEP</td><td>X, Z</td><td>IN GRV</td><td>Z</td></tr>
</table></td></tr>
</table>

3-26-4 Selection of a measurement type

The following measurement types are provided for the workpiece measurement unit.

- Outside-diameter measurement (**OUTER X**, **OUTER Y**)
  .... To measure the outside-diameter of machined workpiece.
- Inside-diameter measurement (**INNER X**, **INNER Y**)
  .... To measure the inside-diameter of machined workpiece.
- Groove width measurement (**X GRV**, **Y GRV**, **Z GRV**, **IN GRV**)
  .... To measure the width of groove or other recesses.
- Protrusion width measurement (**X WIDTH**, **Y WIDTH**, **Z WIDTH**, **IN WIDTH**)
  .... To measure the width of protrusion or other convexities.
- Step distance measurement (**+X STEP**, **–X STEP**, **+Y STEP**, **–Y STEP**, **+Z STEP**, **–Z STEP**)
  .... To measure the step distance of machined workpiece
- External measurement (**EXT MILL**, **EXT TURN**)
  .... To read the measurement data of an external measuring unit.
  (During this unit, the data is only read in for compensating the tool data registered in the NC unit; no machine action takes place.)

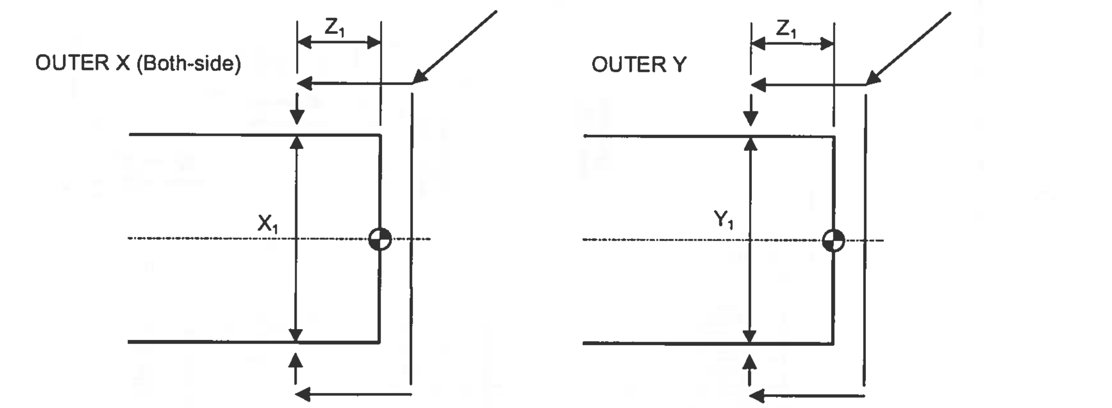
1. **Outside-diameter measurement**

Select **OUTER X** to measure any two X-axial points on the outside-diameter section of the workpiece. Select **OUTER Y** to measure any two Y-axial points on the outside-diameter section of the workpiece.

When performing the **OUTER X** function for a workpiece such as a shaft workpiece, it is possible to conduct tool corrections by measuring only one point on the outside-diameter section. This is referred to as single-side measurement. In this case, a predefined reference point is also measured and then the results are compared with those of the above-measured point to calculate the tool offset value.

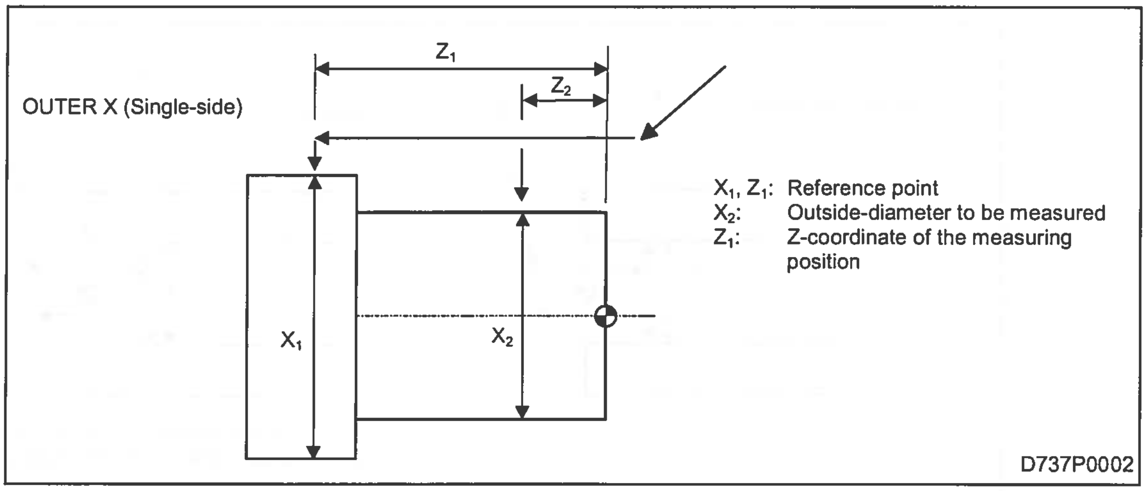
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	OUTER X 0	X_1	◆	Z_1	X_2	◆	Z_2	T_1	T_2	S	Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	OUTER Y	◆	Y_1	Z_1	◆	◆	◆	T_1	T_2	S	Z



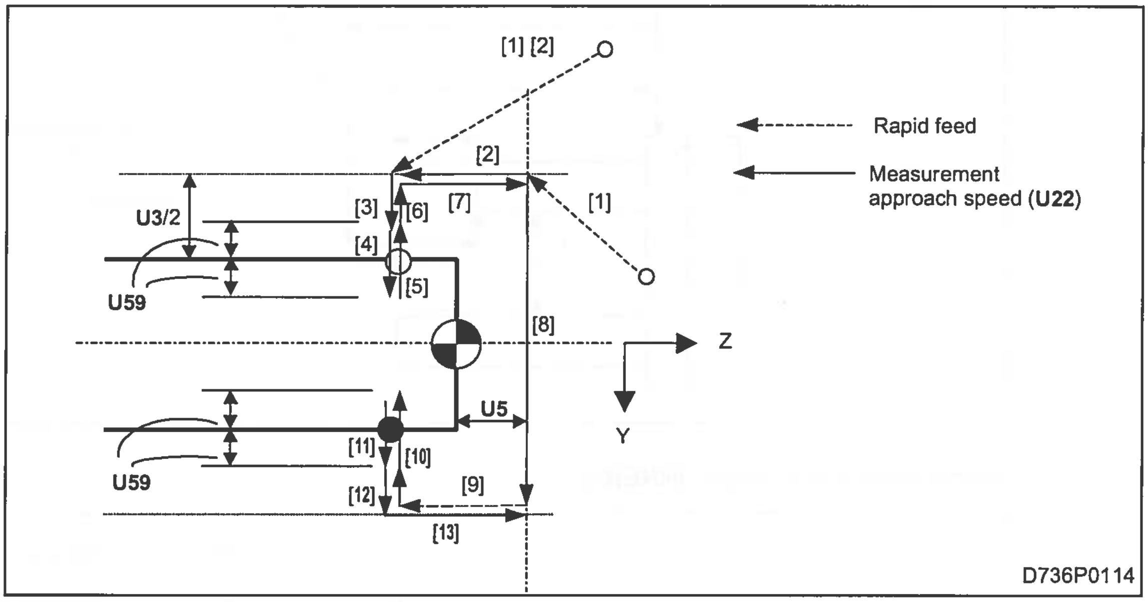
X_1, Y_1: Outside-diameters to be measured
Z_1: Z-coordinate of the measuring position

D736P0113



Set "0" as the reference position in the case of single-side measurement.

[Measurement movement (**OUTER Y**)]



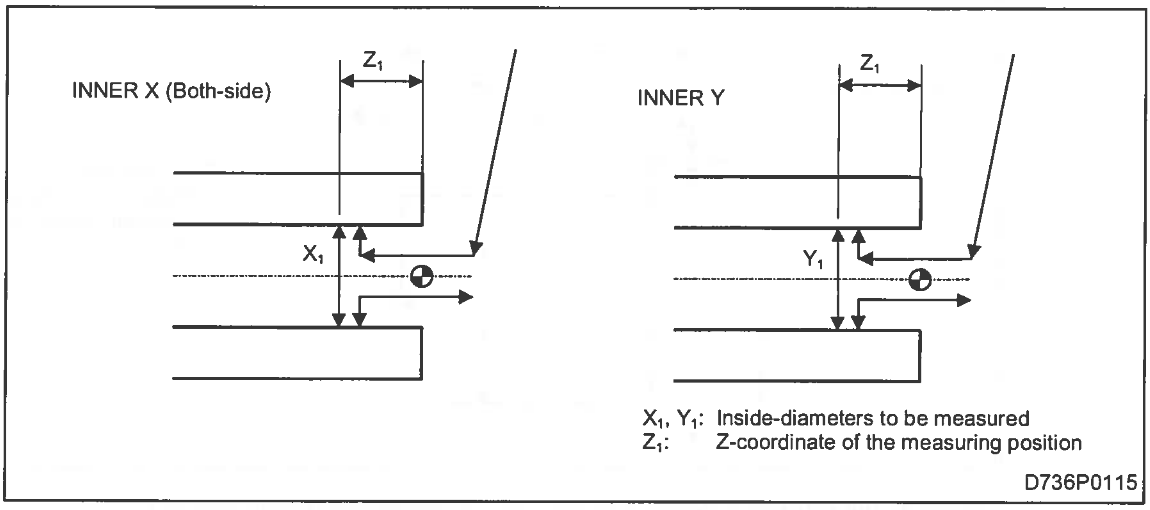
2. Inside-diameter measurement

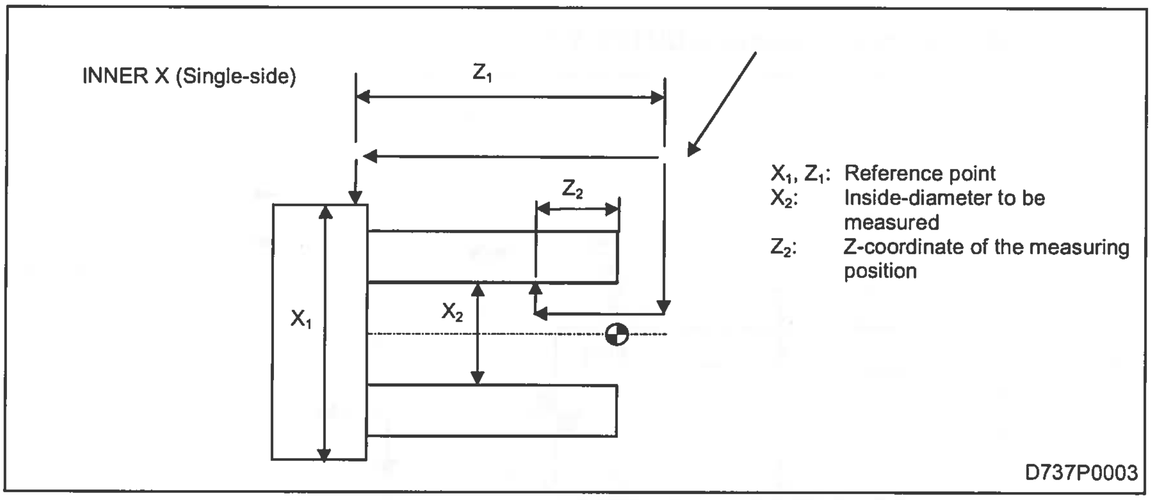
Select **INNER X** to measure any two X-axial points on the inside-diameter section of the workpiece. Select **INNER Y** to measure any two Y-axial points on the inside-diameter section of the workpiece.

For **INNER X** tool correction is also possible by measuring only one point on the inside-diameter section. This is referred to as single-side measurement. In this case, a predefined reference point is also measured and then the results are compared with those of the above-measured point to calculate the tool offset value.

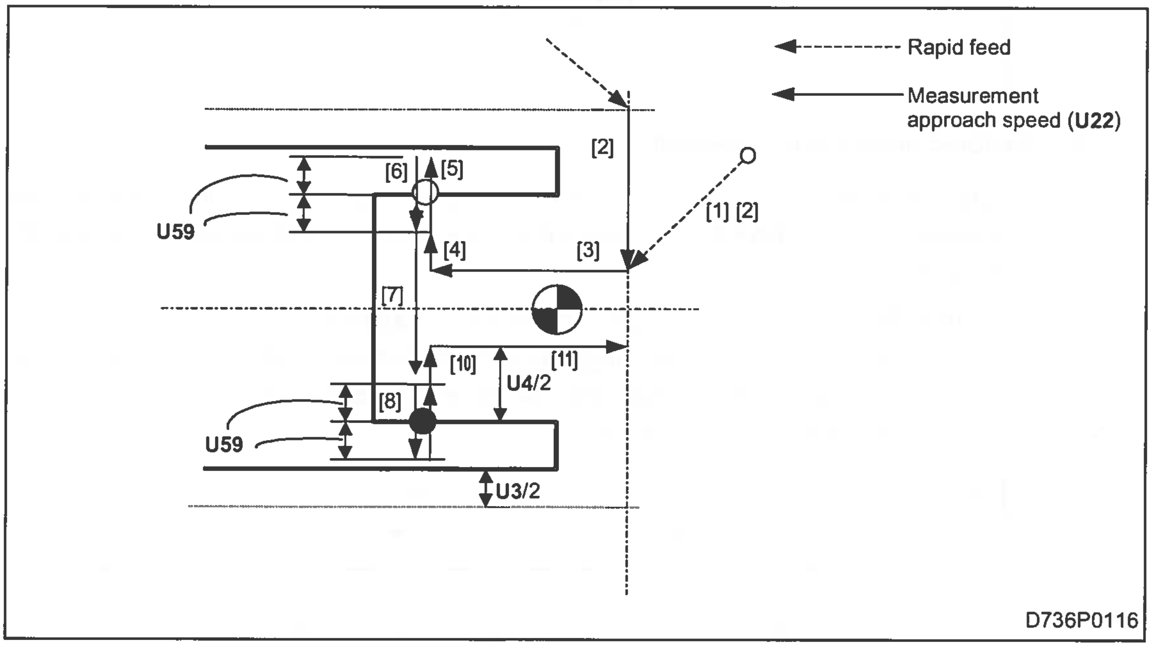
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	INNER X 0	X_1	◆	Z_1	X_2	◆	Z_2	T_1	T_2	S	Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	INNER Y	◆	Y_1	Z_1	◆	◆	◆	T_1	T_2	S	Z





[Measurement movement (**INNER**)]

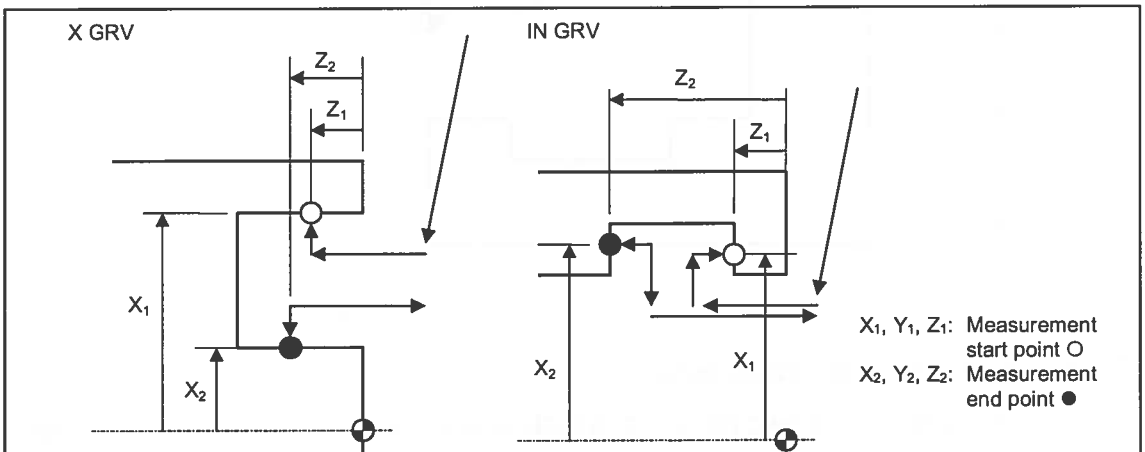


3. Groove width measurement

For **X GRV**, **Y GRV**, and **Z GRV** measurements are performed on X-axial, Y-axial, and Z-axial groove widths, respectively.
For **IN GRV**, the groove width at the inside diameter side is measured.

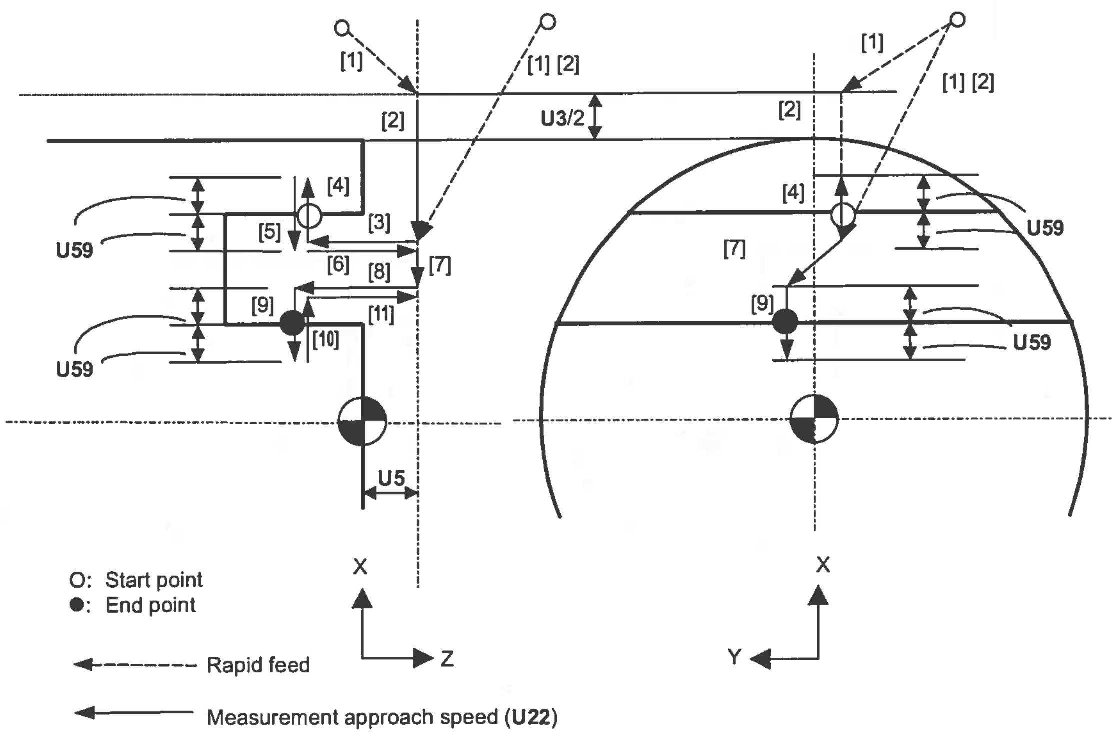
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	X GRV	X_1	Y_1	Z_1	X_2	Y_2	Z_2	T_1	T_2	S	Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	IN GRV	X_1	Y_1	Z_1	X_2	Y_2	Z_2	T_1	T_2	S	Z



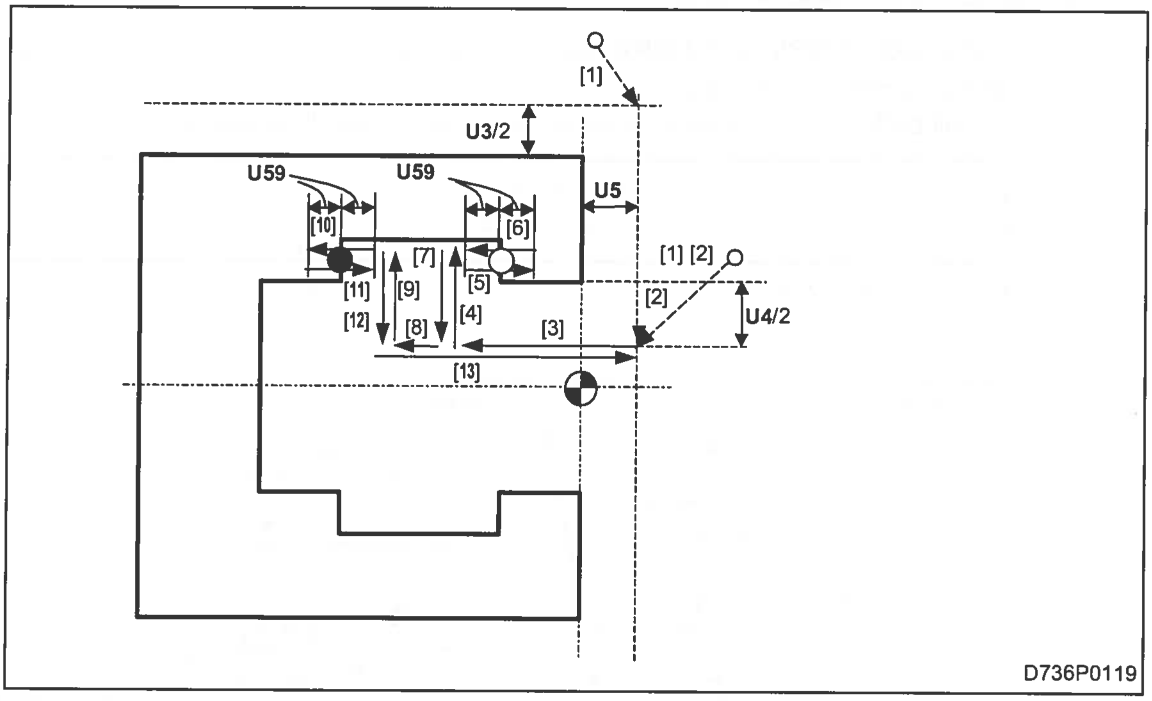
D736P0117

[Measurement movement (**X GRV**)]



D736P0118

[Measurement movement (**IN GRV**)]

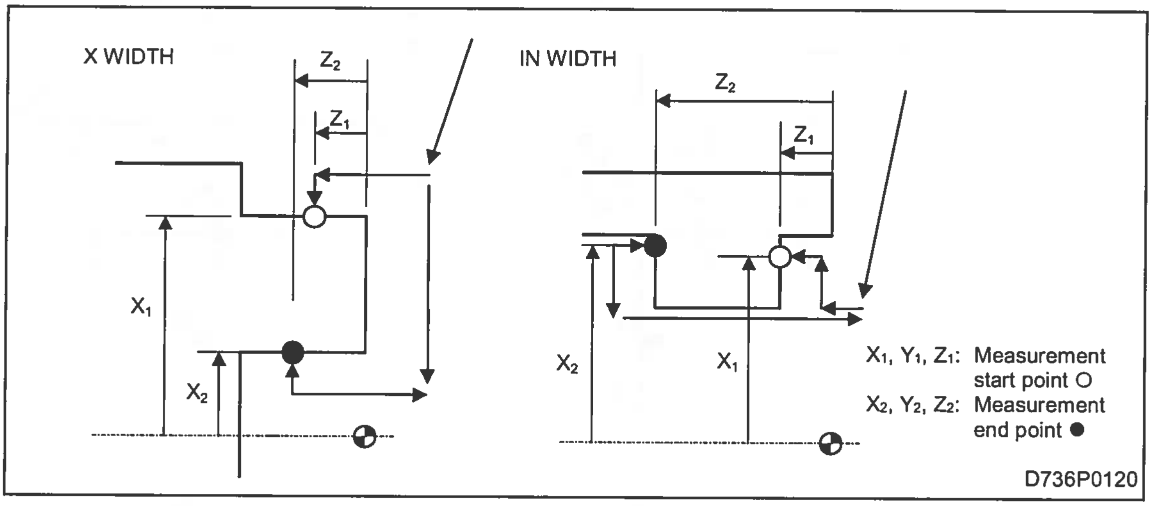


4. **Protrusion width measurement**

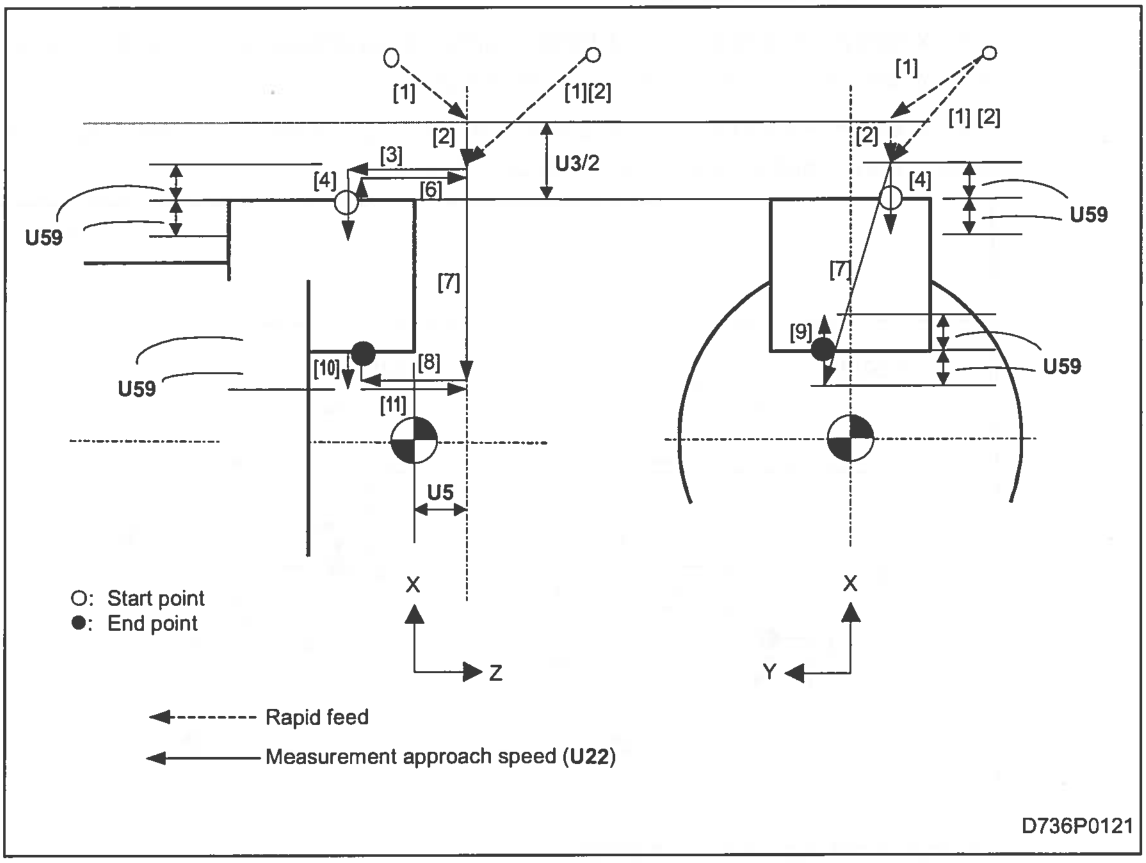
For **X WIDTH**, **Y WIDTH**, and **Z WIDTH** measurements are performed on X-axial, Y-axial, and Z-axial protrusion widths, respectively.
For **IN WIDTH**, the protrusion width at the inside diameter side is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	X WIDTH	X_1	Y_1	Z_1	X_2	Y_2	Z_2	T_1	T_2	S	Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	IN WIDTH	X_1	Y_1	Z_1	X_2	Y_2	Z_2	T_1	T_2	S	Z

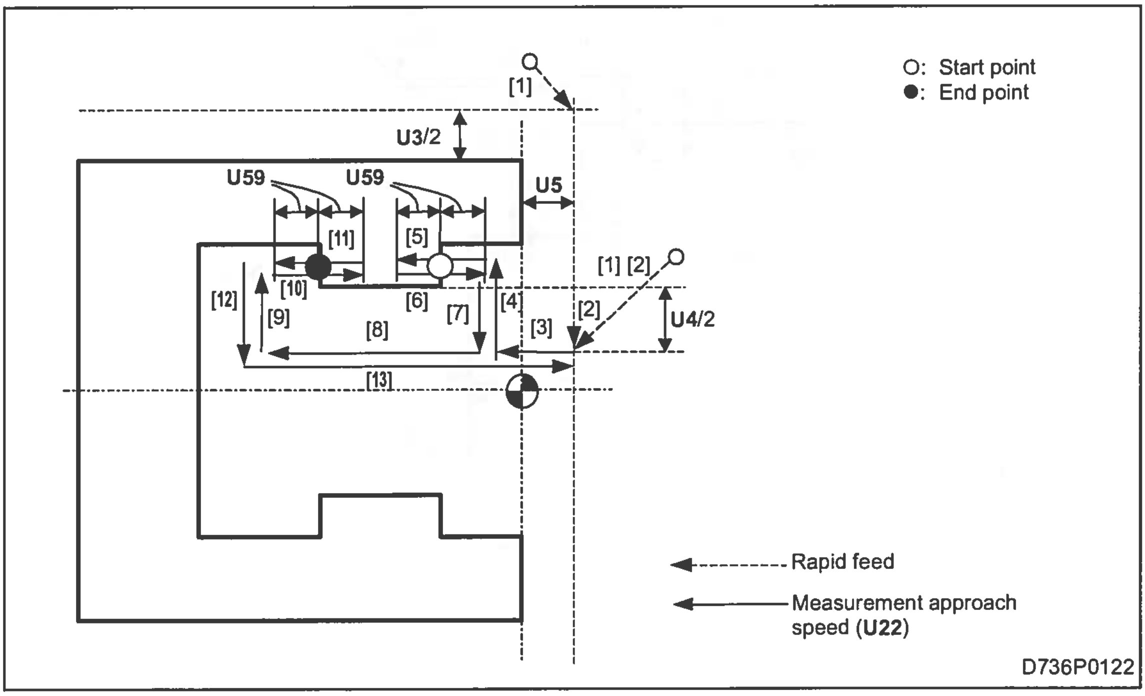


[Measurement movement (**X WIDTH**)]



Note: The safety profile clearance back (**U6**) is used for measurement at the No. 2 spindle side.

[Measurement movement (**IN WIDTH**)]

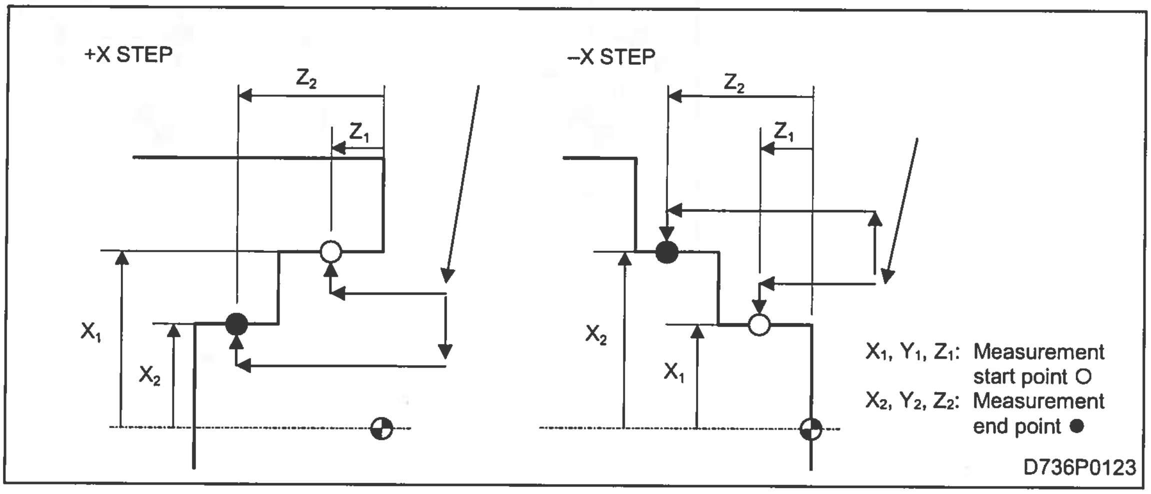


5. Step distance measurement

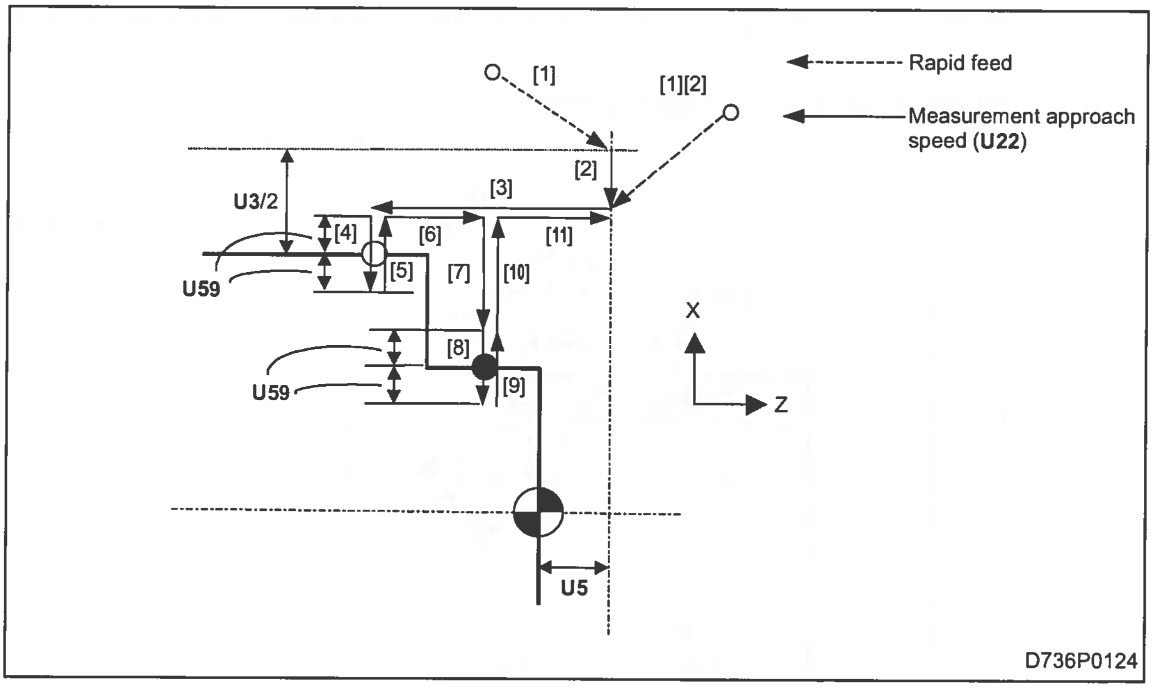
For **+X STEP**, **+Y STEP**, and **+Z STEP**, surface level differences in the plus directions of the X-axis, Y-axis, and Z-axis, respectively, are measured.

For **–X STEP**, **–Y STEP**, and **–Z STEP**, surface level differences in the minus directions of the X-axis, Y-axis, and Z-axis, respectively, are measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	+X STEP	X_1	Y_1	Z_1	X_2	Y_2	Z_2	T_1	T_2	S	Z



[Measurement movement (**–X STEP**)]



6. External measurement

For **EXT MILL**, data that has been measured using an external measuring unit is read and the measured data is incorporated into milling tool data.

For **EXT TURN**, measured data is incorporated into turning tool data.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	EXT MILL	P	N	Z_1	◆	◆	◆	T_1	T_2	◆	◆

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	EXT TURN	P	N	Z_1	◆	◆	◆	T_1	T_2	◆	◆

P: Select an offset item from the menu below.

WEAR X	WEAR Z	TOOL DIAMETER							

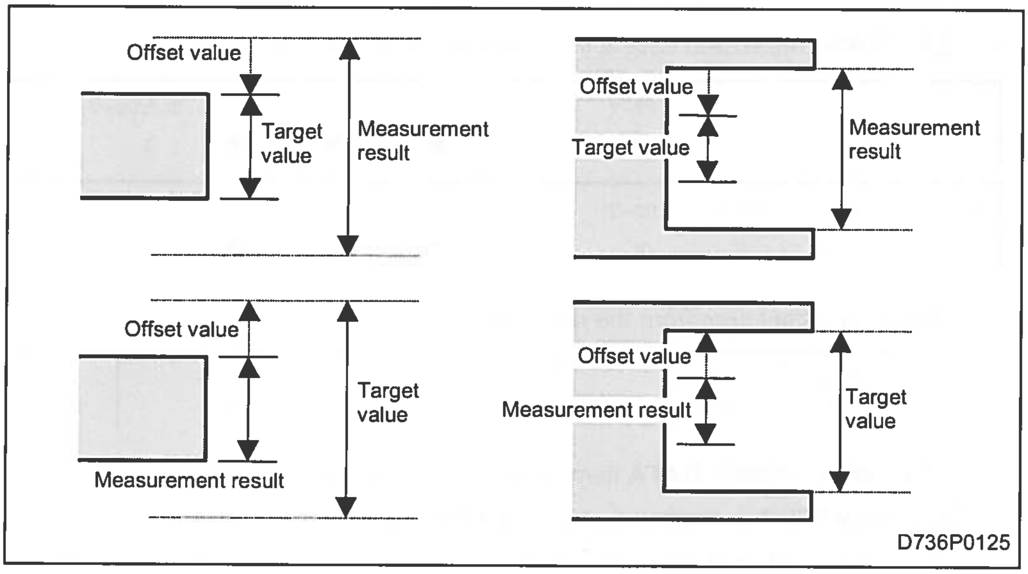
- For **EXT MILL, COMP. DATA** item on the unit line is invalid.
- **TOOL DIAMETER** is displayed only for **EXT MILL**.

