Return to Main Menu

PROGRAMMING MANUAL

for

MAZATROL MATRIX

(For INTEGREX IV)

MAZATROL Program

MANUAL No. : H740PA0031E

Serial No. :

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

IMPORTANT NOTICE

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

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

10. 2006

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, Virtual Machining, and other functions, if provided.
- Make sure that the appropriate feed rate and rotational speed are designated for the particular machining requirements. Always understand that since the maximum usable feed rate and rotational speed are determined by the specifications of the tool to be used, those of the workpiece to be machined, and various other factors, actual capabilities differ from the machine specifications listed in this manual. If an inappropriate feed rate or rotational speed is designated, the workpiece or the tool may abruptly move out from the machine.
- Before executing correction functions, fully check that the direction and amount of correction are correct. Unexpected operation of the machine will result if a correction function is executed without its thorough understanding.
- Parameters are set to the optimum standard machining conditions prior to shipping of the machine from the factory. In principle, these settings should not be modified. If it becomes absolutely necessary to modify the settings, perform modifications only after thoroughly understanding the functions of the corresponding parameters. Modifications usually affect any program. Unexpected operation of the machine will result if the settings are modified without a thorough understanding.

Remarks on the cutting conditions recommended by the NC

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.
- The system of G-code and M-code commands differs, especially for turning, between the machines of INTEGREX e-Series and the other turning machines.
  Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

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

- For the machines of INTEGREX e-Series, programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA program. However, for example, when the B-axis is rotated through 180 degrees around the Y-axis to implement machining with the turning spindle No. 2, the plus side of the X-axis in the programmed coordinate system faces downward and if the program is created ignoring this fact, the resulting movement of the tool to unexpected positions may incite collisions.
  To create the program with the plus side of the X-axis oriented in an upward direction, use the mirror function of the WPC shift unit or the mirror imaging function of G-code command (G50.1, G51.1).
- After modifying the tool data specified in the program, be sure to perform the tool path check function, the Virtual Machining function, and other functions, and confirm that the program operates properly. The modification of tool data may cause even a field-proven machining program to change in operational status.
  If the user operates the machine without being aware of any changes in program status, interference with the workpiece could arise from unexpected operation.
  For example, if the cutting edge of the tool during the start of automatic operation is present inside the clearance-including blank (unmachined workpiece) specified in the common unit of the MAZATROL program, care is required since the tool will directly move from that position to the approach point because of no obstructions being judged to be present on this path.
  For this reason, before starting automatic operation, make sure that the cutting edge of the tool during the start of automatic operation is present outside the clearance-including workpiece specified in the common unit of the MAZATROL program.

CAUTION

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

Operations

WARNING

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

CAUTION

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

BEFORE USING THE NC UNIT

Limited Warranty

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

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

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

Operating Environment

1. Ambient temperature

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

2. Relative humidity

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

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

Keeping the Backup Data

Note: Do not attempt to delete or modify the data stored in the following folder.
Recovery Data Storage Folder: D:\MazakBackUp

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

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

CONTENTS

Page

1 MAZATROL PROGRAM CONFIGURATION

1-1 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
      - End milling-step
      - Pocket milling
      - Pocket milling-mountain
      - Pocket milling-valley
      - End milling slot
  - Turning unit
    - Bar-materials machining
    - Copy-machining
    - Corner-machining
    - Facing
    - Threading
    - Grooving
    - Drilling (turning)
    - Tapping (turning)
    - Mill-turning
- Manual program machining unit
- End unit
- M-code unit
- Subprogram unit
- Coordinate measuring unit
- Workpiece measuring unit
- Tool measuring unit
- Head selection unit
- Workpiece transfer unit
- Process end unit
- Simultaneous machining unit
- Two-workpiece machining unit
- Add-in MAZATROL unit

Data to be set in the units listed above is classified into the following four 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

```
A  UNo. MAT.  OD-MAX   ID-MIN   LENGTH   WORK FACE   ATC MODE  RPM   LTUR DIA
    0    FC     70.       0.      97.        2          0     3000

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

B  SNo.TOOL NOM-ϕ  No.  #  HOLE-ϕ HOLE-DEP PRE-DIA PRE-DEP RGH   DEPTH C-SP    FR   M  M  M
    1 CTR-DR  12.  A        10.      ◆          ◆       ◆      90.   SPOT   25    0.1
    2  DRILL  10.           10.     20.         0.      100  DRIL T   5.    63    0.1

C  FIG PTN   SPT-R/x  SPT-C/y  SPT-Z  SPT-Y    NUM.    ANGLE   Q     R
    1   PT     0.        0.      0.     0.       ◆        ◆     ◆     0

A  UNo. UNIT MODE    POS-B  POS-C SRV-A SLOT-WID  BTM   WAL FIN-A FIN-R PAT.
    2    SLOT  ZY      ◆       90.   10.    20.     4     4    0.     0.   0

B  SNo. TOOL   NOM-ϕ  No.  #  APRCH-1 APRCH-2 TYPE  AFD   DEP-A  DEP-R C-SP  FR    M  M  M
    F1 END MILL  20. A           ?       ?     CW   G01     ◆      ◆    120  0.13

C  FIG PTN    SPT-R     Z       Y       R/th    I   J   P  CNR  RGH
    1  LINE    25.      20.     20.
    2  LINE     ◆       20.    –20.
```

A: Unit data

B: Tool sequence data

C: Shape sequence data

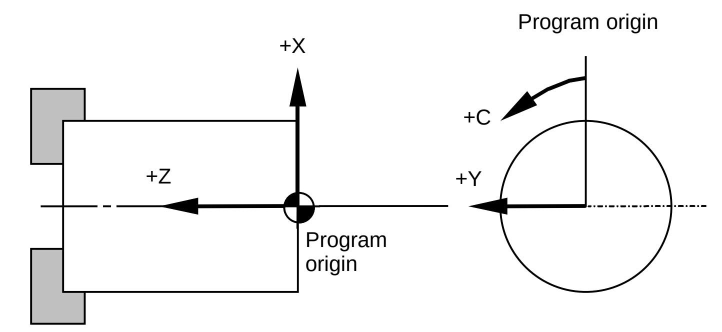
Specific details and setting procedures of each data are described in Chapter 3. Here (Chapter 1), you should understand what types of units and data constitute a program.

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.

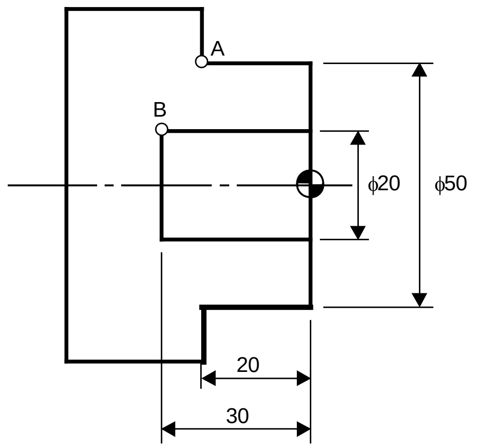
For MAZATROL programs, the following coordinate system is used to define machining patterns:



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 manual program machining units (MANL PRG) 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 chapter 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.

3-1 Procedure for Program Creation

(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	BARRIER INFORM.	WPC MSR	TOOL PATH	PROCESS CONTROL	PROGRAM LAYOUT	HELP	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” refers to a number assigned to each program to distinguish one program from another. A combination of up to 32 alphanumeric characters: 0 to 9 and A to Z, including the symbols “_”, “.”, “+” and “–”, can be used for a work number.

Note 1: If a work number is composed of figures alone, it should be a natural number between 1 and 99999999.

Note 2: A program name should not begin with a dot (.).

- 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    ATC MODE      RPM    LTUR DIA
 0
```

Cursor

This line denotes the common unit.

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

- See Section 3-2, "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 → C → A → B → C in order.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING	MANUAL PROGRAM			END	SHAPE CHECK	>>>	··· A
			M M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPIECE SHAPE	>>>	··· B
SELECT HEAD	TRANSFER WORKPICE	PROCESS END					SIMUL.	2 WORKPC MODE	>>>	··· C

(6) From the menus A, B and C, 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   ATC MOCE    RPM    LTUR DIA

 0   CBN STL  100.    0.     100.      2.          0       3000     120.

UNo.  UNIT   PART  POS-B  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 1000 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 200 lines of shape sequence data can be registered per unit.

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:

Turning
- BAR unit
- CPY unit
- CORNER unit
- FACING unit
- THREAD unit
- T. GROOVE unit
- T. DRILL unit
- T. TAP unit
- MILLTURN unit

Other units
- MMS unit
- WORK MES unit
- TOOL MES unit
- TRANSFER unit

Milling
- 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
- STEP unit
- POCKET unit
- PCKT MT unit
- PCKT VLY unit
- SLOT unit

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

3-2 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.

3-2-1 Setting unit data (common data)

UNo.	MAT.	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM	LOW TURR
0	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]

[1] MAT

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

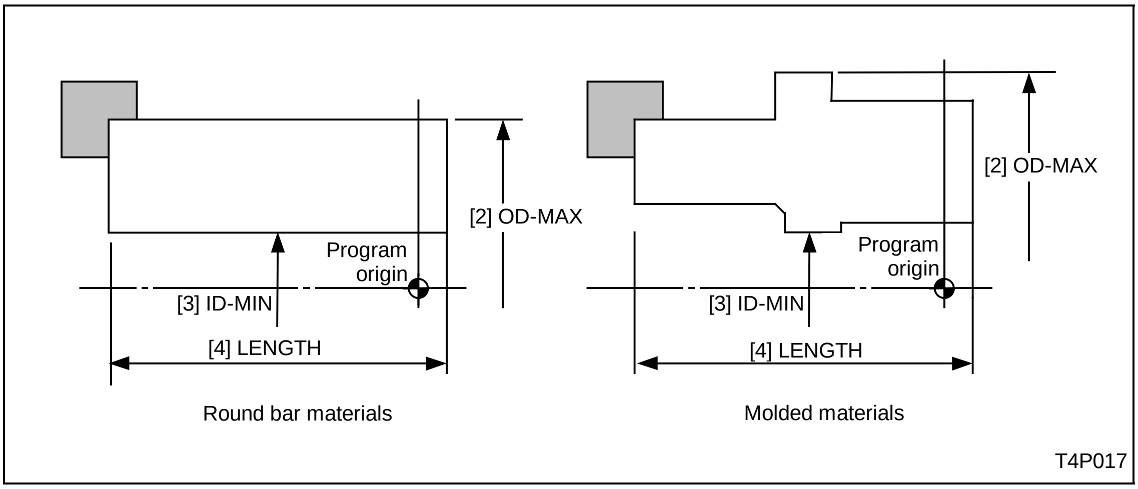
CST IRN	DUCT IRN	CBN STL	ALY STL	STNLESS	ALUMINUM	L.C.STL	AL CAST		

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 - PERCENTAGE** 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

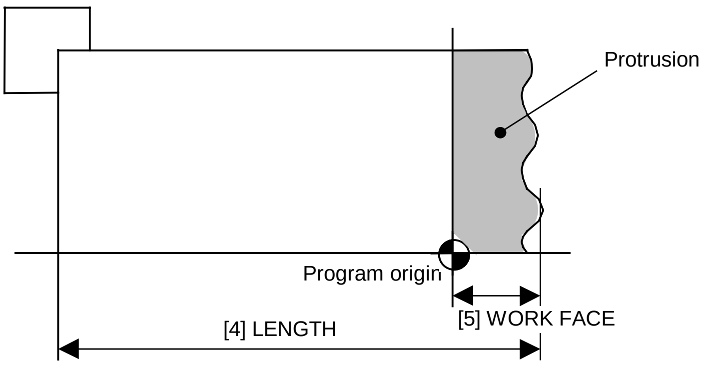
Set the maximum outside diameter, minimum inside diameter, and maximum length, respectively, of the workpiece.



- 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.



T4P019

- 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] ATC MODE

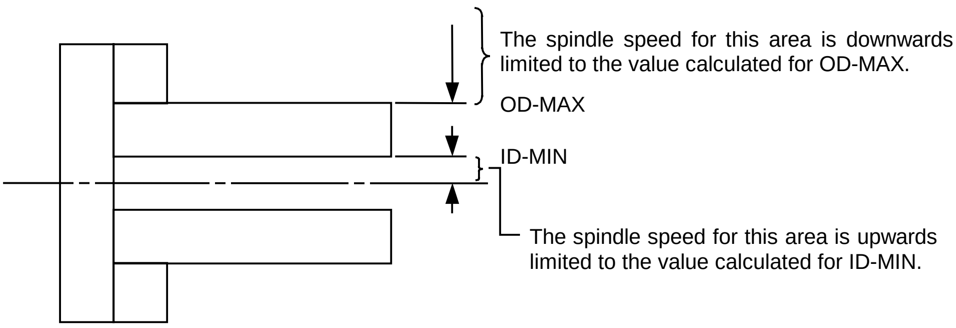
Specify how to retract the axes before ATC.

- Enter 0 to move the axes one by one from the machining end point to the ATC position.
- Enter 1 to move the axes all together from the machining end point to the ATC position.

[7] 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 surface speed control is cancelled for extra-workpiece area in order to reduce the machining time.

[8] LOW TURR

For a machine equipped with upper and lower turrets, enter a safe outside-diameter value for the lower turret. See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for further details.

3-3 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. This unit of base data, therefore, has no relation to units of machining on the front and back faces, since the tool path for such units are created merely on the basis of the settings in the common unit.
This unit need not be set for round-bar materials.
Press the **[WORKPICE SHAPE]** menu key to select the materials shape unit.

3-3-1 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.

3-3-2 Setting sequence data

```
UNo.      UNIT
 *     MATERIAL ***
FIG     PTN    SPT-X  SPT-Z  FPT-X  FPT-Z            RADIUS
 1      [1]     [2]    [3]    [4]    [5]               [6]
```

[1] PTN

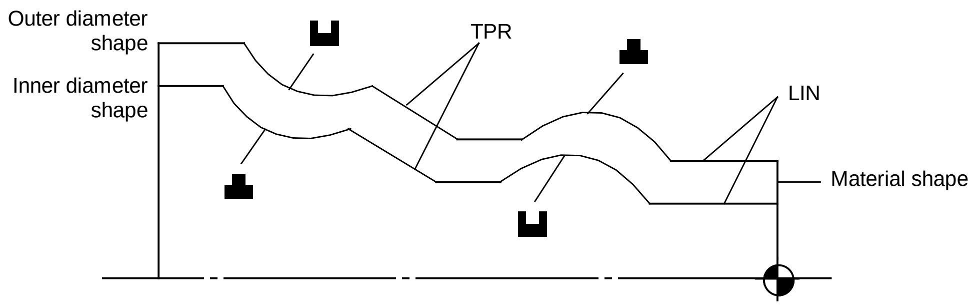
The following menu will be displayed when the cursor is placed at this item.

LIN	TPR								SHAPE END

Select the type of shape from the above menu.

The data of the displayed menu denote the following shapes:

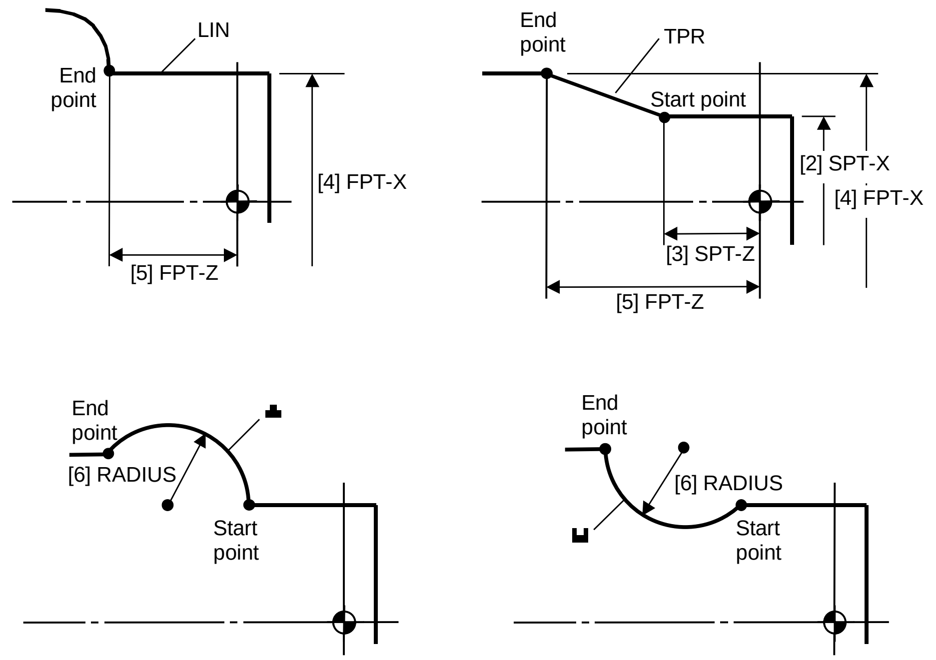
- LIN : Line parallel to the center line of the workpiece
- TPR : Line not parallel to the center line of the workpiece (Taper line)
- : Convex arc
- : Concave arc



T4P021

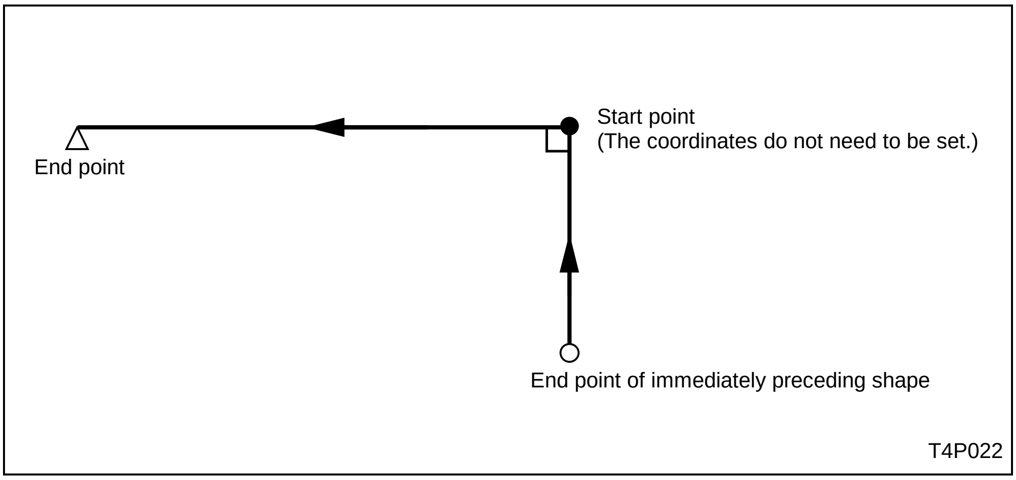
[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z, [6] RADIUS

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 .

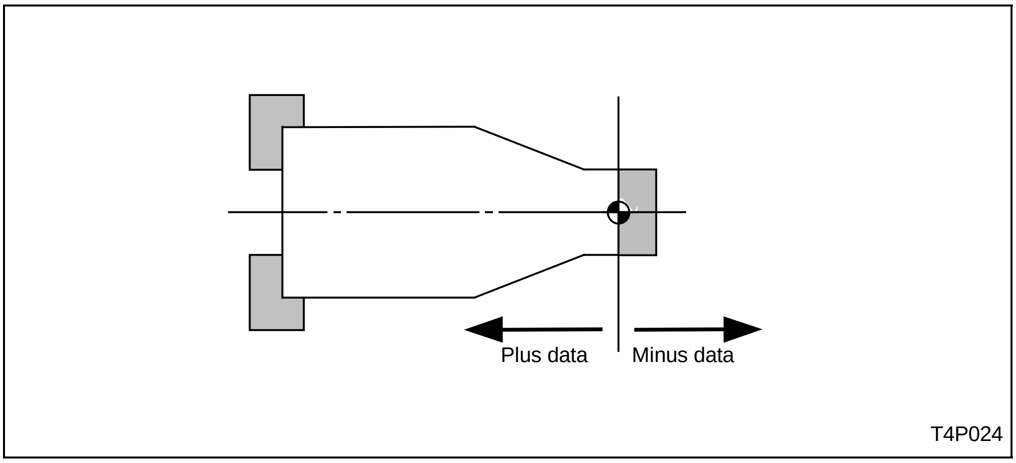


T4P023

- 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.



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



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
 FIG       PTN     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
 FIG       PTN     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.

Note 1: Although a maximum of 200 lines of shape sequence data can be set in one materials shape unit or turning unit, the maximum usable number of shape sequence data lines may be less than 200 when corner R/C is defined for a complex shape. In that case, alarm **723 EXCEEDS NUMBER OF SHAPES** will be displayed, even before the maximum usable number of shape sequence data lines is reached.

Note 2: If the maximum usable number of shape sequence data lines is exceeded, alarm **723 EXCEEDS NUMBER OF SHAPES** will be displayed during tool path checking, shape checking, shape drawing, or automatic operation.

3-4 Types of the Milling Unit

The milling unit is available in the following three types :

- Point machining unit ...... used for drilling of holes (Section 3-5)
- Line machining unit .... used for a contour machining (Section 3-6)
- Face machining unit ...... used for machining an area and machining form (Section 3-7)

Each milling unit includes tool sequence and shape sequence.

3-4-1 Planes to be machined and machining methods

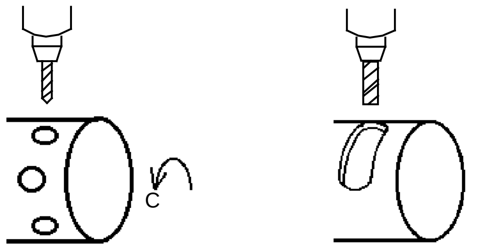
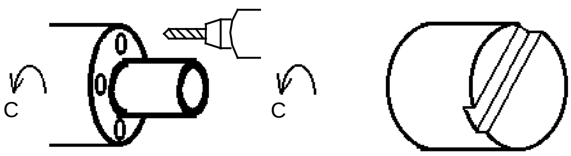
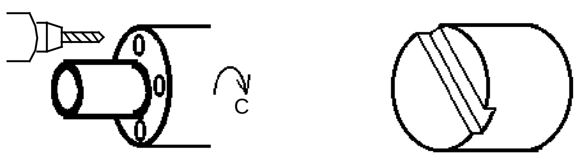
Data items for setting the plane to be machined and for setting the machining method exist in all point, linear, and face machining unit data. These data items are displayed as **MODE**, **POS-B**, and **POS-C**.

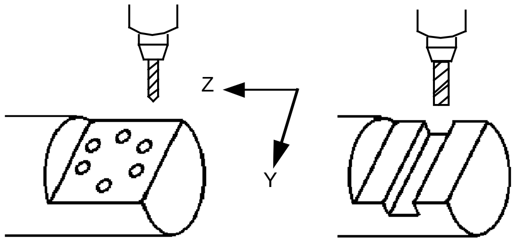
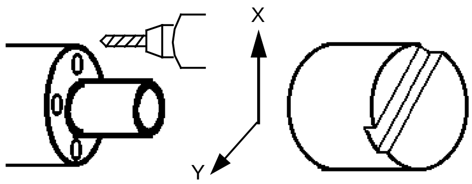
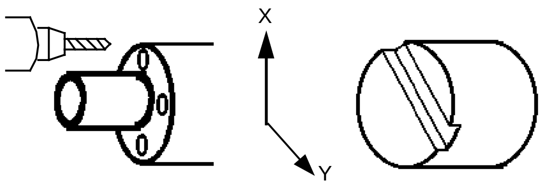
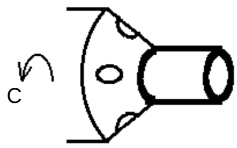
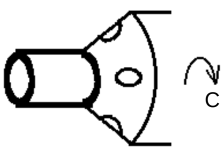
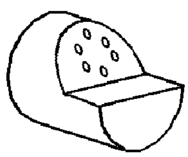
Specify the desired face and method under the **MODE**, **POS-B**, and **POS-C** columns.

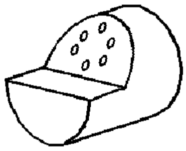
```
UNo.      UNIT       MODE    POS-B      POS-C      DIA      DEPTH    CHMF
        DRILLING      [1]         [2]        [3]
```

[1] MODE

Select the machining method.

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) 
$\boxed{XC}$	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.

Mode	Description
ZY	Plane of cylinder can be machined into the desired shape as specified in the Z-Y coordinate system. (Y-axial machining) 
XY	Edges can be machined into the desired shape as specified in the X-Y or R-C coordinate system. (Y-axial machining) 
$\boxed{\text{XY}}$	Rear plane can be machined into the desired shape as specified in the X-Y or R-C coordinate system. (Y-axial machining) 
/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 edge side.
$\boxed{\text{/C}}$	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.

Mode	Description
[/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 [**XC**], [**XY**], [**/C**], [**/Y**] mode can be selected for a machine model capable of back machining.

Note: For the line machining unit, the **/C** or [**/C**] mode cannot be selected. The **ZC**, **XC**, [**XC**], **/C** or [**/C**] mode cannot be selected for a face machining unit.

<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 [**XC**] mode. (See Note 1.)
   It is not possible to use **/C**, [**/C**], **ZY**, **XY,** [**XY**], /**Y**, or [**/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).

<u>[2] POS-B</u>

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.

<u>[3] POS-C</u>

Specify the position of the C-axis.

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

3-5 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.

3-5-1 Types of point machining units

As shown below 12 types of point machining units are available:

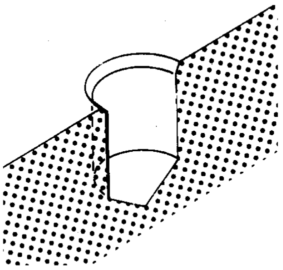
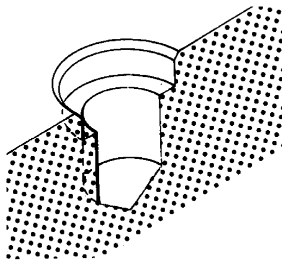
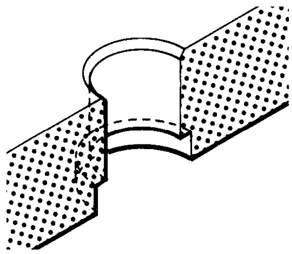
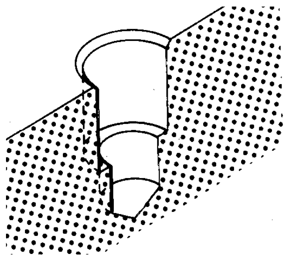
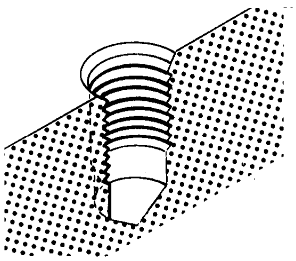
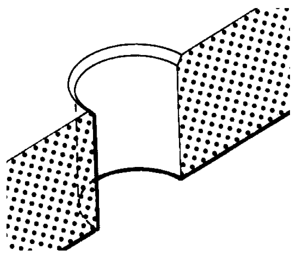
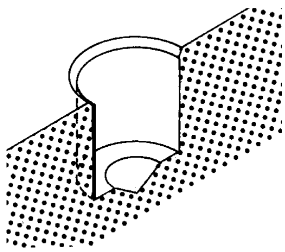
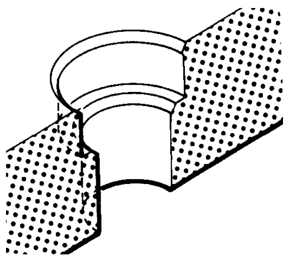
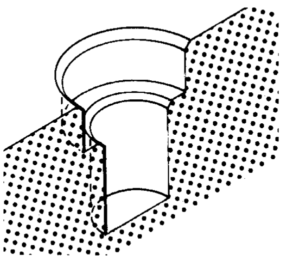
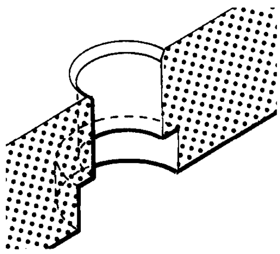
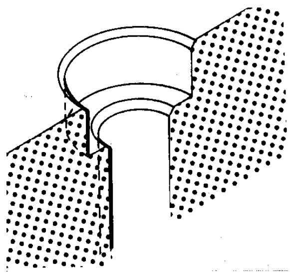
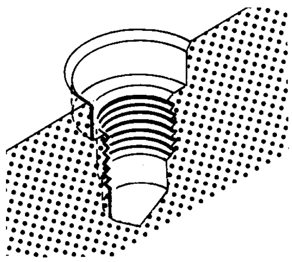
1. Drilling	2. Counterbore machining	3. Inversed faced hole machining	4. Reaming
 NM210-00532	 NM210-00533	 NM210-00534	 NM210-00535
5. Tapping	6-(1) Boring of through hole	6 (2) Boring of non-through hole	6 (3) Boring of stepped through hole
 NM210-00536	 NM210-00537	 NM210-00538	 NM210-00539