N: Enter the number of the section to be measured using an external measuring unit.

Z_1: Enter the target value for the section which is to be measured using an external measuring unit.

3-26-5 Offset value and the direction of offset

1. Outside- and inside-diameter measurement



Target value X (Y) = Starting position X (Y)

Measurement result X
  = | (Measurement point #1 – Measurement point #2) |/2 – 2 × Touch sensor stylus ball radius

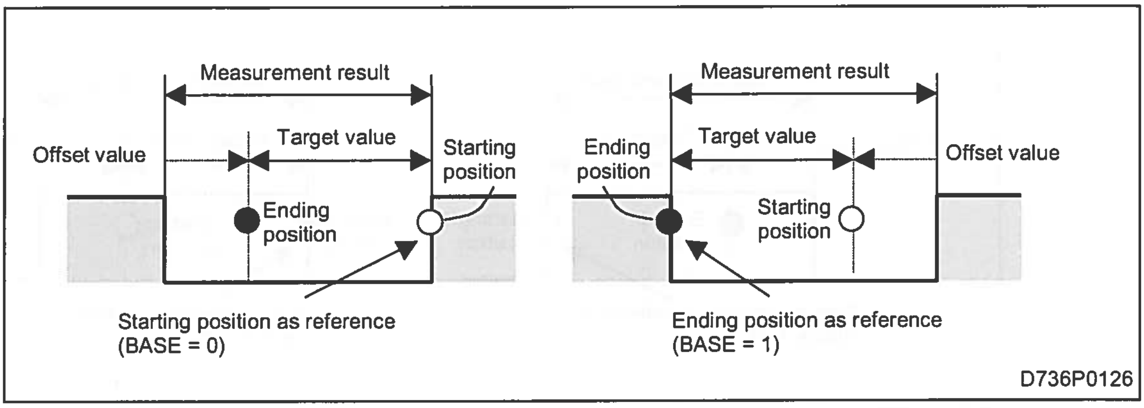
Measurement result Y
  = | (Measurement point #1 – Measurement point #2) | – 2 × Touch sensor stylus ball radius

Offset value X (Y) = Target value X (Y) – Measurement result X (Y)

[Offset for outside- and inside-diameter measurement]

	Measuring direction	Offset
Turning tool	X	Wear offset X = Wear offset X + Offset X
	Y	Wear offset X = Wear offset X + Offset Y

2. Groove width and inner groove width measurements



Target value X (YZ) = | Starting position X (YZ) – Ending position X (YZ) |

Measurement result X
= | Measurement point #1 – Measurement point #2 | + 4 × Touch sensor stylus ball radius

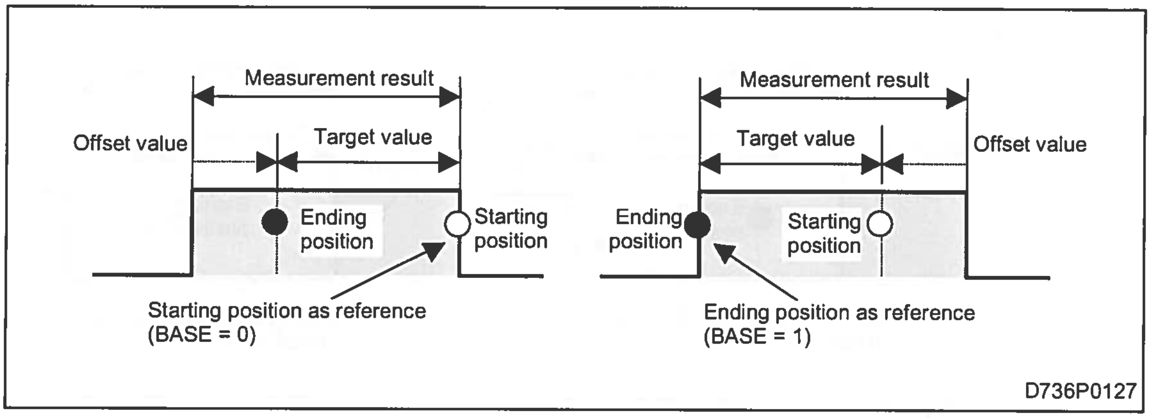
Measurement result Y (Z)
= | Measurement point #1 – Measurement point #2 | + 2 × Touch sensor stylus ball radius

Offset value X (YZ) = Target value X (YZ) – Measurement result X (YZ)

[Offset for groove width measurement]

	Measuring direction	Offset	Starting position as reference (BASE = 0)	Ending position as reference (BASE = 1)
Turning tool	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset X =	Wear offset X – (Offset Y × 2)	Wear offset X + (Offset Y × 2)
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z
Milling tool	X	Tool diameter =	Tool diameter + Offset X	
	Y	Tool diameter =	Tool diameter + (Offset Y × 2)	
	Z	Tool diameter =	Tool diameter + (Offset Z × 2)	
	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset Y =	Wear offset Y – Offset Y	Wear offset Y + Offset Y
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z

3. Protrusion width and inner protrusion width measurements



Target value X (YZ) = | Starting position X (YZ) – Ending position X (YZ) |

Measurement result X
= | Measurement point #1 – Measurement point #2 | – 4 × Touch sensor stylus ball radius

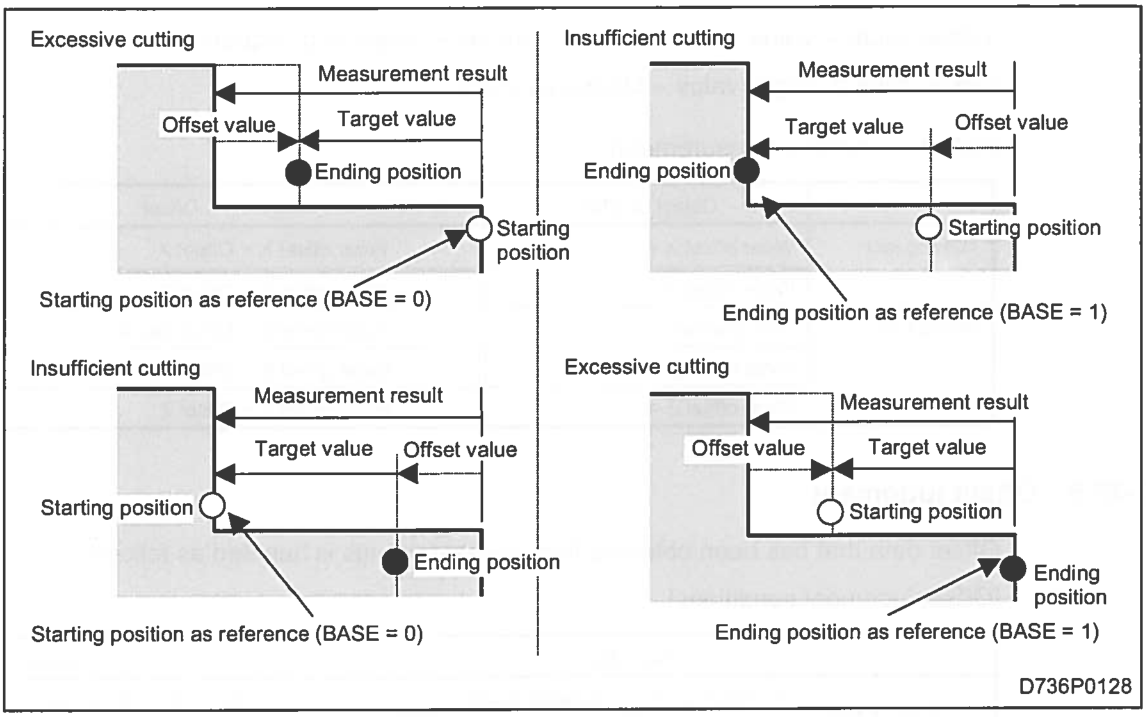
Measurement result Y (Z)
= | Measurement point #1 – Measurement point #2 | – 2 × Touch sensor stylus ball radius

Offset value X (YZ) = Target value X (YZ) – Measurement result X (YZ)

[Offset for protrusion width measurement]

	Measuring direction	Offset	Starting position as reference (BASE = 0)	Ending position as reference (BASE = 1)
Turning tool	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset X =	Wear offset X – (Offset Y × 2)	Wear offset X + (Offset Y × 2)
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z
Milling tool	X	Tool diameter =	Tool diameter – Offset X	
	Y	Tool diameter =	Tool diameter – (Offset Y × 2)	
	Z	Tool diameter =	Tool diameter – (Offset Z × 2)	
	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset Y =	Wear offset Y – Offset Y	Wear offset Y + Offset Y
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z

4. Step distance measurement



Target value X (YZ) = | Starting position X (YZ) – Ending position X (YZ) |

Measurement result X (YZ) = | Measurement point #1 – Measurement point #2 |

Offset value X (YZ) = Target value X (YZ) – Measurement result X (YZ)

[Offset for step distance measurement]

	Measuring direction	Offset	The reference position is in a plus direction with respect to the other position.	The reference position is in a minus direction with respect to the other position.
Turning tool	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset X =	Wear offset X – (Offset Y × 2)	Wear offset X + (Offset Y × 2)
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z
Milling tool	X	Tool diameter =	Tool diameter – Offset X	Tool diameter + Offset X
	Y	Tool diameter =	Tool diameter – (Offset Y × 2)	Tool diameter + (Offset Y × 2)
	Z	Tool diameter =	Tool diameter – (Offset Z × 2)	Tool diameter + (Offset Z × 2)
	X	Wear offset X =	Wear offset X – Offset X	Wear offset X + Offset X
	Y	Wear offset Y =	Wear offset Y – Offset Y	Wear offset Y + Offset Y
	Z	Wear offset Z =	Wear offset Z – Offset Z	Wear offset Z + Offset Z

5. External measurement

Target value = Value specified in the workpiece measuring sequence

Offset value = Target value – Measured value

[Offset for remote measurement]

	Object of offset	Offset
Turning tool	Wear offset X =	Wear offset X + Offset X
	Wear offset Z =	Wear offset Z + Offset Z
Milling tool	Tool diameter =	Tool diameter + Offset value / 2
	Wear offset X =	Wear offset X + Offset X
	Wear offset Z =	Wear offset Z + Offset Z

3-26-6 Offset judgment

Offset data that has been obtained from measurements is handled as follows:

[Offset judgment conditions]

Condition	Result
Offset value > $\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U58}$	An alarm is displayed.
$\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U58} \geq$ Offset value $\geq \frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U57}$	The offset is performed.
$\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U57} >$ Offset value $> -\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U57}$	The offset is not performed.
$-\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U57} \geq$ Offset value $\geq -\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U58}$	The offset is performed.
$-\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{U58} >$ Offset value	An alarm is displayed.

Parameter **U57**: Lower-limit value of the measurement tolerance

Parameter **U58**: Upper-limit value of the measurement tolerance

Tolerance upper/Tolerance lower: Settings in the program

Offset value: Calculated from the target value, the measurement result, and tolerance upper/tolerance lower.

3-27 Tool Measuring Unit (TOOL MES)

This unit measures tool data using TOOL EYE during automatic operation and then automatically corrects the tool wear offset value.

3-27-1 Procedure for selecting tool measuring unit

(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING MACH-ING	WORKPICE SHAPE	END	MILLING MANUAL P	MANUAL PROGRAM	PLANE AUTO SET	>>>

(2) Press the **[>>>]** menu key. The following menu will be displayed.

	M CODE	SUB PROGRAM	SEPARATE PROCESS	TRANSFER WORKPICE	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	SIMUL.	>>>

(3) Press the **[TOOL MEASURE]** menu key.

3-27-2 Setting the unit data

1. Setting TOOL MES unit data

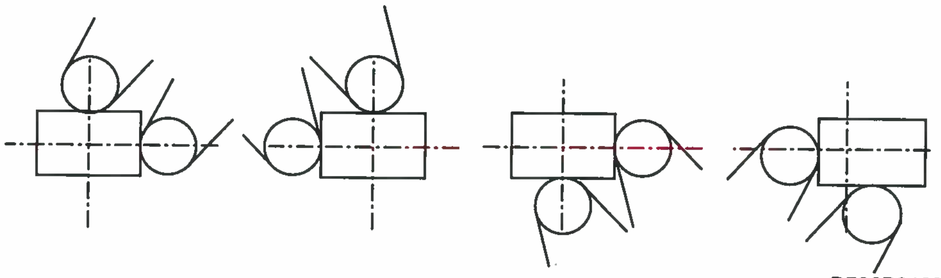
UNo.	UNIT	OFS	OFS-TOOL	OUTPUT
	TOOL MES			

Cursor position	Description
OFS	Select from the menu whether the measurement results are to be used to correct tool data. YES | NO
OFS-TOOL	Enter the name, nominal diameter (nominal size), identification code and turret number of the tool to be measured.
OUTPUT	Select whether the measurement results are to be sent to external equipment. 0: No output 1: Output to a text file on the HDD 2: Output to a serial printer via an RS-232C interface **Note:** Specify output items in parameter **P112**.

3-27-3 Setting the sequence data

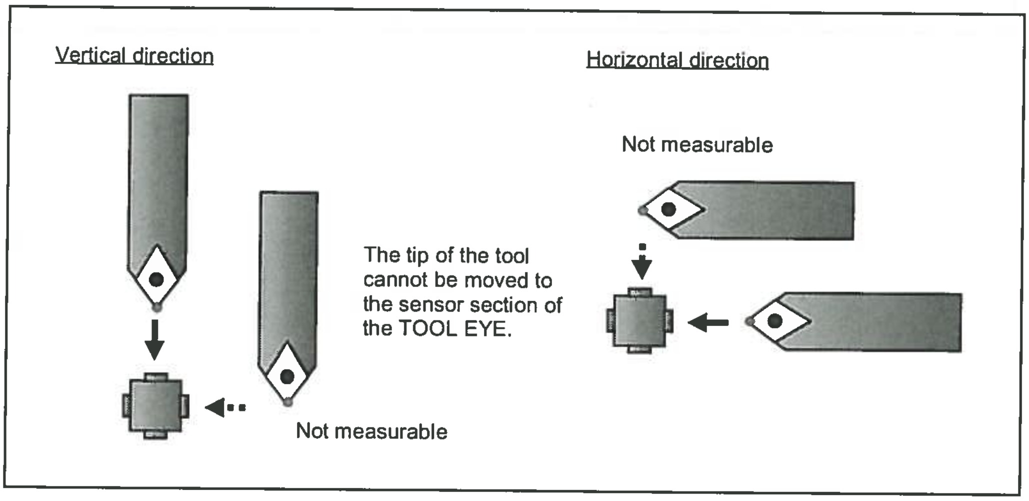
1. Setting TOOL MES sequence data

SNo.	PTN	T-LIM-X	T-LIM-Z	TOOLEYE
1				

Cursor position	Description
PTN	Select a tool measuring pattern from the menu. TOOL EYE #1 | TOOL EYE #2 | TOOL EYE #3 | TOOL EYE #4 Specify the following measuring directions for the TOOL EYE: #1. O. D., Forward #2. O. D., Reverse #3. I. D., Forward #4. I. D., Reverse  D736P0129
T-LIM-X	Enter the X-axial tolerance upper-limit value. If the X-axial tolerance is not entered, X-axial measurement will not occur.
T-LIM-Z	Enter the Z-axial tolerance upper-limit value. If the Z-axial tolerance is not entered, Z-axial measurement will not occur.
TOOLEYE	Enter 0 to retract the TOOL EYE after execution of the measuring unit, or enter 1 if it is not to be retracted. When the noses of multiple tools are to be measured in succession, the measuring time can be minimized by entering 1, since the TOOL EYE does not need to be extended or retracted each time. However, enter 0 for the last tool whose nose is to be measured using this unit. If 1 remains entered, the next machining unit will be executed with TOOL EYE extended.

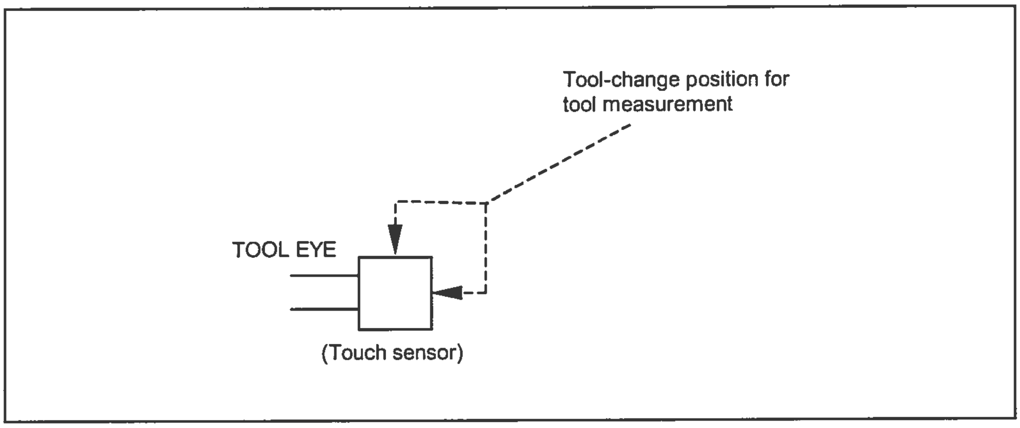
Note: The measurement may not be possible according to the particular type of tool.

Example: A sharply pointed cutting tool with a cutting angle less than 90 degrees



3-27-4 Measuring patterns

1. Tool path during TOOL EYE measurement



Note: The moment that the tool-change position for tool measurement is reached, the TOOL EYE begins to advance. This must be considered when setting the parameter for tool measurement tool-change position to prevent collision between TOOL EYE and tool.

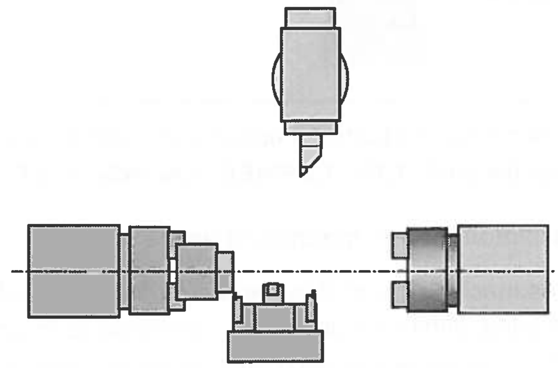
- NOTE -

4 LOWER-TURRET CONTROL FUNCTIONS

This chapter describes the programming methods that use the lower turret (turret 2) mounted under twin-turret machine specifications.

There are three types of machining which uses the lower turret: independent machining with the lower turret, simultaneous machining with the upper and lower turrets, and balanced cutting with the upper and lower turrets.

1. Independent machining with the lower turret



D737P0004

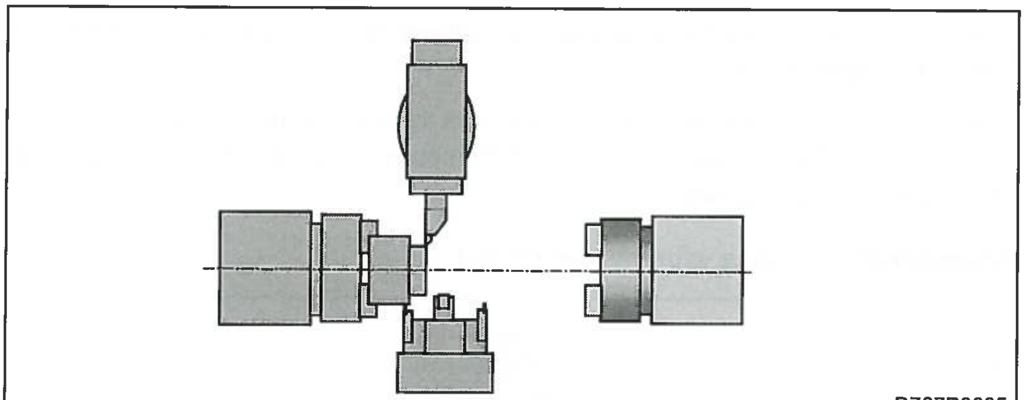
To execute machining that uses only the lower turret, select one of the tools mounted in the lower turret and then specify turret 2 (marked “◤”).

```
UNo.   MAT                  OD-MAX  ID-MIN   LENGTH  WORK FACE        RPM    TR2-DIA
 0   CST IRN                 100.     0.      40.      0.             2000
UNo.   UNIT      PART       CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR          OUT        100.      0.      0.2      0.2
SNo.     TOOL       NOM.   #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X   FIN-Z   C-SP    FR    M M
 R 1  GENERAL  OUT  45. A       0     5.       ◆          ◆       ◆       ◆      120   0.45
 F 2  GENERAL  OUT  30.   ◤     ◆     ◆        ◆          ◆      0.      0.      196   0.1
 FIG   SHP     S-CNR        SPT-X    SPT-Z    FPT-X    FPT-Z  F-CNR/$ RADIUS/th  RGH
  1   LIN                               ◆       80.      55.               ◆        4
UNo.   UNIT    COUNTER  RETURN  WK.No.  CONT.   NUM.    SHIFT
 2   END
```

◤: Identification mark for the lower turret (turret 2)

The above program is intended to perform the UNo. 1 – SNo. R1 process as roughing with the upper turret (turret 1) and then perform the UNo. 1 – SNo. F2 process as finishing with the lower turret (turret 2).

2. Simultaneous machining with the upper and lower turrets



D737P0005

In the simultaneous machining unit both the upper and lower turrets are used at the same time. This unit is valid only for the **BAR**, **CPY**, **CORNER**, **FACING**, and **T. GROOVE** unit.

A. Specification of the simultaneous machining unit

To set the simultaneous machining unit, it is necessary first to specify the turning spindle speed for simultaneous machining. While simultaneous machining is in progress, the turning spindle speed is kept constant and peripheral speed constant control does not work.
Since multiple sets of simultaneous machining may occur, it is also necessary to specify the simultaneous machining group number to identify those sets of simultaneous machining.

Specify the above two values in the simultaneous machining unit.

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	TR2-DIA
0	CST IRN	100.	0.	40.	0.	2000	

UNo.	UNIT	SIMUL.No.	RPM
1	SIMULTAN	50	2000

Machining identified by simultaneous machining group number 50 occurs at a speed of 2000 min^{-1}.

B. Specification in the turning units

Specify the turrets to be used for each turning unit, and the respective simultaneous machining group numbers.

```
UNo.  MAT              OD-MAX  ID-MIN  LENGTH  WORK FACE       RPM   TR2-DIA
 0   CST IRN            100.    0.      40.      0.            2000
UNo.  UNIT    SIMUL.No.     RPM
 1   SIMULTAN    50        2000
UNo.  UNIT     PART      CPT-X     CPT-Z    FIN-X    FIN-Z
 2   BAR       OUT       100.       0.      0.2      0.2
SNo.    TOOL        NOM.   #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X   FIN-Z   C-SP   FR   M M
R 1  GENERAL  OUT  45. A  50   0    5.       ◆          ◆      ◆       ◆      ◆    0.45
F 2  GENERAL  OUT  30. E       ◆    ◆        ◆          ◆      0.      0.     196  0.1
FIG  SHP    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th   RGH
 1   LIN               ◆        ◆       80.      55.                ◆        4
UNo.   UNIT      PART       PAT.   No.  PITCH          WIDTH  FINISH
 3   T.GROOVE    OUT         0      1    0              10.     ◆
SNo.    TOOL       NOM.   #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X   FIN-Z   C-SP   FR   M M
F 1  GROOVE  OUT  10.  ◤50   ◆     2.       ◆          ◆      ◆       ◆      ◆    0.1
FIG            S-CNR      SPT-X    SPT-Z    FPT-X    FPT-Z    CNR     ANG     RGH
 1        C 1.            100.      70.      90.      70.     C 1.
```

Specify the simultaneous machining group number and the turret.

The above program is intended to perform simultaneously the UNo. 2 – SNo. R1 process as bar materials O.D. roughing with the upper turret (turret 1) and the UNo. 3 – SNo. F1 process as grooving with the lower turret (turret 2) and then perform the UNo. 2 – SNo. F2 process as finishing with the upper turret.

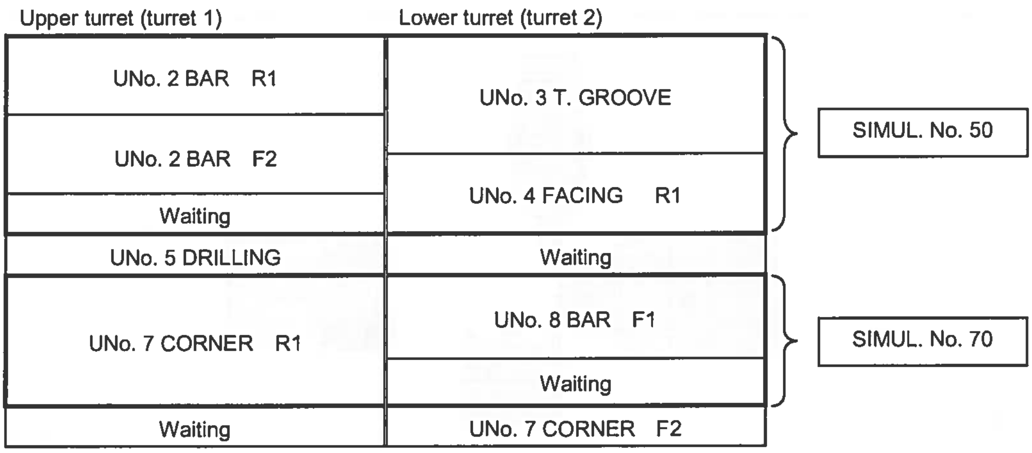
C. Example of multiple sets of simultaneous machining

Multiple sets of simultaneous machining can be performed using multiple simultaneous machining units.

A sample program is shown below.

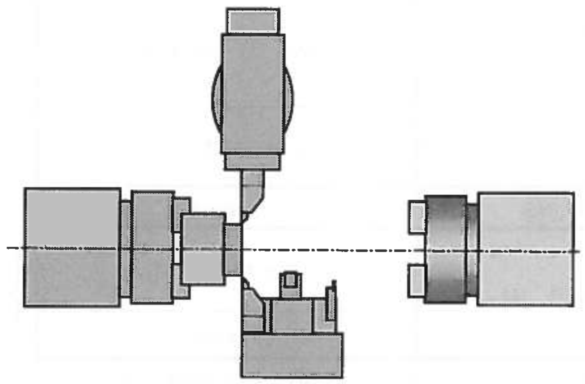
```
UNo.  MAT              OD-MAX  ID-MIN  LENGTH  WORK FACE      RPM   TR2-DIA
 0   CST IRN            100.     20.     100.     1.           2000
UNo.  UNIT    SIMUL.No.     RPM
 1   SIMULTAN     50        2000
UNo.  UNIT   PART    CPT-X    CPT-Z   FIN-X   FIN-Z
 2   BAR     OUT     100.      0.      0.2     0.2
SNo.   TOOL       NOM.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR   M M
R 1 GENERAL  OUT  45. A 50  0    5.      ◆        ◆    ◆     ◆     ◆   0.45
F 2 GENERAL  OUT  30. E 50  ◆    ◆       ◆        ◆    0.    0.   196  0.1
FIG  SHP   S-CNR   SPT-X   SPT-Z   FPT-X   FPT-Z  F-CNR/$ RADIUS/th  RGH
 1   LIN             ◆       ◆      80.     55.              ◆         4
UNo.  UNIT     PART     PAT.  No. PITCH    WIDTH   FINISH
 3   T.GROOVE  OUT       0     1   0.       10.      ◆
SNo.   TOOL      NOM.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR   M M
F 1 GROOVE  OUT  10. ◤ 50  ◆    2.      ◆        ◆    ◆     ◆     ◆   0.1
FIG        S-CNR   SPT-X   SPT-Z   FPT-X   FPT-Z    CNR    ANG     RGH
 1          C 1.    100.    70.     90.     70.     C 1.
UNo.   UNIT     PART               FIN-Z
 4   FACING     FACE                 0
SNo.   TOOL      NOM.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR   M M
R 1 GENERAL EDGE 10. ◤50   ◆   0.2      ◆        ◆    ◆     ◆     ◆   0.1
FIG                SPT-X   SPT-Z   FPT-X   FPT-Z                         RGH
 1                  80.     1.      20.     0.                            4
UNo.  UNIT    MODE  POS-B  POS-C  DIA   DEPTH  CHMF
 5   DRILLING  ZC     ◆      ◆     5.    20.    0.
SNo.  TOOL   NOM-ϕ HOLE-ϕ HOLE-DEP PRE-DIA PRE-DEP  RGH  DEPTH  C-SP  FR    M M
 1   DRILL    5.    5.      20.      0.     100    PCK1 T  2.5    25  0.089
FIG  SHP  SPT-R/x  SPT-C/y   SPT-Z    SPT-Y   NUM.  ANGLE  Q  R
 1   ARC    50.      0.       90.      0.      4     90.   0  1
UNo.   UNIT    SIMUL.No.     RPM
 6   SIMULTAN     70        1500
UNo.  UNIT    PART    CPT-X    CPT-Z
 7   CORNER   OUT      0.2      0.2
SNo.   TOOL      NOM.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR   M M
R 1 GENERAL  OUT  50. F  70  0    5.      ◆        ◆    ◆     ◆     ◆   0.45
F 2 GENERAL  OUT  50. ◤      ◆    ◆       ◆        ◆    0.    0.   196  0.2
FIG                SPT-X   SPT-Z   FPT-X   FPT-Z  F-CNR/$                RGH
 1                  70.     20.     90.     10.
UNo.  UNIT    PART    CPT-X    CPT-Z   FIN-X   FIN-Z
 8   BAR      IN       20.      0.      0.2     0.2
SNo.   TOOL      NOM.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR   M M
F 1 GENERAL  IN  15. A ◤70  ◆    ◆       ◆        ◆    0.    0.   196  0.1
FIG  SHP   S-CNR   SPT-X   SPT-Z   FPT-X   FPT-Z  F-CNR/$ RADIUS/th  RGH
 1   LIN             ◆       ◆      30.     10.              ◆         4
UNo.  UNIT   COUNTER  RETURN  WK.No. CONT.  NUM.  SHIFT
 9   END
```

For the above machining program, the timing chart is as follows:



The above machining sequence can be edited on the **PROCESS LAYOUT** display.

3. Balanced cutting with the upper and lower turrets



D737P0006

The same shape can be created using both the upper and lower turrets at the same time. This is referred to as balanced cutting and can be used in the **BAR**, **CPY**, and **CORNER** unit.

The loads on the tools can be reduced to half by causing the upper and lower turrets to act exactly the same. Thus, the feed rate can be increased by a factor of two.

A. Programming method

UNo.	UNIT		PART		CPT-X	CPT-Z	FIN-X	FIN-Z				
2	BAR		OUT		100.	45.	0.2	0.2				
SNo.	TOOL		NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M M
R 1	GENERAL	OUT	50. C	□	0	5.	◆	◆	◆	◆	120	0.45	
F 2	GENERAL	OUT	50. A		◆	◆	◆	◆	0.	0.	196	0.2	

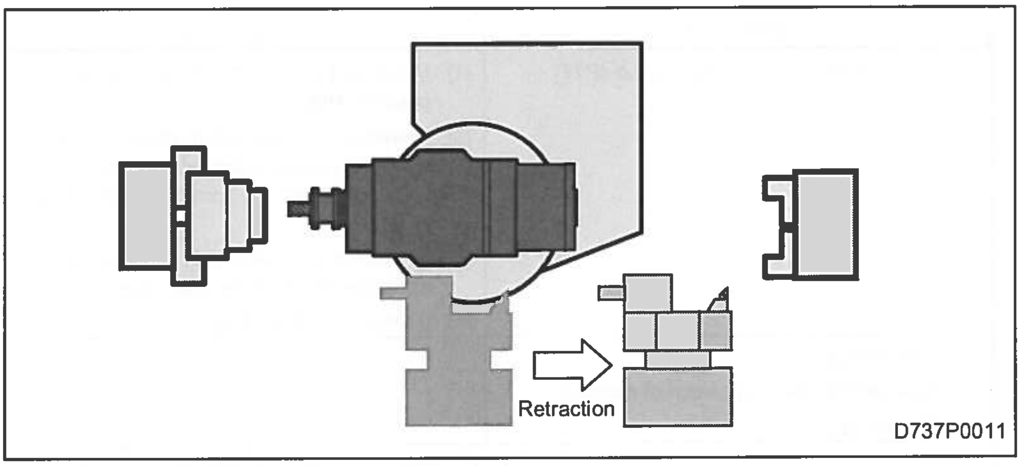
Move the cursor to the simultaneous machining number input column of the tool sequence data.

Pressing the **[BALANCE FEED 2]** menu key first and then the enter key adds a lower-turret tool sequence for balanced cutting.

UNo.	UNIT		PART		CPT-X	CPT-Z	FIN-X	FIN-Z					
2	BAR		OUT		100.	45.	0.2	0.2					
SNo.	TOOL		NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M M
R 1	GENERAL	OUT	50. C	B2	0	5.	◆	◆	◆	◆	120	0.45	
R 2	GENERAL			B2		◆	◆	◆	◆	◆	◆	◆	
F 3	GENERAL	OUT	50. A		◆	◆	◆	◆	0.	0.	196	0.2	

- **B2** means balanced cutting at twice a normal cutting feed rate.
- Since balanced cutting at twice a normal cutting feed rate has been specified, actual machining operates at twice the feed rate specified in the program. The feed rate in the above example is 0.45 × 2 = 0.9 mm/rev.
- Balanced cutting can be specified for the machining portion **(PART) OUT** and **OUT** of the roughing process of the **BAR**, **CPY**, and **CORNER** unit.

4. Retraction of the lower turret



When performing upper-turret machining operations near the rotational center of the turning spindle, interference between the upper and lower turrets can be avoided by retracting the lower turret.

A. Programming method

The use of the tool sequence menu allows the following two retraction positions to be selected for the lower turret:

TURRET 2 POS.1	TURRET 2 POS.2								

◤1: After the return of the turret to its X2/Z2 axial second home position, retraction tool 1 is indexed and then the turret is retracted to fixed position 1 in the X2/Z2 axial direction.

◤2: After the return of the turret to its X2/Z2 axial second home position, retraction tool 2 is indexed and then the turret is retracted to fixed position 2 in the X2/Z2 axial direction.

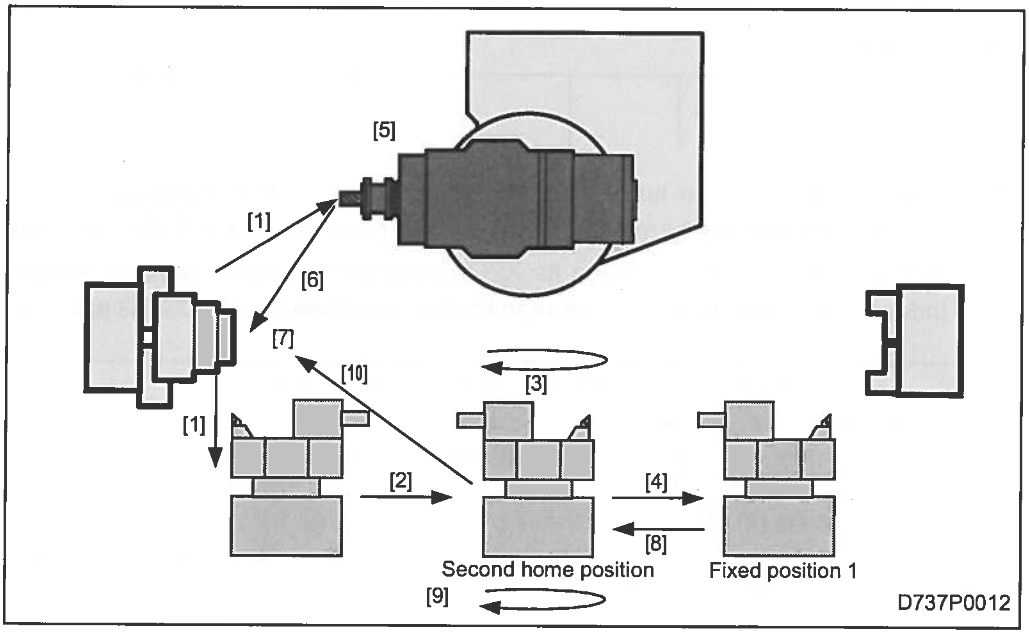
```
UNo.  UNIT      MODE    POS-B   POS-C    DIA    DEPTH    CHMF
 1   DRILLING    XY       ◆       45.     10.     30.      0.
SNo.  TOOL    NOM-ϕ  #  HOLE-ϕ HOLE-DEP PRE-DIA PRE-DEP  RGH    DEPTH   C-SP   FR    M M
 1   T-DRILL   10.  ◤1   10.     30.      0.      100   DRILTT   5.      36   0.294
FIG  SHP   SPT-R/x   SPT-C/y  SPT-Z   CX/PX   CY/PY  F  M  N ANGLE   P  Q  R
 1   LIN     3.       -50.     0.      25.      ◆     ◆ 4 ◆   0.      ◆  0  1
UNo.  UNIT       PART      PAT.  No. PITCH    WIDTH  FINISH
 2   T.GROOVE    OUT        0     1   0.       10.     ◆
SNo.    TOOL       NOM.   #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z  C-SP   FR    M M
F 1  GROOVE  OUT   10.   50   ◆     2.       ◆        ◆     ◆     ◆     ◆    0.1
FIG        S-CNR      SPT-X    SPT-Z    FPT-X    FPT-Z    CNR     ANG     RGH
 1         C 1.        130.     70.      120.     70.     C 1.
UNo.  UNIT   COUNTER  RETURN  WK.No.  CONT.  NUM.  SHIFT
 3    END
```

During drilling with the upper turret, the lower turret is retracted to fixed position 1.