6-(4) 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

3-5-2 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Presse the **[POINT MACH-ING]** menu key.

➔ The following unit menu will be displayed.

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.

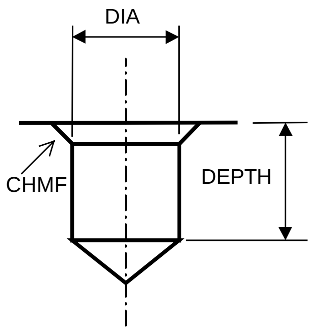
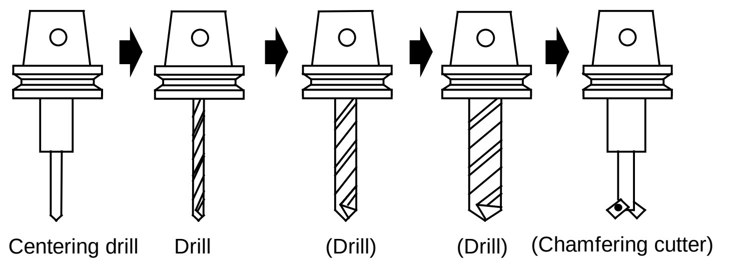
BORING	BORING	BORING	BORING						

Remark: For the function of the **[HI SPD. DRL. USE]** menu key, refer to the Subsection 3-5-4, “Automatic tool development for cemented carbide drills”.

3-5-3 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	Tool sequence
	

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

<u>Automatic tool development</u>

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.

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

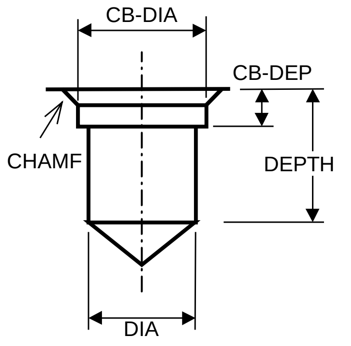
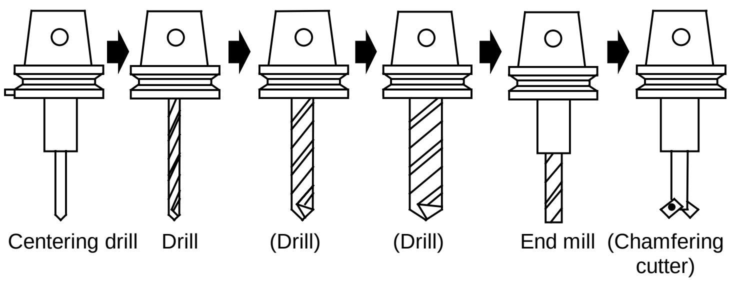
The bold codes represent parameter addresses.

Note: In the following cases the alarm **416 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	 D740PA031

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

<u>Automatic tool development</u>

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.

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 cassses: CHMF = 0 DIA + (DEPTH × 2) ≥ CB-DIA + (CHMF × 2) < **D13**

The bold codes represent parameter addresses.

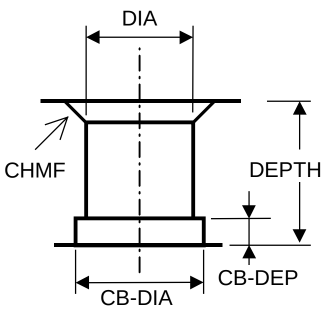
Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.

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

3. Inversed faced hole machining unit (RGH BCB)

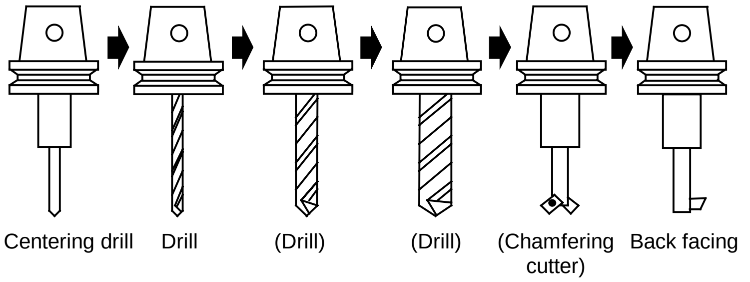
This unit is selected for machining a hole with an inversed faced hole.

RGH BCB unit



M3P089

Tool sequence



D740PA032

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

<u>Automatic tool development</u>

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.

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 **416 AUTO PROCESS IMPOSSIBLE** will be displayed.

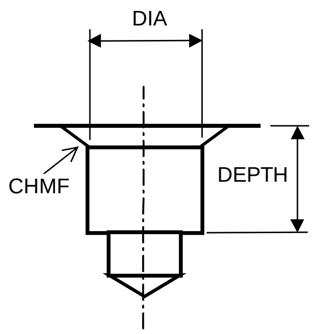
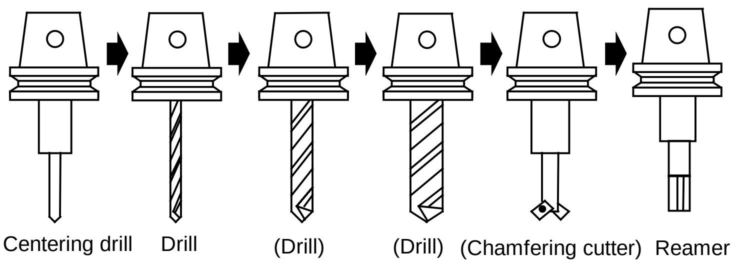
- **CB-DIA** < **DIA**
- **DEPTH** < **CB-DEP**
- **DEPTH** < **CHMF**

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	 D740PA033

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

<u>Automatic tool development</u>

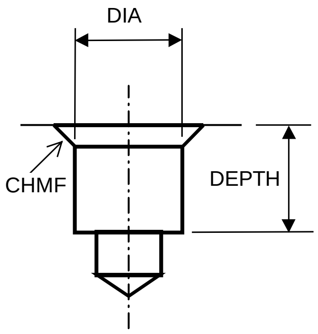
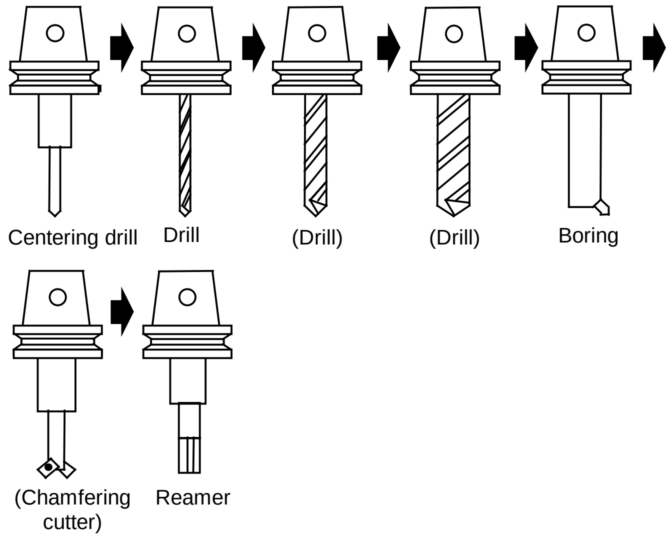
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

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 **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH** < **CHMF**

B. Case of preceding process = boring

REAMING unit	Tool sequence
 M3P093	 D740PA034

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

<u>Automatic tool development</u>

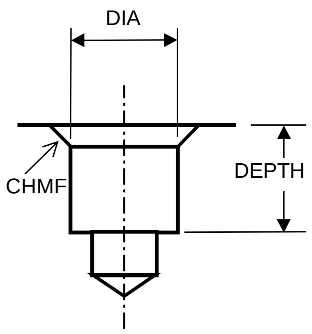
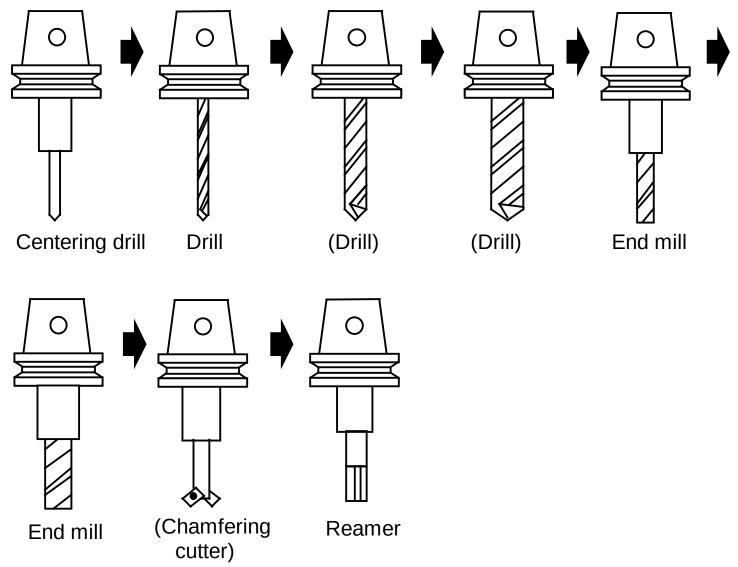
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

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 **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH** < **CHMF**

C. Case of preceding process = end mill

REAMING unit	Tool sequence
 M3P095	 D740PA035

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

<u>Automatic tool development</u>

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.

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 – **D37** ≤ **D8**: Development of one tool **D8** < DIA – **D37** ≤ **D9**: Development of two tools **D9** < DIA – **D37** ≤ **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 × 2) ≤ **D2** – **D4** CHMF = 0
Reamer	Development is always executed.

The bold codes represent the parameter addresses.

Note: In the following case the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- **DEPTH** < **CHMF**

5. Tapping unit (TAPPING)

Select this unit for performing tapping.

<Setting the nominal diameter of unified thread>

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

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

<Setting the nominal diameter of pipe thread>

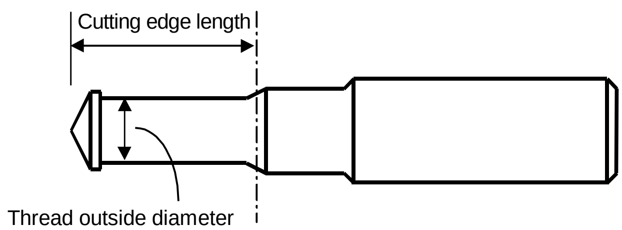
Example 1: For PT 3/8 thread:
Press the **[E (1/8) EIGHTH]** menu key, and then press the keys 3 and INPUT in this order.

Example 2: For PF 1 thread:
Press the keys 1 and INPUT in this order.

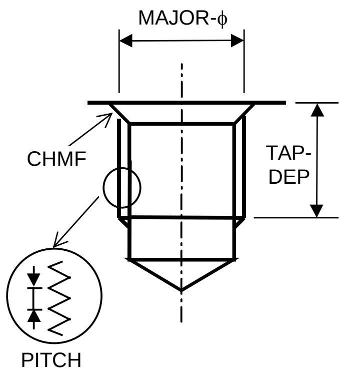
Note 1: The thread depths of PT screws or PS screws are set automatically according to MAZAK specifications.

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**.

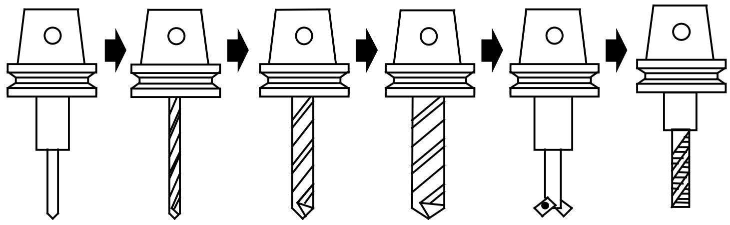


TAPPING unit



M3P097

Tool sequence



D740PA036

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.

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.

Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.

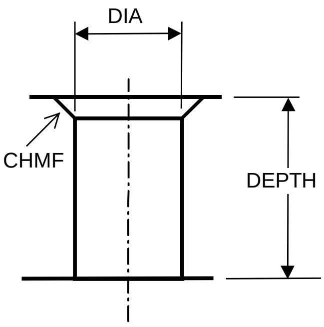
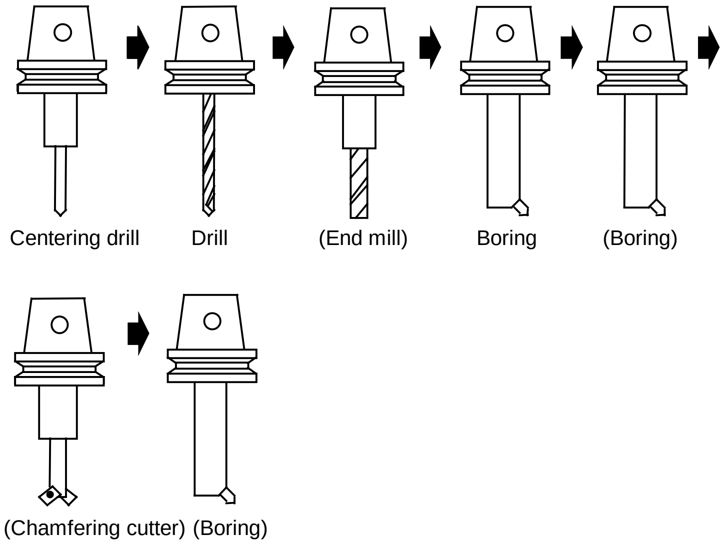
- **TAP-DEP** < **CHMF**
- Case of designation of threading other than the JIS standard threading (however, this can be used for forced insertion).

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	Tool sequence
	

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

<u>Automatic tool development</u>

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.

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.

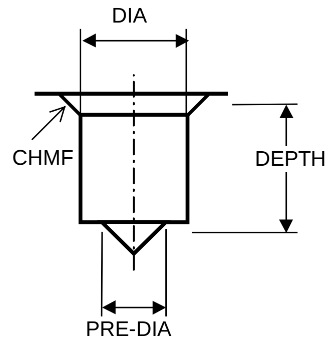
Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.

- Diameter of faced hole < **DIA**
- **DEPTH** < Depth of faced hole
- **DEPTH** < **CHMF**

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

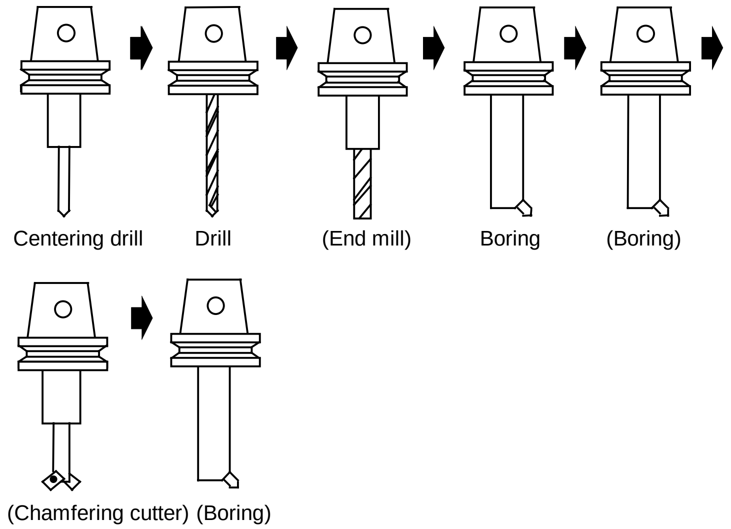
Select this unit for performing boring of non-through holes.

BORE S1 unit



M3P0101

Tool sequence



D740PA037

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

<u>Automatic tool development</u>

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.

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 **416 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

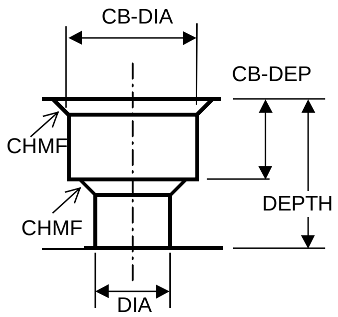
A: **DIA** – 6.0 (in case of **DIA** – 6.0 < **D8)** or

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

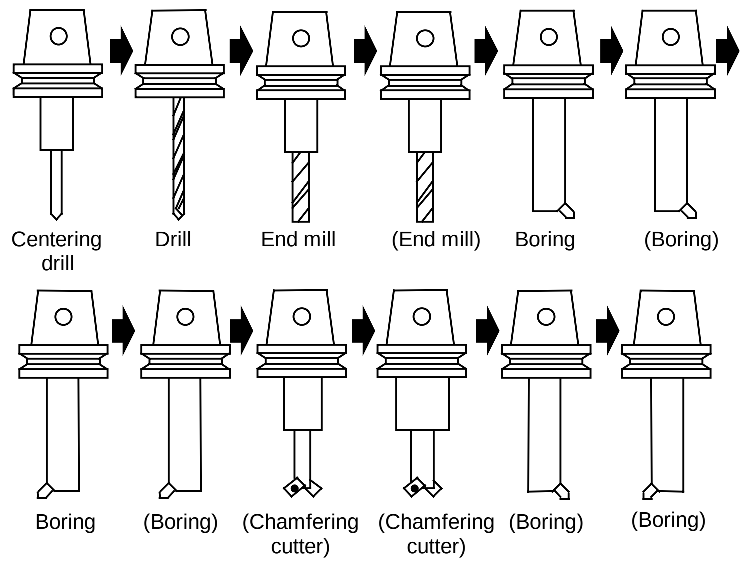
C. Stepped through hole boring unit (BORE T2)

Select this unit for performing stepped through hole boring.

BORE T2 unit



Tool sequence



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

<u>Automatic tool development</u>

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.

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
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.

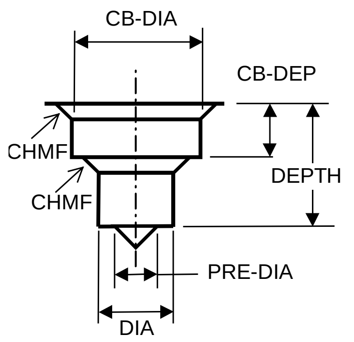
Note: The alarm **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

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

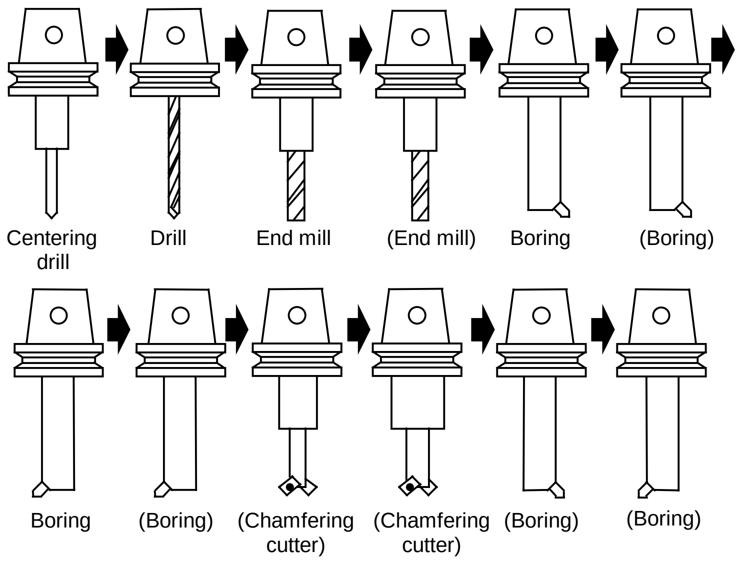
D. Stepped non-through hole boring unit (BORE S2)

Select this unit for performing stepped non-through boring.

BORE S2 unit	Tool sequence



M3P0104



D740PA038

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

<u>Automatic tool development</u>

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.

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 **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

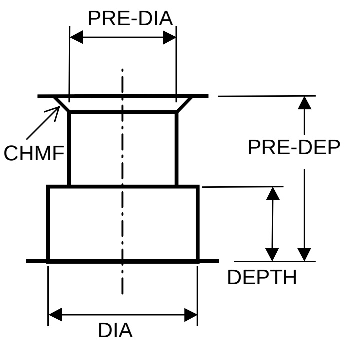
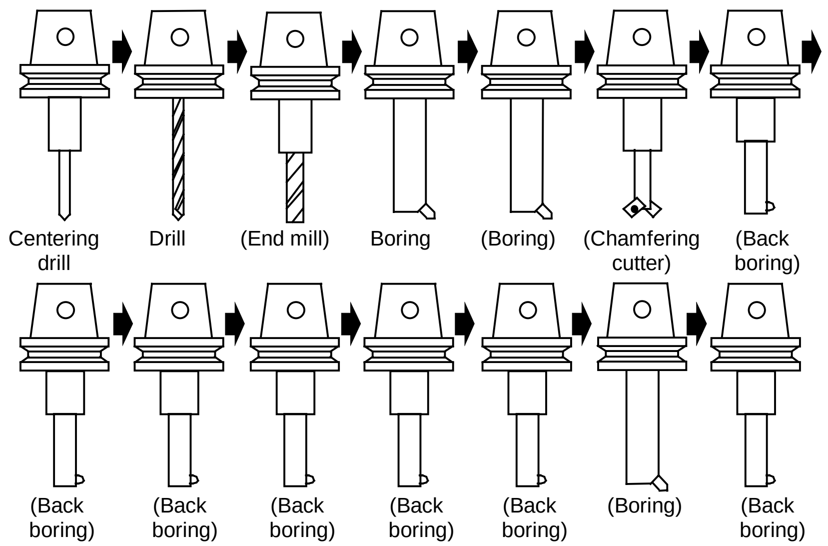
- **CB-DIA** < **DIA**
- **DIA** ≤ **PRE-DIA**
- **DEPTH** < **CB-DEP**
- **CB-DEP** < **CHMF** (**CB**)
- (**CB-DIA** – **DIA**)/2 < **CHMF**
- (**DEPTH** – **CB-DEP**) < **CHMF**
- **DIA** ≤ 6.0
- **DEPTH** < **CHMF**
- B ≤ 0

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	 D740PA039

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

<u>Automatic tool development</u>

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.

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 **416 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.

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

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

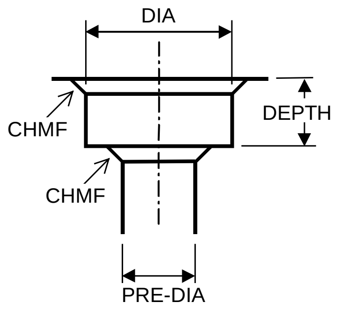
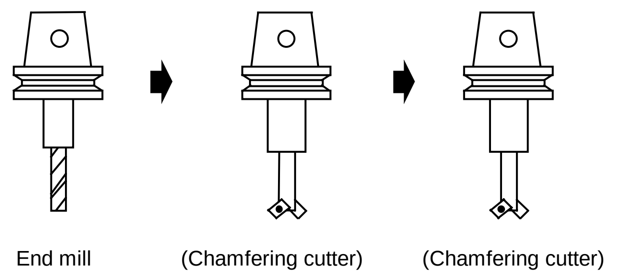
8. Circular milling unit (CIRC MIL)

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

A. Circular milling cycle

CIRC MIL unit	Tool sequence
 M3P108	 D740PA040

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

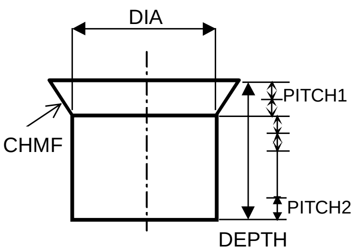
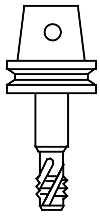
<u>Automatic tool development</u>

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.

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 **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:
- **DIA** < **PRE-DIA**
- **DEPTH** < **CHMF**
- (**DIA** – **PRE-DIA**)/2 < **CHMF** (pre-hole)

B. Tornado milling cycle

CIRC MIL unit	Tool sequence
 D735P0063	 End mill (Thread mill) D740PA041

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

<u>Automatic tool development</u>

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.

Tool	Development patterns
End mill	Development is always executed.

Note 1: The alarm **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:

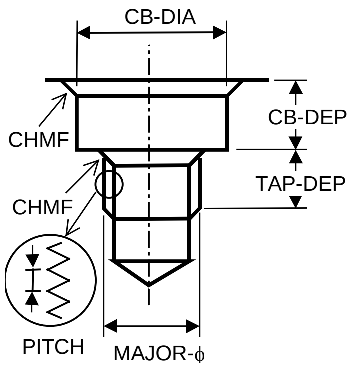
- **DIA** < **PRE-DIA**
- **DEPTH** < **CHMF**
- (**DIA** – **PRE-DIA**)/2 < **CHMF** (pre-hole)

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

9. Counterbore-tapping unit (CBOR-TAP)

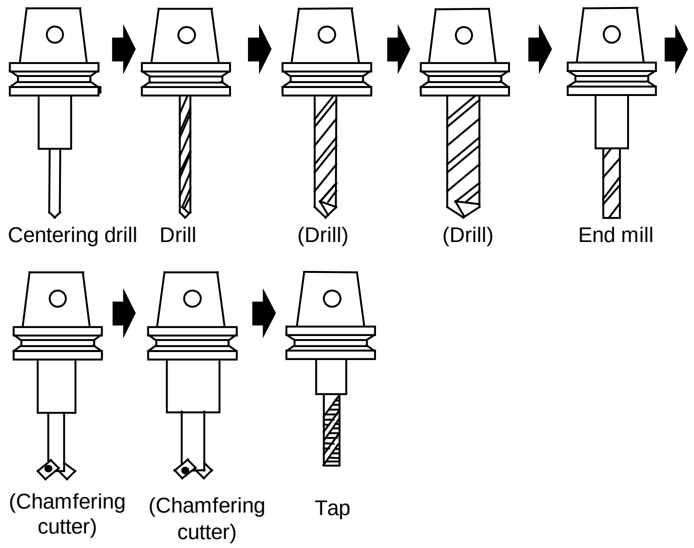
Select this unit for machining a tapped hole with a counterbore (faced hole).

CBOR-TAP unit



M3P110

Tool sequence



D740PA042

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

<u>Automatic tool development</u>

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.

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.

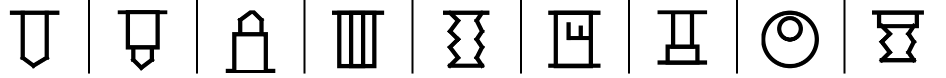
Note: The alarm **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:
- **CB-DIA** < **MAJOR-ϕ**
- (**CB-DIA** – **MAJOR-ϕ**)/2 < **CHMF** (threaded hole)
- **PRE-DEP** < **CHMF** (faced hole)
- **TAP-DEP** < **CHMF** (threaded hole)

3-5-4 Automatic tool development for carbide drills

The Subsection 3-5-3 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	No.	#	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	M
1	DRILL	○			○	○	○	○	DRIL	○					
2	CHAMFER	○			○	○	○	○	◆	○					

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	No.	#	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	M
1	DRILL	○			○	○	○	○	DRIL	○					
2	CTR-DR	○			○	◆	◆	◆	90°	◆					

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

◆ : Data are not necessary to be set here.

3-5-5 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.

○: 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	—	—	—	○	—
Unified thread	—	—	—	○	—
Pipe thread (PT)	○	○	○	○	○
Pipe thread (PF)	○	○	—	○	—
Pipe thread (PS)	○	○	○	○	○

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=8000 ;<M1> Diameter of Prehole(1/10000mm)          ←—— Pre-hole diameter
PRE_DIA_2=9000 ;<M1.1> Diameter of Prehole(1/10000mm)        ←—— Pre-hole diameter
            ⋮
            ⋮
[UN]
PRE_DIA_1=15000;<No.1-64UN> Diameter of Prehole(1/10000mm)  ←—— Pre-hole diameter
PRE_DIA_2=18000;<No.2-56UN> Diameter of Prehole(1/10000mm)  ←—— Pre-hole diameter
            ⋮
            ⋮
```