Note: The retraction tool is always indexed at a lower-turret retraction tool changing position (second home position), not the normal lower-turret tool changing position designated with parameter **P68**.

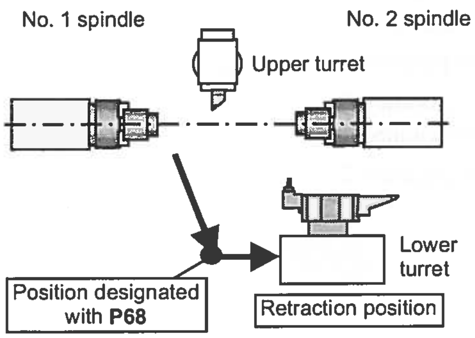
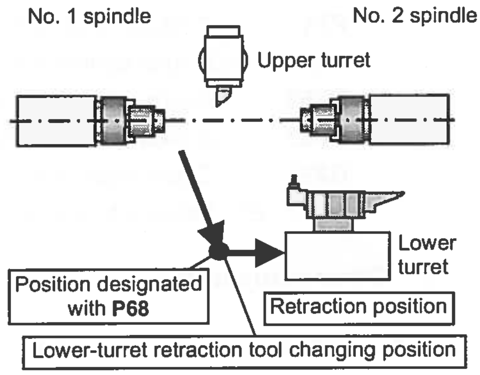
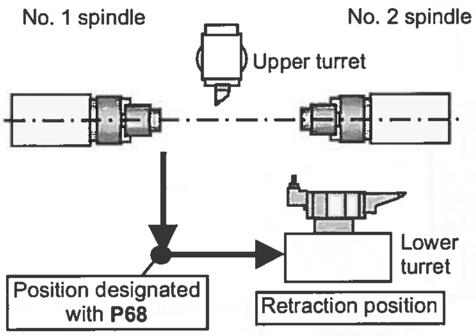
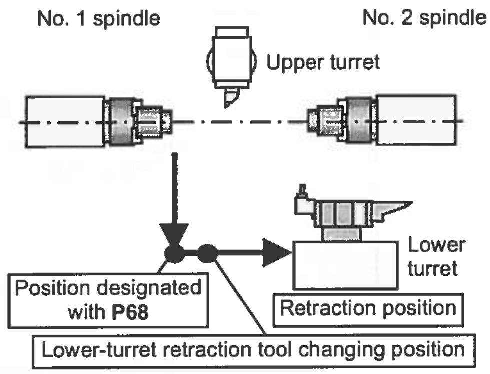
B. Description of operation

Upper turret	Lower turret
[1] Movement to tool-rotating position **(P17)**	[1] Movement to lower-turret tool changing position **(P68)** [2] Movement to lower-turret retraction tool changing position (second home position) in accordance with the command from the upper turret [3] Retraction tool indexing in accordance with the command from the upper turret [4] Retraction to fixed position 1
[5] Tool change [6] Approach to starting position of machining [7] Machining	
	⇩ Before machining with the lower turret is started following completion of the retraction:
	[8] Movement to lower-turret retraction tool changing position (second home position) [9] Tool indexing (Note) [10] Approach, followed by machining



Note: Even if the currently indexed tool and the tool to be used for next machining are the same (no tool change is conducted), the lower turret will move temporarily to the lower-turret retraction tool changing position (second home position) and the next unit will be executed.

<Examples of operation>

	Retraction of the lower turret from the state where it is not in a retraction position	
	If the retraction tool and the current tool are the same	If the retraction tool and the current tool are different tools
P68 = 3 (Note 1)	 The X- and Z-axes move to the **P68**-designated position and then move to the retraction position.	 The X- and Z-axes move to the **P68**-designated position, then move to the lower-turret retraction tool changing position (second home position)*, and after tool changing, move to the retraction position. *See Note 3.
P68 = 6 (Note 2)	 The X- and Z-axes move to the **P68**-designated position and then move to the retraction position.	 The X- and Z-axes move to the **P68**-designated position, then move to the lower-turret retraction tool changing position (second home position), and after tool changing, move to the retraction position.

Note 1: Lower-turret tool changing position
X-axis: Machine home position, Z-axis: Machine home position (Default setting)

Note 2: Lower-turret tool changing position
X-axis: Machine home position, Z-axis: Previous ending position of machining

Note 3: Since the **P68**-designated position is the second home position, the lower turret does not move in this case.

Note 4: No operation will occur if started with the lower turret present at its retraction position and a retraction tool remaining selected.

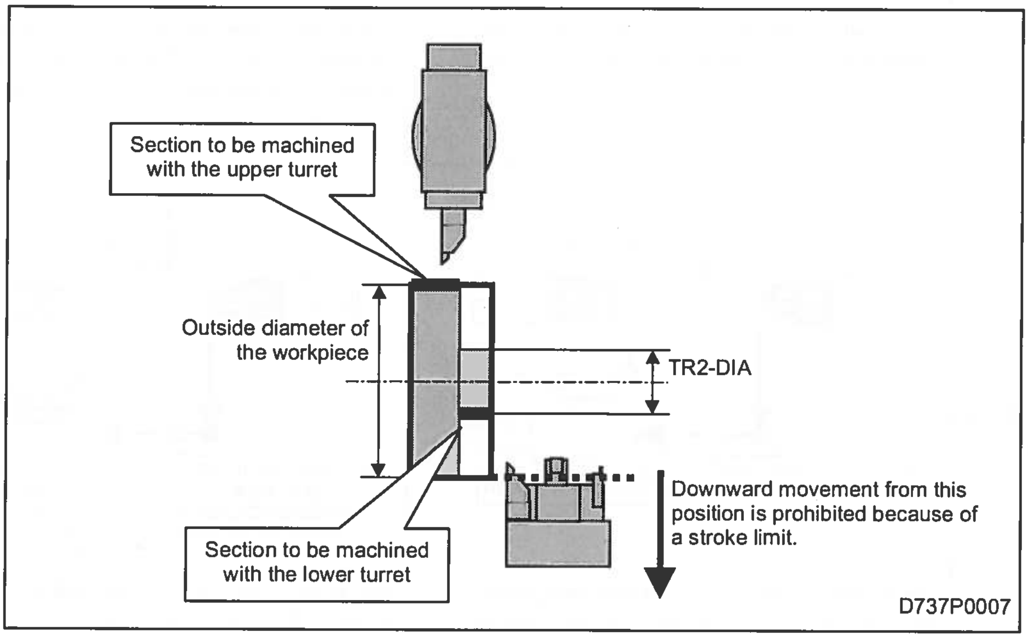
C. Description of parameters

Parameters relating to the lower-turret retraction function are listed below. See the Parameter List for further details.

P70: Tool number of retraction tool 1
P71: Tool number of retraction tool 2
B287: X-axis coordinate of fixed position 1
B288: Z-axis coordinate of fixed position 1
B289: X-axis coordinate of fixed position 2
B290: Z-axis coordinate of fixed position 2
A5 (X, Z): Second home position

5. Other setup items

A. TR2-DIA in common unit



As shown in the above view, when a large-size workpiece is machined, the lower turret may come into the outside-diameter section of the workpiece.

In this case, although the lower turret may attempt moving out of the outside-diameter section of the workpiece for safety reasons during the start of machining with the upper turret, operation comes to a software-limited stroke alarm stop because of a stroke limit.

In such a case, specify **TR2-DIA** in the common unit as the safe outside-diameter value for the lower turret. The lower turret judges the setting of **TR2-DIA** to be a safe position, and when the upper turret performs the machining operation, the lower turret moves to this position and does not suffer the stroke limit.

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	TR2-DIA
0	CST IRN	280.	0.	55.	0.	2000	130.

In other cases, no data needs to be specified in **TR2-DIA**.

B. Turret selection for spindle-synchronous machining

For spindle-synchronous machining, select a turret in the head selection unit. That is, select the turret for which the surface velocity (revolutions) of the turning spindle is to be programmed.

Select the same turret for subsequent machining units as well.

UNo.	UNIT	TYPE	HEAD	SPDL	TURRET
1	HEAD	SYNC	1	◆	◤

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
2	BAR	OUT	100.	45.	0.2	0.2

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	50.	◤	0	5.	◆	◆	◆	◆	120	0.45		
F 2	GENERAL OUT	70.	◤	◆	◆	◆	◆	0.	0.	196	0.2		

Note 1: For simultaneous machining, the above does not always apply since the turning spindle speed is to be programmed in the simultaneous machining unit.

Note 2: For balanced cutting, select the upper turret in the head selection unit.

- NOTE -

5 TPC DATA SETTING

5-1 Operating Procedure for Setting TPC (Tool-Path Control) Data

(1) First, place the cursor at the unit data line of the unit for which the TPC data is to be set.

Example: To set TPC data for the bar-materials machining unit (**BAR**) (UNo. 1):

```
UNo.   MAT              OD-MAX   ID-MIN   LENGTH   WORK FACE         RPM    TR2-DIA
 0   CST IRN             100.      0.      40.       0.              2000
UNo.  UNIT     PART      CPT-X      CPT-Z     FIN-X    FIN-Z
 1   BAR       OUT        100.        0.        0.       0.
SNo.   TOOL       NOM.  # PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X  FIN-Z  C-SP   FR    M M
R 1  GENERAL  OUT  45. A    0    5.       ◆        ◆     ◆      ◆      300  0.45
FIG  SHP   S-CNR      SPT-X     SPT-Z     FPT-X     FPT-Z   F-CNR/$ RADIUS/th  RGH
 1   LIN   C  3.        ◆         ◆        60.       50.               ◆        4
 2   TPR               80.       50.      100.        ?
UNo.   UNIT      PART      DIA
 2   T.DRILL     FACE      18.
SNo.   TOOL     NOM.  #  PAT.  DEP-1 DEP-2/NUM.  DEP-3  FIN-X    FIN-Z   C-SP   FR    M M
R 1  T-DRILL     18.  1   1     18.      3.        3.     ◆        ◆       24   0.25
FIG                            SPT-Z                FPT-Z
 1                              3.
```

Place the cursor at this line.

(2) Press the menu selector key, and then the **[TPC]** menu key.

- Pressing the **[TPC]** menu key indicates the **TPC** display for the unit specified at step (1). For the case of the example above, the following **TPC** display will be presented on the screen.

```
UNo.  UNIT   PART        CPT-X     CPT-Z     FIN-X     FIN-Z
 1     BAR    ***          ***       ***       ***       ***                  <- A

PARAMETER                  U3        U4        U5        U6
                           ■         ■         ■         ■

                           U33       U34       K1        K4        K5
                           ■         ■         ■         ■         ■

ROUGH               ROTATE POSITION     P17■    U1■     U2■    X  □   Z  □
CUTTING             APPROACH RELAY POINT    [AUTO]   ESCAPE RELAY POINT    [AUTO]
               □             X      Z                        X       Z
               □      1      □      □                  1     □       □
               □      2      □      □                  2     □       □
 OFS           □      3      □      □                  3     □       □

FINISH              ROTATE POSITION     P17■    U1■     U2■    X  □   Z  □
CUTTING             APPROACH RELAY POINT    [AUTO]   ESCAPE RELAY POINT    [AUTO]
               □             X      Z                        X       Z
               □      1      □      □                  1     □       □
               □      2      □      □                  2     □       □
 OFS           □      3      □      □                  3     □       □
```

TPC data

(3) Set the data in indicated item or change the data displayed in indicated item.

- On line (A) above, unit data that was set on the **PROGRAM** display will be displayed as it is. The data cannot be changed on the **TPC** display.
- Data that is preset in related parameters will be displayed at items marked with ■. Parameters denoting the distance are usually preset in 0.001 mm (0.0001 inch) steps, but on the **TPC** display, they will be displayed in 1 mm (1 inch) steps.

  Example: Data preset in parameter **U3**: 5000 (in 0.001 mm steps)
  ↓
  U3 data displayed on the **TPC** display: 5.000 (in 1 mm steps)

  Data being displayed at these items can be changed to any other data.
  If changes are made to the data, the corresponding unit will have its parameter settings overriden with the new data.
  The parameter settings will not change even if the displayed data is changed on the **TPC** display.
- Items marked with □ can be filled with data as required.
  To specify relay points for the tool approach or escape (return) path, first move the cursor to the item "**[AUTO]**" of the required data section, and then press the **[MANUAL]** menu key. The item marked with the cursor will change over to "**[MANU]**" and you can set required data for the relay points.
- See the next section for the contents of each TPC data. The same items are displayed except the items for line of **PARAMETER** in any machining unit.

Note 1: The following menu is displayed while the **TPC** display remains on the screen:

						TPC END	TPC CANCEL		

Pressing the **[TPC END]** menu key calls up the **PROGRAM** display anew.

Note 2: Setting or changing TPC data displays "+" mark on the left side of the corresponding unit number.
For units whose TPC data has been set or changed, six blocks of program memory (maximum) will be used.

If TPC data has not been set:

UNo.	MAT	OD-MAX	ID-MIN
0	CBN STL	***	***
UNo.	UNIT	PART	CPT-X
1	BRA	OUT	***
SNo.	TOOL	NOM.	#
F1	GENERAL OUT	***	*
FIG	SHP	S-CNR	SPT-X
1	LIN	***	***
UNo.	UNIT	PART	CPT-X
2	BAR	IN	***
SNo.	TOOL	NOM.	#
F1	GENERAL IN	***	*
FIG	SHP	S-CNR	SPT-X
1	TPR	***	***
UNo.	UNIT	COUNTER	
3	END	***	

+ mark

If TPC data has been set for unit No. 2:

UNo.	MAT	OD-MAX	ID-MIN
0	CBN STL	***	***
UNo.	UNIT	PART	CPT-X
1	BRA	OUT	***
SNo.	TOOL	NOM.	#
F1	GENERAL OUT	***	*
FIG	SHP	S-CNR	SPT-X
1	LIN	***	***
UNo.	UNIT	PART	CPT-X
+2	BAR	IN	***
SNo.	TOOL	NOM.	#
F1	GENERAL IN	***	*
FIG	SHP	S-CNR	SPT-X
1	TPR	***	***
UNo.	UNIT	COUNTER	
3	END	***	

Note 3: Carry out the following procedure to cancel the entire TPC data that has been set (or changed):

1) Press **[TPC CANCEL]** menu key.
2) Set "–9999".

The entire current TPC data is cancelled and initial TPC data is displayed on the **TPC** display. Also, the + mark on the **PROGRAM** display is deleted.
This procedure, of course, only cancels the TPC data for the respective unit.

Note 4: After TPC data has been set (or changed), the + mark will become ! mark if you update the unit data.
In that case, you must carry out the procedure above (described in Note 3) to temporarily initialize the TPC data. Subsequently, you can set (or change) the desired TPC data once again. An alarm will occur if you make an attempt to execute the program with the ! mark displayed.

5-2 Description of Each TPC Data Item of Turning Unit and Measurement Unit

1. TPC data for turning units

```
UNo.  UNIT   PART   #    CPT-X    CPT-Z    FIN-X    FIN-Z                    (a)
 1    BAR    OUT    1    100.     0.       0.2      0.1

PARAMETER      U3     U4     U5     U6
               5.     5.     2.     2.                                        (b)
               U33    U34    K1     K4     K5
               1.     2.     100    100    100

ROUGH      ROTATE POSITION  P17/P68  U1/U97  U2/U98  X            Z          (c)
           APPROACH RELAY POINT [AUTO]       ESCAPE RELAY POINT [AUTO]
                X      Z                          X      Z
           1                                 1
      (d)  2          (e)                    2          (f)
OFS        3                                 3

FINISH     ROTATE POSITION  P17/P68  U1/U97  U2/U98  X            Z          (c)
           APPROACH RELAY POINT [AUTO]       ESCAPE RELAY POINT [AUTO]
                X      Z                          X      Z
           1                                 1
      (d)  2          (e)                    2          (f)
OFS        3                                 3

BALANCE    ROTATE POSITION  P68      U97     U98     X            Z          (c)
CUT TR2    APPROACH RELAY POINT [AUTO]       ESCAPE RELAY POINT [AUTO]
                X      Z                          X      Z
           1                                 1
      (d)  2          (e)                    2          (f)
OFS        3                                 3
```

(a) Unit data for which the **TPC** display is called up. The data cannot be changed with the TPC data displayed on the screen.

(b) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display.

Refer to the separate Parameter List for details of parameter data.
Parameters denoting the distance are usually set in 0.001 mm (or 0.0001 in.) steps, but they will be displayed here in 1 mm (or 1 in.) steps.

Data shown as **(c)**, **(d)**, **(e)**, and **(f)** above, can be set for each process.

To perform balanced-cutting operations with a machine having a lower turret, enter upper-turret machining data in the **ROUGH** area, and lower-turret (**TR2**) machining data, in the **BALANCE CUT TR2** area.

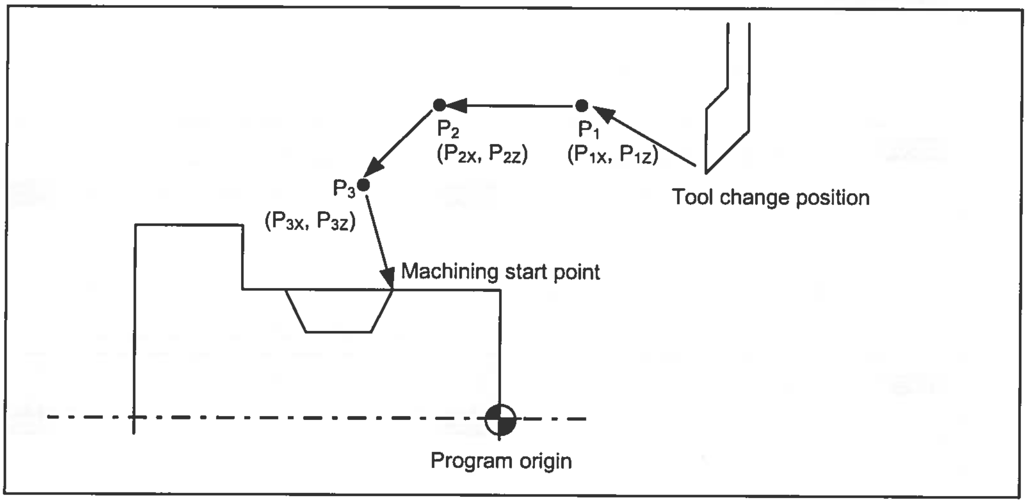
(c) Data related to the position of tool change (turret rotation) for each roughing or finishing unit.

- For items **X** and **Z**, specify the coordinates (in the machine coordinate system) of the required fixed point in mm (or in.).
- Refer to the Parameter List for details of **P17, P68, U1** (X), **U97** (X), **U2** (Z), and **U98** (Z).

(d) Data to be executed prior to the machining.

- For **OFS** item, enter an offset-No. of the data registered on the **TOOL OFFSET** display as required. For tool positioning, the data specified here will be taken into account in addition to **TOOL SET** and **WEAR COMP.** data on the **TOOL DATA** display.

(e) Use this section to modify the approach path so that interference does not occur.
To modify the path, first set the cursor at "**[AUTO]**" of the required section and then press the **[MANUAL]** menu key, to display "**[MANU]**".



For approaching path from the tool change position through relay points P_1, P_2 and P_3 to the machining start point, as shown above, set data as follows:

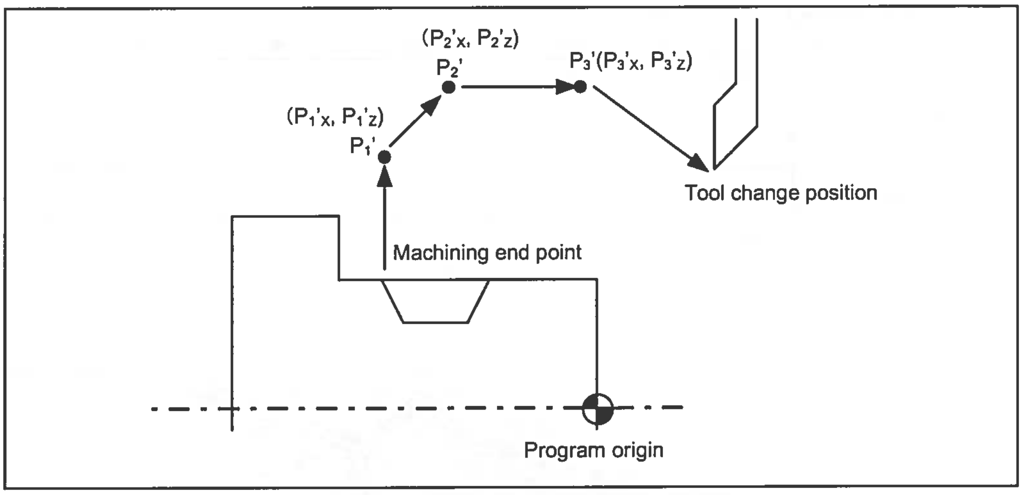
```
APPROACH RELAY POINT [MANU]
```

	X	Z	M	S	
1	P_{1X}	P_{1Z}			← Path through P_1
2	P_{2X}	P_{2Z}			← Path through P_2
3	P_{3X}	P_{3Z}			← Path through P_3

Specify the position using the program coordinate system.

- Enter X-coordinates in diameter values.
- For Z-axis positions on the right of the program origin, enter negative values except for the **FACING** unit for which a negative Z-coordinate denotes a position on the left of the program origin.

(f) Use this section to modify the escape path so that interference does not occur.
Refer to the description in (e) for details on data setting.



ESCAPE RELAY POINT [MANU]

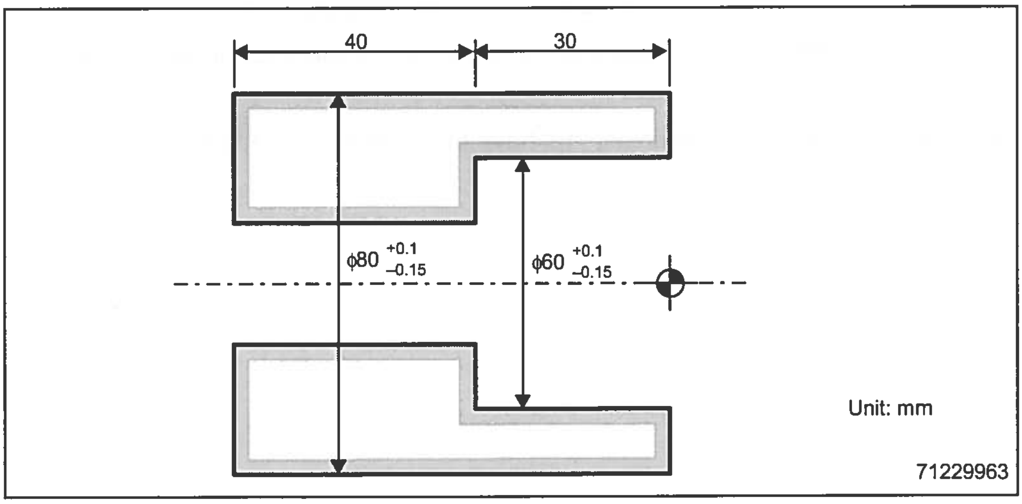
	X	Z	M	S	
1	P_1'x	P_1'z			← Path through P_1'
2	P_2'x	P_2'z			← Path through P_2'
3	P_3'x	P_3'z			← Path through P_3'

Note 1: In **TOOL MES** unit, the tool change position is determined by the parameter **A6**. Therefore, the tool change position (**ROTATE POSITION**) given on the **TPC** display will be ignored.

Note 2: The manually input data for relay points are not cancelled by changing "**[MANU]**" over to "**[AUTO]**". They will automatically be restored by changing "**[AUTO]**" back to "**[MANU]**" again. To change the data, therefore, first clear the displayed data with the data cancellation key and then input new data as required.
Machine operation is always carried out according to the data setting on the **TPC** display.

2. TPC data for measurement units

A. Program examples and tool paths for WORK MES



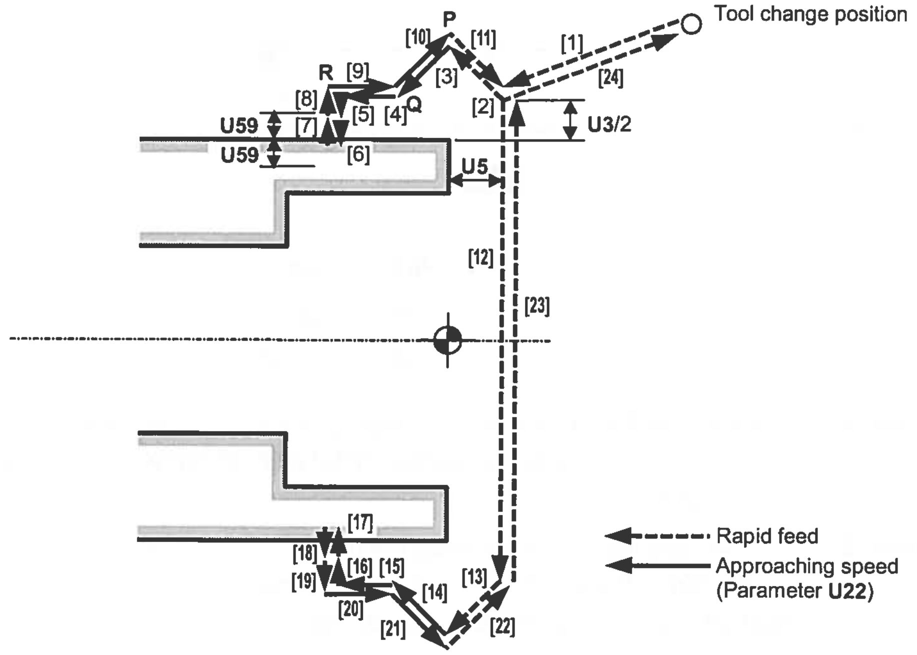
Example 1:

PROGRAM display

```
UNo.  UNIT      OFS  OFS-TOOL             COM.DATA  SNS-TOOL        OUTPUT
**    WORK MES  YES  GENERAL  OUT  20 A   ◆         TOL  SENS  OUT  1
```

TPC display

```
PARAMETER     U3      U4      U5      U6      P95
              2.      2.      2.      2.      6.
              U57     U58     U59
              25      50      1.

       ROTATE POSITION  P17    U1          U2     X      Z
       APPROACH RELAY POINT [AUTO]    ESCAPE RELAY POINT
  M              X       Z               X      Z
  M         1  120.      0.            110.    40.
  GR        2  110.     20.            110.    20.
  OFS       3  110.     40.            120.     0.
```



71229664

U3: Safety clearance from stock material (O. D.)

U5: Safety clearance from stock material (End face)

U59: Workpiece measurement stroke
When the tool moves through the workpiece measurement stroke that was set as parameter **U59**, the tool assumes the measuring speed that was set as parameter **U21**.

On the tool paths [6] and [17] the axis stops moving at the position where the touch sensor has turned on.

P, **Q**, **R**: Intermediate relay points that have been set on the **TPC** display

Example 2:

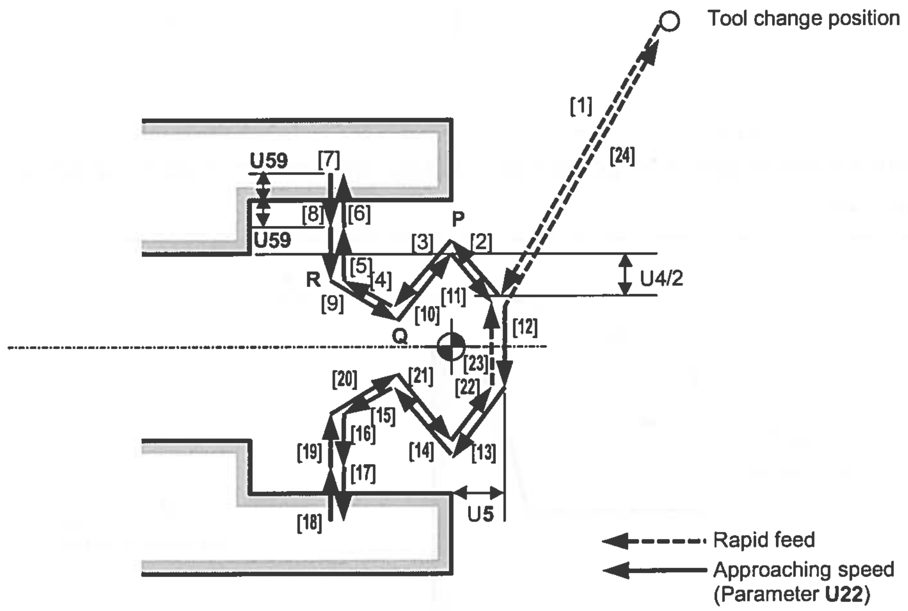
PROGRAM display

UNo.	UNIT	OFS	OFS-TOOL	COM.DATA	SNS-TOOL	OUTPUT
**	WORK MES	YES	GENERAL IN 20 A	◆	TOL SENS IN	1

TPC display

PARAMETER	U3	U4	U5	U6	P95
	2.	2.	2.	2.	6.
	U57	U58	U59		
	25	50	1.		

		ROTATE POSITION	P17	U1	U2	X	Z
		APPROACH RELAY POINT [AUTO]			ESCAPE RELAY POINT		
M		X	Z		X	Z	
M	1	70.	0.		60.	40.	
GR	2	50.	20.		50.	20.	
OFS	3	60.	40.		70.	0.	



71229291

U4: Safety clearance from stock material (I. D.)

U5: Safety clearance from stock material (End face)

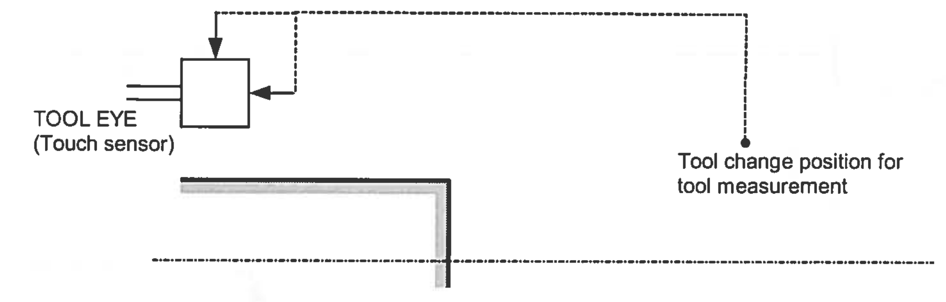
U59: Workpiece measurement stroke
When the tool moves through the workpiece measurement stroke that was set as parameter **U59**, the tool assumes the measuring speed that was set as parameter **U21**.

On the tool paths [6] and [17] the axis stops moving at the position where the touch sensor has turned on.

P, **Q**, **R**: Intermediate relay points that have been set on the **TPC** display

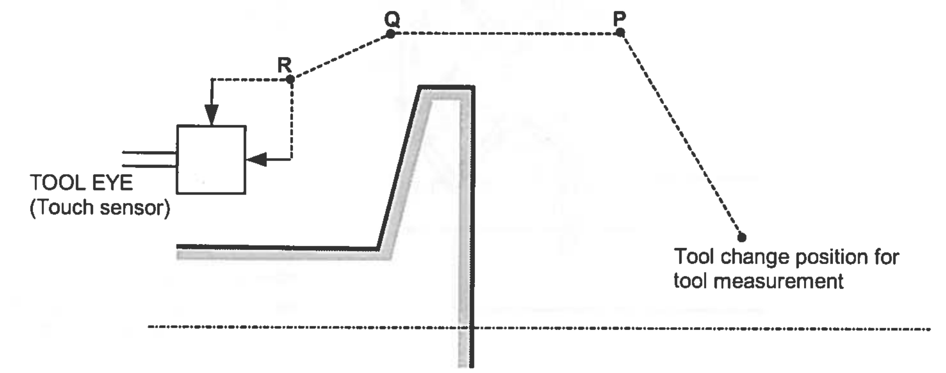
B. Tool paths for TOOL MES

When intermediate relay points do not exist (that is, when intermediate relay points are not set on the **TPC** display)



71229309-1

When intermediate relay points exist (that is, when intermediate relay points are set on the **TPC** display)



71229309-2

Note 1: During tool measurement, the TOOL EYE will move out to the required position when the tool arrives at the tool change position. Therefore, the tool change position in which the tool will not come into contact with the **T**OOL EYE when it has moved out, must be specified using parameter **A6**.

Note 2: Do not set the intermediate relay points in the **TPC** display for the single-side measurement of step distance, groove width, inside-diameter groove width, protrusion width, outside-diameter, and inside-diameter.

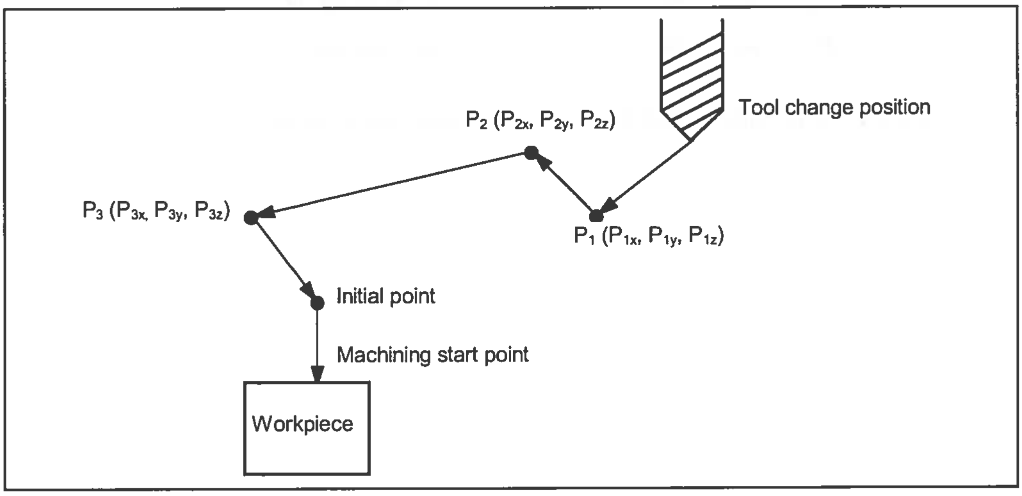
5-3 Description of Each TPC Data Item of Milling Unit

```
UNo.  UNIT     MODE  POS-B  POS-C  CB-DIA  CB-DEP  CHMF  BTM  DIA  DEPTH  } (a)
 4   RGH CBOR   ZC     ◆      ◆     20.     5.

PARAMETER   U3         U4         U5         U6        P17               
            2.         ◆          2.         2.        3.                
            D1         D3         D16        D17       D19       K43     
            0.         0          0          0.        0         10.      } (b)
            D42        D91        D92        D45       D46               
            0.         00000000   00000000   0.4       0.4               

RELAY POINT
APPROACH RELAY POINT [MANU]            ESCAPE RELAY POINT [MANU]
       X          Y          Z                X          Y          Z
 1                                      1
 2               (c)                    2               (d)
 3                                      3
```

(a) Unit data for which the **TPC** display is called up. The data cannot be changed with the TPC data displayed on the screen.

(b) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display.
Refer to the separate Parameter List for details of parameter data.
Parameters denoting the distance are usually set in 0.1 mm (or 0.01 in.) steps, but they will be displayed here in 1 mm (or 1 in.) steps.

(c) Use this section to modify the approach path so that interference does not occur.
To modify the path, first set the cursor at "**[AUTO]**" of the required section and then press the menu key **[MANUAL]**, to display "**[MANU]**". Finally enter the coordinates of the relay points on three lines (1, 2 and 3) in the desired order of relaying.