B. Text data items

- Pre-hole diameter (Setting unit: 1/10000 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	1/10000 mm	1000	9999000

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 **416 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=7000 ;<M1>       Diameter of Prehole(1/10000mm)
PRE_DIA_2=9000 ;<M1.1>     Diameter of Prehole(1/10000mm)
         ⋮
         ⋮
```

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 1/10000 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=97280  ;Diameter(1/10000mm)               ←—— Tap outside diameter
THREAD_1=280      ;Number of Thread(1/10Thread)      ←—— Total threads
DEPTH_1=156000    ;Depth(1/10000mm)                  ←—— Thread depth
PRE_DIA_1=82000   ;Diameter of Prehole(1/10000mm)    ←—— Pre-hole diameter
PRE_DEP_1=184100  ;Depth of Prehole(1/10000mm)       ←—— Pre-hole depth
            ⋮
            ⋮
[PF]
;PF 1/8
DIAMETER_1=97280  ;Diameter(1/10000mm)               ←—— Tap outside diameter
THREAD_1=280      ;Number of Thread(1/10Thread)      ←—— Total threads
PRE_DIA_1=88600   ;Diameter of Prehole(1/10000mm)    ←—— Pre-hole diameter
            ⋮
            ⋮
[PS]
;PS1/8
DIAMETER_1=97280  ;Diameter(1/10000mm)               ←—— Tap outside diameter
THREAD_1=280      ;Number of Thread(1/10Thread)      ←—— Total threads
DEPTH_1=155000    ;Depth(1/10000mm)                  ←—— Thread depth
PRE_DIA_1=85000   ;Diameter of Prehole(1/10000mm)    ←—— Pre-hole diameter
PRE_DEP_1=183100  ;Depth of Prehole(1/10000mm)       ←—— Pre-hole depth
            ⋮
            ⋮
```

B. Text data items

- Tap outside diameter (Setting unit: 1/10000 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: 1/10 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: 1/10000 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: 1/10000 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: 1/10000 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:\nm64mdata” 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	1/10000 mm	10	999990
Total threads	THREAD	1/10 threads	26	2147483647
Thread depth*	DEPTH	1/10000 mm	10	9999990
Pre-hole diameter*	PRE_DIA	1/10000 mm	100	9999000
Pre-hole depth*	PRE_DEP	1/10000 mm	100	9999000