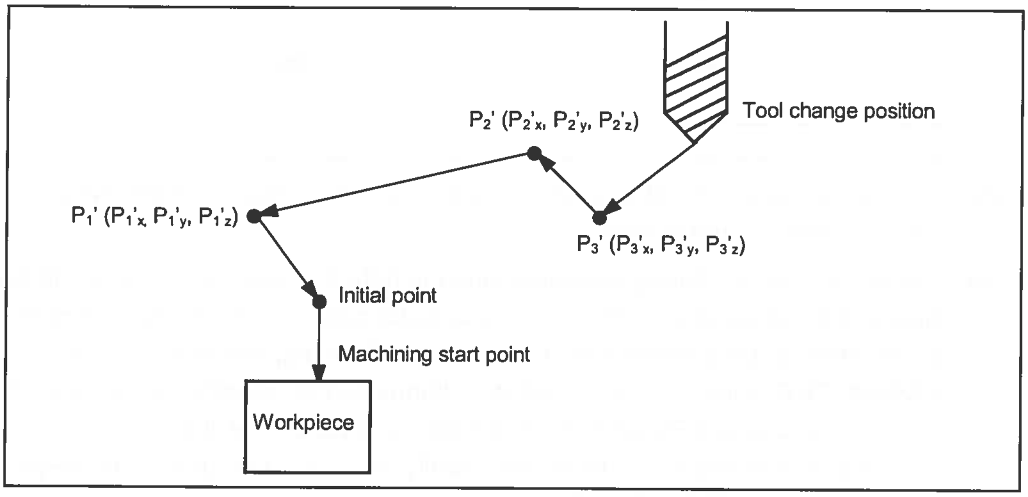
For approaching path from the tool change position through relay points P_1, P_2 and P_3 to the machining start point, as shown above, set data as follows:

APPROACH RELAY POINT [MANU]

	X	Y	Z	
1	P_{1X}	P_{1y}	P_{1Z}	.... Path through P_1
2	P_{2X}	P_{2y}	P_{2Z}	.... Path through P_2
3	P_{3X}	P_{3y}	P_{3Z}	.... Path through P_3

Specify the position using the program coordinates system.

(d) Use this section to modify the escape path so that interference does not occur. Refer to the description in **(c)** for details on data setting.



ESCAPE RELAY POINT [MANU]

	X	Y	Z	
1	$P_1'x$	$P_1'y$	$P_1'z$	.... Path through P_1'
2	$P_2'x$	$P_2'y$	$P_2'z$	.... Path through P_2'
3	$P_3'x$	$P_3'y$	$P_3'z$	.... Path through P_3'

Specify the position using the program coordinates system.

6 PROGRAM EDITING

This chapter describes operating procedures for editing programs already created. It also describes the various editing functions of the NC unit.

A. Operating procedures for editing a MAZATROL program

(1) Call up the **PROGRAM** display.

- Press the display selector key and then the **[PROGRAM]** menu key.

(2) Specify the work number of the program to be edited.

- After pressing the **[WORK No.]** menu key, specify the work number.
  The work-No. can also be specified in the work-Nos. listing window which will appear after pressing the above menu key. Using the cursor keys, place the cursor on the desired work-No. and press the input key.
- The selected program will be displayed on the screen.

(3) Press the **[PROGRAM]** menu key.

- Now, program data setting is possible; you can move the cursor to the desired position in the program using the cursor keys. Unless you press this menu key, you cannot change the program data being displayed; the cursor will only move vertically through the left end of each line even if you press the cursor keys.

(4) Edit the program.

(5) When necessary corrections or other edits to the program are completed, press the **[PROGRAM COMPLETE]** menu key.

B. Program editing functions

Pressing the menu changeover key with the **PROGRAM** display remaining on the screen displays the following menu:

PROGRAM COMPLETE	FIND	COPY	INSERT	ERASE		TPC	CALCULAT	SHAPE CHECK	HELP
	1	2	3	4					

You can use functions 1 to 4 above to carry out the editing operations listed below.

No.	Menu item	Function
1	FIND	To search for the intended unit in the program
2	COPY	To copy the data in unit of program, unit or machining shape (sequence)
3	INSERT	To insert one blank line for unit or sequence into any position on the **PROGRAM** display
4	ERASE	To delete a specific unit or sequence existing in the program currently being displayed on the **PROGRAM** display

6-1 Unit Search

(1) Press the **[FIND]** menu key.

(2) Set the number of the unit to be searched for.

- A blinking cursor will appear at the first character position of the selected unit data.

Example: If unit number 10 is set:
A blinking cursor will appear at the position indicated below.

```
UNo.   MAT              OD-MAX  ID-MIN  LENGTH  WORK FACE       RPM
 0  CST IRN               50.      0.     100.     0            2000
 :
 :
UNo.   UNIT      PART     PAT.   No.  PITCH       WIDTH  FINISH
 10  T.GROOVE    OUT       0      1    0.          10.     ◆
SNo.  TOOL       NOM.   #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR   M M
 F 1  GROOV  OUT 45.  A    ◆      2.       ◆          ◆      ◆      ◆    105  0.1
 FIG         S-CNR    SPT-X   SPT-Z   FPT-X   FPT-Z   CNR    ANG     RGH
 1                     50.     45.     40.     45.
 :
 :
UNo.   UNIT      COUNTER   WK.No.  CONT.   NUM.    SHIFT
 15  END           0                 0       0
```

Cursor

- An alarm will be displayed if the selected unit does not exist.

Note: The last unit in the program being displayed will be searched for if you only press the input key without setting a unit number during step (2) above.

6-2 Copy

(1) Using the cursor keys, select the position where the copied data are to be placed.

- For program or unit copy, place the cursor on the line of the unit data which shall follow the copied data.
- For machining-shape copy, place the cursor at the first line of the shape sequence data.

Example 1: To copy program or unit data into the position before unit No. 2:

Work No.100

```
UNo.   MAT               OD-MAX  ID-MIN  LENGTH  WORK FACE      RPM
 0   CBN STL               80.     40.     90.      0.          2000
UNo.  UNIT     PART     CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR       OUT       80.      0.       0.       0.
SNo.    TOOL      NOM.  # PAT.  DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
 R 1 GENERAL  OUT  60. B    0     5.        ◆        ◆      ◆      ◆    120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z  F-CNR/$  RADIUS/th   RGH
 1   LIN  C  0.1      ◆        ◆       60.      45.                ◆
UNo.  UNIT      PART     PAT.   No.  PITCH       WIDTH   FINISH
 2   T.GROOVE    OUT       0      1    0.           5.       ◆
SNo.   TOOL     NOM.   #  PAT.  DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
 F 1 GROOVE OUT  30. E     ◆      2.        ◆        ◆      ◆      ◆    105   0.1
FIG        S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z    CNR     ANG      RGH
 1                   60       45.      56.
```

Place the cursor on this line.

Example 2: To copy machining-shape data into UNo. 4:

```
UNo.  UNIT     PART     CPT-X    CPT-Z    FIN-X    FIN-Z
 4   BAR       OUT       80.      0.       0.       0.
SNo.    TOOL      NOM.  #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
 R 1 GENERAL  OUT  60. B     0     5.       ◆        ◆      ◆      ◆    120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z  F-CNR/$  RADIUS/th   RGH
 1
```

Cursor

(2) Press the **[COPY]** menu key.

6-2-1 Program copy

To copy the entirety (except for UNo. 0 and **END** unit) of a registered program other than that currently being displayed.

(3) Press the **[PROGRAM COPY]** menu key.

(4) Set the work number of the program to be copied.

Example: If work number 101 is set:

Work No. 101

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	CBN STL	80.	40.	90.	0.	1500
UNo.	UNIT	#1	#2	#3	#4	
1	M-CODE	8				
....						
....						
UNo.	UNIT	PART	CPT-X	CPT-Z		
8	BAR	IN	40.	0.		
....						
....						
UNo.	UNIT	#1	#2	#3	#4	
13	M-CODE	9				
UNo.	UNIT	COUNTER	RETURN	WK.No.	CONT.	NUM.
14	END	1	0		0	0

Data to be copied

The program of work number 100 will become:

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	
0	CBN STL	80.	40.	90.	0.	2000	
UNo.	UNIT	PART	CPT-X	CPT-Z			
1	BAR	OUT	80.	0.			
SEQ	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	
1	LIN	C 1.	◆	◆	60.	45.	
UNo.	UNIT	#1	#2	#3	#4		
2	M-CODE	8					
....							
....							
UNo.	UNIT	PART	CPT-X	CPT-Z			
9	BAR	IN	40.	0.			
....							
....							
UNo.	UNIT	#1	#2	#3	#4		
14	M-CODE	9					
UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FINISH
15	T.GROOVE	OUT	0	1	0	5.	0.

Copied data

6-2-2 Unit copy

To copy only a specific unit existing in the program being displayed or in any other program pre-registered in the NC unit

(3) Press the **[UNIT COPY]** menu key.

(4) Set the work number of the program that includes the unit to be copied.

- Simply press the input key for the program being displayed.

(5) Set the number of the unit to be copied.

- The whole data (including sequence data) of the selected unit will be copied in the same position as for program copy.

Note: The common unit cannot be copied.

6-2-3 Machining-shape copy

To copy only a specific set of machining-shape data existing in the program being displayed

(3) Press the **[SEQUENCE COPY]** menu key.

(4) Set the number of the unit that includes the machining-shape data to be copied.

Example: If unit number 3 is set:

```
UNo.  UNIT    PART    CPT-X    CPT-Z    FIN-X    FIN-Z
 3    BAR     OUT      80.      0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
R 1  GENERAL  OUT  60. B     0     5.        ◆         ◆      ◆      ◆     120   0.45
FIG   SHP    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$  RADIUS/th   RGH
 1   LIN   C  1.        ◆        ◆       50.      20.                 ◆
 2   TPR   C  1.       60.      20.      70.      40.
 3   凸                70.      40.      80.      80.
UNo.  UNIT    PART    CPT-X   CPT-Z    SRV-X    SRV-Z    FIN-X    FIN-Z
 4    CPY     OUT      80.      5.       5.       0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
R 1  GENERAL  OUT  60. A     ◆     5.        ◆         ◆      ◆      ◆     120   0.45
FIG   SHP    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$  RADIUS/th   RGH
 1
```

Cursor

```
UNo.  UNIT    PART    CPT-X    CPT-Z    FIN-X    FIN-Z
 3    BAR     OUT      80.      0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
R 1  GENERAL  OUT  60. B     0     5.        ◆         ◆      ◆      ◆     120   0.45
FIG   SHP    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$  RADIUS/th   RGH
 1   LIN   C  1.        ◆        ◆       50.      20.                 ◆
 2   TPR   C  1.       60.      20.      70.      40.
 3   凸                70.      40.      80.      80.
UNo.  UNIT    PART    CPT-X   CPT-Z    SRV-X    SRV-Z    FIN-X    FIN-Z
 4    CPY     OUT      80.      5.       5.       0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR    M M
R 1  GENERAL  OUT  60. A     ◆     5.        ◆         ◆      ◆      ◆     120   0.45
FIG   SHP    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$  RADIUS/th   RGH
 1   LIN   C  1.        ◆        ◆       50.      20.                 ◆
 2   TPR   C  1.       60.      20.      70.      40.
 3   凸                70.      40.      80.      80.
```

Machining-shape copy is available only for a unit of the same type.

(Example)
BAR ↔ CPY
T. DRILL ↔ T. TAP

For programming a series of units, for example, for tapping:

UNo. 1 **T. DRILL** (Spot-drilling)
UNo. 2 **T. DRILL** (Pilot-drilling)
UNo. 3 **T. TAP** (Tapping)

Use this function to copy the shape data of UNo. 1 for setting the sequence data of UNo. 2 and UNo. 3.

6-3 Line Insertion

(1) Using the cursor keys, select the position where you want to insert a blank line.

- Place the cursor on the line of the data that shall follow the inserted line.

Example 1: To insert a new unit into the position before unit number 2:

```
UNo.  UNIT     PART     CPT-X     CPT-Z     FIN-X     FIN-Z
 1  BAR         OUT       80.        0.        0.        0.
SNo.   TOOL     NOM.     #   PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP    FR    M M
R 1  GENERAL  OUT 60. B       0     5.       ◆        ◆     ◆     ◆    120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th   RGH
 1  LIN  C   1.       ◆        ◆        60.      45.                 ◆
UNo.   UNIT      PART      PAT.   No. PITCH      WIDTH   FINISH
 2  T.GROOVE     OUT          0      1    0.          5.       ◆
SNo.   TOOL     NOM.     #   PAT.  DEP-1 DEP-2/NU    DEP-3 FIN-X FIN-Z C-SP    FR    M M
F 1  GROOVE   OUT 30. E       ◆      2.      ◆        ◆     ◆     ◆    105   0.2
FIG          S-CNR    SPT-X    SPT-Z    FPT-X    F        CNR      ANG      RGH
 1                    60       45.      56.
```

Place the cursor on this line.

Example 2: To insert a new sequence into the position before the first sequence of UNo. 1:

```
UNo.  UNIT     PART     CPT-X     CPT-Z     FIN-X     FIN-Z
 1  BAR         OUT       80.        0.        0.        0.
SNo.   TOOL     NOM.     #   PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP    FR    M M
R 1  GENERAL  OUT 60. B       0     5.       ◆        ◆     ◆     ◆    120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th   RGH
 1  LIN  C   1.       ◆        ◆        60.      45.                 ◆
UNo.    UNIT      PART      PAT.   No. PITCH     WIDTH   FINISH
 2  T.GROOVE      OUT         0      1    0.         5.       ◆
SNo.   TOOL     NOM.     #   PAT.  DEP-1 DEP-2/NUM. DEP-  IN-X FIN-Z C-SP    FR    M M
F 1  GROOVE   OUT 30. E       ◆      2.      ◆        ◆     ◆     ◆    105   0.2
FIG          S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z             ANG      RGH
 1                    60       45.      56.      45.
```

Place the cursor on this line.

(2) Press the **[INSERT]** menu key.

(3) Press the input key.

- One blank line will be inserted.

In case of example 1:

```
UNo.  UNIT    PART    CPT-X    CPT-Z    FIN-X    FIN-Z
 1  BAR      OUT      80.       0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP   FR    M M
 R 1  GENERAL  OUT 60. B    0     5.      ◆        ◆     ◆     ◆   120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th  RGH
 1  LIN  C   1.      ◆        ◆       60.      45.              ◆
 2
UNo.    IT       PART      PAT.   No.  PITCH     WIDTH   FINISH
 2  T.G   VE      OUT        0     1    0.        5.      ◆
SNo.  TO        NOM.    #  PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z C-SP   FR    M M
 F 1  GROOV        T 30. E    ◆     2.       ◆          ◆     ◆     ◆   105   0.2
FIG                  SPT-X    SPT-Z    FPT-X    FPT-Z     CNR      ANG      RGH
 1                    60       45.      56.      45.
```

One blank line (sequence data line) is inserted here.

Press the **[SHAPE END]** menu key, and the inserted line will be changed into a unit data line.

```
UNo.  UNIT    PART    CPT-X    CPT-Z    FIN-X    FIN-Z
 1  BAR      OUT      80.       0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP   FR    M M
 R 1  GENERAL  OUT 60. B    0     5.      ◆        ◆     ◆     ◆   120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th  RGH
 1  LIN  C   1.      ◆        ◆       60.      45.              ◆
UNo.  UNIT
 2
UNo.  UN          PART      PAT.   No.  PITCH     WIDTH   FINISH
 3  T.GR    E      OUT        0     1    0.        5.      ◆
SNo.  TO        NOM.    #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP   FR    M M
 F 1  GROOV       T 30. E    ◆     2.      ◆        ◆     ◆     ◆   105   0.2
FIG                  SPT-X    SPT-Z    FPT-X    FPT-Z     CNR      ANG      RGH
 1                    60       45.      56.      45.
```

One blank line (unit data line) is inserted here.

In case of example 2:

```
UNo.  UNIT    PART    CPT-X    CPT-Z    FIN-X    FIN-Z
 1  BAR      OUT      80.       0.       0.       0.
SNo.  TOOL     NOM.    #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP   FR    M M
 R 1  GENERAL  OUT 60. B    0     5.      ◆        ◆     ◆     ◆   120   0.45
FIG  SHP   S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$ RADIUS/th  RGH
 1
 2  LI   C   1.      ◆        ◆       60.      45.              ◆
UNo.  U           PART      PAT.   No.  PITCH         WIDTH FINISH
 2  T.GR          OUT        0     1    0.            5.     ◆
SNo.  TO        NOM.    #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z  C-SP   FR    M M
 F 1  GROOV        T 30. E    ◆     2.      ◆        ◆     ◆     ◆    105   0.2
FIG                  SPT-X    SPT-Z    FPT-X    FPT-Z     CNR      ANG      RGH
 1                    60       45.      56.      45.
```

One blank line (sequence data line) is inserted here.

6-4 Line Deletion

(1) Place the cursor on the line to be deleted.

- To delete the entire data of a unit (sequence data included), place the cursor on the line of the unit data.
- To delete the data of a specific sequence, place the cursor on the data line of the sequence.

```
UNo.  UNIT     PART                    FIN-Z
 1  FACING   FACE                       0.
SNo.   TOOL       NOM.    # PAT. DEP-1 DEP-2/NUM.  DEP-3  FIN-X FIN-Z C-SP   FR    M M
R 1 GENERAL  EDGE 45. A     ◆    5.       ◆          ◆      ◆     ◆    104   0.4
FIG                       SPT-X    SPT-Z    FPT-X    FPT-Z                    RGH
 1                         100.     1.       0.       0.
UNo.  UNIT     PART     CPT-X    CPT-Z    FIN-X    FIN-Z
 2  BAR     OUT          100.      0.       0.       0.          To delete the unit
SNo.   TOOL       NOM.   #  PAT. DEP-1 DEP-2/NUM.  DEP-3  FIN-X FIN-Z C-SP   FR    M M
R 1 GENERAL  OUT  45. A     0    5.       ◆          ◆      ◆     ◆    120  0.45
FIG  SHP    S-CNR     SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$  RADIUS/th   RGH
 1  LIN  C  3.          ◆        ◆       60.      50.
 2  TPR                 80.      50.     100.      ?             To delete the sequence
UNo.   UNIT     PART    DIA
 3  T.DRILL   FACE     18.
```

(2) Press the **[ERASE]** menu key.

(3) Press the input key.

- In case of unit deletion:

```
UNo.  UNIT     PART                    FIN-Z
 1  FACING   FACE                       0.
SNo.   TOOL       NOM.    # PAT. DEP-1 DEP-2/NUM.  DEP-3  FIN-X FIN-Z C-SP   FR    M M
R 1 GENERAL  EDGE 45. A     ◆    5.       ◆          ◆      ◆     ◆    104   0.4
FIG                       SPT-X    SPT-Z    FPT-X    FPT-Z                    RGH
 1                         100.     1.       0.       0.
UNo.   UNIT     PART    DIA
 2  T.DRILL   FACE     18.
```

- In case of sequence deletion:

```
UNo.  UNIT    PART                FIN-Z
 1   FACING   FACE                 0.
SNo.  TOOL       NOM.     #  PAT.  DEP-1 DEP-2/NUM.  DEP-3  FIN-X FIN-Z  C-SP   FR    M M
 R 1 GENERAL  EDGE  45. A       ◆      5.       ◆        ◆      ◆      ◆      104   0.4
 FIG                      SPT-X    SPT-Z    FPT-X    FPT-Z                          RGH
 1                        100.      1.       0.       0.
UNo.  UNIT    PART     CPT-X    CPT-Z    FIN-X    FIN-Z
 2   BAR      OUT      100.      0.       0.       0.
SNo.  TOOL      NOM.     #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z C-SP   FR    M M
 R 1 GENERAL  OUT  45. A      0      5.       ◆        ◆      ◆       ◆     120   0.45
 FIG  SHP    S-CNR     SPT-X    SPT-Z    FPT-X    FPT-Z  F-CNR/$  RADIUS/th   RGH
 1   LIN  C   3.         ◆        ◆        60.      50.                ◆
UNo.   UNIT     PART     DIA
 3   T.DRILL    FACE     18.
```

Note: The common unit cannot be deleted.

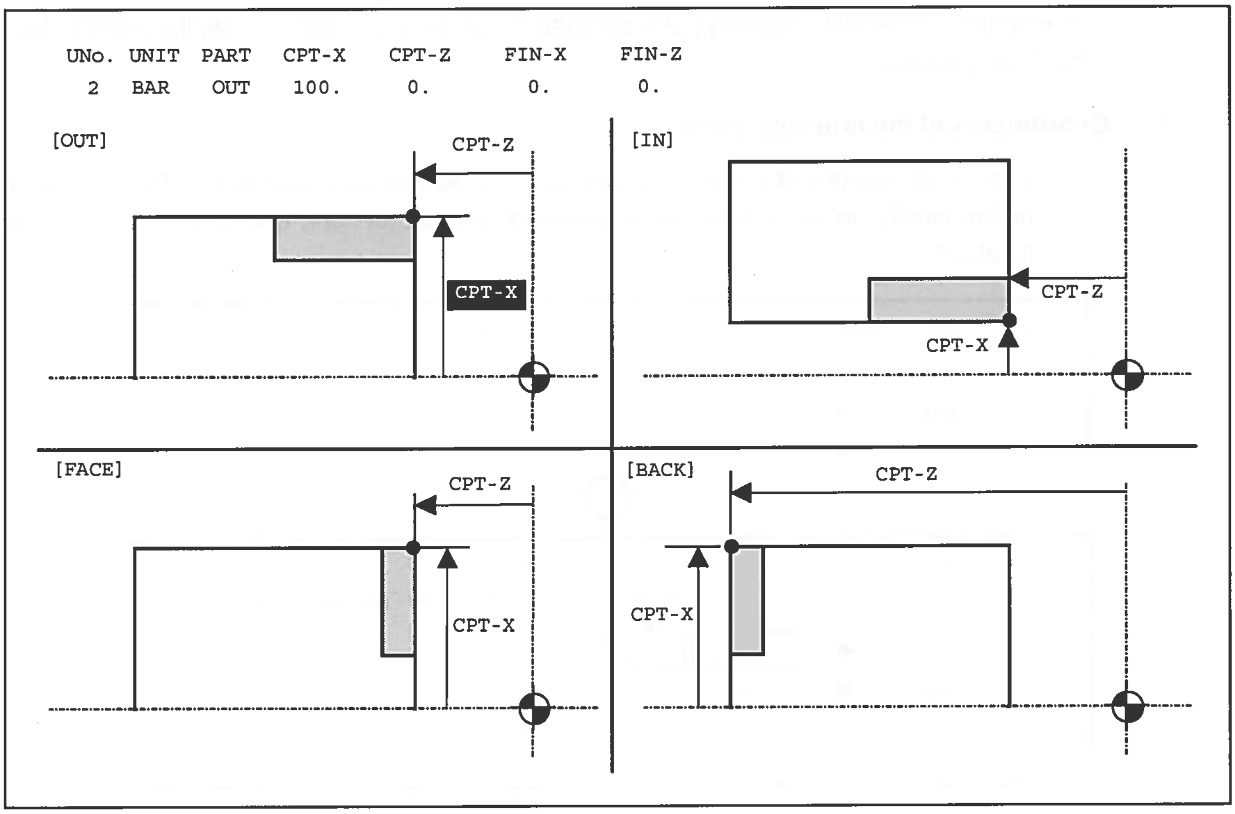
- NOTE -

7 PROGRAM CREATING/EDITING FUNCTIONS

7-1 Help Function

A help function is provided in the NC unit to give an illustrated description of program data. Help window shown below will be called up by pressing the **[HELP]** menu key with the cursor placed on a unit data item on the **PROGRAM** display. In the illustration the display of the respective item is highlighted according to the cursor position.

Example: Help window for the bar-materials machining unit (**BAR**)



In the Help window, you can check details of the data to be set.

Note 1: Not all types of data can be plotted in the Help window. See the relevant section of this manual if you are placed at a loss what type of data to set in the program.

Note 2: Items which will be auto-set and those which will have an illustration on the menu display may not be indicated in the Help window.

7-2 Automatic Crossing-Point Calculation Function

Automatic crossing-point calculation function for the NC system is to compute unknown coordinates of a point of intersection on an arbitrary form and to automatically enter the result in a program.

7-2-1 Automatic crossing-point calculation in the line and face machining units

A crossing-point of arbitrary form is automatically calculated in the line and face machining units.

In the description below machining unit for ZY mode (setting for **MODE** in unit data) is explained as example. Automatic crossing-point calculation can be also used in a similar manner for other machining modes.

1. Coordinates of the crossing-point

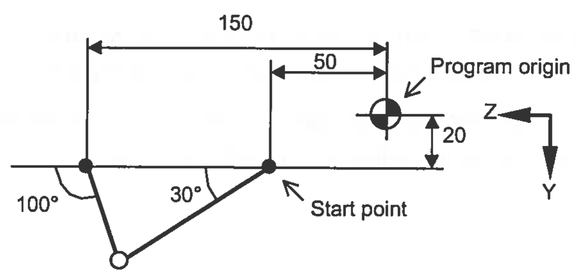
Even if coordinates of a crossing-point are unknown as illustrated below, the NC system will automatically obtain it from the coordinates of the start and end points and from angles involved.

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	10.	50.	20.						
2	LINE	◆	?	?	30.					
3	LINE	◆	150.	20.	100.					

⇩

FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	10.	50.	20.						
2	LINE	◆	140.76	72.4						
3	LINE	◆	150.	20.	30.					
					100.					

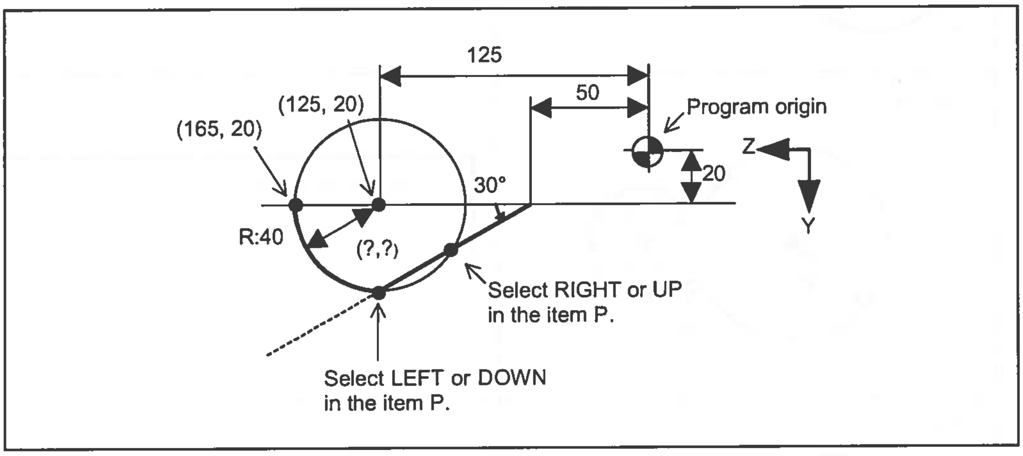
Displayed in yellow



After checking the plane, return to the **PROGRAM** display again and the coordinates so automatically obtained as a crossing-point will be displayed in yellow.

Note: When unknown coordinates of a crossing-point are automatically obtained in a combination of a line with an arc or of two arcs, do not fail to enter P. (Select the position of crossing-point.)

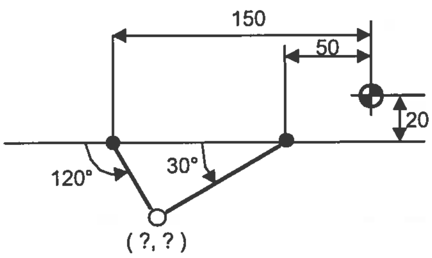
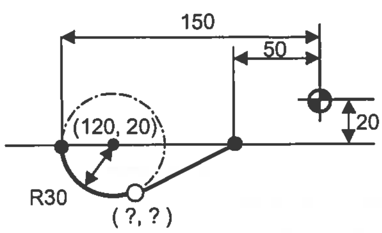
FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR	RGH
1	LINE	10.	50.	20.						
2	LINE	◆	?	?	30.					
3	CW	◆	165.	20.	40.	125.	20.			



Remark: To obtain the coordinates of the crossing-point of the right side, press the **[RIGHT]** or **[UP]** menu key.

2. Examples of automatic crossing-point calculation

A crossing-point is automatically calculated for combinations of line with line, line with arc and arc with arc as shown in the examples below.

<table>
<tr><th>Pattern</th><th>Shape</th><th>Shape sequence</th></tr>
<tr><td>LINE
|
LINE</td><td>
</td><td>
<table>
<tr><th>FIG</th><th>SHP</th><th>SHIFT-R</th><th>Z</th><th>Y</th><th>RADIUS/th</th><th>I</th><th>J</th><th>P</th><th>CNR</th></tr>
<tr><td>1</td><td>LINE</td><td></td><td>50.</td><td>20.</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>2</td><td>LINE</td><td></td><td>?</td><td>?</td><td>30.</td><td></td><td></td><td></td><td></td></tr>
<tr><td>3</td><td>LINE</td><td></td><td>150.</td><td>20.</td><td>120.</td><td></td><td></td><td></td><td></td></tr>
</table>
</td></tr>
<tr><td>LINE
|
ARC
(Contacting)</td><td>
</td><td>
<table>
<tr><th>FIG</th><th>SHP</th><th>SHIFT-R</th><th>Z</th><th>Y</th><th>RADIUS/th</th><th>I</th><th>J</th><th>P</th><th>CNR</th></tr>
<tr><td>1</td><td>LINE</td><td></td><td>50.</td><td>20.</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>2</td><td>LINE</td><td></td><td>?</td><td>?</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>3</td><td>CW</td><td></td><td>150.</td><td>20.</td><td>30.</td><td>120.</td><td>20.</td><td></td><td></td></tr>
</table>
</td></tr>
</table>

Pattern	Shape	Shape sequence

LINE | ARC (Intersecting)

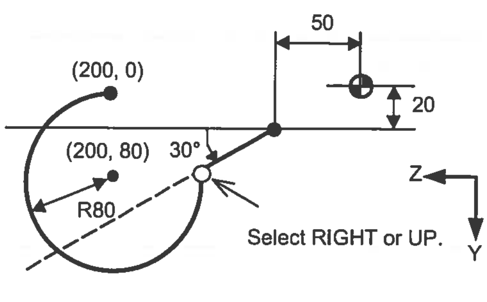


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR
1	LINE		50.	20.					
2	LINE		?	?	30.			R	
3	CW		200	0.	80.	20	80		

ARC | ARC

Closed

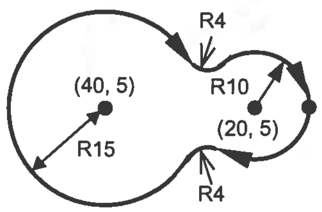


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR
1	CW		?	?	10.	20.	5.	D	R4
2	CW		?	?	15.	40.	5.	U	R4

Open

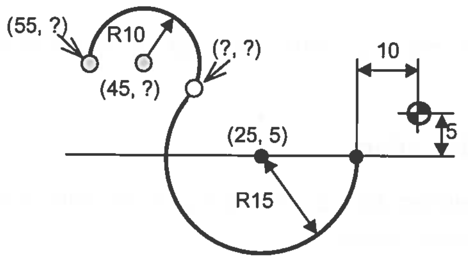


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR
1	LINE		10.	5.					
2	CW		?	?	15.	25.	5.	U	
3	CCW		55.	?	10.	45.	?		

ARC | LINE | ARC

Closed

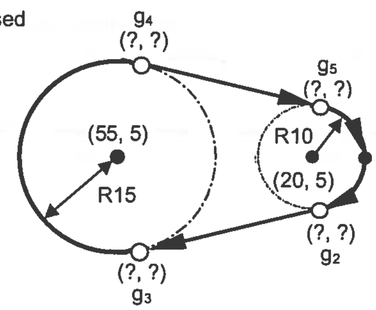


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR
1	LINE		?	?					
2	CW		?	?	10.	20.	5.		
3	LINE		?	?					
4	CW		?	?	15.	55.	5.		

ARC | ARC | ARC

Open

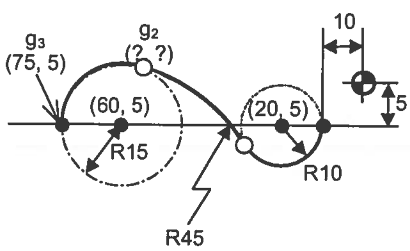


FIG	SHP	SHIFT-R	Z	Y	RADIUS/th	I	J	P	CNR
1	LINE		10.	5.					
2	CW		?	?	10.	20.	5.		
3	CCW		?	?	45.				
4	CCW		75.	5.	15.	60.	5.		

●: Both Z and Y coordinates are known (i, j in the case of the center of an arc).

○: Both Z and Y coordinates are not known (i, j in the case of the center of an arc).

7-2-2 Automatic crossing-point calculation function in the turning unit

When a **TPR**, ■ or ■ shape is to be defined on the sequence line of the bar-materials machining unit (**BAR**) or the copy-machining unit (**CPY**), or when an oblique groove, isopodic trapezoidal groove, or tapered groove shape is to be defined on the sequence line of the groove-machining unit (**T. GROOVE**), you can make the NC unit automatically calculate any unknown coordinates of the start point or end point of that shape.
Automatic calculation may be performed within one sequence or it may span over two sequences.
Conditions for automatic calculation are as follows.

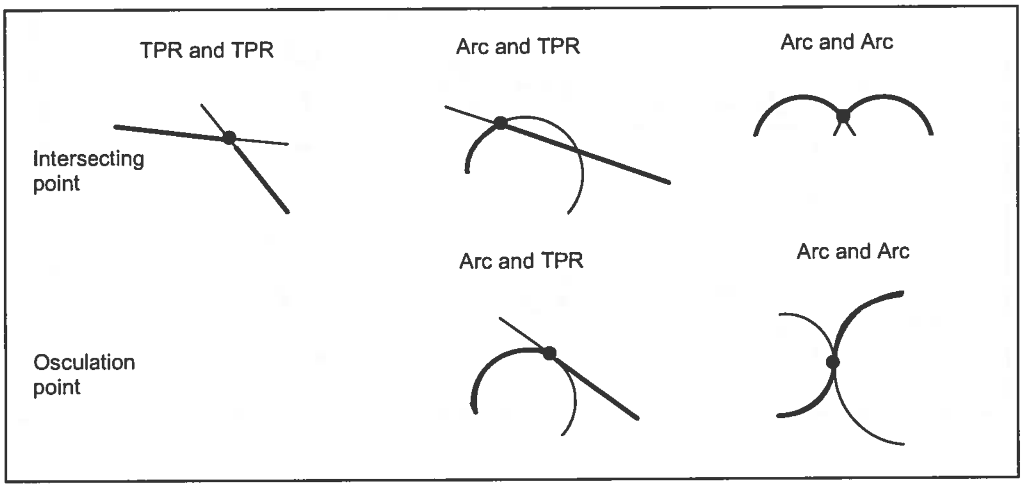
- Automatic calculation within one sequence

Unit	Shape pattern	Conditions
BAR or **CPY**	1. **TPR**	One of the items **SPT-X**, **SPT-Z**, **FPT-X** and **FPT-Z** is unknown; tapering angle known.
	2. Arc	One item of the data pair (**SPT-X**, **SPT-Z**) or (**FPT-X**, **FPT-Z**) is unknown; center coordinates and radius of arc known.
T. GROOVE	3. -	One of the items **SPT-X**, **SPT-Z**, **FPT-X** and **FPT-Z** is unknown; tapering angle known.

- Automatic calculation over two sequences

Unit	Shape pattern	Conditions
BAR or **CPY**	4. Intersection of two **TPR**s	X- and Z-coordinates of the intersecting point of two taperings are unknown; two angels of tapering known.
	5. Intersection of **TPR** and arc	X- and Z-coordinates of the intersecting point of tapering and arc are unknown; tapering angle and center coordinates and radius of arc known.
	6. Osculation of **TPR** and arc	X- and Z-coordinates of the osculation point of tapering and arc are unknown; center coordinates and radius of arc, or tapering angle and radius of arc, are known.
	7. Intersection of two arcs	X- and Z-coordinates of the intersecting point of two arcs are unknown; center coordinates and radii of both arcs known.
	8. Osculation of two arcs	X- and Z-coordinates of the osculation point of two arcs are unknown; center coordinates and radius of one arc, and radius of the other arc are known.

- "Intersecting point" refers to a non-smoothly crossing point. Press the **[INTER PT]** menu key for an unknown intersecting point.
- "Osculation point" refers to a smoothly crossing point. Press the **[CONT PT]** menu key for an unknown osculation point.

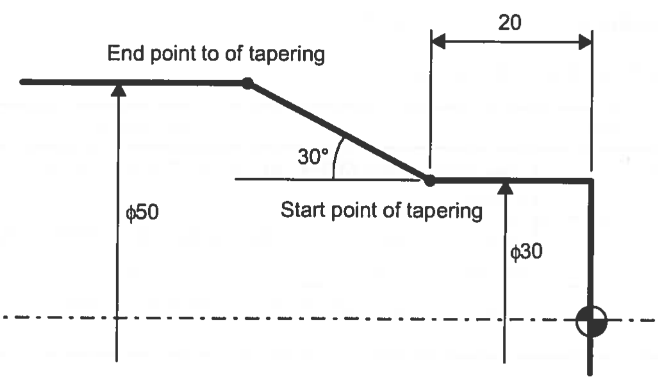


- Automatic calculation can also be performed in grafically checking the programmed data on the **TOOL PATH** or **SHAPE CHECK** display and the result is entered in a program.

Given below is the procedure of data setting for automatic calculation in cases 1 to 8 shown in the table above.

1. **If start or end point of tapering is unknown.**

Example: **FPT-Z** of tapering is unknown.



T4P288

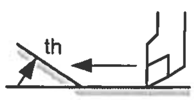
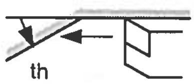
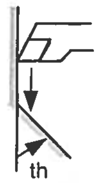
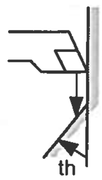
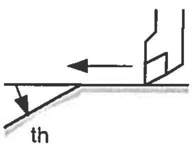
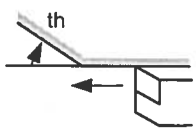
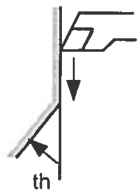
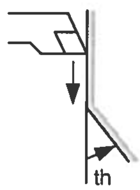
Set data as follows:

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
*	BAR	OUT	*****	*****	*****	*****

FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th
1	LIN		◆	◆	30.	20.		◆
2	TPR		30.	20.	50.	?		30.

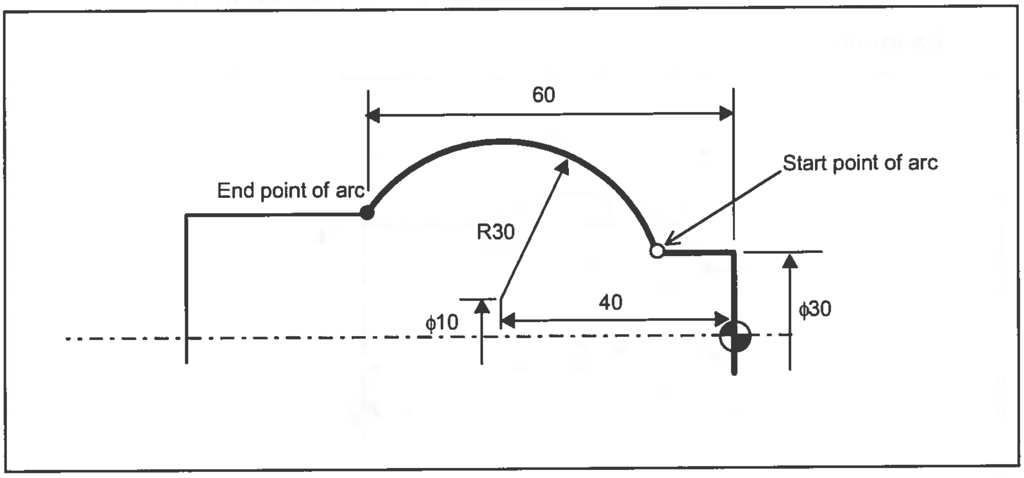
Press the **[INTER PT]** menu key for the unknown **FPT-Z**.
Enter the tapering angle, 30°, for **RADIUS/th.**

Note: Enter positive angle value to designate upward tapering, or negative value for downward tapering.

Section to be machined / Sign	OUT (OUT)	IN (IN)	FACE (FACE)	BACK (BACK)
th: Positive value				
th: Negative value				

2. If start or end point of arc is unknown.

Example: **SPT-Z** and **FPT-X** of convex arc is unknown.



Set data as follows.

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z		
*	BAR	OUT	***	***	***	***		
FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th
1	▄		30.	?	?	60.		30.
2	CTR	◆	10.	40.	◆	◆	◆	◆

Press the **[INTER PT]** menu key for the unknown **SPT-Z** and **FPT-X**.
Enter the radius of the convex arc, 30, for **RADIUS/th**.
For the sequence data line next to that of convex arc, first press the **[CENTER]** menu key and then enter the X- and Z-coordinates of the arc center in **SPT-X** and **SPT-Z**, respectively.

<Supplement>

1. Enter the X-coordinate with minus sign for a center below the workpiece center line; likewise the Z-coordinate for a center on the right of program origin.

   Example:

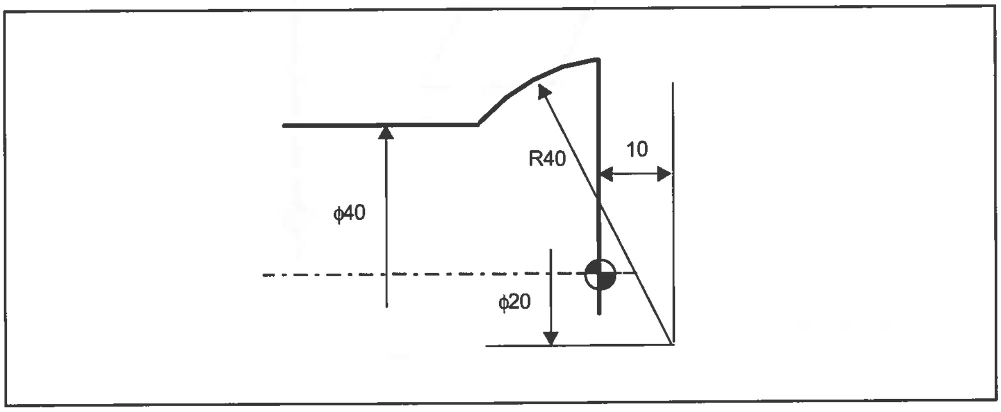


FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	▄		?	0.	40.	?		40.	
2	CTR	◆	-20.	-10.	◆	◆	◆		←

2. In general, an arc and a line cross each other at two points. To specify which one is to be set, use the menu keys **[UP ↑]**, **[DOWN ↓]**, **[LEFT←]** or **[RIGHT→]** on the **CTR** sequence line at the **RADIUS/th** item for unknown **SPT** or at **ROUGH** for **FPT**.

   Example:

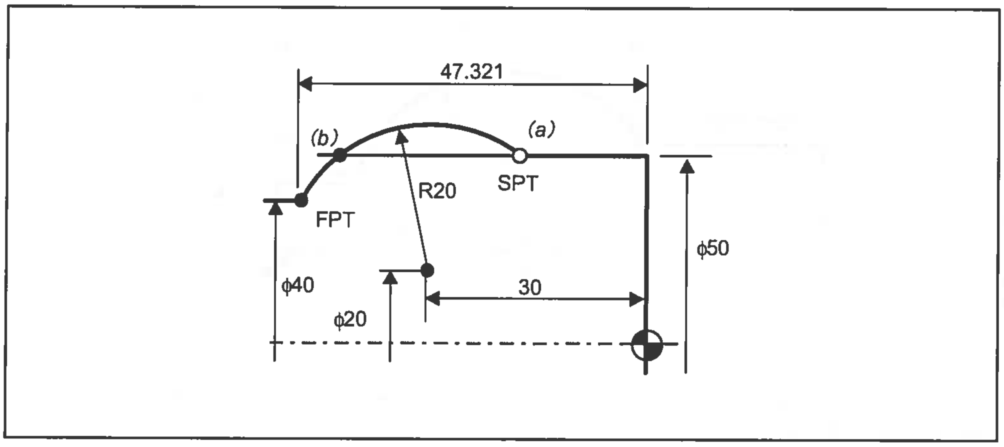


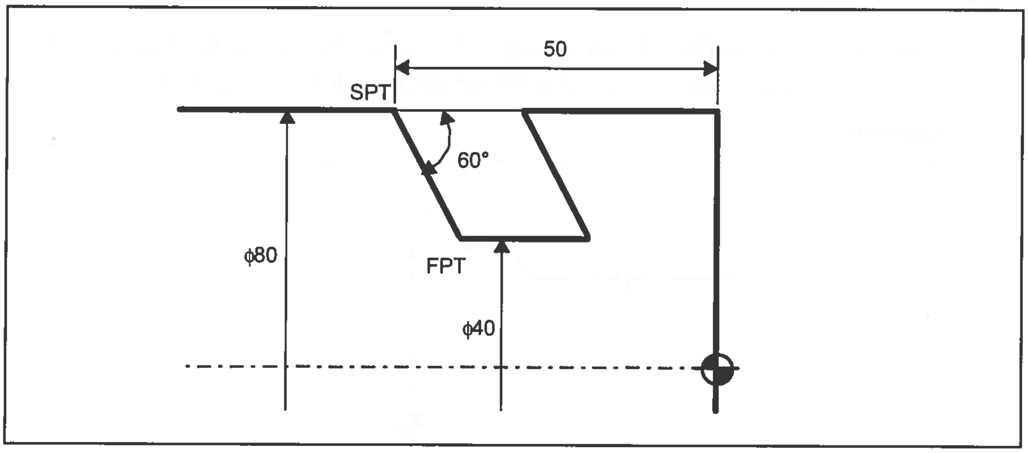
FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	■		50.	?	40.	47.321		20.	
2	CTR	◆	20.	30.	◆	◆	◆	→	

To specify (*a*) for calculation of **SPT-Z**, press the **[RIGHT→]** menu key at **RADIUS/th** since the one point (*a*) lies on the right of the other possible point (*b*).

3. **If start or end point of tapered shape is unknown (for T. GROOVE unit).**

As for the case 1, one of the items **SPT-X** to **FPT-Z** can be auto-set if the tapering angle is clearly known.

Example: **FPT-Z** of tapering is unknown.



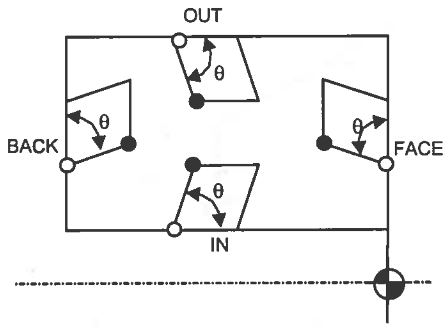
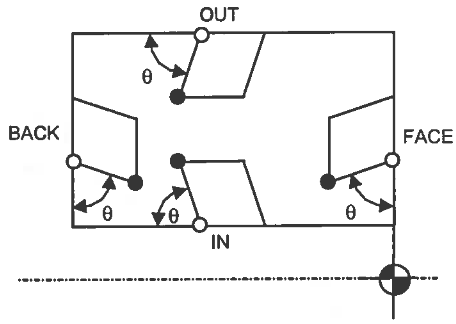
Set data as follows:

UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FINISH
*	T.GROOVE	OUT	0	1	0.	30.	◆

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
F1	GROOVE OUT	50.	B	◆	2.	◆	◆	◆	◆	105	0.2		

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	CNR	ANG	RGH
1		80.	50.	40.	?		60.	

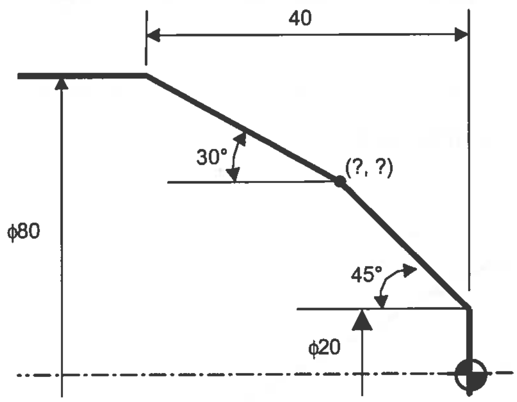
For the grooving pattern #0, the **ANGLE** data must be entered as a positive or negative value according to the direction of the respective tapering.

Enter positive values for **ANGLE** (θ).	Enter negative values for **ANGLE** (θ).
	

For the patterns #1 to #3, the sign of the **ANGLE** data is insignificant.

4. If intersecting point of two taperings is unknown.

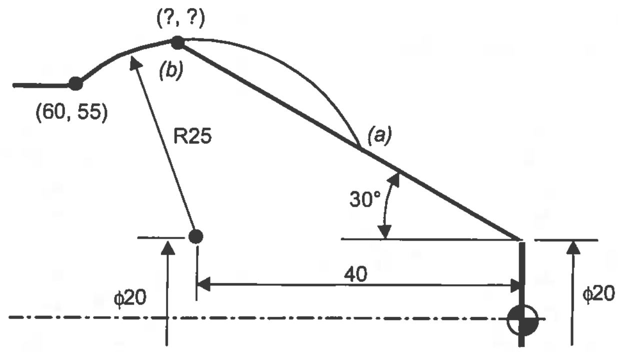
Example:



SHP	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS/th
TPR	20.	0.	?	? *1	45. *2
TPR	?	? *1	80.	40.	30. *3

*1. Press the [INTER PT] menu key for unknown coordinates of the intersecting point of two taperings.
*2. Enter the tapering angle.
*3. Enter the tapering angle.

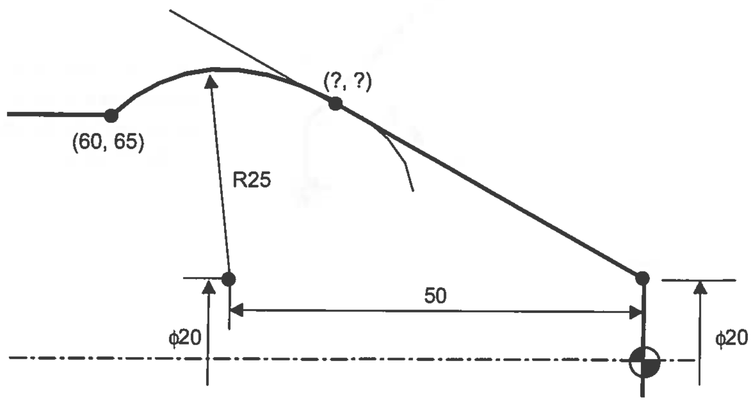
5. If intersecting point of tapering and arc is unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	?	? *1		30. *2	
■		?	? *1	60.	55.		25. *3	
CTR	◆	20.	40. *4	◆	◆	◆	← *5	

*1. Press the **[INTER PT]** menu key for unknown coordinates of the intersecting point of tapering and arc (■).
*2. Enter the tapering angle.
*3. Enter the radius of arc.
*4. Enter the coordinates of arc center.
*5. To specify (*b*) from among the two intersecting points of tapering and arc, press the **[LEFT ←]** (or **[UP ↑]**) menu key.

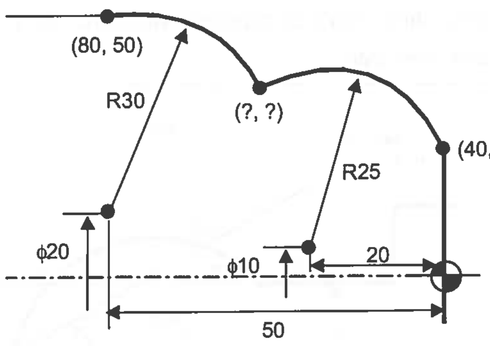
6. If osculation point of tapering and arc is unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	? Ō	? Ō *1			
■		? Ō	? Ō *1	60.	65.		25. *2	
CTR	◆	20.	50. *3	◆	◆	◆		

*1. Press the **[CONT PT]** menu key for unknown coordinates of the osculation point of tapering and arc (■).
*2. Enter the radius of arc.
*3. Enter the coordinates of arc center.

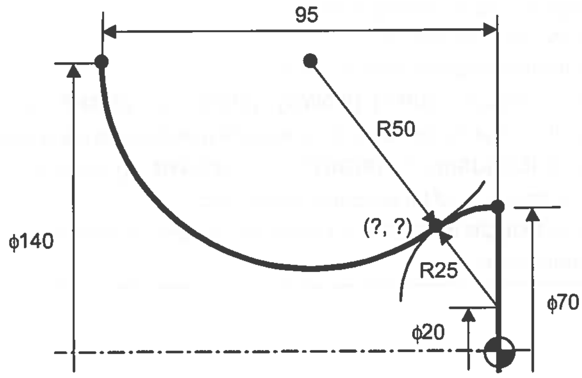
7. If intersecting point of two arcs is unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
		40.	0.	?	? *1		25. *2	
TPR	◆	10.	20. *3	◆	◆	◆	↑ *6	
		?	? *1	80.	50.		30. *4	
CTR	◆	20.	50. *5	◆	◆	◆		

*1. Press the **[INTER PT]** menu key for unknown coordinates of the intersecting point of two convex arcs.
*2. Enter the radius of arc.
*3. Enter the coordinates of arc center.
*4. Enter the radius of arc.
*5. Enter the coordinates of arc center.
*6. To specify the upper one of the two possible intersecting points, press the **[UP ↑]** menu key in response to the message **INTERSEC POS OF FINAL POINT?.**

8. If osculation point of two arcs is unknown.



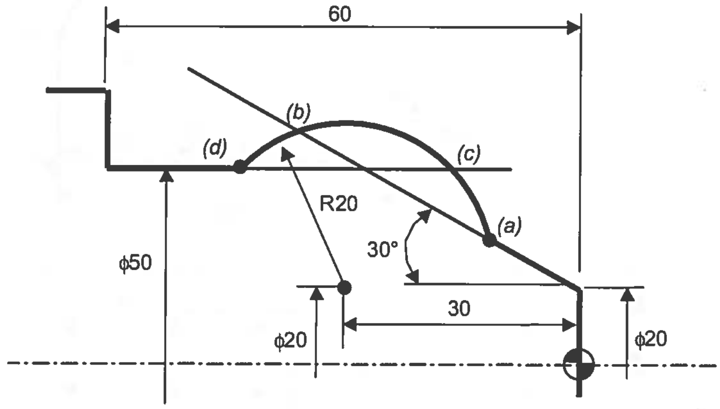
SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
		70.	0.	? Ō	? Ō *1		25. *2	
CTR		20.	0. *3	◆	◆	◆		
	◆	? Ō	? Ō *1	140.	95.		50. *4	

*1. Press the **[CONT PT]** menu key for unknown coordinates of the osculation point of convex and concave arcs.
*2. Enter the radius of convex arc.
*3. Enter the center coordinates of convex arc.
*4. Enter the radius of concave arc.

9. Supplement

In cases 5 to 8, the following unknown intems can also be auto-set.

Example: For intersecting point of tapering and arc, **SPT-X** or **-Z** of tapering and **FPT-X** or **-Z** of arc are unknown.



SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH
TPR		20.	0.	?	? *1		30. *2	
■		?	? *1	50.	? *3		20. *4	
CTR	◆	20.	30. *5	◆	◆	◆	→ *6	← *7
LIN		◆	◆	50.	60.			

*1. Press the **[INTER PT]** menu key for unknown coordinates of the intersecting point of tapering and convex arc.

*2. Enter the tapering angle.

*3. Press the **[INTER PT]** menu key for unknown **FPT-Z** of the convex arc.
In general, even an unknown coordinate of arc end point can be calculated with the intersecting point of tapering and arc remaining unknown.

*4. Enter the radius of convex arc.

*5. Enter the center coordinates of convex arc.

*6, 7 Press the menu key **[UP ↑]**, **[DOWN]** , **[LEFT ←]** or **[RIGHT →]** at the items **RADIUS/th**. and **ROUGH** to specify one of the two possible intersecting points of arc and tapering.
Press at **RADIUS/th** the **[RIGHT]** → (or **[DOWN ↓]**) menu key to specify (*a*) from among the two intersecting points of tapering and convex arc.
Press at **ROUGH** the **[LEFT ←]** menu key to specify (*d*) from among the two intersecting points of arc and straight line.

7-3 Automatic Cutting-Conditions Setting Function

For machining units except for manual program machining units (**MANUAL. P** and **M-MANUAL**), the items of cutting conditions can be automatically set upon specifying a tool for the respective unit. The automatic setting is performed using the data registered on the **CUTTING CONDITION** displays and other various parameters (refer to the table shown below for details of the calculation expressions).

If the programmed feedrate or surface velocity is modified using the VFC function, the new modified value will be stored together with the corresponding basic conditions (machining mode, materials type of workpiece and tool, outside diameter and length of workpiece) into the system memory. Those modified values will then be given priority in the next and subsequent auto-settings (and displayed in reverse form) if the basic conditions agree with the stored ones.

Example: For **BAR OUT** roughing

Before VFC

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	CBN STL	100.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45. A		0	3.	◆	◆	◆	◆	100	0.3		

After VFC

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	CBN STL	100.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45. A		0	3.	◆	◆	◆	◆	120	0.33		

Registered with “cermet” on the **TOOL DATA**

After modification of programmed data using the VFC function during machining, the new values of cutting conditions (marked with ▁▁) will be stored together with the basic conditions (＿). If a program, such as (A) shown below, is subsequently created, those new values will be ‘auto-set’ since all basic conditions agree with the jointly stored ones. For program (B), which has different basic conditions, the auto-setting function will set values normally calculated using the fixed expressions.

(A)

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	CBN STL	100.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45.	A	0		◆	◆	◆	◆				

Pressing the **[AUTO SET]** menu key.

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	CBN STL	100.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45.	A	0	3.	◆	◆	◆	◆	120	0.33		

Stored data displayed in reverse form

(B)

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	ALUMINUM	30.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45.	A	0		◆	◆	◆	◆				

Pressing the **[AUTO SET]** menu key.

No.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM
0	ALUMINUM	30.	0.	40.	0.	2000

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	45.	A	0	3.	◆	◆	◆	◆	300	0.45		

Data calculated using fixed expressions displayed.

Note: For internally checking the basic conditions for equality, the data of **OD-MAX** and **LENGTH** are roughly managed under parametrically specified classification into four groups. If the three classifying values for **LENGTH** data are 30, 60 and 110 mm (four groups: 0 to 30, 30 to 60, 60 to 110, and from 110 upwards), for example, then the data "75" and "90" will be managed here as equal to each other.

Calculation expressions of auto-setting function

Unit	Calculation expressions
BAR CPY CORNER FACING	C-SP (R) = R-SPD in CUT. COND. (TURN.) display × [R-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-SPD% (TOOL) in CUT. COND. (MAT.) display/100] C-SP (F) = F-SPD in CUT. COND. (TURN.) display × [F-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [F-SPD% (TOOL) in CUT. COND. (MAT.) display/100] FR (R) = R-FEED in CUT. COND. (TURN.) display × [R-FEED% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-FEED% (TOOL) in CUT. COND. (MAT.) display/100] DEP (R) = R-DEPTH in CUT. COND. (TURN.) display × [R-DEP% (WRKP.) in CUT. COND. (MAT.) display/100 × [R-DEP% (TOOL) in CUT. COND. (MAT.) display/100]
T. GROOVE	C-SP (R) = R-SPD in CUT. COND. (TURN.) display × [R-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-SPD% (TOOL) in CUT. COND. (MAT.) display/100] C-SP (F) = F-SPD in CUT. COND. (TURN.) display × [F-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [F-SPD% (TOOL) in CUT. COND. (MAT.) display/100] FR = R-FEED in CUT. COND. (TURN.) display × [R-FEED% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-FEED% (TOOL) in CUT. COND. (MAT.) display/100] DEP = R-DEPTH in CUT. COND. (TURN.) display × [R-DEP% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-DEP% (TOOL) in CUT. COND. (MAT.) display/100]
THREAD	HGT = Thread pitch × **K24**/10000 (when OUT/FACE/BACK is selected for metric threads) HGT = Thread pitch × **K25**/10000 (when IN is selected for metric threads) HGT = Thread pitch × **K26**/10000 (when OUT/FACE/BACK is selected for inch threads) HGT = Thread pitch × **K27**/10000 (when IN is selected for inch threads) NUMBER = (see the description given afterwards) V = F-SPD in CUT. COND. (TURN.) display × [F-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [F-SPD% (TOOL) in CUT. COND. (MAT.) display/100] DEPTH = (see the description given afterwards)
T. DRILL	DEP-1 = Hole diameter (DRL-DIA) × **K17**/100 DEP-2 = **U44**/A A = 1000 for metric system DEP-3 = **U46**/A 10000 for inch system V = R-SPD in CUT. COND. (TURN.) display x [R-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-SPD% (TOOL) in CUT. COND. (MAT.) display/100] FEED = R-FEED in CUT. COND. (TURN.) display × [R-FEED% (WRKP.) in CUT. COND. (MAT.) display/100] × [R-FEED% (TOOL) in CUT. COND. (MAT.) display/100]
T. TAP	PITCH = Data based on JIS (Japan Industrial Standards); depends on the nominal diameter of the thread. V = F-SPD in CUT. COND. (TURN.) display × [F-SPD% (WRKP.) in CUT. COND. (MAT.) display/100] × [F-SPD% (TOOL) in CUT. COND. (MAT.) display/100]

Details on calculation expressions for **THREAD** unit

- **NUMBER**

#0, $\boxed{\#0}$ Metric $\dfrac{3.4 \times \text{LEAD}}{\text{MULTI}} + 3.59$

Inch $\dfrac{3.4 \times \text{LEAD} \times 25.4}{\text{MULTI}} + 3.59$

The first decimal is rounded off.

#1, $\boxed{\#1}$ $N = (D - a)/D_1$ (N: any decimals are cut away)

If $|(D - a)/N - D_1| > 0.000475$, then $N = N + 1$

#2, $\boxed{\#2}$ $N = (D - a)^2/D_1$ (N: any decimals are cut away)

If $\left|\dfrac{D - a - D_1 \times \sqrt{N}}{N}\right| > 0.000475$, then $N = N + 1$

a: Finishing allowance for threading (parameter)
D: **HGT** (programmed data)
D_1: **DEPTH** (programmed data)
N: **NUMBER** (solution to be sought)

- **DEPTH**

#0, $\boxed{\#0}$ If **NUMBER** is even:

$$D_1 = \frac{8 \times D - 2\,(N + 2) \times a/2}{3 \times N - 1}$$

If **NUMBER** is odd:

$$D_1 = \frac{2\,(N - 2) \times (4 \times N \times D - (N + 1)^2 \times a/2)}{(N - 1) \times (3 \times N^2 - 4 \times N - 1)}$$

#1, $\boxed{\#1}$ $D_1 = (D - a)/N$

#2, $\boxed{\#2}$ $D_1 = (D - a)/\sqrt{N}$

7-4 Desk Calculator Functions

When entering shapes (sequence data) for a MAZATROL program, add/subtract/multiply/divide operations and calculations using trigonometric functions and/or square roots can be carried out by selecting **[Calculator]** from the menu bar **[Window]**.

Enter a calculation expression and press the input key one time. The calculation result will then be displayed in the data input area at the bottom right of the display.

If the result is correct, press the input key once again. The particular data will then be set at the cursor position. If the result is not correct, enter the correct calculation expression after pressing the data cancellation key (this deletes the entire expression) or the clear key (this deletes character by character).

In the menu, the asterisk sign (∗) means multiplication and the slash sign (/) means division.

7-5 Tool Data Window

A tool data window can be displayed by pressing the **[TOOL DAT WINDOW]** menu key while the cursor remains set at a **NOM.** (**NOM-ϕ**), **C-SP** or **FR** item for the machining unit. Only the tools corresponding to the current machining unit or the tool sequence are selected and displayed in the window.

- Press the page key to view the next page.
- Pressing the menu key once again closes the window. Moving the cursor to an item of other data type also closes it.

Example 1: Turning tool

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	100.	0.	0.	0.

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R 1	GENERAL OUT	□											

TOOL DATA

TNo.	TOOL		NOM. NOM-Ø		FW/RV R/L	ACT-Ø NOSE-R	GRV CUT	DEPTH ANGLE	TIP-WID EDG-ANG
1	GENERAL	OUT	1.	D	RG	0.8			
1	GENERAL	OUT	1.	A	RG	0.8	95.		80.
1	GENERAL	OUT	1.	B	RG	0.8	95.		80.
2	GENERAL	OUT	2.	A	LF	0.4	93.		55.
2	GENERAL	OUT	2.	B	LF	0.4	95.		55.
4	GENERAL	OUT							
7	GENERAL	OUT	7.	B	RG	0.4	107.		55.
8	GENERAL	OUT	8.	B	RG	0.4	105.		55.

1/ 3

Example 2: Milling tool

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R		
3	LINE CTR	ZY	◆	45.	5.	10.	7	0.068	◆		
SNo.	**TOOL**	**NOM-ϕ**	**APRCH-1**	**APRCH-2**	**TYPE**	**AFD**	**DEP-A**	**DEP-R**	**C-SP**	**FR**	**M M**
R 1	END MILL	▯									

TOOL DATA

TNo.		TOOL	NOM. NOM-Ø		FW/RV R/L	ACT-Ø NOSE-R	GRV DEPTH CUT ANGLE	TIP-WID EDG-ANG
22	←	END MILL	11.	A		11.		
22	↓	END MILL	11.	B		11.		
22	→	END MILL	11.	C		11.		
23	←	END MILL	10.	A		10.		
23	↓	END MILL	10.	B		10.		
24	←	END MILL	12.	A		12.		
24	↓	END MILL	12.	B		12.		
24	→	END MILL	12.	C		12.		

1/ 2

7-6 Tool File Window

A tool file window can be displayed by pressing the **[TOOL. F WINDOW]** menu key while the cursor remains set at a **NOM-ϕ** item in the tool sequence data for the end mill, face mill, chamfering cutter, or ball end mill of the milling unit. Only the data for tools corresponding to the current tool sequence are selected from the tool file data registered on the **TOOL FILE** display and displayed in the window.

- Press the page key to view the next page.
- Pressing the menu key once again closes the window. Moving the cursor to an item of other data type also closes it.

Example:

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R		
3	LINE CTR	ZY	◆	45.	5.	10.	7	0.068	◆		
SNo.	**TOOL**	**NOM-ϕ**	**APRCH-1**	**APRCH-2**	**TYPE**	**AFD**	**DEP-A**	**DEP-R**	**C-SP**	**FR**	**M M**
R 1	END MILL	▯									

TOOL FILE

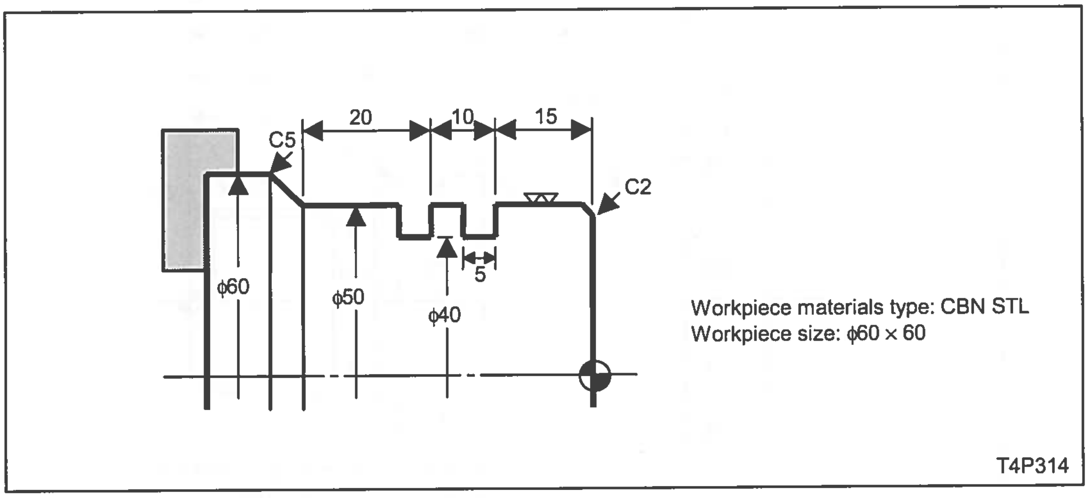
	TOOL NAME	NOM-Ø		MATERIAL	DEPTH	TEETH
1	END MILL	7.	A	HSS	12.	4
2	END MILL	7.	B	HSS	12.	4
3	END MILL	7.	C	HSS	12.	4
4	END MILL	10.	A	COAT.HSS	3.	2
5	END MILL	10.	B	COAT.HSS	3.	2
6	END MILL	10.	H	HSS	20.	4
7	END MILL	10.	V	HSS	20.	4
8	END MILL	10.	W	CARBIDE	20.	4
9	END MILL	11.	A	CARBIDE	3.	2
10	END MILL	11.	B	COAT.CBD	3.	2

1/2

8 SAMPLE PROGRAMS

Example 1: 2-axes machining (**BAR, T. GROOVE**)

Machining drawing

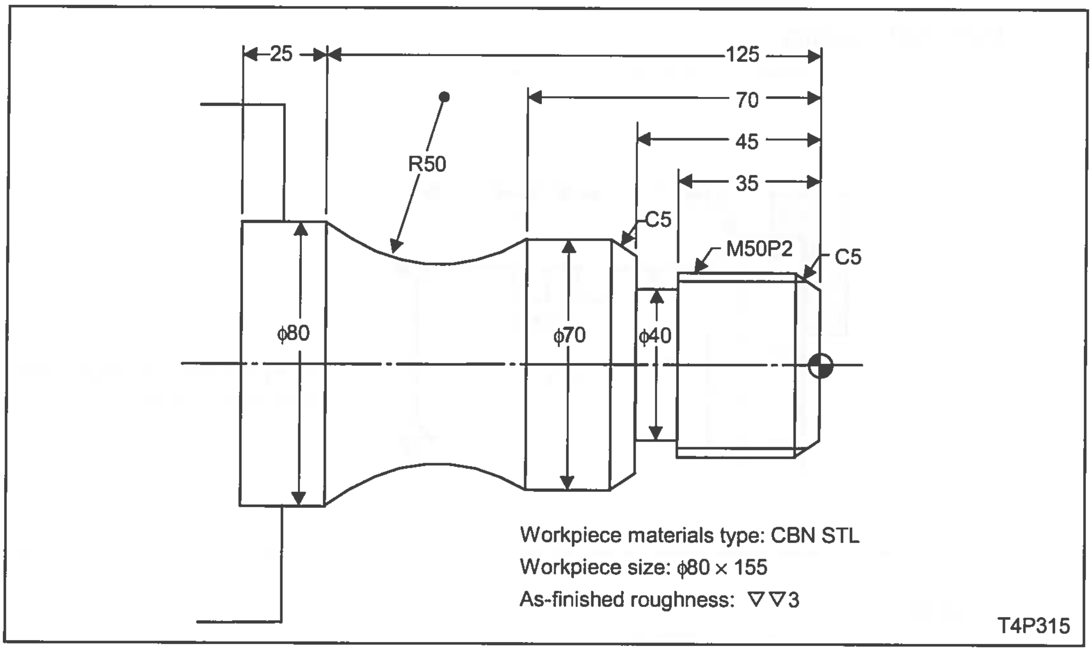


Program

```
UNo.   MAT      OD-MAX  ID-MIN  LENGTH  WORK FACE   RPM
 0     CBN STL   60.     0.      60.       0.       2000
UNo.   UNIT PART CPT-X  CPT-Z  FIN-X   FIN-Z
 1      BAR OUT   60.    0.     0.2     0.1
SNo.    TOOL       NOM.  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR   M  M
 R1   GENERAL OUT  0.5A   0     2.5      ◆          ◆      ◆      ◆    130   0.3
 F2   GENERAL OUT  0.1A   ◆     ◆        ◆          ◆      0      0    200   0.1
FIG     SHP      S-CNR  SPT-X  SPT-Z   FPT-X   FPT-Z  F-CNR/$  RADIUS/th   RGH
 1      LIN      C 2.    ◆      ◆       50.     50.    C 5.       ◆        ▼▼4
UNo.    UNIT      PART   PAT.   No.    PITCH   WIDTH  FINISH
 2    T.GROOVE    OUT     0      2      10.     5.      ◆
SNo.    TOOL      NOM.    PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X  FIN-Z   C-SP   FR    M M
 F2   GROOVE OUT  3.A      ◆     2.       ◆        ◆     0.     ◆      120   0.08
FIG     S-CNR        SPT-X     SPT-Z   FPT-X    FPT-Z      CNR      ANG      RGH
 1                    50.       20.     40.      20.
UNo.    UNIT       COUNTER    RETURN  WK.No.  CONT.   NUM.    SHIFT
 3      END           0          0              0       0       0.
```

Example 2: 2-axes machining (**FACING, BAR, T. GROOVE, THREAD**)