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 **416 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=101170   ;Diameter(1/10000mm)
THREAD_1=270        ;Number of Thread(1/10Thread)
DEPTH_1=110000      ;Depth(1/10000mm)
PRE_DIA_1=84300     ;Diameter of Prehole(1/10000mm)
PRE_DEP_1=170000    ;Depth of Prehole(1/10000mm)
         ⋮
         ⋮
```

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 1/10000 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 62500, a nominal drill diameter of 6.3 may be displayed as its auto-set value.

3-5-6 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.

Table 3-1 Tool sequence data

	TOOL	NOM-φ			No.	#	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	M
CTR DR	○	○	○	○	○	○	○	◆	◆	◆	○	◆	○	○	○	○	○
DRILL	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○
CHAMFER	○	○	○	○	○	○	○	○	○	○	◆	○	○	○	○	○	○
END MILL	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○
BCK FACE	○	○	○	○	○	○	○	○	◆	○	◆	◆	○	○	○	○	○
REAMER	○	○	○	○	○	○	○	○	◆	◆	○	○	○	○	○	○	○
TAP	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○
BOR BAR	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○
B-B BAR	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○	○
Reference	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	15	15

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

Remark 1: For setting of each data item refer to 1 to 15 below.

Remark 2: If **[TAPPING CYCLE]** menu item is selected for **PRE-DIA**, there is no need to set data in **PRE-DEP**.

1. 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. NOM-φ (Nominal diameter)

Used to specify the nominal diameter of the tool by means of numeric keys.

Note: The alarm **434 NO ASSIGNED TOOL IN TOOL FILE** is given if the tool entered has not been previously recorded in the **TOOL FILE** display.

3. NOM-φ (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	HEAVY TOOL	>>>

In order to designate a heavy tool, first of all press the **[HEAVY TOOL]** menu key to reverse the menu display and then select the desired menu key in the menu thus displayed.

4. NOM-ϕ (Turret selection)

For the machine with the lower turret, select the turret in which the tool to be used is mounted. The following menu is displayed (if **[SET UPPER TURRET]** is selected, the column will remain blank, and if **[SET LOWER TURRET]** is selected, "◤" will be displayed). See Section 5, LOWER-TURRET CONTROL FUNCTIONS, for further details:

SET UPPER TURRET	SET LOWER TURRET								

5. No. (Priority No.)

Assign priority levels in the order of machining. The following menu is displayed. A press of a menu key displays the menu item in reverse mode, allowing a priority number to be assigned.

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN			PRI.No. ALL ERAS	SUB PROG PROC END	
	(a)		(b)	(c)			(d)	(e)	

The function of menu item (a) to (e) is described below:

Menu item	Function
(a)	Select to conduct subsequent-machining.
(b)	Select to change the priority number for the tool within the particular process. If the cursor is present at a blank space, assign a new number in a usual manner. Entry of an existing priority number displays alarm **420 SAME DATA EXISTS**.
(c)	Select to assign a priority number to the tool to be used repeatedly in the particular process. Alarm **420 SAME DATA EXISTS** will be displayed if the assigned priority number has already been set on any other unit line.
(d)	Selection of this item displays message **ALL ERASE (PROC:0, PROG:1)?**. Setting 0 will erase the priority numbers preassigned to the tool to be used repeatedly in the process. Setting 1 will erase the priority numbers preassigned to the tool to be used repeatedly in the program.
(e)	Select to terminate the process with the subprogram unit.

For details see Chapter 4, "PRIORITY FUNCTION FOR THE SAME TOOL."

6. # (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. For details see Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS."

LOWER TURRET POS.1	LOWER TURRET POS.2								

7. HOLE-ϕ (Diameter of machining hole)

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 of the hole to an interference. Enter 999 if there is no interference.

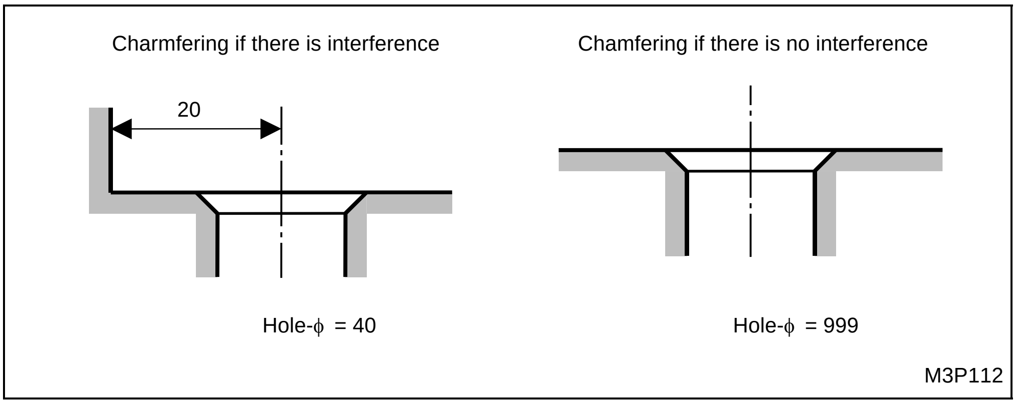


Fig. 3-2 Specification of diameter of machining hole for chamfering cutter

8. HOLE-DEP (Depth of machining hole)

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.

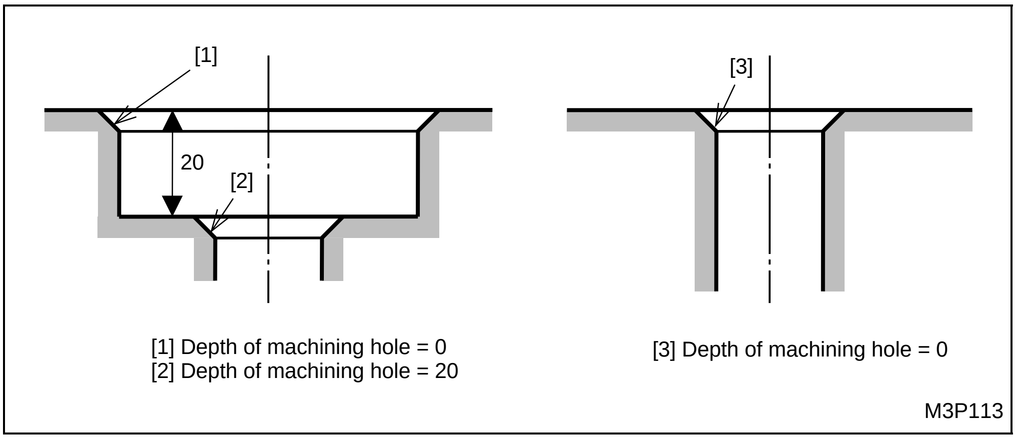
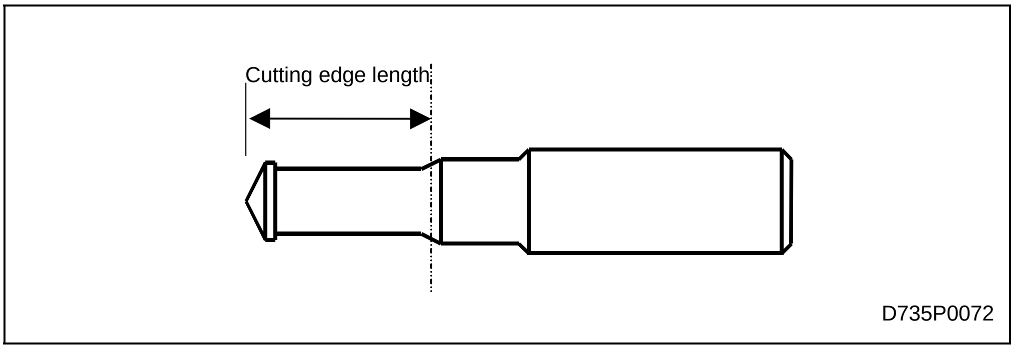


Fig. 3-3 Specification of depth of machining hole

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**.



9. PRE-DIA (Diameter of pre-hole)

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 3-5-7 “Tool path of the point machininig unit”, “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 3-5-7 “Tool path of the point machininig unit”, “7. Tap”.

10. PRE-DEP (Depth of the pre-hole)

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 3-5-7 “Tool path of the point machininig unit”, “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 **D50** parameter “Pre-hole machining feed” is set for **[PLANET CYCLE]** automatically.

11. RGH (Cutting surface roughness)

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 3-5-7 "Tool path of the point machininig unit", "2. Drill".

Note 3: Enter the duration of the dwell time for the tapping (invalid for synchronous tapping).
In automatic tool development mode, **FIX** is selected. In this case, the dwell time is set by parameter **D22**.

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.

12. DEPTH (Cutting 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.

13. C-SP (Surface speed)

To auto-set a surface 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 - W. MAT./T. MAT.** display.

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

HSS AUTO	CARBIDE AUTO								

Data can also be set using the numeric keys.

14. FR (Feedrate)

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

15. M (M-code)

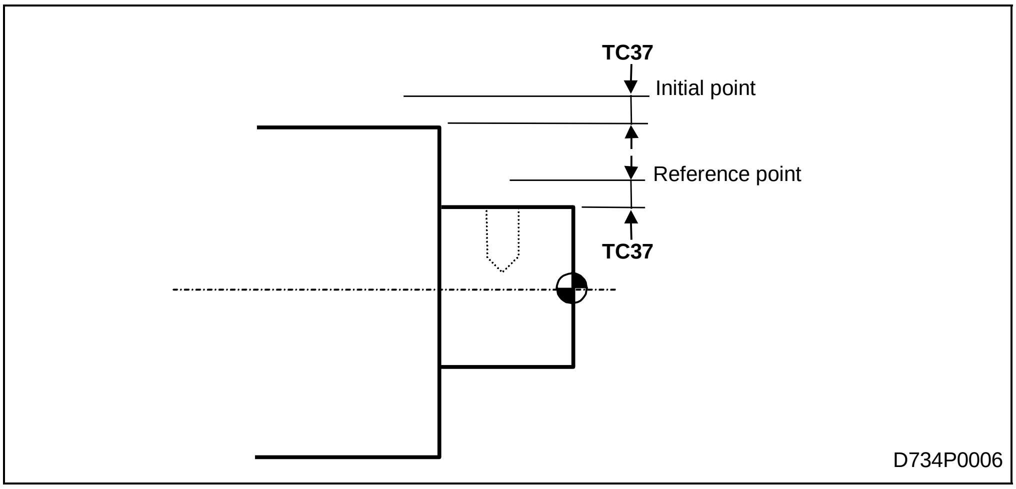
Set the required M-code(s) to be output immediately after mounting the tool onto the spindle in the ATC mode. A maximum of up to three M-codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu.

3-5-7 Tool path of the point machining unit

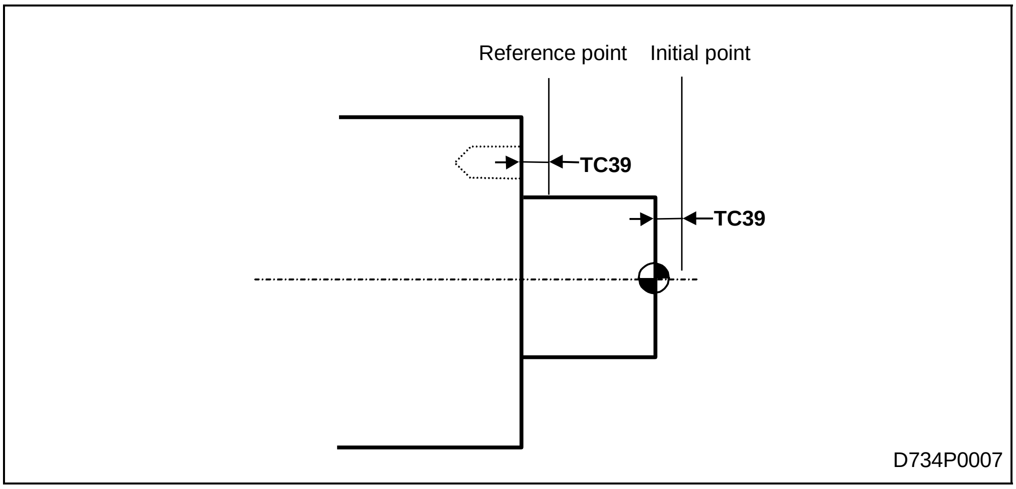
This section shows the path of each tool used during execution of a point machining unit.

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

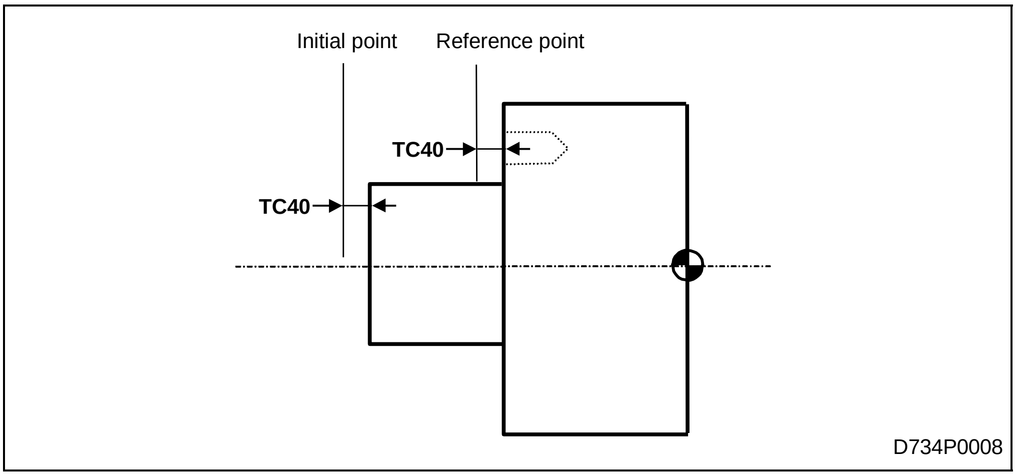
- When the selected mode is **ZC** or **ZY**



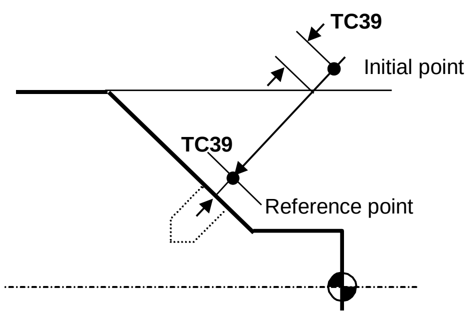
- When the selected mode is **XC** or **XY**



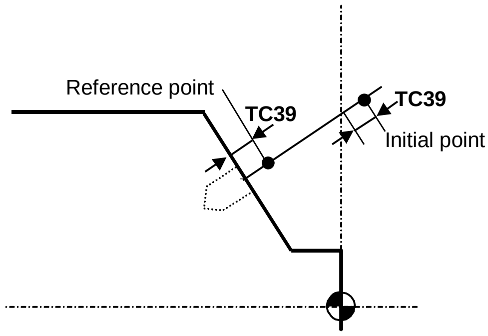
- When the selected mode is $\boxed{\textbf{XC}}$ or $\boxed{\textbf{XY}}$



- When the selected mode is ***/C*** or ***/Y***

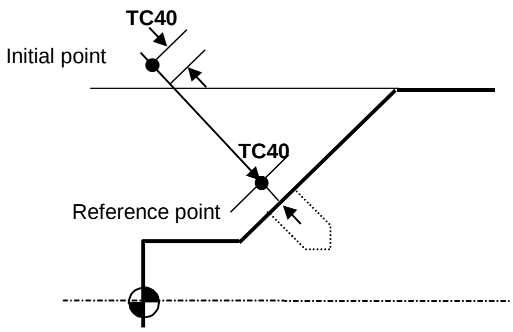


D734P0009



D734P0010

- When the selected mode is $\boxed{/C}$ or $\boxed{/Y}$

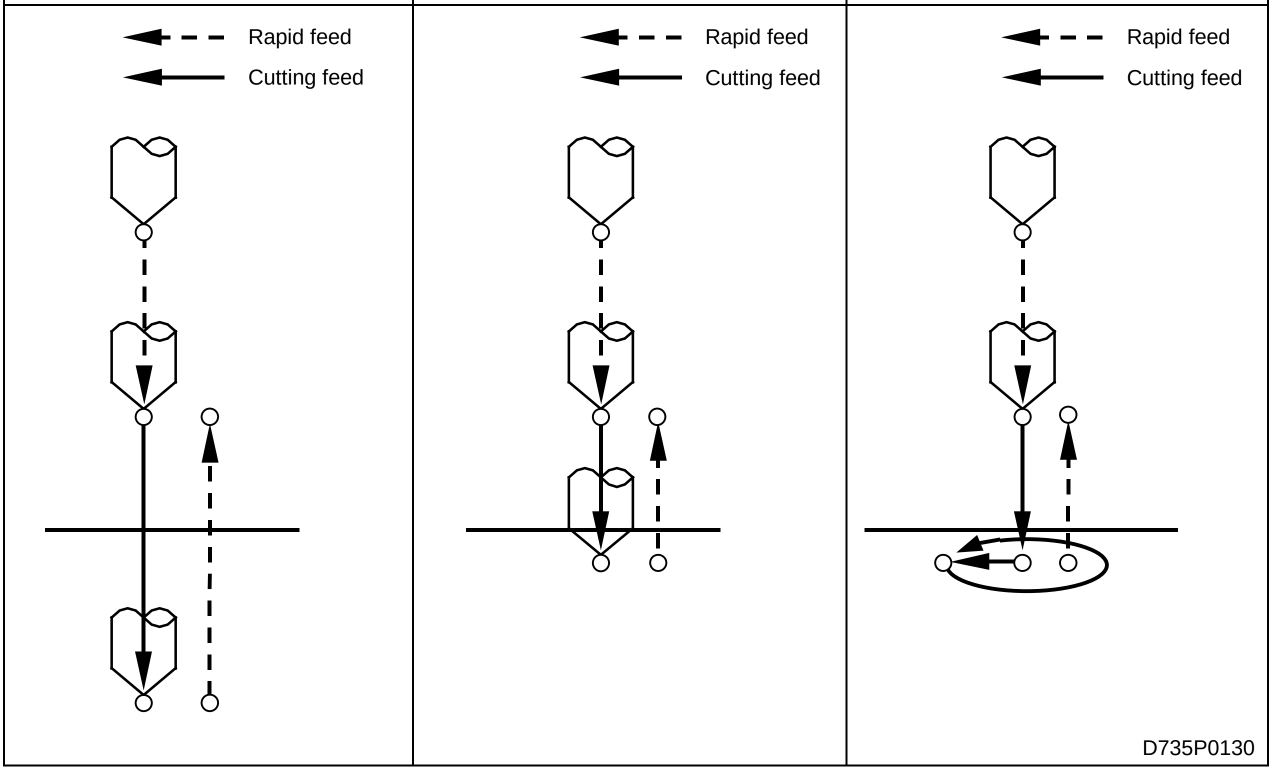


D734P0011

1. Centering drill

The cycle of machining with a centering drill is available in the following three types.

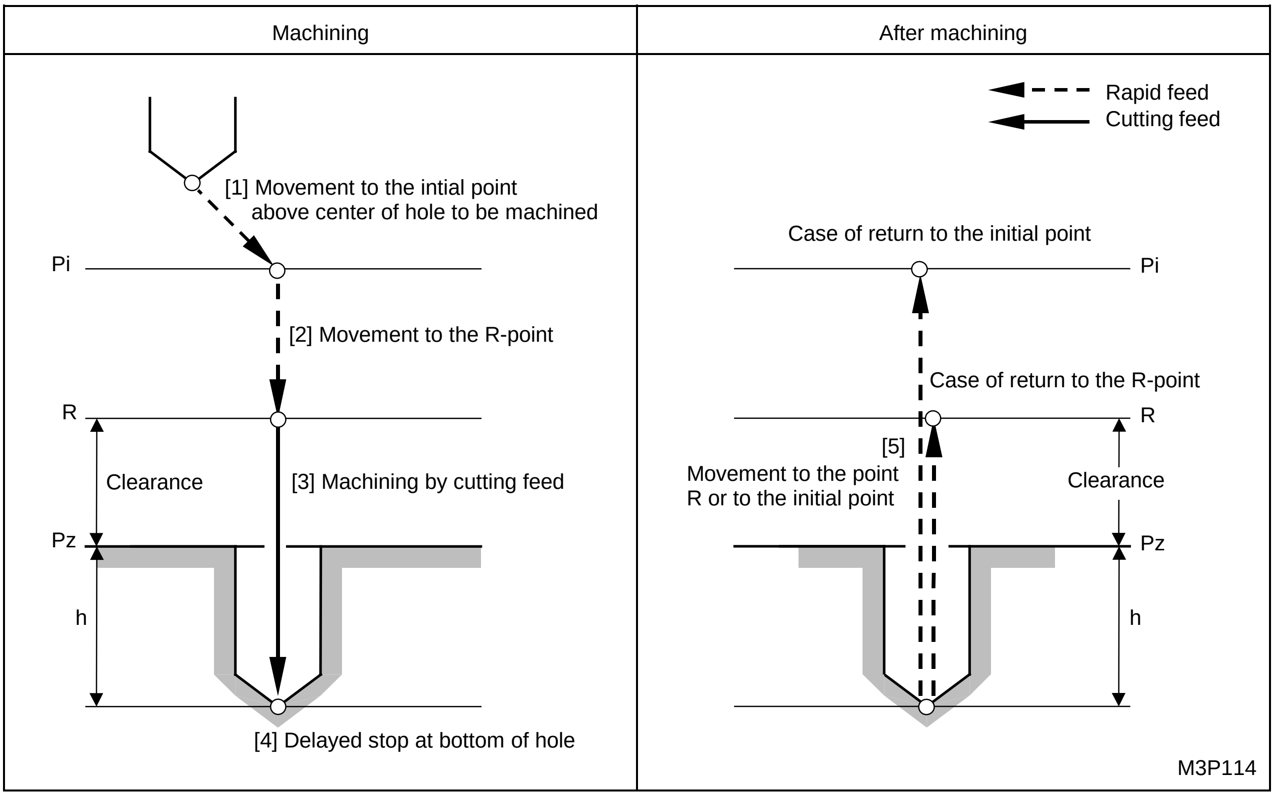
A Drilling cycle	Chamfering cycle	
	B Cycle 1	C Cycle 2



Remark: Two types of chamfering cycles are provided: “Cycle 1”, which only moves the tool in the Z-axial direction during machining, and “Cycle 2”, which moves the tool in X- and Y-axial directions in addition to the Z-axial direction.
Which of the two cycles is to be used for actual machining is automatically selected during operation.

For details of the tool paths in the two cycles, see Items A to C below.

A. Centering drilling cycle



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

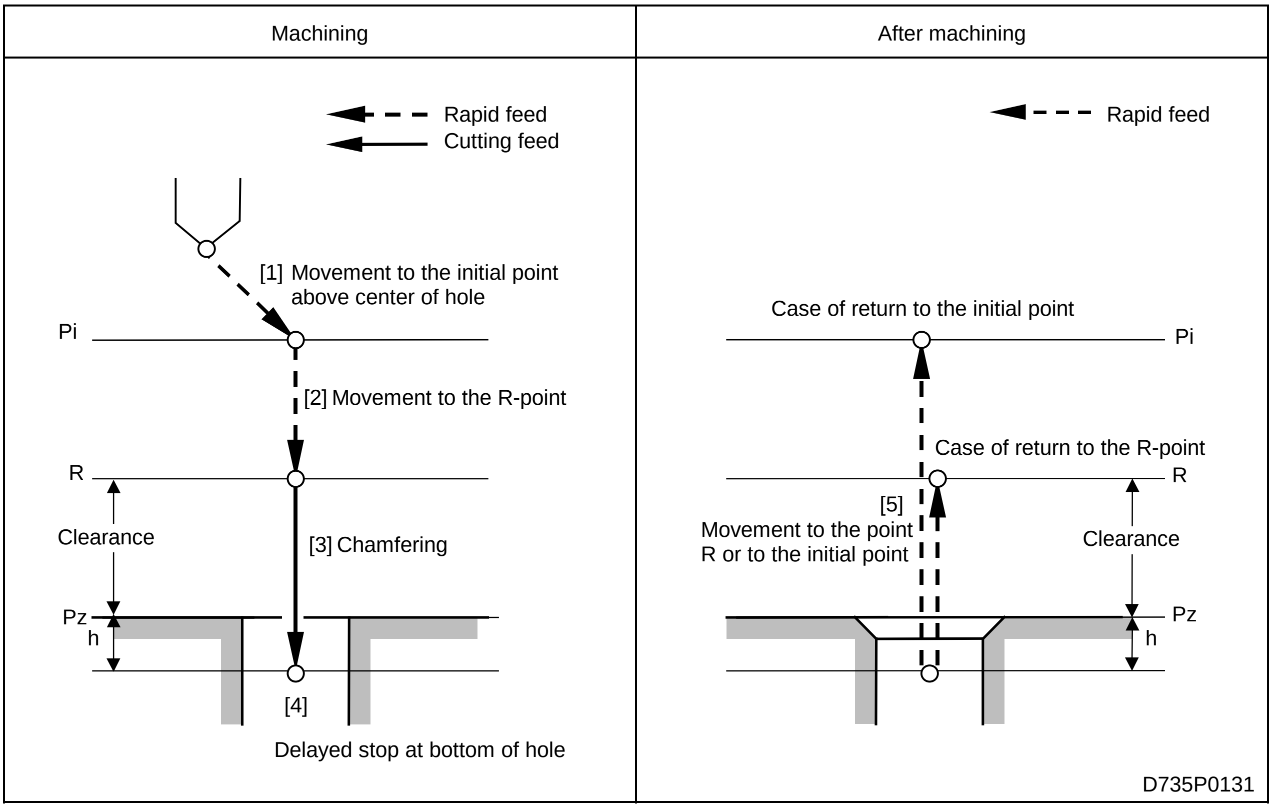
R: Safety clearance above the point Pz

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 **LENG 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 axial feed at bottom of hole is set by the parameter **D3**.

B. Cycle 1 of chamfering cycle



The bold codes represent the parameter addresses.

Pi: Initial point

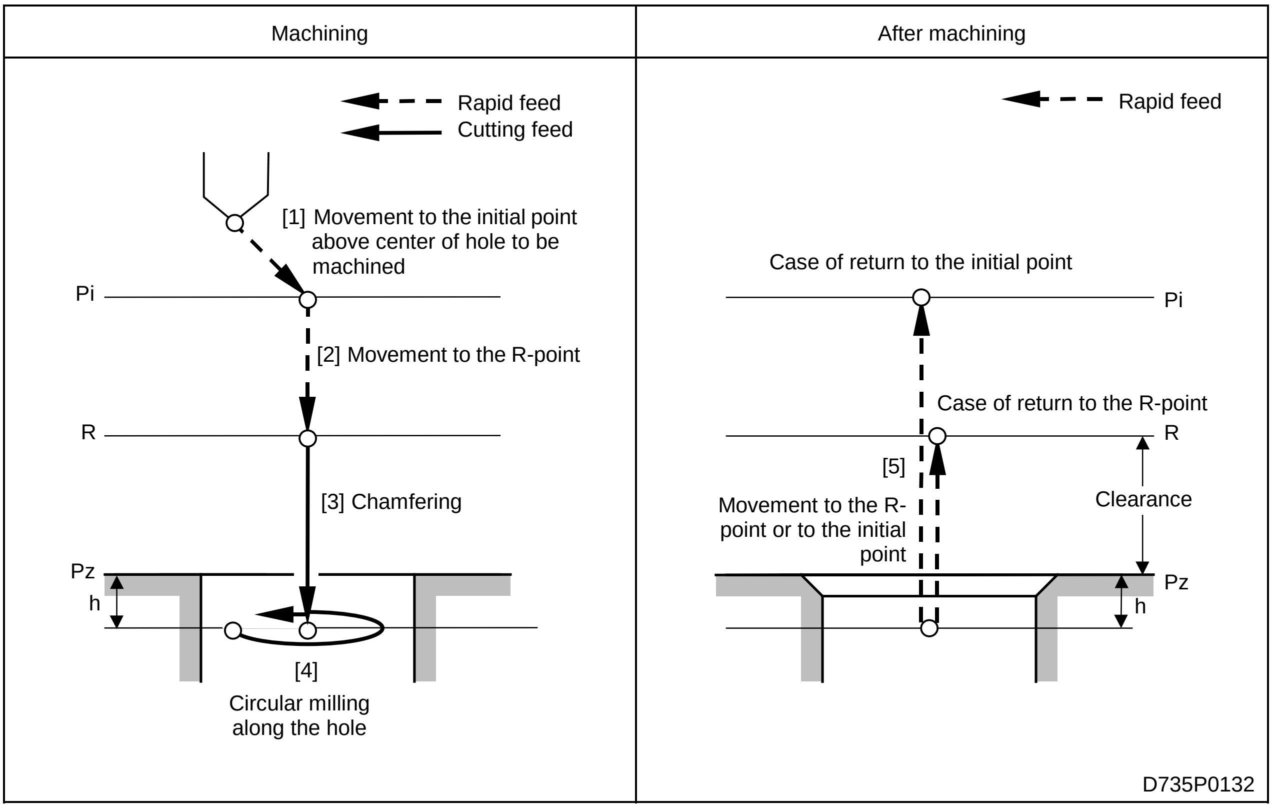
Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

h: Optimum distance to be automatically calculated by the data **PRE-DIA** and **RGH** in the tool sequence and also the data **CHMF** in the point machining unit.

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

C. Cycle 2 of chamfering cycle



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following condition is fulfilled, R before machining will be equaled to the parameter **D42**.
However, R after machining is always equaled to the (safety) clearance.
- Case where the bit 7 of parameter **D91** is 1.

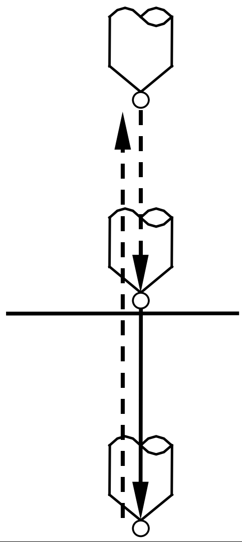
h: The optimum distance is automatically calculated by the data **PRE-DIA** and **RGH** of the tool sequence and also the data **CHMF** in the point machining unit.

Note: For the circular milling, refer to the paragraph dealing with 4. End mill, C. Cycle 3.

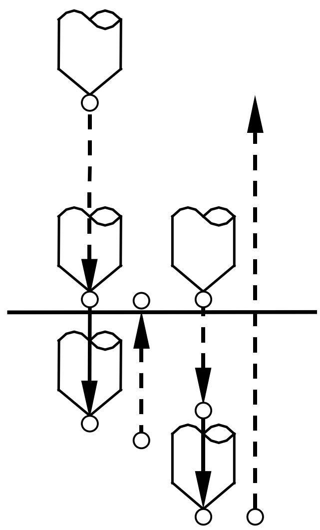
2. Drill

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

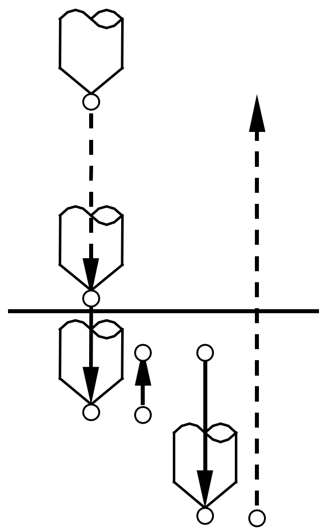
A. DRILLING CYCLE



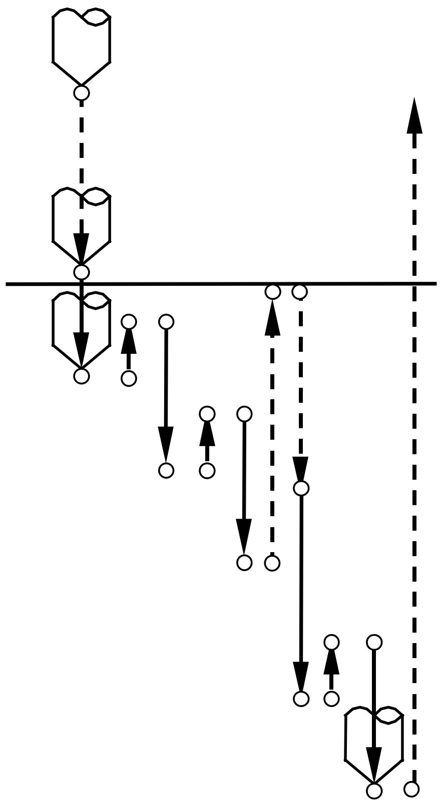
B. Deep-hole drilling cycle
(PECKING CYCLE 2)



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



D. Very deep-hole drilling cycle (PECKING CYCLE 3)



E. Auto-pecking cycle of the cutting load detection type
(AUTOPECK CYCLE)

The cutting load torque of the drill is continually monitored during the auto-pecking cycle of the cutting load detection type. This pecking cycle will be performed only when required, which prevents tool breakage and reduces machining time.

F. Deep-hole drilling cycle for gradual depth reduction
(DECREME. PECKING CYCLE 2)

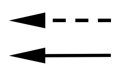
In this machining cycle, the cutting depth is decremented with respect to a normal deep-hole drilling cycle each time the workpiece is cut.

G. High speed deep-hole drilling cycle for gradual depth reduction (DECREME. PECKING CYCLE 1)