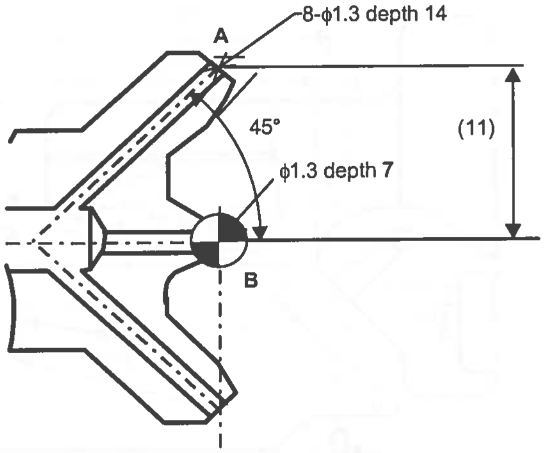
Machining drawing



Program

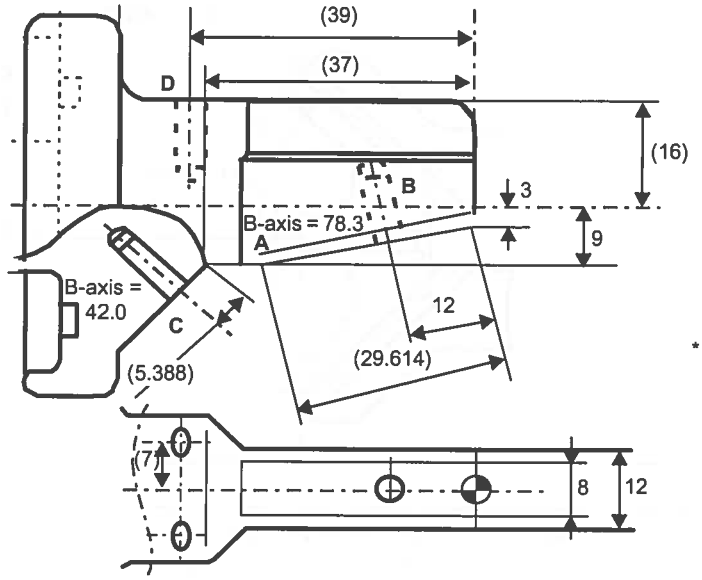
```
UNo.     MAT      OD-MAX ID-MIN LENGTH  WORK FACE   RPM
 0     CBN STL    80.     0.     155.       5.      2000
UNo.   UNIT     PART                     FIN-Z
 1    FACING    FACE                      0.1
SNo.      TOOL        NOM.       PAT.   DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z   C-SP    FR  M  M
 R1   GENERAL EDGE    0.5A        ◆      2.        ◆        ◆      ◆      ◆     120.    0.3
 F2   GENERAL EDGE    0.5B        ◆      ◆         ◆        ◆      ◆      0.    160.    0.1
FIG                          SPT-X  SPT-Z  FPT-X  FPT-Z                 RGH
 1                            80.    5.     0.     0.                   ▼▼3
UNo.   UNIT     PART   CPT-X    CPT-Z    FIN-X    FIN-Z
 2      BAR     OUT     80.      0.       0.2      0.1
SNo.      TOOL        NOM.   PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z   C-SP     FR     M  M
 R1   GENERAL OUT     0.5A    0     2.5       ◆          ◆      ◆      ◆     130     0.3
 F2   GENERAL OUT     0.1B    ◆     ◆         ◆          ◆      0      0     200     0.1
FIG      SHP      S-CNR   SPT-X  SPT-Z  FPT-X  FPT-Z  F-CNR/$  RADIUS/th  RGH
 1       LIN      C 5.     ◆      ◆      50.    45.                ◆      ▼▼3
 2       LIN      C 5.     ◆      ◆      70.    70.                ◆      ▼▼3
 3       ⌒                 70.    70.    80.    125.               50.    ▼▼3
UNo.   UNIT     PART    PAT.     No.    PITCH   WIDTH   FINISH
 3   T.GROOVE   OUT      0        1      0.      10.      ◆
SNo.     TOOL        NOM.        PAT.  DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP    FR  M  M
 F2   GROOVE OUT    3.A           ◆     2.        ◆        ◆      0.     ◆     120   0.08
FIG      S-CNR          SPT-X     SPT-Z    FPT-X    FPT-Z       CNR       ANG    RGH
 1                       50.       45.      40.      45.
UNo.   UNIT     PART    CHAMF    LEAD     ANG    MULTI    HGT
 4    THREAD    OUT      0        2       55       1     1.299
SNo.     TOOL        NOM.        PAT.  DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M  M
 1    THREAD OUT     1.A          0      ◆       10        ◆      ◆      ◆     120    ◆
FIG                          SPT-X  SPT-Z  FPT-X  FPT-Z
 1                            50.    0.     50.    38.
UNo.   UNIT          COUNTER  RETURN  WK.No.  CONT.  NUM.  SHIFT
 5     END              0        0              0      0     0.
```

Example 3: Point milling



	UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM						
	0	FC	24.	0.	37.	0.	10000						
Milling of eight oblique holes (See A of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	DIA	DEPTH	CHMF					
	1	DRILLING	/C	45.	◆	1.3	14.	0.					
	SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
	1	CTR-DR	4.	1.3	◆	◆	◆	90°	◆	10	0.1		
	2	DRILL	1.3	1.3	14.	0.	100	PCK2 T	0.65	9	0.019		
	FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANGLE	Q	R				
	1	ARC	11.	0.	0.	8	45	0	1				
Milling of center hole (See B of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	DIA	DEPTH	CHMF					
	2	DRILLING	XC	◆	◆	1.3	7.	0.					
	SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
	1	CTR-DR	4.	1.3	◆	◆	◆	90°	◆	10	0.1		
	2	DRILL	1.3	1.3	7.	0.	100	PCK2 T	0.65	9	0.019		
	FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANG	Q	R				
	1	PT	0.	0.	0.	◆	◆	◆	0				
	UNo.	UNIT	COUNTER	RETURN	WK.No.	CONT.	NUM.	SHIFT					
	3	END											

Example 4: Point/face milling

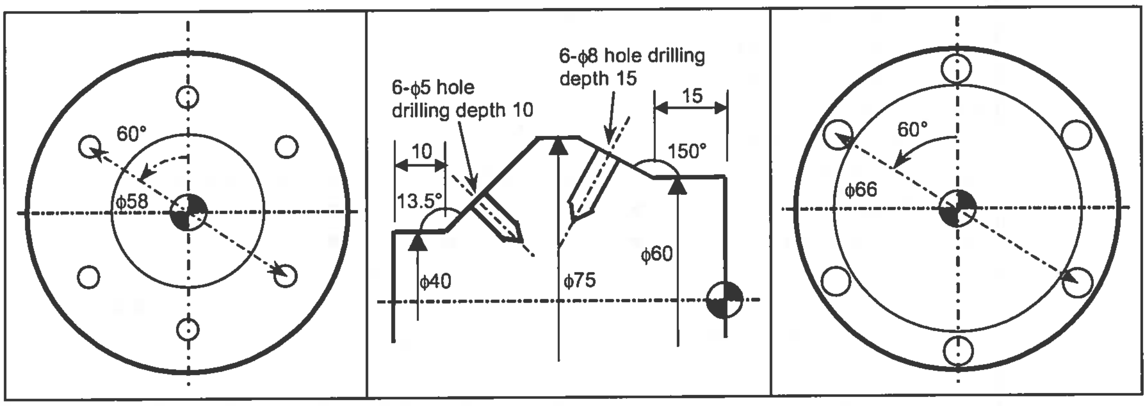


* All the holes are 4-M4, 10 deep and 12 in depth of prepared hole.

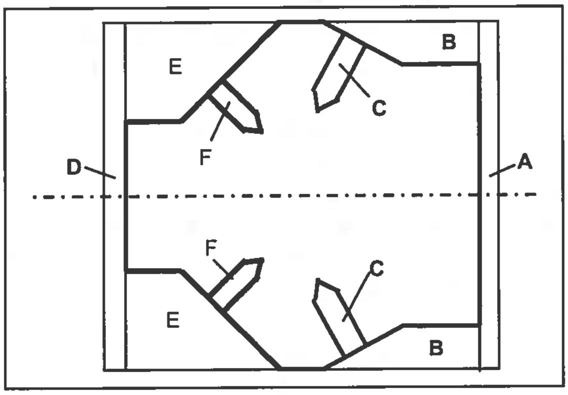
	UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM								
	0	FC	58.	0.	320.	0	10000								
Milling of the oblique face (See A of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R					
	1	LINE CTR	/Y	78.3	180.	2.	6.	1	0.	◆					
	SNo.	TOOL	NOM-ϕ	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M		
	R 1	END MILL	20. A	?	?	◆	G01	2.	◆	100	0.2				
	FIG	SHP	SHIFT-Z	SHIFT-R	X	Y	RADIUS/th	I	J	P	CNR	RGH			
	1	LINE	0.	3.	0.	0.									
	2	LINE	◆	◆	29.614	0.									
Oblique tapping in the groove (See B of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	NOM-	MAJOR-ϕ	PITCH	TAP-DEP	CHMF					
	2	TAPPING	/Y	78.3	180.	M 4.	4.	0.7	10.	1.					
	SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M		
	1	CTR-DR	4.	10	◆	◆	◆	90°	◆	10	0.1				
	2	DRILL	3.4	3.4	12.	0.	100	T	1.7	9	0.037				
	3	TAP	M 4. J	4.	10.	TAP	◆	FIX P	0.7	10	0.7				
	FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
	1	PT	0.	3.	12.	0.	◆	◆	◆	◆	◆	◆	0	◆	0
Oblique tapping (See C of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	NOM-	MAJOR-ϕ	PITCH	TAP-DEP	CHMF					
	3	TAPPING	/Y	42.0	180.	M 4.	4.	0.7	10.	1.					
	SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M		
	1	CTR-DR	4.	10	◆	◆	◆	90°	◆	10	0.1				
	2	DRILL	3.4	3.4	12.	0.	100	PCK1 T	1.7	9	0.037				
	3	TAP	M 4. J	4.	10.	TAP	◆	FIX P	0.7	10	0.7				
	FIG	SHP	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
	1	LINE	37.	9.	5.388	-7.	14	◆	◆	2	◆	90.	◆	0	1
Tapping (See D of the figure above.)	UNo.	UNIT	MODE	POS-B	POS-C	NOM-	MAJOR-ϕ	PITCH	TAP-DEP	CHMF					
	4	TAPPING	ZC	◆	◆	M 4.	4.	0.7	10.	1.					
	SNo.	TOOL	NOM-ϕ	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DDP	RGH	DEPTH	C-SP	FR	M	M		
	1	CTR-DR	4.	10	◆	◆	◆	90°	◆	10	0.1				
	2	DRILL	3.4	3.4	12.	0.	100	PCK1 T	1.7	9	0.037				
	3	TAP	M 4. J	4.	10.	TAP	◆	FIX P	0.7	10	0.7				
	FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	SPT-Y	NUM.	ANGLE	Q	R					
	1	PT	16.	0.	39.	0.	◆	◆	◆	0					
	UNo.	UNIT	COUNTER	RETURN	WK.No.	CONT.	NUM.	SHIFT							
	5	END													

Example 5: Workpiece transfer

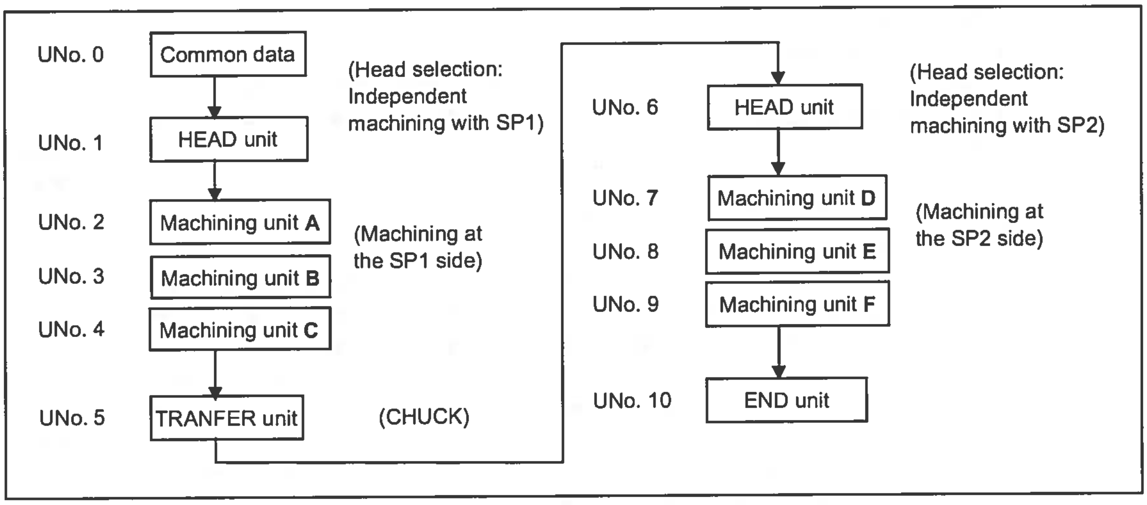
Machining drawings



Machining outline



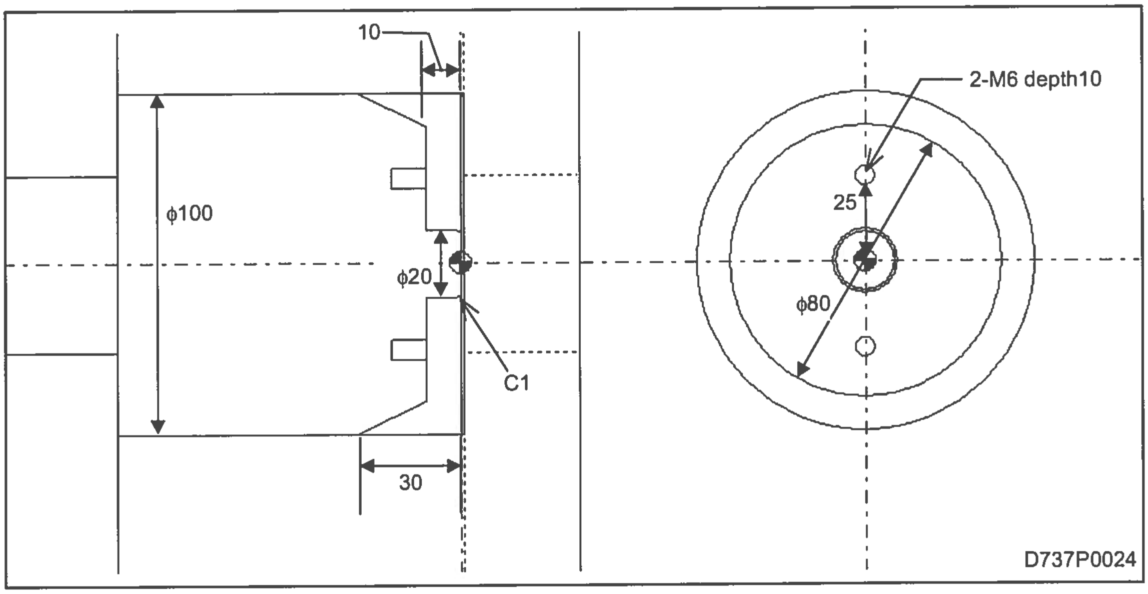
Program outline



Program

```
UNo.  MAT       OD-MAX  ID-MIN  LENGTH WORK FACE  RPM
 0    CBN STL   75.     0.      62.    1.         1500
UNo.  UNIT      TYPE    HEAD    SPDL
 1    HEAD      SIN     1       ◆
UNo.  UNIT      PART    FIN-Z
 2    FACING    FACE    0.1
SNo.  TOOL          NOM.  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR    M  M
 R1   GENERAL EDGE  0.5A  ◆     2.     ◆           ◆      ◆      ◆      120   0.25
 F2   GENERAL EDGE  0.5B  ◆     ◆      ◆           ◆      0.     0.     160
FIG                       SPT-X  SPT-Z  FPT-X  FPT-Z               RGH
 1                        75.    1.     0.     0.                  ▼▼4
UNo.  UNIT  PART  CPT-X  CPT-Z            FIN-X  FIN-Z
 3    BAR   OUT   75.    0.               0.2    0.1
SNo.  TOOL         NOM.  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR    M  M
 R1   GENERAL OUT  0.5A  0     2.5    ◆           ◆      ◆      ◆      150   0.25
 F2   GENERAL OUT  0.1B  ◆     ◆      ◆           ◆      0.     0.     250
FIG   SHP   S-CNR  SPT-X  SPT-Z  FPT-X  FPT-Z  F-CNR/$  RADIUS/th  RGH
 1    LIN          ◆      ◆      60.    15                ◆          ▼▼4
 2    TPR          60     15.    75.    27.99.                       ▼▼4
UNo.  UNIT      MODE  POS-B  POS-C       DIA  DEPTH  CHMF
 4    DRILLING  /C    60.    ◆           8.   15.    0.
SNo.  TOOL    NOM-ϕ  HOLE-ϕ  HOLE-DEP  PRE-DIA  PRE-DEP  RGH   DEPTH  C-SP  FR     M   M
 1    CTR-DR  20.    10.     ◆         ◆        ◆        90°   ◆      25    0.09
 2    DRILL   8.     8.      15.       0.       100      DRIL  T 4.   25    0.129
FIG   SHP   SPT-R/x  SPT-C/y  SPT-Z   NUM.  ANGLE  Q  R
 1    ARC   33.      0.       20.196  6     60.    0  0
UNo.  UNIT      SETUP-No.  HEAD  SPDL  PUSH  CHUCK
 5    TRANSFER  CHUCK      1     1→2   1     0     ◆
UNo.  UNIT      TYPE    HEAD    SPDL
 6    HEAD      SIN     2       ◆
UNo.  UNIT      PART    FIN-Z
 7    FACING    BACK    0.1
SNo.  TOOL          NOM.  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR   M  M
 R1   GENERAL EDGE  0.5C  ◆     1.     ◆           ◆      ◆      ◆      150   0.3
FIG                       SPT-X  SPT-Z  FPT-X  FPT-Z                RGH
 1                        75.    -61.   0.     -60.                 ▼▼4
UNo.  UNIT PART  CPT-X  CPT-Z  FIN-X  FIN-Z
 8    BAR  OUT   75.    60.    0.2    0.1
SNo.  TOOL         NOM.  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR   M  M
 R1   GENERAL OUT  0.5C  0     1.5    ◆           ◆      ◆      ◆      150   0.3
 F2   GENERAL OUT  0.1D  ◆     ◆      ◆           ◆      0.     0.     196   0.
FIG   SHP   S-CNR  SPT-X  SPT-Z  FPT-X  FPT-Z  F-CNR/$  RADIUS/th  RGH
 1    LIN          ◆      ◆      40.    50.               ◆          ▼▼4
 2    TPR          40.    50.    75.    32.5              45.        ▼▼4
UNo.  UNIT      MODE  POS-B  POS-C       DIA  DEPTH  CHMF
 9    DRILLING  /C    135.   ◆           5.   10.    0.3
SNo.  TOOL    NOM-ϕ  HOLE-ϕ  HOLE-DEP  PRE-DIA  PRE-DEP  RGH   DEPTH   C-SP  FR     M   M
 1    CTR-DR  20.    5.6     ◆         ◆        ◆        90°   ◆       25    0.09
 2    DRILL   5.     5.      10.       0.       100      DRIL  T 2.5   25    0.088
FIG   SHP   SPT-R/x  SPT-C/y  SPT-Z  NUM.  ANGLE  Q  R
 1    ARC   29.      0.       41.    6     60.    0  0
UNo.  UNIT  COUNTER  RETURN  WK.No.  CONT.  NUM.  SHIFT
 10   END
```

Example 6: Machining with upper and lower turrets



UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	TR2-DIA
0	CST IRN	100.	0.	101.	1.	2000	

UNo.	UNIT	PART	FIN-Z
1	FACING	FACE	0.1

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1	GENERAL EDGE	10.	◤	◆	1.5	◆	◆	◆	◆	150	0.15		
F2	GENERAL EDGE	10.	B	◆	◆	◆	◆	◆	0.	200	0.		

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	RGH
1	100.	1.	0.	0.	▼▼3

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
2	BAR	OUT	100.	0.	0.2	0.1

SNo.	TOOL	NOM.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1	GENERAL OUT	5.	◤	1	1.5	◆	◆	◆	◆	150	0.17		
F2	GENERAL OUT	10.	B	◆	◆	◆	◆	0.	0.	200	0.		

FIG	SHP	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	RADIUS/th	RGH
1	LIN	C 1.	◆	◆	20.	10.		◆	▼▼3
2	TPR		80.	10.	100.	30.			▼▼3

UNo.	UNIT	MODE	POS-B	POS-C	NOM-	MAJOR-ϕ	PITCH	TAP-DEP	CHMF
3	TAPPING	XC	◆	◆	M 6.	6.	1.	10.	1.5

SNo.	TOOL	NOM-ϕ	#	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	DTR-DR	12.	◤	8.134	◆	◆	◆	90°	◆	59	0.09		
2	DRILL	5.1	D	5.1	12.	0.	100	DRIL T	2.55	25	0.08		
3	TAP	M6.	◤	6.	10.	TAP	◆	FIX	P1.	39	1.		

FIG	SHP	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANGLE	Q	R
1	ARC	25.	0.	10.	2	180.	0	0

UNo.	UNIT	COUNTER	RETURN	WK.No.	CONT.	NUM.	SHIFT
4	END		1				

- NOTE -

9 THREE-DIGIT G-FORMAT

9-1 Outline

The three-digit G-format is a format of expressing MAZATROL program data and other NC data. The various types of data within the NC unit are each assigned to a specific "three-digit G + address + data" set. Use of the data input/output functions based on the three-digit G-format allows the NC-stored data to be managed under the same environment as those of EIA/ISO programs.
Data that have been output to external units in the three-digit G-format can be edited using a personal computer, and if the edited data are restored into the NC unit, the corresponding original data within the NC unit will be auto-modified according to the required edition.

9-2 Detailed Description

1. MAZATROL program data

- Unit data have an assigned specific three-digit G-code for each unit.
- Tool sequence data succeed the block of the three-digit G-code of the unit data and are positioned between code G424, which denotes the beginning of the sequence data, and code G425, which denotes the end of the sequence data.
- The shape data, if present, succeed the block of the three-digit G-code of sequence data and are positioned between code G420, which denotes the beginning of the shape data, and code G421, which denotes the end of the shape data.
- The TPC data, if present, succeed the block of the three-digit G-code of the unit data and are positioned between code G422, which denotes the beginning of TPC data, and code G423, which denotes the end of TPC data.
- The order of output of the machining program data is predetermined. That is, unit data, TPC data (barrier information included), sequence data and shape data are output in that order. Do not change the order.
- Machining set-up information is keyed to the address that immediately succeeds code G426.
- Process layout information is keyed to the address that immediately succeeds code G427.
- The output data of machining programs may include additional lower digits which are not displayed on the screen. This is the case, for example, with the values of approach point or crossing point which are automatically calculated and internally used by the NC unit. Such data should not therefore be modified with respect to the displayed data.

Example:

```
SNo.   TOOL       NOM-ϕ        APRCH-1     APRCH-2
 1   END MILL     10.A          1.234       2.345
```

Displayed data

Output data

```
N1T15D10.S1  X&1.2345  Y&2.3455 ~
```

Output longer by one digit than displayed.

2. **Data on the TOOL OFFSET, TOOL DATA, TOOL FILE, PARAMETER, MACRO VARIABLE, CUTTING COND. and WORK OFFSET displays, etc.**

   - The code G10 is used to input/output the above data.

   The codes for data identification are listed up and described in detail on the following pages.

9-3 Three-Digit G-Format of MAZATROL Program

1. **Program number and program name**

   In the three-digit G-format input/output of MAZATROL programs, the number and name of a program is described in the following format:

(Without program name)

EOB	*Program No.* O99999999	(	M	G	3	–	1	4	1	)	EOB	*Description in 3-digit G-format*	EOB	EOR %

Identifier

(With program name)

EOB	*Program No.* O99999999	(	M	G	3	–	1	4	1	:	*Program name*	)	EOB	*Description in 3-digit G-format*	EOB	EOR %

Identifier

- Program number
  The program number is assigned following "O".
- Identifier
  The code "(MG3-141)" succeeding the program number identifies a MAZATROL program described in the three-digit G-format of the M640MT Pro.
- Program name
  The program name is assigned in the parentheses with the identifier separated by a colon.
  The maximum available number of characters is usually 32 for naming a program stored in the NC memory.
  An excess in characters will be given away.

2. Unit

- Common unit G300

UNo.	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	RPM	MPX	FIN-LENGTH	TR2-DIA	SIMUL	Program info.
U	( )	X	I	Z	C	S	D	H	J	L	K

- End unit G301

UNo.	COUNTER	RETURN	WK No.	CONT.	NUM.	SHIFT
U	C	D	W	E	K	I

- Sub-program unit G303

UNo.	WK No.	NUM.
U	W	L

- Turning manual program mode unit G313

UNo.	TOOL	Tool shape	NOM-ϕ/(Nominal Diameter/Size)	ID code	CHANGE-PT	GEAR	Turret	#
U	T (*1)	&T (*2)	D	S (*3)	L	M	K (*4)	Q (*5)

* For nominal diameter of taps and taps for turning, refer to "Tapping unit".

(*1)Tool name table

T:	Tool name	T:	Tool name	T:	Tool name
1	CTR-DR	11	BOR BAR	36	T-DRILL
2	DRILL	12	B-B BAR	37	T-TAP (M)
3	REAMER	13	CHAMFER	38	T-TAP (UN)
4	TAP (M)	14	FCE MILL	39	T-TAP (PT)
5	TAP (UN)	15	END MILL	40	T-TAP (PF)
6	TAP (PT)	16	OTHER	41	T-TAP (PS)
7	TAP (PF)	19	BAL EMIL	42	T-TAP (SPECIAL)
8	TAP (PS)	33	GENERAL	43	SPECIAL
9	TAP (OTHER)	34	GROOVE	44	TOL SENS
10	BCK FACE	35	THREAD		

(*2)Tool shape table

&T:	Tool shape
1	OUT OUTER DIAMETER
2	IN INNER DIAMETER
3	EDGE EDGE
4	IN INNER (BAK)
5	EDGE EDGE (BAK)
17	0001
18	0002
19	0003
20	0004
21	0005
22	0006
23	0007
24	0008
25	0009

(*3)Tool ID code

S:	ID code	S:	ID code
0	none		
1	A	–1	A
:	:	:	:
8	H	–8	H
9	J	–9	J
:	:	:	:
13	N	–13	N
14	P	–14	P
:	:	:	:
24	Z	–24	Z

(*4)Turret type table

K:	Type
0	Turret 1
1	Turret 2

(*5)Setting for lower turret

#:	Setting
1 - 99	Simultan. machining No.
101	BALANCE FEED 2
110	TURRET 2 POS. 1
111	TURRET 2 POS. 2

- Milling manual program mode unit G305

UNo.	TOOL	NOM-ϕ/(Nominal Diameter/Size)	ID code	#	CHANGE-PT	GEAR
U	T	D	S	Q	L	M
				110: TURRET 2 POS. 1		
				111: TURRET 2 POS. 2		

* For nominal diameter of taps and taps for turning, refer to "Tapping unit."

- M-code unit G302

UNo.	Control flag	#	M1	M2	M3	M4	M5	M6	M7	M8	M9	M10	M11	M12
U	A	K	MA	MB	MC	MD	ME	MF	MG	MH	MI	MJ	MK	ML
	Bit0=1:#4 secondary miscellaneous	0:TR1												
	Bit0=0: #4 M-code	1:TR2												
	Bit1=1: #8 secondary miscellaneous													
	Bit1=0: #8 M-code													
	Bit2=1: #12 secondary miscellaneous													
	Bit2=0: #12 M-code													

(*) M-codes (m) for the opposite side are distinguished by an addition of 10000 to the original code number.

- Coordinates measuring unit G304

UNo.	TOOL	Tool shape	NOM-ϕ	ID code	#
U	T	&T	D	S	W
					110: TURRET 2 POS. 1
					111: TURRET 2 POS. 2

Refer to "Turning manual program mode unit".

- Simultaneous unit G312

UNo.	SIMUL. No.	RPM
U	L	S

- Materials shape unit G307

UNo.	Shape
U	E
	1: OUT
	3: IN

- Workpiece transfer unit G309

UNo.	PAT.	HEAD	SPDL	PUSH	CHUCK	SETUP-No.
U	P	H	S	J	K	L
	0: None	<Case of CHUCK>	0: Spindle stop	0: With pressing	0: Chuck open	
	1: CHUCK	3: 1→2	1: Spindle forward	1: Without pressing	1: Chuck close	
	2: BAR	4: 2→1	2: Spindle reverse			
	3: MOVE	<Case of BAR>	3: Spindle orient			
		1: HEAD1	4: C-axis positioning			
		2: HEAD2	5: Spindle mov. keep			

- Head selection unit G310

UNo.	TYPE	HEAD	SPDL	TURRET
U	P	H	L	K
	0: None	0: None	0: THE OTHER SPDL SYNCRO	0: TR1
	1: SINGLE	1: HEAD1	1: THE OTHER SPDL STOP	1: TR2
	2: SYNCH.	2: HEAD2		
	3: CROSS			

- Workpiece measuring unit G308

UNo.	OFS	OFS-TOOL	Ofs. tool shape	Ofs. tool/ Nom. dia./size	Ofs. tool ID	Turret	COMP. DATA	SNS-TOOL	SNS-TOOL shape	Sensor Nom. dia.	Sensor ID code	#	OUTPUT
U	H	T	&T	D	I	P	J	K	L	R	S	W	Q
	0: YES					0: TR1	0: DIAMETER					110: TURRET 2 POS. 1	0: No output
	1: NO					1: TR2	1: WEAR COMP.					111: TURRET 2 POS. 2	1:Output to HDD
													2:Output to RS232C

- Tool measuring unit G311

UNo.	OFS	Tool	Tool shape	Nom.dia./size	ID code	TURRET	#	OUTPUT
U	H	T	&T	D	S	K	W	Q
	0: YES					0: TR1	110: TURRET 2 POS. 1	0: No output
	1: NO					1: TR2	111: TURRET 2 POS. 2	1:Output to HDD
								2:Output to RS232C

- Drilling unit G350

UNo.	MODE			POS-B	POS-C	DIA	DEPTH	CHMF
U	Q			Y	W	D	H	C
	65:ZC	69:XY	73:/Y					
	66:XC	70:XY	74:/Y					
	67:XC	71:/C						
	68:ZY	72:/C						

- Counterbore machining unit G351

UNo.	MODE	POS-B	POS-C	CB-DIA	CB-DEP	CHMF	BTM	DIA	DEPTH
U	Q	Y	W	&D	&H	C	F	D	H

- Inversed faced hole machining unit G352

UNo.	MODE	POS-B	POS-C	CB-DIA	CB-DEP	DIA	DEPTH	CHMF
U	Q	Y	W	&D	&H	D	H	C

- Reaming unit G353

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	PRE-REAM
U	Q	Y	W	D	H	C	A
							1: Drilling
							2: Boring
							3: End milling

- Tapping unit G354

UNo.	MODE	POS-B	POS-C	NOM-	MAJOR -ϕ	PITCH	TAP-DEP	CHMF
U	Q	Y	W	*	E	P	H	C

Tap screw type A:	
1	M
2	UNn
3	UN
4	PT
5	PF
6	PS
7	OTHER

Tap fraction B:	
1	1/2
2	1/4
3	1/8
4	1/16

Nominal diameter D:
Nominal diameter 2 V:

Example:

M10.	A1D10.
UNn 1-2	A2D1V2
UN 1H-2	A3D1V2B1
PT 2Q	A4D2B2

- Back boring unit G355

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	BTM	WAL	PRE-DIA	PRE-DEP	CHMF	WAL
U	Q	Y	W	D	H	I	J	&D	&H	C	&J

- Circular milling unit G356

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	TORNA.	BTM	PRE-DIA	CHMF	PITCH1	PITCH2
U	Q	Y	W	D	H	C	K	I	&D	&C	E	F
							0: CIRCUL					
							1: TORNADE					
							2: HIGH AC.					

- Counterbore-tapping unit G357

UNo.	MODE	POS-B	POS-C	NOM-	MAJOR-ϕ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM
U	Q	Y	W	*	E	P	H	C	&D	&H	&C	I

Refer to "Tapping unit".

- Through hole boring unit G358

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	WAL
U	Q	Y	W	D	H	C	J

- Non-through hole boring unit G359

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	BTM	WAL	PRE-DIA
U	Q	Y	W	D	H	C	I	J	E

- Stepped through hole boring unit G360

UNo.	MODE	POS-B	POS-C	CB-DIA	CB-DEP	CHMF	BTM	WAL	DIA	DEPTH	CHMF	WAL
U	Q	Y	W	&D	&H	&C	&I	&J	D	H	C	J

- Stepped non-through hole boring unit G361

UNo.	MODE	POS-B	POS-C	CB-DIA	CB-DEP	CHMF	BTM	WAL	PRE-DIA	DIA	DEPTH	CHMF	BTM	WAL
U	Q	Y	W	&D	&H	&C	&I	&J	E	D	H	C	I	J

- Central linear machining unit G362

UNo.	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A
U	Q	Y	W	Z	R	F	&Z

- Right-hand linear machining unit G363
- Left-hand linear machining unit G364
- Outside linear machining unit G365
- Inside linear machining unit G366

UNo.	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R
U	Q	Y	W	Z	R	F	&Z	&R

- Right-hand chamfering unit G367
- Left-hand chamfering unit G368
- Outside chamfering unit G369
- Inside chamfering unit G370

UNo.	MODE	POS-B	POS-C	INTER-Z	INTER-R	CHMF
U	Q	Y	W	I	J	C

- Face milling unit G371
- End milling-top unit G372

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	FIN-A
U	Q	Y	W	Z	I	&Z

- Pocket milling unit G374

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R	INTER-R	CHMF
U	Q	Y	W	Z	I	J	&Z	&R	K	C

- End milling-slot unit G377

UNo.	MODE	POS-B	POS-C	SRV-A	SLOT-WID	BTM	WAL	FIN-A	FIN-R	PAT.
U	Q	Y	W	Z	D	I	J	&Z	&R	F
										0: line
										1: arbitry

- Bar-materials machining unit G320

UNo.	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
U	E	X	Z	&X	&Z

0: None
1: OUT (Outside-diameter open type)
2: OUT (Outside-diameter middle type)
3: IN (Inside-diameter open type)
4: IN (Inside-diameter middle type)
5: FACE (Front-face open type)
6: FACE (Front-face middle type)
7: BACK (Back-face open type)
8: BACK (Back-face middle type)

- Copy-machining unit G321

UNo.	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	FIN-X	FIN-Z
U	E	X	Z	I	J	&X	&Z

Refer to "Bar-materials machining unit".

- Corner-machining unit G322

UNo.	PART	FIN-X	FIN-Z
U	E	&X	&Z

Refer to "Bar-materials machining unit".

- Facing unit G323

UNo.	PART	FIN-Z
U	E	&Z

Refer to "Bar-materials machining unit".

- Threading unit G324

UNo.	PART	CHAMF	LEAD	ANG	MULTI	HGT
U	E	C	K	D	R	H

0: None
1: 45 degrees
2: 60 degrees

Refer to "Bar-materials machinig unit".

- Grooving unit G325

UNo.	PART	PAT.	No.	PITCH	WIDTH	F.allowance (Grooving pattern #0 - #3) /Overshoot (Grooving pattern #4, #5)
U	E	I	K	F	J	Z

0: #0 (Right-angled or oblique)
1: #1 (Isosceles trapezoidal)
2: #2 (Right-tapered)
3: #3 (Left-tapered)
4: #4 (Right-corner cut-off)
5: #5 (Left-comer cut-off)

Refer to "Bar-materials machining unit".

- Turning drilling unit G326

UNo.	PART	DIA
U	E	D

Refer to "Bar-materials machining unit".

- Turning tapping unit G327

UNo.	PART	NOM-DIA	PITCH
U	E	*	F

Refer to "Bar-materials machining unit".
Refer to "Tapping unit".

3. Machining sequence

```
G420U_    Beginning of sequence data (U: unit No.)
N1
N2
 :        Sequence data
 :
G421      End of sequence data
```

- Manual program sequence
- Mill manual program mode sequence

SNo.	G	DATA-1 address	DATA-2 address	DATA-3 address	DATA-1	DATA-2	DATA-3	RADIUS/ VARIABLE	RPM	surf. vel.	FEED (/min)	FEED (/rev)	M-code	B-code	OFS
N	G	J	K	L	X (&X)*	Y (&Y)*	Z (&Z)*	R	S	V	F	E	M	B	H

66: B	87: W	89: Y
67: C	88: X	90 Z

(*) &X, &Y, &Z : An incremental data

- Coordinate measuring sequence

SNo.	PTN	X	Y	Z	C	R	D/L	K	Measuring dir.	Approach/Escape dir.
N	A	X	Y	Z	C	R	D	K	W	V
	3: Z FACE								0: CW	0: Z direction
	14: C FACE								1: CCW	1: X direction
	15: C GROOVE									
	16: C STEP									

- Sub-program sequence

SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
N	?	?	?	?	?	?

Set address and data as specified, e.g. "X123.456."
(if macro variable has been specified, set "X#100.")

- Materials shape sequence

SNo.	SHP	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
N	A	X	Z	&X	&Z	I
	1: LIN					
	2: TPR					
	3:					
	4:					
	5: CTR					

- Workpiece measuring sequence

SNo.	PTN	Type	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM–	BASE	Approach
N	A	P	X	Y	Z	&X	&Y	&Z	V	W	Q	L
	1: OUTER X	0: Both-side									0: SPT	0: Z dir.
	2: OUTER Y	1: Single-side									1: FPT	1: X dir.
	3: INNER X											
	4: INNER Y											
	5: X GRV											
	6: Y GRV											
	7: Z GRV											
	8: X WIDTH											
	9: Y WIDTH											
	10: Z WIDTH											
	11: +X STEP											
	12: –X STEP											
	13: +Y STEP											
	14: –Y STEP											
	15: +Z STEP											
	16: –Z STEP											
	17: IN WIDTH											
	18: IN GRV											

- External measurement

SNo.	PTN	COMP.DATA	MEASURING POINT	TARGET DATA	T LIM+	T LIM–
N	A	I	J	K	V	W
	19: EXT MIL	0: WEAR X				
	20: EXT TURN	1: WEAR Z				
		2: DIAMETER				

- Tool measuring sequence

SNo.	PTN	TOLERANCE X	TOLERANCE Z	TOOL EYE
N	A	V	W	Q
	2: TOOL EYE #1			0: Retract
	3: TOOL EYE #2			1: Not retract
	4: TOOL EYE #3			
	5: TOOL EYE #4			

- Point machining shape sequence ZC, XC, /C

FIG	SHP	Control flag	SPT-R/x	SPT-C/y	SPT-Z	SPT-Y	NUM.	ANGLE	Q	R
N	A	E	I/X	J/Y	Z	V	M	D	Q	R
	1: PT	Bit0=1: SPT-X input								
	2: LIN	Bit0=0: SPT-R input								
	3: SQR	Bit1=1: SPT-Y input								
	4: GRD	Bit1=0: SPT-th input								
	6: ARC									

- Point machining shape sequence ZY

FIG	SHP	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANGLE	P	Q	R
N	A	Z	Y	I	&Z	&Y	F	M	K	D	L	Q	R

- Point machining shape sequence XY

FIG	SHP	Control flag	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
N	A	E	I/X	J/Y	Z	&Z	&Y	F	M	K	D	L	Q	R

- Point machining shape sequence /Y

FIG	SHP	SHFT-Z	SHFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANGLE	P	Q	R
N	A	Z	I	X	Y	&Z	&Y	F	M	K	D	L	Q	R

- Linear/face (fixed) machining shape sequence ZC

FIG	SHP	P1Z/CZ	P1C/CC	P3Z/R	P3C	SPT-R	CNR	Type of shape (fixed/arbitrary)
N	A	Z	J	&Z	&J	I	R/C	B (0: fixed)
	16: SQR							
	17: CIR							

- Linear/face (fixed) machining shape sequence XC
- Linear/face (fixed) machining shape sequence XY

FIG	SHP	Control flag	P1Rx/CRx	P1Cy/CCy	P3Rx/R	P3Rx/R	SPT-Z	CNR	Type of shape
N	A	E	I/X	J/Y	&I/&X	&J/&Y	Z	R/C	B

- Linear/face (fixed) machining shape sequence ZY

FIG	SHP	P1Z/CZ	P1Y/CY	P3Z/R	P3Y	SPT-R	CNR	Type of shape
N	A	Z	Y	&Z	&Y	I	R/C	B

- Linear/face (fixed) machining shape sequence /Y

FIG	SHP	SHFT-Z	SHFT-R	P1X/CX	P1Y/CY	P3X/R	P3Y	CNR	Type of shape
N	A	Z	I	X	Y	&Z	&Y	R/C	B

- Linear/face (arbitrary) machining shape sequence ZC

FIG	SHP	Z	C	SPT-R	RADIUS/th	I	J	P	CNR	RGH/ Feedrate set flag
N	A	Z	J	I	&R/&C	V	W	L	R/C/K	E
	32: LINE							1: LEFT		
	33: CW							2: RIGHT		
	34: CCW							3: DOWN		
	35: FIG-SH							4: UP		
	36: CW-SH									
	37: CCW-SH									
	38: REP-EN									

RGH code/ Feedrate	SPT	Type of shape	?mark	AUTO SET
F	S	B (1: arbitrary)	Q	H

- Linear/face (arbitrary) machining shape sequence XC
- Linear/face (arbitrary) machining shape sequence XY

FIG	SHP	Control flag	R/x	C/y	SPT-Z	RADIUS/th	I	J	P	CNR	RGH/ Feedrate set flag
N	A	P	I/X	J/Y	Z	&R/&C	V/&X	W/&Y	L	R/C/K	E

RGH code/ Feedrate	SPT	Type of shape	?mark	AUTO SET
F	S	B	Q	H

- Linear/face (arbitrary) machining shape sequence ZY

FIG	SHP	Z	Y	SPT-R	RADIUS/th	I	J	P	CNR	RGH/ Feedrate set flag
N	A	Z	Y	I	&R/&C	V	W	L	R/C/K	E

RGH code/ Feedrate	SPT	Type of shape	?mark	AUTO SET
F	S	B	Q	H

- Linear/face (arbitrary) machining shape sequence /Y

FIG	SHP	SHIFT-Z	SHIFT-R	X	Y	RADIUS/th	I	J	P	CNR	RGH/ Feedrate set flag
N	A	Z	I	X	Y	&R/&C	V	W	L	R/C/K	E

RGH code/ Feedrate	SPT	Type of shape	?mark	AUTO SET
F	S	B	Q	H

- Bar-materials machining/Copy machining sequence

FIG	SHP	S-CNR	SPT-X	SPT-Z	SPT INTER PT	F-CNR/ $	FIN-X	FIN-Z	FIN INTER PT
N	A	C/R	X	Z	K	&C/&R/Q	&X	&Z	L
	0: None				0: None				0: None
	1: LIN				1: UP				1: UP
	2: TPR				2: DOWN				2: DOWN
	3:				3: LEFT				3: LEFT
	4:				4: RGT				4: RGT
	5: CTR								

RADIUS/th	FGH/ Feedrate set flag	RGH code/ Feedrate	INTER PT flag
I/J	E	F	H
	0: ROUGHNESS		bit0=1: SPT-X?
	1: FEEDRATE/rev		bit1=1: SPT-Z?
			bit2=1: FIN-X?
			bit3=1: FIN-Z?
			bit4=1: SPT-X CONT?
			bit5=1: SPT-Z CONT?
			bit6=1: FIN-X CONT?
			bit7=1: FIN-Z CONT?

- Corner-machining/Edge-machining unit sequence

FIG	SPT-X	SPT-Z	F-CNR/ $	FIN-X	FIN-Z	FGH/Feedrate set flag	RGH code/ Feedrate
N	X	Z	&C/&R/Q	&X	&Z	E	F

- Threading unit sequence

FIG	SPT-X	SPT-Z	FIN-X	FIN-Z
N	X	Z	&X	&Z

- Grooving unit sequence

FIG	S-CNR	SPT-X	SPT-Z	F-CNR	FPT-X	FPT-Z	ANGLE	RGH/ Feedrate flag	RGH code/ Feedrate	INTER PT flag
N	C/R	X	Z	&C/&R	&X	&Z	J	E	F	H

- Turning drilling and turning tapping sequence

FIG	SPT-Z	FPT-Z
N	Z	&Z

4. Tool sequence

`G424U_`	Beginning of tool sequence (U: unit No.)

`N1`
`N2`
⋮
⋮
	Tool sequence data

`G425`	End of tool sequence

- Point machining tool sequence

SNo.	TOOL	NOM-ϕ	ID code	#L	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
N	T	D	S	L	E	H	&E	&H	F	Z	I	J	MA	MB

Turret 2 retracting pos. L:
110: TURRET 2 POS. 1
111: TURRET 2 POS. 2

BOR BAR &E
0: CYCLE 1
1: CYCLE 2
2: CYCLE 3
TAP &E
0: TAP
1: PECK
2: PLANET

For nominal diameter of taps, refer to "Tapping unit".

Refer to "Turning manual program mode unit".

END MILL &H
0: CCW
1: CW
2: CCW
3: CW

CTR-DR F:
0: 90°
1: 118°
2: 60°

DRILL F:
1: DRIL
2: PCK1
3: PCK2
4: AUTO
5: PCK3
18: PCK1
19: PCK2
21: PCK3

REAMER Z:
–1: G00
0: G01
1 - 9999
CTR-DR Z:
0: CTR-DR
1: CHAMF

- Line machining tool sequence

SNo.	TOOL	NOM-ϕ	ID code	#L	APRCH-X	APRCH-Y	TYPE	AFD	DEP-A	C-SP	FR	M	M	R/F
N	T	D	S	L	X	Y/C	Q	F	Z	I	J	MA	MB	B
							7: CW	–1: G00						0: R
							8: CCW	0: G01						1: F
								0.1 - 9.9						

Refer to "Turning manual program mode unit".

Turret 2 retracting pos. L:
110: TURRET 2 POS. 1
111: TURRET 2 POS. 2

If the **[AUTO SET]** function has been used, set "X?." In this case, "&" will be added to the output data, e.g. "X&123.456".

- Face machining tool sequence

SNo.	TOOL	NOM	ID code	#L	APRCH-X	APRCH-Y	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M	R/F
N	T	D	S	L	X	Y/C	Q	F	Z	R	I	J	MA	MB	B

TYPE (Q)
1: 1BI
2: 2BI
3: 1UN
4: 2UN
5: 1BI-S
6: 2BI-S
7: CW
8: CCW

AFD (F)
−1: G00
0: G01
0.1 - 9.9

R/F (B)
0: R
1: F

Refer to "Turning manual program mode unit".

Turret 2 retracting pos. L:
110: TURRET 2 POS. 1
111: TURRET 2 POS. 2

If the **[AUTO SET]** function has been used, set "X?." In this case, "&" will be added to the output data, e.g. "X&123.456".

- Turning tool sequence

SNo.	TOOL	Tool shape	Nom. dia./ size	ID code	Turret	# (*1)	PAT. (*2)	DEP-1	DEP-2/ NUM.	DEP-3	No of cut passes
N	T	&T	D	S	K	L	Q	RA	RB	RC	E

FIN-X	FIN-Z	C-SP	FR	M	M	R/F
X	Y	I	J	MA	MB	F

Refer to "Turning manual program mode unit".

(*1) Setting for lower turret

1 - 99	Simultan. machining No.
101	BALANCE FEED 2
110	TURRET 2 POS. 1
111	TURRET 2 POS. 2

(*2) Machining patterns

Q:	BAR and CORNER units	THREAD unit	T.DRILL unit
0	Perpendicularly upward cutting cycle	#0 STANDARD (Standard pattern)	Drilling cycle (stop hole)
1	High-speed rough-machining cycle	#1 CONST.DEPTH (Constant-threading pattern)	Deep-hole drilling cycle (stop hole)
2	Inside-diameter deep-hole dividing cycle	#2 CONST.AREA (Area-constant pattern)	High-speed deep-hole drilling cycle (stop hole)
3		#0 STANDARD (zigzag threading)	Reaming cycle (stop-hole)
4		#1 CONST.DEPTH (zigzag threading)	Reserve
5		#2 CONST.DEPTH (zigzag threading)	Drilling cycle (through-hole)
6			Deep-hole drilling cycle (through-hole)
7			High-speed deep-hole drilling cycle (through-hole)
8			Reaming cycle (through-hole)

5. TPC

TPC data consists of two types: TPC data (parameter) section and relay-point section. The two types of data are identified as follows:

- The block next to the code G422 (Beginning of TPC data) is always handled as parameter data, and
- Subsequent blocks up to the code G423 (End of TPC data) are handled as relay-point data.

```
G422U_    Beginning of TPC data (U: unit No.)
  :
  :       TPC data
  :
G423      End of TPC data
```

- The block that succeeds G422U0 (beginning of the TPC data in unit No. 0) is handled as barrier information.

A. TPC data

	A	B	C	D	E	F	H	I	J	K	L	M	P	Q	R	S	Y	Z	&C	&D	&E	&X
DRILLING	U3	U4	U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92								
RGH CBOR	U3	U4	U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43						
RGH BCB	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19		D5	D40				
REAMING	U3	U4	U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	
TAPPING	U3	U4	U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92			K75	U62	U63	D32	D22	K21
BK-CBORE	U3	U4	U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	D33
CIRC. MIL	U3	U4	U5	U6	P17			D16	D17	D42	D91			D92	D19	K43						
CBOR-TAP	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	K75	U62	U63	D32	D22	K21
BORE T1	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	
BORE S1	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	
BORE T2	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	
BORE S2	U3		U5	U6	P17	D1	D3	D16	D17	D42	D91	D45	D46	D92	D19	K43	D18	D24	U47	D26	D28	
LINE CTR	U3		U5	U6	P17	U10	E7	K41	E95													
LINE RGT	U3		U5	U6	P17	U10	E7	K41	E95													
LINE LFT	U3		U5	U6	P17	U10	E7	K41	E95													
LINE OUT	U3		U5	U6	P17	U10	E7	K41	E95	E1	E5	E21										
LINE IN	U3		U5	U6	P17	U10	E7	K41	E95	E1	E5	E21										
CHMF RGT	U3		U5	U6	P17	U10		K41	E95				E8	E11								
CHMF LFT	U3		U5	U6	P17	U10		K41	E95				E8	E11								
CHMF OUT	U3		U5	U6	P17	U10		K41	E95	E1		E21	E8	E11								
CHMF IN	U3		U5	U6	P17	U10		K41	E95	E1		E21	E8	E11								
FCE MILL	U3		U5	U6	P17											E12	E15					
TOP EMIL	U3		U5	U6	P17		E7	K41								E13	E97					
STEP	U3		U5	U6	P17	U10	E7	K41		E1	E5	E21				E91	E98	E16				
POCKET	U3		U5	U6	P17	U10	E7	K41		E1	E5	E21			E18	E92						
PCKT MT	U3		U5	U6	P17	U10	E7	K41		E1	E5	E21			E18	E93						
PCKT VLY	U3		U5	U6	P17	U10	E7	K41		E1	E5	E21			E18	E94	E98					
SLOT	U3		U5	U6	P17		E7	K41				E21				E96	U53					
WPC MSR	U3	U4	U5	U6	P95	P20	U61	P122														
WORK MES	U3	U4	U5	U6	P95	U57	U58	U59														
TOOL MES	U3	U4	U5	U6	P95	U57	U58	U60														
TRANSFER						U11	U26	U27	U50													
BAR	U3	U4	U5	U6	P95	U33	U34	K1	K4	K5												
COPY	U3	U4	U5	U6	P95	K6																
CORNER	U3	U4	U5	U6	P95	U33	U34	K1														
FACING	U3	U4	U5		P95	U37	U43															
THREAD	U3	U4	U5	U6	P95	U7	U38	U39	K19													
T. GROOVE	U3	U4	U5	U6	P95	P24	U8	U9	U18	U41	U42											
T. DRILL	U3		U5		P95	U45	K11	K12	K18													
T. TAP	U3		U5		P95	K20	K21															

D1, E2 etc. denote parameter type.

- Relay point (Point/linear/face machining unit)

Relay point type	Relay point setting	X1	Y1	Z1	X2	Y2	Z2	X3	Y3	Z3
A	B	XA	YA	ZA	XB	YB	ZB	XC	YC	ZC
0: APPROACH	0: AUTO									
1: ESCAPE	1: MANU									

- TPC common data

R-tool chng. Position flag	R-tool chng. clearance on X-axis	R-tool chng. clearance on Z-axis	X-coord. of fix. pnt. for roughing	Z-coord. of fix. pnt. for roughing	Offset amount for roughing	F-tool chng. position flag	F-tool chng. clearance on X-axis	F-tool chng. clearance on Z-axis
A	B	C	D	E	F	H	I	J

X-coord. of fix. pnt. for finishing	Z-coord. of fix. pnt. for finishing	Offset amount for finishing	Tool chng. Position flag for low turret	Tool chng. clearance on X2-axis for low turret	Tool chng. clearance on Z2-axis for low turret	X2-coord. of fix. pnt. for low turret	Z2-coord. of fix. pnt. for low turret	Offset amount for low turret tool
K	L	M	P	Q	R	S	Y	Z

Unit	M-code 1	M-code 2	Gear No.	M-code 1 (No. 2 spindle)	M-code 2 (No. 2 spindle)	Gear No. (No. 2 spindle)
Workpiece transfer	MA	MB	MC	MD	ME	MF
Tool/Coordinate measuring	MA	MB	MC			
Workpiece measuring	MD	ME	MF			

- Relay point (turning unit)

Relay point setting	Relay point P1X	Relay point P1Z	Relay point P2X	Relay point P2Z	Relay point P3X	Relay point P3Z	TPCdata No.
B	C	D	E	F	H	I	J
0: AUTO							
1: MANU							

6. Machining set-up data (G426)

L	Data type	Data item and related address							
	Set-up data	Z-OFFSET	C-OFFSET						
0	HEAD1	Z	C						
1	HEAD2	Z	C						
	Set-up data	MES. INTERVAL					Parts COUNTER		
		WORK	TOOL	EXTERNAL	Z-OFFSET	C-OFFSET			
2		A	B	C	D	E	F		
	Set-up data	JAW No.	GRIP DIA.	QUILL EXT.	REF. TO T/S	CHUCK	TAIL BARRI.	TAILSTOCK	
3	HEAD1	H	I	K	M	N	P	Q	
4	HEAD2	H	I	K	M	N	P	Q	
	Transfer data	Z1	Z2	C1	C2	Z-OFFSET	C-OFFSET	Z2	TNo.
5	No. 1	A	B	C	D	E	F	J	K
6	No. 2	A	B	C	D	E	F	J	K
7	No. 3	A	B	C	D	E	F	J	K
8	No. 4	A	B	C	D	E	F	J	K
9	No. 5	A	B	C	D	E	F	J	K
10	No. 6	A	B	C	D	E	F	J	K
11	No. 7	A	B	C	D	E	F	J	K
12	No. 8	A	B	C	D	E	F	J	K
	NC tail position	TAIL POS.1	TAIL POS.2						
13		A	B						

7. Process layout data (G427)

UNo.	Unit ID code (*1)	PART/MODE/ PTN/RPM (*2)	SNo.	TOOL	Tool shape	NOM-ϕ/ (Nominal Diameter/Size)
U	C	Q	N	T	&T	D

ID code	R/F	Turret	#	Approach relay point setting	Escape relay point setting	TRS mode	WNo.	HEAD	Pattern
S	L	E	F	H	I	J	W	K	P
	0:R			0: AUTO	0: AUTO				
	1:F			1: MANU	1: MANU				

(*1)

ID	Unit	ID	Unit	ID	Unit
1	Common	37	Back boring	64	Central linear machining
4	End	38	Circular milling	65	Right-hand linear machining
5	Subprogram	39	Counterbore tapping	66	Left-hand linear machining
6	Milling manual program machining	40	Through hole boring	67	Outside linear machining
7	M-code	41	Non-through hole boring	68	Inside linear machining
8	Coordinate measurement	42	Stepped through hole boring	80	Right-hand chamfering
16	Materials shape	43	Stepped non-through hole boring	81	Left-hand chamfering
17	Workpiece measurement	48	Bar-materials machining	82	Outside chamfering
18	Transfer unit	49	Copy-machining	83	Inside chamfering
19	Head select unit	50	Corner-machining	96	Face milling
20	Tool measurement	51	Edge-machining	97	End milling-top
21	Simultaneous unit	52	Threading	98	End milling-step
32	Drilling	53	Grooving	99	Pocket milling
33	Counterbore machining	54	Drilling (turning mode)	100	Pocket milling-mountain
34	Inversed faced hole machining	55	Tapping (turning mode)	101	Pocket milling-valley
35	Reaming	56	Manual program machining	102	End milling-slot
36	Tapping				

(*2)

- Turning unit

PART	
Q	
0: None	5: FACE (Front-face open type)
1: OUT (Outside-diameter open type)	6: FACE (Front-face middle type)
2: OUT (Outside-diameter middle type)	7: BACK (Back-face open type)
3: IN (Inside-diameter open type)	8: BACK (Back-face middle type)
4: IN (Inside-diameter middle type)	

- Milling unit

MODE			
Q			
0: None	67: XC	70: XY	73: /Y
65: ZC	68: ZY	71: /C	74: /Y
66: XC	69: XY	72: /C	

- Workpiece measuring unit
- External measuring unit

PTN			
Q			
0: None	6: Y GROOVE	12: –X STEP	18: INNER GROOVE
1: OUTER X	7: Z GROOVE	13: +Y STEP	19: EXT MIL
2: OUTER Y	8: X WIDTH	14: –Y STEP	20: EXT TURN
3: INNER X	9: Y WIDTH	15: +Z STEP	
4: INNER Y	10: Z WIDTH	16: –Z STEP	
5: X GROOVE	11: +X STEP	17: INNER WIDTH	

- Tool measurement unit

PTN	
Q	
0: None	4: TOOL EYE #3
2: TOOL EYE #1	5: TOOL EYE #4
3: TOOL EYE #2	

9-4 Various Data Description Using G10

"G10" is normally used to express the other various data than program data, and address "L" that follows denotes the type of the data.

```
G10L_
 ↑
 └──────── Data type
```

1. Tool data

A. TOOL DATA 1

- Milling tool

```
G10L41H_T_C_D_S_I_E_P_F_J_K_(_)Q_X_R_A_N_W_
```

Head	TNo.	TOOL NAME	NOM-ϕ	ID CODE	DIRCTION	ACT-ϕ	FW/RW	EDG-ANG	MAX. ROT.	LENG COMP
H	T	C	D	S	I	E	P	F	J	K

MATERIAL	TIP POS	TAP TYPE	REC. FEED	BORDER	STATUS1	STATUS2
( )	Q	X	R	A	N	W

- Turning tool

```
G10L41H_T_C_&C_D_S_I_E_P_F_A_(_)Q_X_R_N_W_
```

Head	TNo.	TOOL NAME	SHAPE	NOM.	ID CODE	DIRECTION	ACT-ϕ	R/L	EDG-ANG	CUT ANGLE	MATERIAL
H	T	C	&C	D	S	I	E	P	F	A	( )

INDEX ANG	TAP TYPE	REC. FEED NOSE-R	STATUS1	STATUS2
Q	X	R	N	W

C:	TOOL
1	CTR-DR
2	DRILL
3	REAMER
4	TAP (M)
5	TAP (UN)
6	TAP (PT)
7	TAP (PF)
8	TAP (PS)
9	TAP (OTHER)
10	BCK FACE
11	BOR BAR
12	B-B BAR
13	CHAMFER
14	FCE MILL
15	END MILL
16	OTHER
17	CHIP VAC
18	TOL SENS
19	BAL EMIL
33	GENERAL
34	GROOVE
35	THREAD
36	T.DRILL
37	T. TAP (M)
38	T. TAP (UN)
39	T. TAP (PT)
40	T. TAP (PF)
41	T. TAP (PS)
42	T. TAP (OTHER)
43	SPECIAL

&C:	TOOL (Section to be machined)
1	OUT OUTER DIAMETER
2	IN INNER DIAMETER
3	EDG EDGE
4	IN INNER (BAK)
5	EDG EDGE (BAK)
17	0001
18	0002
19	0003
20	0004
21	0005
22	0006
23	0007
24	0008
25	0009

X	Tap function
	0: FLOAT
	1: FIX

S:	ID code
0	
1	A
:	:
8	H
9	J
:	:
13	N
14	P
:	:
24	Z
–1	A
:	:
–8	H
–9	J
:	:
–13	N
–14	P
:	:
–24	Z

P:	R/L, FW/RW
0	None
1	←RIGHT
2	→RIGHT
3	←LEFT
4	→LEFT
5	←
6	→

Nominal diameter of tap and turning tap

A:	Tap screw type
1	M
2	UNn
3	UN
4	PT
5	PF
6	PS
7	OTHER

B:	Tap fraction
1	1/2
2	1/4
3	1/8
4	1/16

Nominal diameter D:
Nominal diameter 2V :

Example

M10.	A1D10.
UNn 1-2	A2D1V2
UN 1H-2	A3D1V2B1
PT 2Q	A4D2B2

B. TOOL DATA 2

```
G10L42H_T_A_B_X_Y_Z_&X_&Y_&Z_C_D_E_I_J_K_M_N_P_Q_F_R_S_V_(_)
```

HEAD	TNo.	TOOL SET X	TOOL SET Z	WEAR COMP X	WEAR COMP Y	WEAR COMP Z	MAX WEAR X	MAX WEAR Y	MAX WEAR Z
H	T	A	B	X	Y	Z	&X	&Y	&Z

EASY COMP X	EASY COMP Y	EASY COMP Z	CONS. COMP X	CONS. COMP Y	CONS. COMP Z	LIFE (TIME)	USED (TIME)	LIFE (NUMBE)	USED (NUMBE)
C	D	E	I	J	K	M	N	P	Q

HOLDER No.	WIDTH	GROUP No.	OFS No.	COMMENT/ IDNo.
F	R	S	V	( )

2. Tool offset

A. Geometric offset

G10L14P_N_X_Z_R_Y_Q_

Tool offset No.	Spindle No.	OFFSET X	OFFSET Z	NOSE R	OFFSET Y	DIRCTN
P	N	X	Z	R	Y	Q
	1: Spindle 1					
	2: Spindle 2					

B. Wear compensation

G10L15P_N_X_Z_R_Y_

Tool offset No.	Spindle No.	COMP. X	COMP. Z	NOSE R	COMP. Y
P	N	X	Z	R	Y
	1: Spindle 1				
	2: Spindle 2				

3. Tool file

A. End mill, face mill and ball-end mill

G10L49P_C_D_S_(_)R_H_A_

Tool file No.	TOOL	NOM-ϕ	ID code	MAT.	DEPTH	TEETH.	ANG.
P	C	D	S	( )	R	H	A

B. Chamfering cutter

G10L49P_C_D_S_E_(_)A_

Refer to "Tool data".

Tool file No.	TOOL	NOM-ϕ	ID code	MIN-ϕ	MAT.	ANG.
P	C	D	S	E	( )	A

4. Chuck jaw data

G10L69P_M_A_B_C_D_E_N_

JAW No.	TYPE	Dimension A	Dimension B	Dimension C	Dimension D	Dimension E	Jaw code/ NAME
P	M	A	B	C	D	E	N/( )
	1: OD JAW						
	2: ID JAW						

5. Cutting condition

A. Cutting condition (WORK MAT)

G10L52P_(_)

Material No.	WORK MAT
P	( )

B. Milling cutting condition (TOOL MAT, C-SP, FR)

G10L_P_S_F_(_)

DRILL	CTR-DR	REAMER	TAP	BOR BAR	B-B BAR	BCK FACE	CHAMFER	END MILL	FCE MILL	BAL EMIL	OTHER
G10L53	G10L54	G10L55	G10L56	G10L57	G10L58	G10L59	G10L60	G10L61	G10L62	G10L63	L10L64

No.	SPD	FR	TOOL MAT.
P	S	F	( )

C. Cutting condition for turning

G10L65P_A_B_C_D_

No.	R-SPD	F-SPD	R-FEED	R-DEP.
P	A	B	C	D

D. Cutting condition for turning (WORK MATERIAL PERCENTAGE)

G10L66P_A_B_C_D_

No.	R-SPD	F-SPD	R-FEED	R-DEP.
P	A	B	C	D

E. Cutting condition for turning (TOOL MATERIAL PERCENTAGE)

G10L67P_A_B_C_D_(_)

No.	R-SPD	F-SPD	R-FEED	R-DEP.	TOOL MAT
P	A	B	C	D	( )

F. Cutting condition parameter

G10L68CA_Z_ Cutting condition parameter CA1 to CA108
G10L68CB_Z_ Cutting condition parameter CB to CB108
G10L68CC_Z_ Cutting condition parameter CC to CC108

Parameter address	Setting
CA/CB/CC	Z

6. Workpiece offset

A. Standard

```
G10L2 P_N_X_Z_C_Y_B_
```

Workpiece offset No.	Spindle No.	1st axis	2nd axis	3rd axis	4th axis	5th axis
P	N	X	Z	C	Y	B
0: COORD SHIFT	1: Spindle 1					
1: G54	2: Spindle 2					
:						
6: G59						

7. Parameter

A. User parameter

G10L50	User parameter setting start
P1Z11111111	11111111 is set to parameter P1 (example).
P2Z00000000	
:	
G11	User parameter setting end

Format	Parameter	Format	Parameter
P_Z_	P1 to P126	D_Z_	D to D108
K_Z_	K to K112 (Note)	E_Z_	E to E108
U_Z_	U to U108		

Note: For K81 to K96
K_Z_ N_ ; Z: Code N: Number

B. Machine parameter

G10L51	Machine parameter setting start
A1N1Z38000	38000 is set to parameter A1 (1st axis) (example).
A2N1Z0	
:	
G11	Machine parameter setting end

Format	Parameter	Format	Parameter
A_N_Z_	A1 to A44 (N: Axis No. 1 to 10)	RL_Z_	RL1 to RL48
B_N_Z_	B1 to B306 (N: Axis No. 1 to 2)	RB_Z_	RB1 to RB88
BS_N_Z_	BS1 to BS44 (N: Axis No. 1 to 10)	SV_N_Z_	SV1 to SV96 (N: Axis No. 1 to 10)
TIM_Z_	TIM1 to TIM80	SP_N_Z_	SP1 to SP384 (N: Axis No. 1 to 2)
RS_Z_	RS1 to RS80	O_Z_	O1 to O112

C. Pitch error compensation

```
G10L47          Pitch error compensation setting start
N1A1B0C1D0E0
#1N1Z-300       –300 is set to #1 (1st axis) (example).
:
G11             Pitch error compensation setting end
```

Axis No.	H1	H2	H3	H4	H5	Variable No.	Compensation value
NP,	A	B	C	D	E	#	Z
11: No.1						1: #1	
:						:	
10: No.10						4096: #4096	

Note: Parameter data should not be changed until you have fully understood the meaning of the respective parameter.

8. Tool life

G10L30 P_Q_N_T_F_A_B_C_D_S_

Group No.	Priority No.	Spindle No.	TNo.	OFFSET No.	LIFE (TIME)	LIFE (CNT)	USED (TIME)	USED (CNT)	Status
P	Q	N	T	F	A	B	C	D	S
		1: Spindle 1							
		2: Spindle 2							

9. Macro variable

A. Common variable

G10L44N1#100=100 100 is set to #100 of the spindle 1 (N: Spindle No. 1 or 2).

B. Common variable (for checking)

G10L45N1#100=100 100 is set to #100 of the spindle 1 (N: Spindle No. 1 or 2).

10. Maintenance check

A. Regular check item

G10L70P_T_C_Y_M_D_( )

Check No.	Target time	Current time	Year	Month	Day	Check item
P	T	C	Y	M	D	( )

B. Long-term check item

G10L70P_( )

Check No.	Check item
P	( )

10 SHAPE CORRECTION FUNCTION

10-1 Function General

10-1-1 Shape Correction Function

Machining at rapid feed will generally degrade a shape accuracy in proportion to the feed rate. For example, delayed follow-up of smoothing circuit and servo system occurs in circular cutting, making a smaller workpiece than commanded circle radius.

Our CNC unit allows conventional shape errors to be reduced by using shape correction function. The shape correction function is composed of the following six functions:

1. Pre-interpolation acceleration/deceleration
2. Optimal corner deceleration
3. Arc-cutting velocity clamping
4. Precise vector interpolation
5. Feedforward control
6. Soft acceleration/deceleration control

These functions are valid for EIA/ISO programs as well, and the following description includes information on EIA/ISO programs.

In circular cutting, shape errors from smoothing circuit can be reduced by pre-interpolation acceleration/deceleration and those from delayed follow-up of servo system can be reduced by feedforward control.

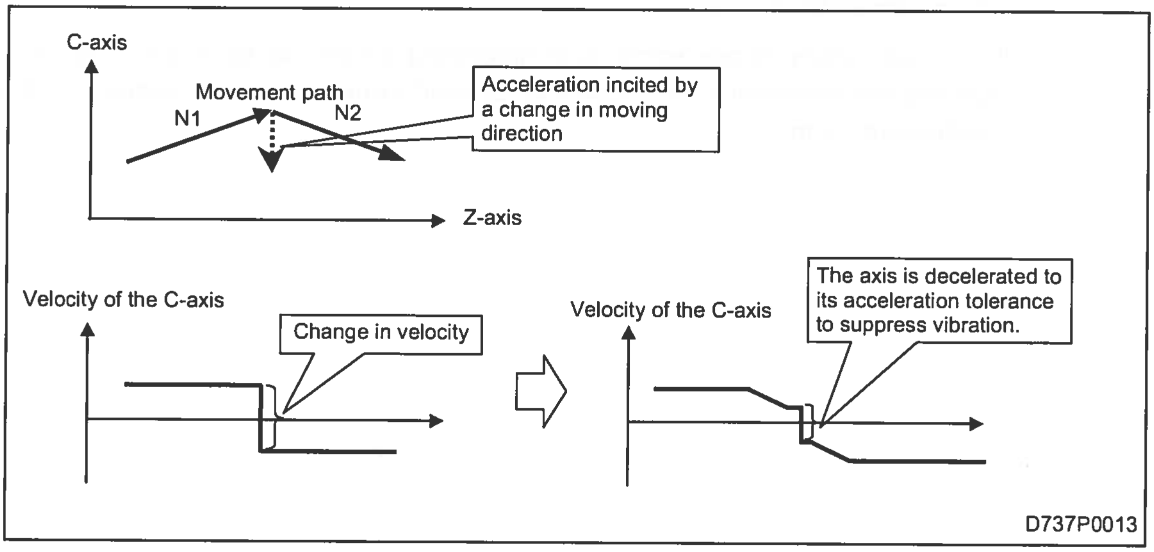
10-1-2 Rotational-axis shape-correcting function

Even when the move commands that cover the rotational axes are issued, the rotational-axis shape-correcting function will improve an error status due to the accuracy of the control system, by achieving the appropriate acceleration according to shape.

When the rotational-axis shape-correcting function is valid, control for the optimum acceleration is conducted instead of the conventional optimum corner deceleration described in Subsection 10-1-1.

	When rotational-axis shape correction is invalid	When rotational-axis shape correction is valid
1	Pre-interpolation acceleration/deceleration	Pre-interpolation acceleration/deceleration
2	Optimal corner deceleration	Optimal acceleration control
3	Arc-cutting velocity clamping	Arc-cutting velocity clamping
4	Precise vector compensation	Precise vector compensation
5	Feedforward control	Feedforward control
6	Soft acceleration/deceleration control	Soft acceleration/deceleration control

The acceleration of each axis that occurs at corners is optimized by this function so that the acceleration tolerance specified by the required parameters is not overstepped.



10-2 Function Details

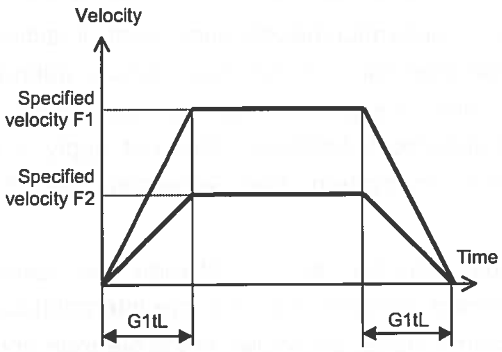
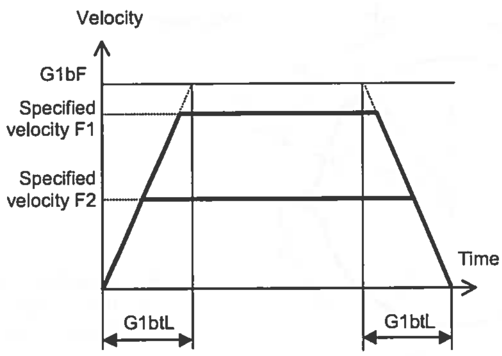
10-2-1 Pre-interpolation acceleration/deceleration

When acceleration/deceleration control is performed on move commands to minimize shocks during the movement start and stop of the machine, conventional acceleration/deceleration control make the corners of block connections round or cause a path error with respect to the command-specified shape.

In this pre-interpolation acceleration/deceleration control mode, acceleration/deceleration control is provided prior to interpolation in order to solve the above problems. Machining based on the programmed-path can be executed with the maximum achievable accuracy by using this pre-interpolation acceleration/deceleration control function.

Acceleration/deceleration time can also be reduced since acceleration/deceleration control is conducted to achieve a fixed gradient.

1. **Basic patterns of acceleration/deceleration control based on linear interpolation commands**

	Acceleration/deceleration waveform patterns	Description
G64 mode (Post-interpolation acceleration/ deceleration)		1) Because of a fixed time-constant type of acceleration/deceleration, as the specified velocity is smaller, the rising and falling edges of the acceleration/deceleration pattern take a more gentle shape. 2) An independent acceleration/deceleration time constant can be set for each axis. However, an error in the programmed path will occur unless the time constants for each axis are set to the same value. G1tL (**BS4**): Linear acceleration/deceleration time constant
G61.1 mode (Pre-interpolation acceleration/ deceleration)		1) Because of a fixed gradient type of acceleration/deceleration, as the specified velocity is smaller, the acceleration/deceleration time is reduced more significantly. 2) One value common to each axis can be set as the acceleration/deceleration time constant. G1bF (B**233**): Target velocity G1btL (**B146**): Acceleration/deceleration time up to the target velocity 3) Use parameter **B147** to set the maximum feed rate in G61.1 mode. **B147**: Clamping velocity for cutting in G61.1 mode

2. Pre-interpolation acceleration/deceleration in rapid-feed G0 command mode

Pre-interpolation acceleration/deceleration is also valid for rapid-feed machine movement during the use of this control function. In that case, acceleration/deceleration control is conducted so that the acceleration of each axis does not exceed the value of parameter **A1** (rapid feed rate)/parameter **BS3** (rapid feed time constant).

When the setting of the G0 interpolation type (parameter **P9**, bit 6) is "Non-interpolation type", even if the command mode is G61.1, post-interpolation acceleration/deceleration will occur.

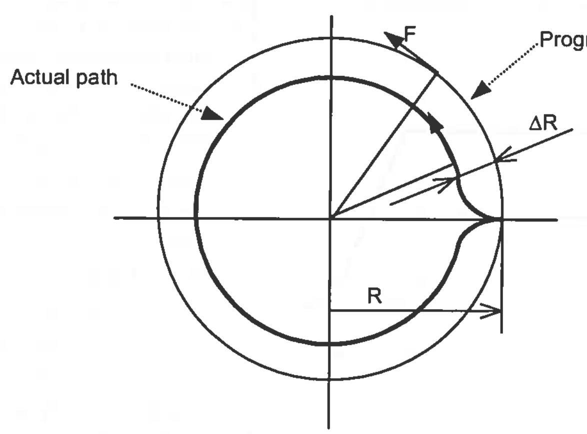
G0 interpolation type	G64 mode	G61.1 mode
Non-interpolation type	Post-interpolation acceleration/ deceleration with fixed time constant	Post-interpolation acceleration/ deceleration with fixed time constant
Interpolation type	Post-interpolation acceleration/ deceleration with fixed time constant	Pre-interpolation Acceleration/ deceleration with fixed gradient

3. Path control based on arc interpolation commands

Under the conventional post-interpolation acceleration/deceleration control scheme, when an arc interpolation command is used, the radius of the arc will be reduced since the actual path sent from the NC unit to the servo will be inner than the programmed path according to the particular amount of dwell of the acceleration/deceleration-use smoothing circuit inside the NC unit.

Under the pre-interpolation acceleration/deceleration control scheme, since interpolation is preceded by acceleration/deceleration control, path errors during acceleration/deceleration processing are removed to draw the programmed arc path more accurately than under the conventional scheme. This statement, however, does not apply to a delay in tracking during position loop control in the servo system. See Subsection 10-2-5 Feed-forward control, for further details.

Comparisons on arc radius reduction errors between the conventional post-interpolation acceleration/deceleration control scheme and the pre-interpolation acceleration/deceleration control scheme using this control mode, are shown in the diagram and table below.



R: Specified radius (mm)
ΔR: Radius error (mm)
F: Cutting feed rate (mm/min)

D737P0014

Theoretically, arc radius reduction error ΔR is calculated as shown in the table below.

G64 mode (Post-interpolation acceleration/deceleration control)	Linear acceleration/deceleration $$\Delta R = \frac{1}{2R}\left(\frac{1}{12}Ts^2 + Tp^2\right)\left(\frac{F}{60}\right)^2$$
G61.1 mode (Pre-interpolation acceleration/deceleration control)	Linear acceleration/deceleration $$\Delta R = \frac{1}{2R}\{Tp^2 \cdot (1-Kf^2)\}\left(\frac{F}{60}\right)^2$$ - The adoption of the pre-interpolation acceleration/deceleration control scheme allows the arc radius reduction error to be reduced since the "Ts" term can be ignored. - The "Tp" section can be offset by assigning 1 to "Kf." See Subsection 10-2-5, Feed-forward control, for further details.

Ts : Acceleration/deceleration time constant within the NC memory (sec)
Tp : Position loop time constant of the servo system (sec)
Kf : Feed-forward control coefficient

10-2-2 Optimal velocity control

The optimal corner deceleration function described in Item 1 below, and the optimal corner acceleration control function described in Item 2 are available as the optimal velocity control function applied to block-to-block connections. When the rotational-axis shape-correcting function is valid, the desired function of the above two can be selected using the following user parameter:

Parameter **P78** = 0: Optimal corner acceleration/deceleration control is applied
= 1: Optimal corner deceleration is applied

Optimal corner deceleration is applied automatically when the rotational-axis shape-correcting function is invalid.

1. Optimal corner deceleration

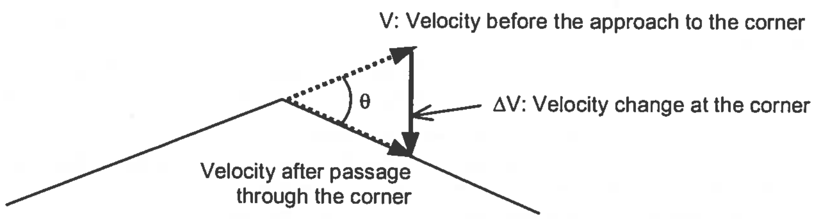
Machining with high edge accuracy can be implemented by calculating the angles of block-to-block connections and performing the acceleration/deceleration control to ensure passage through each corner at the optimal velocity.

During the approach to a corner, the optimal velocity for that corner (namely, the optimal corner velocity) is calculated from the angle with respect to the next block, and after the tool has decelerated beforehand and then passed through the corner, the tool accelerates to the specified velocity.

When blocks are connected smoothly, corner deceleration does not operate. In this case, the criteria for judging whether the connections are smooth can be specified in parameter **B107** (corner deceleration angle).

A. When the corner angle is greater than the setting of parameter B107 (corner deceleration angle)

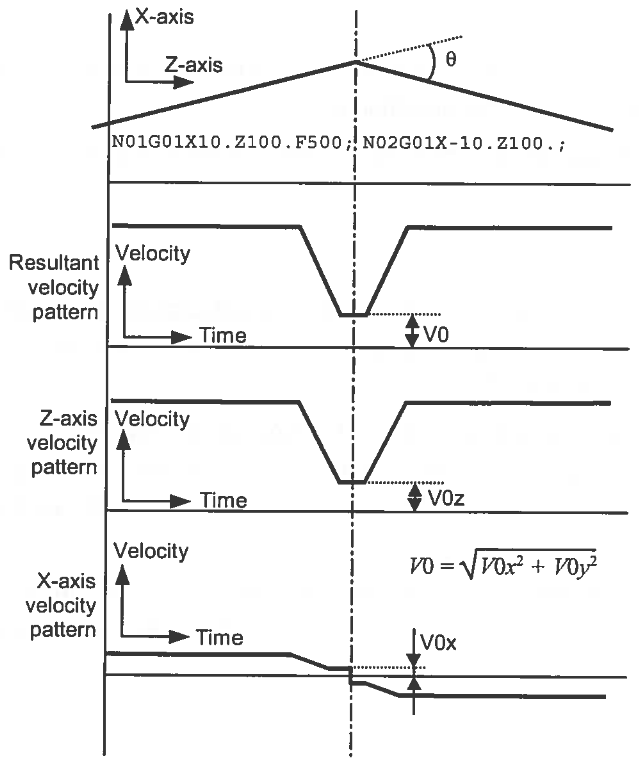
When the tool passes through the corner at a certain velocity of V, acceleration ΔV will occur according to the particular change in the traveling direction.



D737P0015

Corner velocity V is controlled so that ΔV decreases below the pre-interpolation acceleration determined by the setting of the parameter (G1bF: **B233**, G1btL: **B146**).

The velocity patterns in this case are shown below.



The optimal corner velocity V0 is calculated from the pre-interpolation acceleration/deceleration time constant (determined by the related parameters) and corner angle (external angle) θ.

<u>How to calculate V0</u>

$$V0=\frac{\Delta V}{\sqrt{2\cdot(1-\cos\theta)}}$$

$$\Delta V=\frac{G1bF}{G1btl}\times\Delta t$$

Δt: Interpolation cycle time
ΔV: Acceleration

For further reduction in corner velocity Vo (namely, for further improvement of edge accuracy), V0 can be reduced by adjusting the accuracy coefficients within the machining program. See Section 10-3 for further details.

D737P0016

B. Detailed description

1. If the block-to-block angle is greater than or equal to the angle specified in parameter **B107**, that section is judged to be a corner and the tool decelerates.
2. If the setting of parameter **B107** is 0, the corner deceleration angle is judged to be 5°.
3. If the setting of parameter **B107** is more than 90, the corner deceleration angle is judged to be 30°.

C. Precautions

Although the setting range of parameter **B107** (corner deceleration angle) is from 0 to 89°, be careful since, if too great an angle value is set, unusual machine vibration may occur or the as-machined surface may be rough.

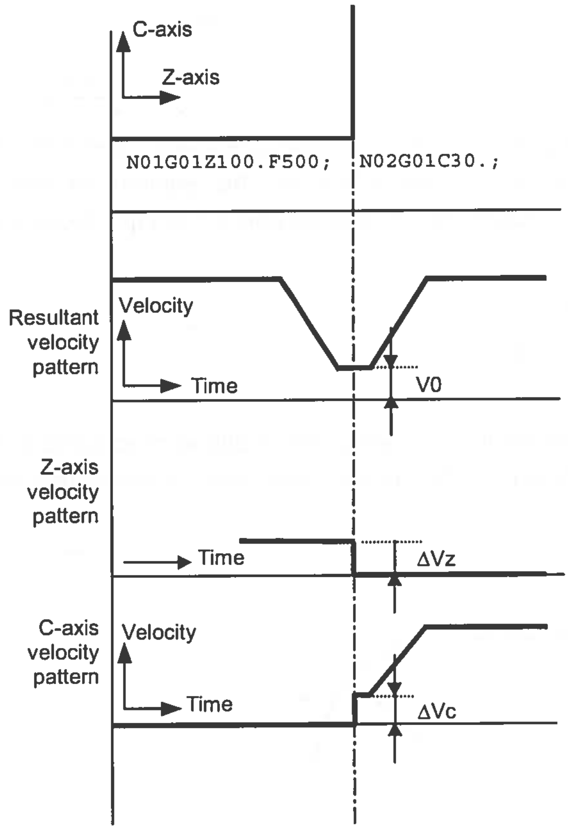
2. Optimal corner acceleration control

Machining with minimum machine vibration and high edge accuracy, can be implemented by evaluating for each axis the acceleration occurring at block-to-block connections, and then performing the acceleration/deceleration control function to ensure passage through each such connection at the optimal velocity.

The optimal deceleration velocity at which the acceleration occurring at each block-to-block connection stays below the maximum allowable acceleration specified by the required parameters is calculated and after the tool has decelerated beforehand and then passed through the corner, the tool accelerates to the specified velocity.

When blocks are connected smoothly, corner deceleration does not operate.

A. Linear interpolation using the Z-axis and the C-axis



Deceleration velocity Vo is controlled so that the acceleration ΔVz of the Z-axis and the acceleration ΔVc of the C-axis at block-to-block connections do not exceed the maximum allowable acceleration Amax_z and Amax_c determined by the settings of the required parameters (**A31**, **A32**, and **A33**), respectively.

Specify corner passage velocity V0 and the maximum allowable acceleration Amax of each axis as follows:

$$V0=\sqrt{\left(\frac{A\max_x}{\Delta Vx}\right)^2+\left(\frac{A\max_z}{\Delta Vz}\right)^2+\left(\frac{A\max_c}{\Delta Vc}\right)^2+\cdots+\left(\frac{A\max_n}{\Delta V_n}\right)^2}$$

$$A\,max=\frac{A31}{A32}\times\frac{1}{60\times1000}$$

Parameter **A31**: Target velocity
Parameter **A32**: Time constant

For further reduction in corner velocity V0 (namely, for further improvement of edge accuracy), V0 can be reduced by adjusting the accuracy coefficients within the machining program. See Section 10-3, for further details.

D737P0017

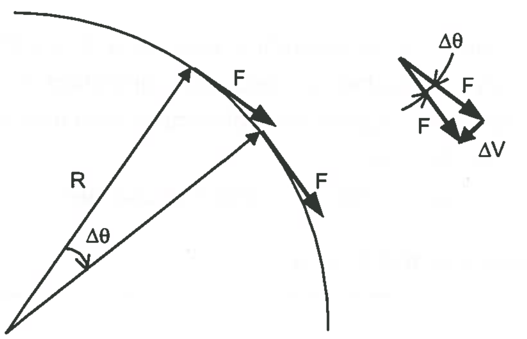
B. Method of setting parameters A31 and A32

Set parameters **A31** and **A32** to the rapid feed rate (**A1**) and G0 time constant (**BS3**).

To adjust the maximum allowable acceleration for some reason (for example, the rigidity of the machine varies from axis to axis), the maximum allowable acceleration during corner passage can be regulated by adjusting the setting of **A32**.

10-2-3 Arc-cutting velocity clamping

During arc interpolation, even during movement at a constant velocity, acceleration occurs since the traveling direction always changes. When the radius of the arc is great enough for the specified velocity, control is conducted in strict accordance with the specified velocity. When the arc radius is relatively small, velocity is clamped so that the acceleration occurring will not exceed the maximum pre-interpolation acceleration calculated from the required parameters. This allows arc cutting at the appropriate feed rate according to the particular radius of the arc.



F: Specified velocity
R: Specified arc radius
Δθ: Angle change per interpolation unit
ΔV: Velocity change per interpolation unit

The tool is fed at such an arc-cutting velocity F' that does not cause ΔV to exceed the maximum pre-interpolation acceleration ΔV'.

$$F' \leq \sqrt{R \times \Delta V' \times 60 \times 1000} \text{ (mm/min)}$$

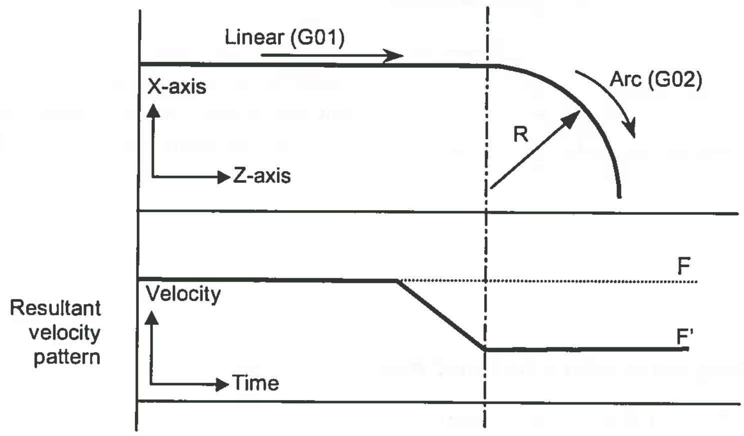
$$\Delta V' = \frac{G1bF \ (mm/\min)}{G1btL \ (m\sec)}$$

D737P0018

If the above expression for F' is assigned to F in the following equation denoting the maximum theoretical arc radius reduction error ΔR described in the section of pre-interpolation acceleration/deceleration, specified radius R will be deleted and ΔR will not depend on R.

$$\Delta R \leq \frac{1}{2R}\left\{Tp^2 \cdot \left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$$
$$\leq \frac{1}{2}\left\{Tp^2 \cdot \left(1 - Kf^2\right)\right\} \times \frac{\Delta V' \times 1000}{60}$$

In other words, the use of arc commands in this control mode allows machining to be executed always within a theoretically fixed radius reduction error range, irrespective of specified velocity F or specified radius R.



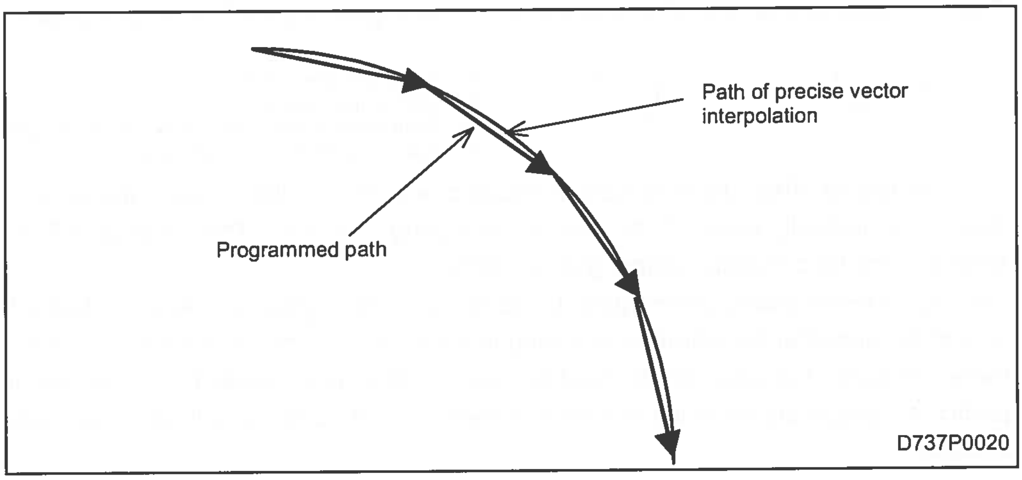
D737P0019

[For further reduction in arc-clamping velocity]

For further reduction in arc-clamping velocity (namely, for further improvement of roundness), the arc-clamping velocity can be reduced by adjusting the accuracy coefficients within the machining program. See Section 10-3, for further details.

10-2-4 Precise vector interpolation

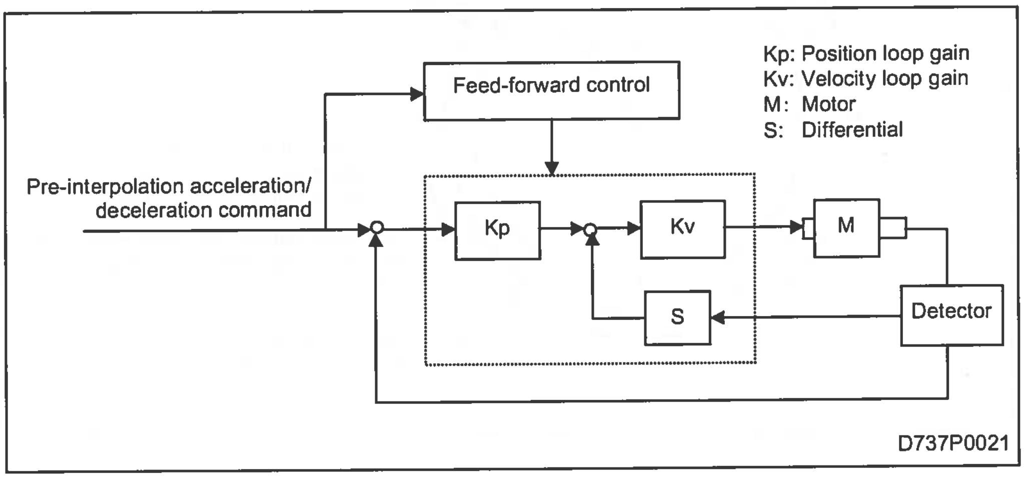
When microsegment commands are issued, if the block-to-block angle is very small and the connections are gentle (in the case that optimal corner deceleration is not applied), even smoother interpolation can be provided using the precise vector interpolation function.



D737P0020

10-2-5 Feed-forward control

The steady-velocity error rate during position loop control of the servo system can be significantly reduced using the feed-forward control function.



D737P0021

A. Reduction of arc radius decrements by feed-forward control

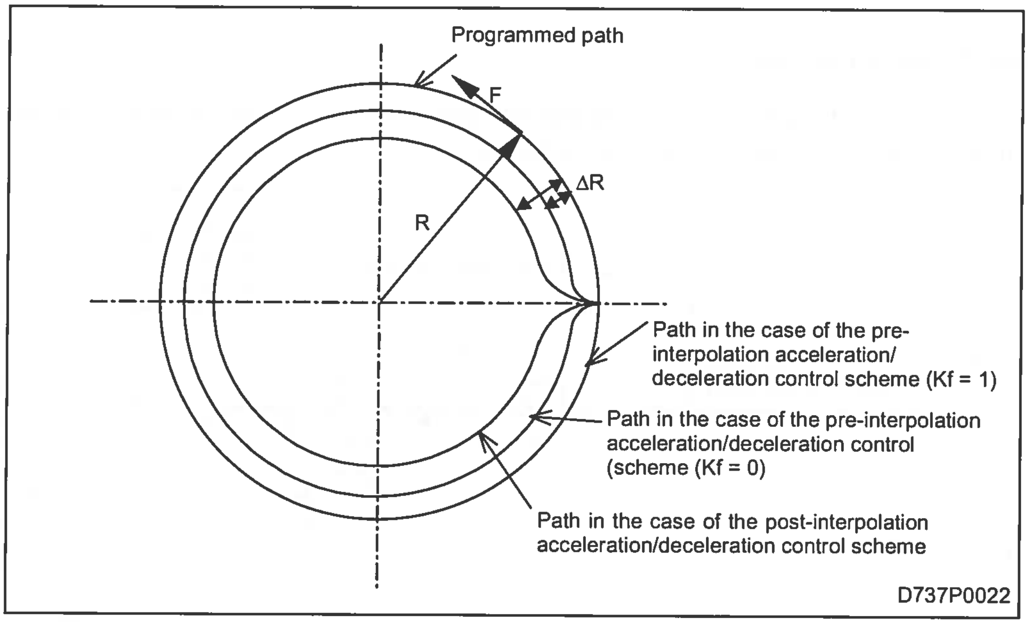
The feed-forward control function allows arc radius decrements to be significantly reduced by combining this control function and the pre-interpolation acceleration/deceleration control scheme described in Subsection 10-2-1.

The arc radius decrement ΔR in this control mode is given by the following equation:

$$\Delta R = \frac{1}{2R}\{Tp^2 \cdot (1-Kf^2)\}\left(\frac{F}{60}\right)^2$$

R: Specified radius (mm)
F: Cutting feed rate (mm/min)
Tp: Position loop time constant for the servo system (sec)
Kf: Feed-forward control coefficient

It is possible to offset delay factors associated with the position loop in the servo system, and thus to theoretically reset ΔR to zero, by assigning 1 to “Kf”. (The value of “Kf” is 1 when the setting of the feed-forward control gain is 100%.)
Too high a feed-forward control gain, however, generates great acceleration during the start and stop of acceleration/deceleration, resulting in unusual machine vibration and/or in overshoot. For these reasons, the appropriate feed-forward control gain needs to be set according to the particular specifications of the machine system so that stable acceleration/deceleration can be achieved.



B. Restrictions

In case of unusual machine vibration, it is necessary to reduce either “Kf” or the acceleration/deceleration time constant or to adjust the servo system. Acceleration/deceleration needs to be stabilized by conducting the three types of adjustments. To suppress machine vibration, “soft acceleration/deceleration control” described below also becomes a valid method.

10-2-6 Soft acceleration/deceleration control

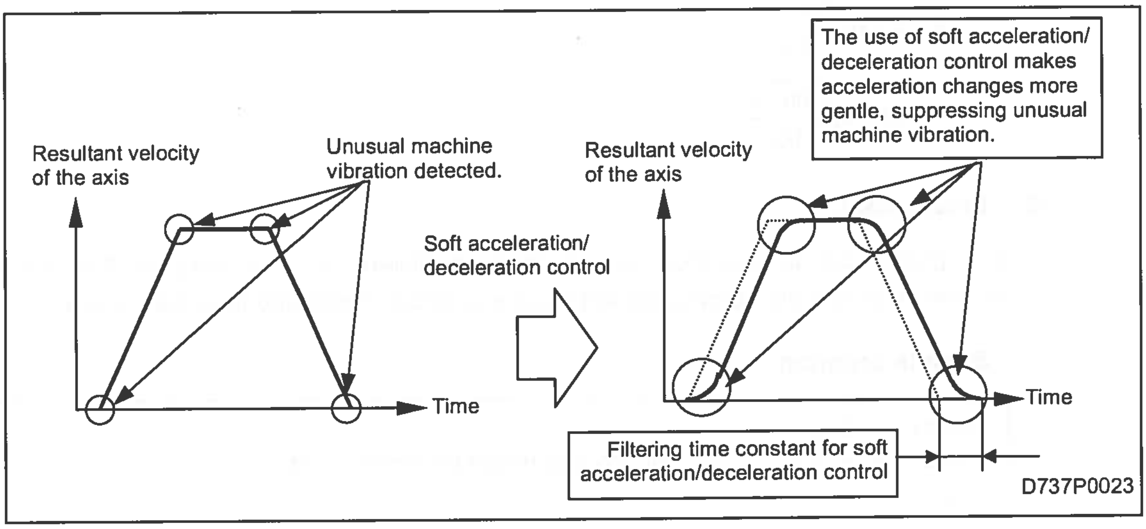
As described up to now in this document, in G61.1 mode, a delay in NC internal operation, associated with the smoothing circuit, is minimized by pre-interpolation acceleration/deceleration control, and a delay relating to the position loop in the servo system is minimized by feed-forward control. For special machine specifications, however, there are the cases that sufficient adjustments cannot be performed with these acceleration/deceleration control parameters or feed-forward control gains alone.

For example, the following cases apply:

1) When the feed-forward control gain is set to a trifle high level to obtain the desired accuracy, unusual machine vibration cannot be suppressed unless an extremely large value is assigned as the acceleration/deceleration time constant.

2) Although the time constant has been set to its permissible minimum value, accuracy does not stay within the permissible range since the feed-forward gain cannot be increased because of unusual machine vibration occurring.

In these cases, although a slight delay occurs, using "soft acceleration/deceleration control" allows unusual machine vibration to be suppressed more effectively than making the gradient gentle by increasing the acceleration/deceleration time constant.



A. Detailed description

Assign either of the following values to parameter **B112** to specify the filtering length for soft acceleration/deceleration control:

0: No filtering
1: 14.2 msec
2: 28.4 msec
4: 56.8 msec
8: 113.6 msec

If a value other than the above is assigned, the filtering length for soft acceleration/deceleration control will be 14.2 msec.

10-3 Accuracy Coefficients (Valid only for EIA programs)

When corner- and arc-cutting commands are issued using the "optimal velocity control" function described in Subsection 10-2-2 or the "arc-cutting velocity clamping" function described in Subsection 10-2-3, automatic deceleration occurs to maintain accuracy. However, accuracy has its limits if the feed-forward control gain cannot be increased.
"Accuracy coefficients" must therefore be programmed for further improved accuracy. The use of these coefficients allows more highly accurate machining to be achieved by reducing velocity during the execution of the corner- and arc-cutting commands.

A. Detailed description

Use program command ",Kk" to set accuracy coefficient "k" (this value can be specified only in EIA programs).

Setting range of "k": –32768 to +99 (1% to 32868%)

This accuracy coefficient is maintained until the mode has been changed to G64. Once this coefficient has been programmed, the tool decelerates during subsequent corner and arc cutting. The corner deceleration velocity V0 and arc radius clamping velocity Vc in G61.1 mode will be changed to the following V0' and Vc' values, respectively:

$$V0' = V0 \times \frac{100-k}{100}$$

$$Vc' = Vc \times \sqrt{\frac{100-k}{100}}$$

B. Precautions

If a plus value is specified as accuracy coefficient "k", the program that includes corner deceleration and arc commands will require a longer machining time than usual.

C. Sample program

```
G0X100.Z100.;
G61.1G1F2000; ....  (This control mode becomes valid.)
U10.W30.;
U5.W30.;
U-5.W30.,K10; ..... (The accuracy coefficient is maintained in the following blocks.)
U-10.W10.;
U-30.W5.;
G64; .............. (This control mode becomes invalid.)
```

10-4 Programming Description

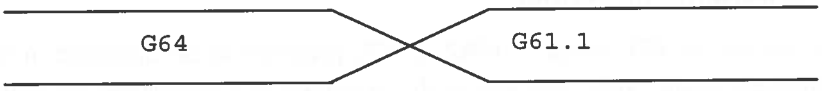
10-4-1 EIA programming command format

Use the following programming command format to make the shape correction mode valid. The G61.1 code can also be used independently:

```
G61.1(Xx Yy Zz Ff);
```

x: X-axis coordinate
y: Y-axis coordinate
z: Z-axis coordinate
f: Feed rate

1. This control mode becomes valid from the move command block that follows the G61.1 command block. However, “non-interpolation” G0 commands are executed in the same manner as in the G64 mode.



2. Use either of the following G-code commands to cancel this control mode. That is, use an other G-code of group 13 to perform the cancellation:

   - G61 (Exact Stop Check mode)
   - G62 (Automatic Corner Override)
   - G64 (Cutting mode)

3. This function is an option. If G61.1 is specified without this option, **708 INCORRECT G-CODE** will be displayed as a program error.

4. Feed rate F is clamped with the parameterized [pre-interpolation cutting-feed clamping velocity (**B147**)] as its upper limit.
   Parameter **B147** specifies the clamping velocity for resultant axes. Therefore, be careful since an independent axis may exceed the automatic rapid feed rate specified in parameter **A1**.

5. For the method of velocity clamping in arc command mode, see the description of “arc-cutting velocity clamping” in Subsection 10-2-3.

A. Sample program

```
G0X100.Z100.;
M200;
G61.1G1F2000; ....   (This control mode becomes valid.)
U10.W30.;
U5.W30.;
U-5.W30.,K10;.....   (The accuracy coefficient is maintained in the following blocks.)
U-10.W10.;
U-30.W5.;
G64;.................   (This control mode becomes invalid.)
M202;
```

B. Precautions

1. During subprogram call, the destination program also inherits the mode. If a different mode has been specified in the destination program, however, this mode will govern.
   The mode will be inherited, even after control has been returned to the main program.
2. Do not include the M200 or M202 commands in the G61.1 mode.
3. Simultaneous interpolation with the cross-machining control axis cannot be provided during G61.1 mode. Simultaneous interpolation with the cross-machining control axis is performed with this control mode being temporarily cancelled.

   Example: Cancel the shape correction mode to perform machining operations with the upper turret and the spindle No. 2 or with the lower turret and the spindle No. 1.

10-4-2 MAZATROL programming format

The use of parameter **P76** allows a MAZATROL program to be operated in this control mode (shape correction mode). Also, the use of the specified M-code allows the selection of whether this control mode is to be made valid/invalid only for a specific tool sequence. This control mode is an option. If this option is not provided, shape correction will be regarded as invalid during operation.

Parameter **P76** = 1: Operation of the MAZATROL program with this control mode valid
= 0: Operation of the MAZATROL program with this control mode invalid

To make this control mode valid/invalid only for a specific tool sequence, enter the M-code value that has been specified in parameter **U67/U68**.

Parameter **U67**: M-code value for making this control mode valid only for the specified tool sequence (121 if 0 is entered)

Parameter **U68**: M-code value for making this control mode invalid only for the specified tool sequence (122 if 0 is entered)

When either of the above two M-codes is included in the tool sequence, this control mode will become valid/invalid after a tool change, and that state will be maintained until the next tool sequence is executed. Neither M-code is transferred to PLC.

A. Sample program

If parameter **P76** = 0 and parameter **U67** = 0:

```
UNo.    UNIT
       LINE RGT

SNo.     TOOL      NOM-ϕ   APRCH-1   APRCH-2              M     M
 R1    END MILL     12
 F2    END MILL      8                                   121

UNo.    UNIT
       POCKET

SNo.     TOOL      NOM-ϕ   APRCH-1   APRCH-2              M     M
 R1    END MILL     20
 F2    END MILL     10                                   121
  3    CHAMFER      40
```

B. Precautions

1. This control mode is temporarily canceled for the turning unit.
2. When the rotational-axis shape-correcting function is invalid, this control mode is temporarily canceled for the ZC, XC, or /C mode (including subsidiary C-axis machining) of the milling unit. In this case, it is ineffective even if the shape correction 'valid' specifying M-code corresponding to parameter **U67** is specified in the tool sequence.
   When the rotational-axis shape-correcting function is valid, this control mode is temporarily canceled only for the ZC, XC, or /C mode that uses a subsidiary C-axis.
3. During subprogram call, the destination program also inherits the mode. If a different mode has been specified in the destination program, however, this mode will govern. After control has been returned to the main program, operation obeys the status of the main program.
4. Accuracy coefficients cannot be specified.

10-4-3 Status change and status display

1. Suspension from outside

Status		G61.1
Initial status	When power is turned on	×
RESET command	M02, M30	×
RESET operations	NC reset key, External resetting	×
Block suspension	Mode change (Automatic/manual), Feed hold	O
Block stop	Single block	O
Emergency stop	Emergency stop button, External emergency stop	O
NC alarm	Servo alarm	O
OT	Hardware overtravel	O
Emergency stop release	Emergency stop button, External emergency stop	×

O: No change ×: Canceled

2. Temporary suspension using a program command

In the following cases, this control mode is temporarily canceled. Modal display on the **POSITION** display is also reset to G64:

1. During rapid feed of the non-interpolation type, this control mode is temporarily canceled.
2. During synchronous tapping, this control mode is temporarily canceled.
3. During measurement (skipping), this control mode is temporarily canceled.
4. During constant surface velocity control, this control mode is temporarily canceled.
5. During threading, this control mode is temporarily canceled.
6. When the rotational-axis shape-correcting function is invalid, during cylindrical interpolation (G7.1) or polar coordinate interpolation (G12.1), this control mode is temporarily canceled.

3. Modal display

In this control mode, "G61.1" is displayed as modal display on the **POSITION** display. During temporary suspension that uses the program command data shown above, "G64" is displayed.

The following modal displays are made during MAZATROL-programmed machining:

- "G64" for the turning unit
- "G61.1" for the tool sequence that includes M121
- "G64" for the tool sequence that includes M122

10-5 Parameters

Related parameter list

<table>
<tr><th colspan="3">Item</th><th>Address</th><th>Unit</th><th>Range</th><th>Contents</th></tr>
<tr><td rowspan="8">Machine parameter</td><td>1</td><td>Pre-interpolation acceleration/ deceleration: Target velocity</td><td>B233</td><td>mm/min</td><td>1 to 999999</td><td>Target velocity for pre-interpolation acceleration/deceleration</td></tr>
<tr><td>2</td><td>Pre-interpolation acceleration/ deceleration: Acceleration/ deceleration time constant</td><td>B146</td><td>msec</td><td>1 to 5000</td><td>Time up to the arrival at the target velocity for pre-interpolation acceleration/deceleration</td></tr>
<tr><td>3</td><td>Pre-interpolation cutting feed clamping velocity</td><td>B147</td><td>mm/min
0.1 inch/min</td><td>0 to 65535
(0: Stops the axis during cutting feed.)</td><td>Cutting feed clamping velocity in G61.1 mode
For normal operating conditions, specify the clamping velocity for the Yt-axis which bears the heaviest load among all axes.</td></tr>
<tr><td>4</td><td>Corner deceleration angle</td><td>B107</td><td>Degree</td><td>0 to 89
If 0 is entered, the angle will be regarded as 5°
If 90 or more is entered, the angle will be regarded as 30°.</td><td>Optimal corner deceleration:
Corner deceleration functions only for angles greater than the entered angle.</td></tr>
<tr><td>5</td><td>Soft acceleration/ deceleration filtering</td><td>B112</td><td>—</td><td>0: No filtering
1: 7.1 msec
2: 14.2 msec
4: 28.4 msec
8: 56.8 msec
Other: 7.1 ms</td><td>Filtering that makes the pattern specified for pre-interpolation acceleration/deceleration gentle</td></tr>
<tr><td>6</td><td>Feed-forward control gain</td><td>A17</td><td>%</td><td>0 to 99</td><td>Feed-forward control gain</td></tr>
<tr><td>7</td><td>Deceleration calculation: Target velocity</td><td>A31</td><td>mm/min</td><td>1 to 999999</td><td rowspan="2">Optimal corner acceleration/deceleration control:
Specify the maximum allowable acceleration value (A31/A32) for each axis that occurs between blocks. Each axis decelerates to a velocity at which the specified acceleration does not exceed its maximum allowable value.</td></tr>
<tr><td>8</td><td>Deceleration calculation: Acceleration/ deceleration time constant</td><td>A32</td><td>msec</td><td>1 to 5000</td></tr>
</table>

Item			Address	Unit	Range	Contents
User parameter	9	Whether to make this control mode valid/invalid in the MAZATROL program	**P76**	—	0/1	0: Makes this control mode invalid in the MAZATROL program. 1: Makes this control mode valid in the MAZATROL program.
	10	Optimal corner acceleration control/ optimal corner deceleration selection	**P78**	—	0/1	Select the type of optimal velocity control for block-to-block connections when the rotational-axis shape-correcting function is valid. 0: Optimal corner acceleration control 1: Optimal corner deceleration
	11	Shape correction valid M-code value during MAZATROL programming	**U67**	—	0 to 65535	The value that has been specified in the parameter is output as an M-code, and this M-code is used to make shape correction valid. If 0 has been specified, however, M121 will be used.
	12	Shape correction invalid M-code value during MAZATROL programming	**U68**	—	0 to 65535	The value that has been specified in the parameter is output as an M-code, and this M-code is used to make shape correction invalid. If 0 has been specified, however, M122 will be used.

The above parameters immediately become valid, provided that NC is reset.

- NOTE -

YAMAZAKI MAZAK CORPORATION
1 Norifune, Oguchi-cho, Niwa-gun,
Aichi-pref., Japan
Phone: Oguchi (0587)95-1131
Facsimile: 0587-95-2717

MAZAK CORPORATION
8025 Production Drive,
Florence, KY41042
U.S.A.
Phone: 859-342-1700
Facsimile: 859-342-1865

YAMAZAKI MAZAK EUROPE, N.V.
Research Park, Grauwmeer 7
3001 Leuven,
BELGIUM
Phone: 16-39-1611
Facsimile: 16-40-0196

YAMAZAKI MAZAK DEUTSCHLAND GmbH
Esslinger Strasse 4-6
D-73037 Goeppingen
GERMANY
Phone: 7161-6750
Facsimile: 7161-675274

YAMAZAKI MAZAK U.K. LTD.
Badgeworth Drive Worcester
WR49NF
UNITED KINGDOM
Phone: 1905-755755
Facsimile: 1905-755001

YAMAZAKI MAZAK SINGAPORE PTE., LTD.
21, Joo Koon Circle,
Jurong, Singapore 629053
Phone: 6862-1131
Facsimile: 6861-9284

URL: www.mazak.com

Printed in Japan