In this machining cycle, the cutting depth is decremented with respect to a high deep-hole drilling cycle each time the workpiece is cut.

H. Very deep-hole drilling cycle for gradual depth reduction
(DECREME. PECKING CYCLE 3)

In this machining cycle, the cutting depth is decremented with respect to a very deep-hole drilling cycle each time the workpiece is cut.



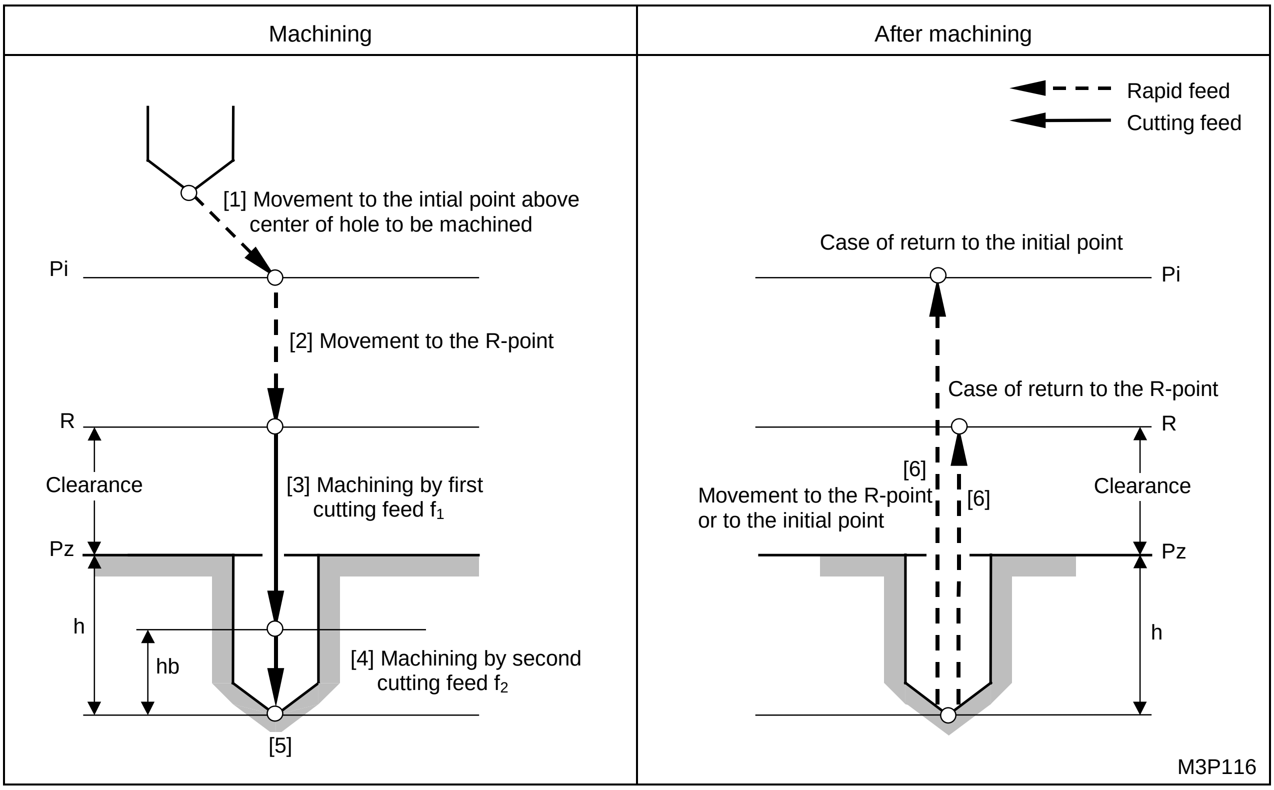
D734P0012

Remark 1: See Items A to H for the tool paths in each cycle.

Remark 2: Specify the decremental cutting depth in parameter **D45**, and the minimum cutting depth, in parameter **D46**.

Remark 3: For both “Very deep-hole drilling cycle” and “Very deep-hole drilling cycle for gradual depth reduction”, three types of machining cycle are available:
Very deep-hole drilling cycle, Very deep stop-hole drilling cycle and Very deep through-hole drlling cycle.

A. Drilling cycle



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

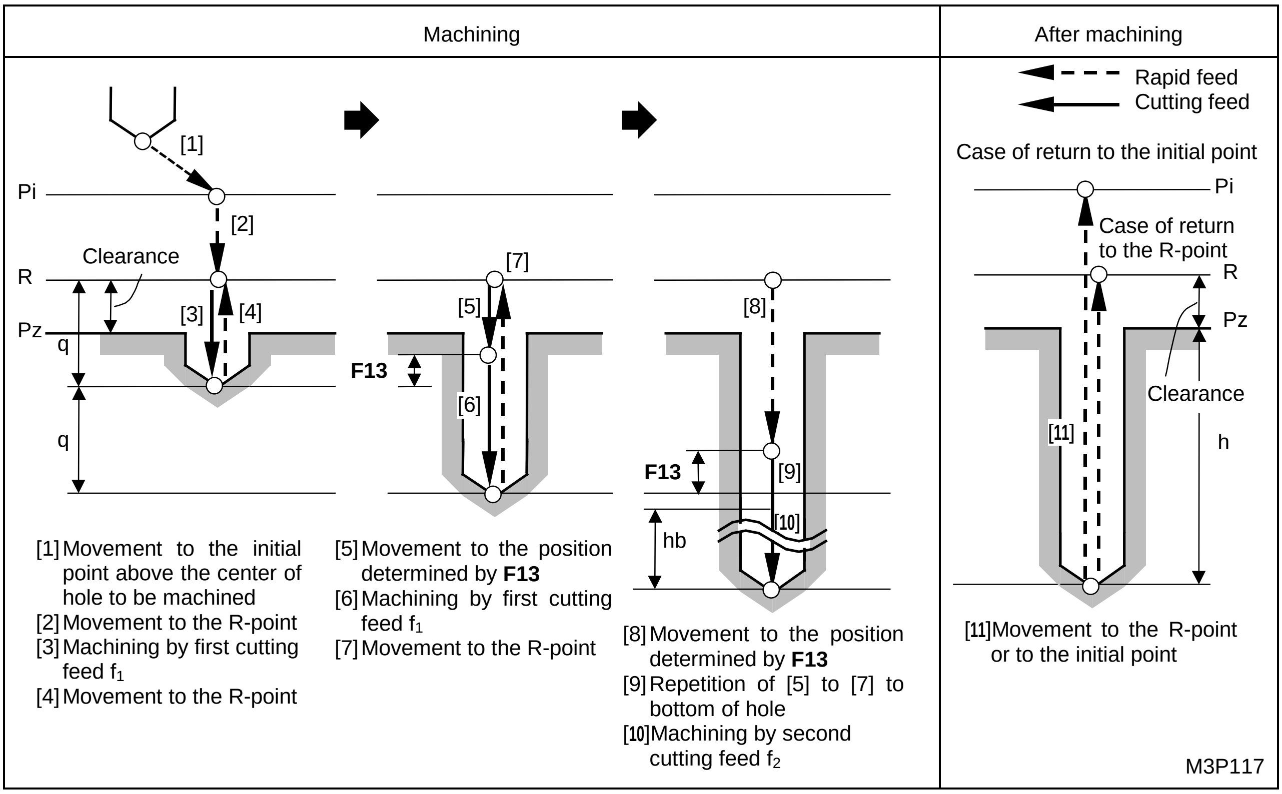
hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

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

B. Deep-hole drilling cycle (PECKING CYCLE 2)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

q: Cutting depth (**DEPTH**) to be entered in the tool sequence data

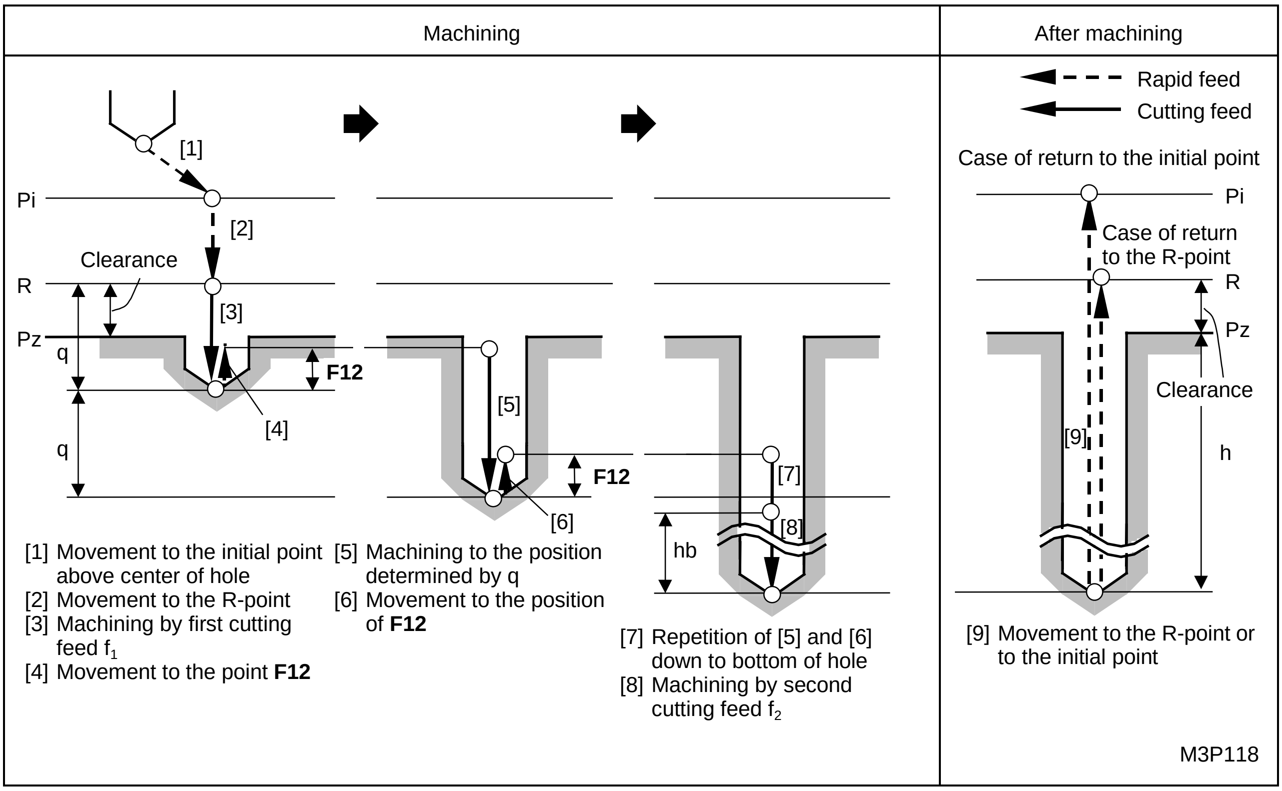
hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

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: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

q: Cutting depth (**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

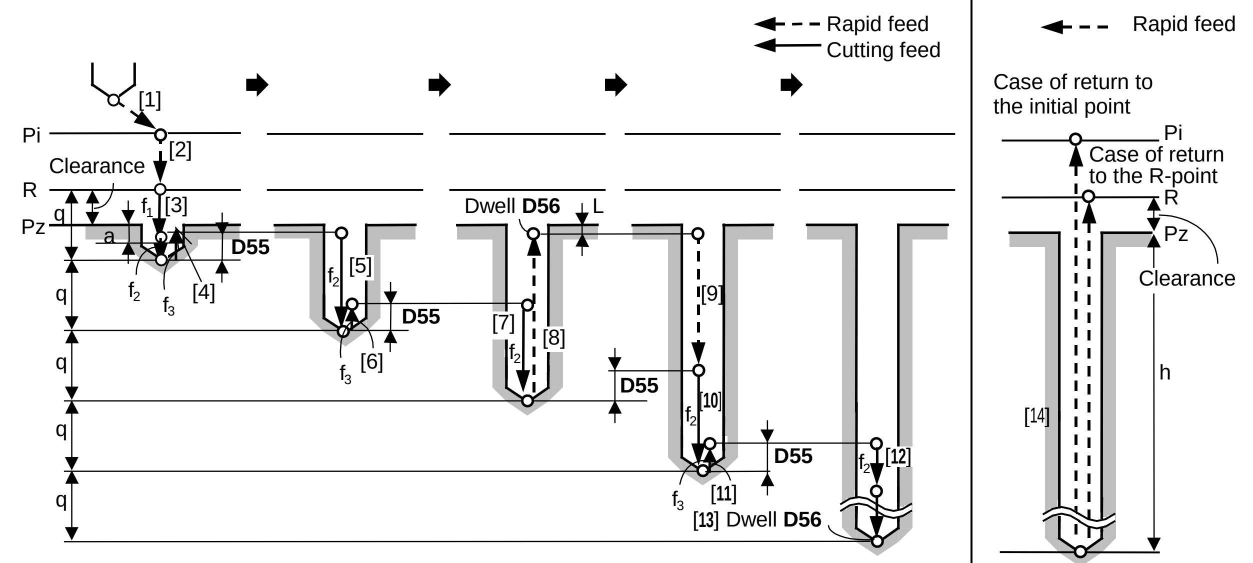
f_2: Feedrate to be modified by the data **PRE-DEP** (feedrate updating rate)

$f_2 = f_1 \times$ Feedrate updating rate

Note: The feed speed on the paths [4] and [6] is 9999 mm/min or 999.9 inch/min for millimeter or inch specification respectively.

D. Very deep-hole drilling cycle (PECKING CYCLE 3)

Machining	After machining



[1] Movement to the initial point
[2] Movement to the R-point
[3] Machining of “a” at feed rate f_1 and movement by q at feedrate f_2
[4] Movement by **D55** to the retraction position at feed rate f_3
[5] Movement by q at feedrate f_2
[6] Movement by **D55** to the retraction position at feedrate f_3
[7] Movement by q at feedrate f_2
[8] After **D53** times peckings, movement to the chip ejection position and wait for **D56** rotations
[9] Movement by **D55** to the advanced position from the previous machining end position
[10] Movement by q at feed rate f_2
[11] Movement by **D55** to the retraction position at feedrate f_3
[12] Repetition of [7] and [9] down to bottom of hole
[13] After movement to bottom of hole, wait for **D56** rotations

[14] Movement to the R-point or to the initial point

D734P0013'

The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.
- 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 **LENG COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

a: Cutting area (Note 3)

q: Cutting depth (**DEPTH**) to be entered in the tool sequence data

f_1: Feedrate (infeed rate) obtained by multiplying “f_2” by the “reduction ratio of the starting speed of cutting” specified in parameter **D54**
where, if **D54** = 0 or if **D54** > 100, then **D54** = 100.

$$f_1 = f_2 \times \frac{\mathbf{D54}}{100}$$

f_2: Feedrate (**FR**) to be set for the tool sequence

f_3: Pecking retraction speed (= setting of parameter **D57**)
where, if **D57** = 0, then **D57** = 1000.

L: Chip ejection distance calculated from the data **ACT-ϕ** (tool diameter: D) and data **LENG COMP.** (tool correction) on the **TOOL DATA** display

L = Data LENG COMP.$-\frac{D}{10}$ (D: Tool diameter)

Note 1: During the “n”th cutting operation, if (q × n) < **D55**, retraction through the **D55**-specified distance does not occur. During machining on the path [3], if the first cutting depth of “q” is greater than or equal to (Clearance at R-point + Cutting area “a”), machining at feedrate “f_1” will occur on the path [3] until (Clearance at R-point + Cutting area “a”) is reached, then retraction through the **D55**-specified distance from that position will occur on the path [4], and the workpiece will be cut to the next cutting position (next cutting depth) on the path [5].

Note 2: The feedrate on the path [8] is “G0 speed × **D52**/100”. (If the input value of **D52** is 0, then **D52** = 100.)

Note 3: Cutting area

Machining pattern: Very deep hole drilling cycle

a = K + R

Where K is **LENG COMP.** (tool correction) on the **TOOL DATA** display and R is the clearance.

Machining pattern: Very deep stop-hole or very deep through-hole drilling cycle

a = D × $\frac{\mathbf{D58}}{100}$

Where D is the tool diameter and **D58** (parameter) is the feedrate updating distance rate at the start of cutting.

If **D58** > 300, **D58** is taken to be 100. If a < R, a is taken to be equal to R.

Under the conditions shown below, alarm **748 CANNOT MAKE T-PATH (CHK DEPTH)** will be issued.

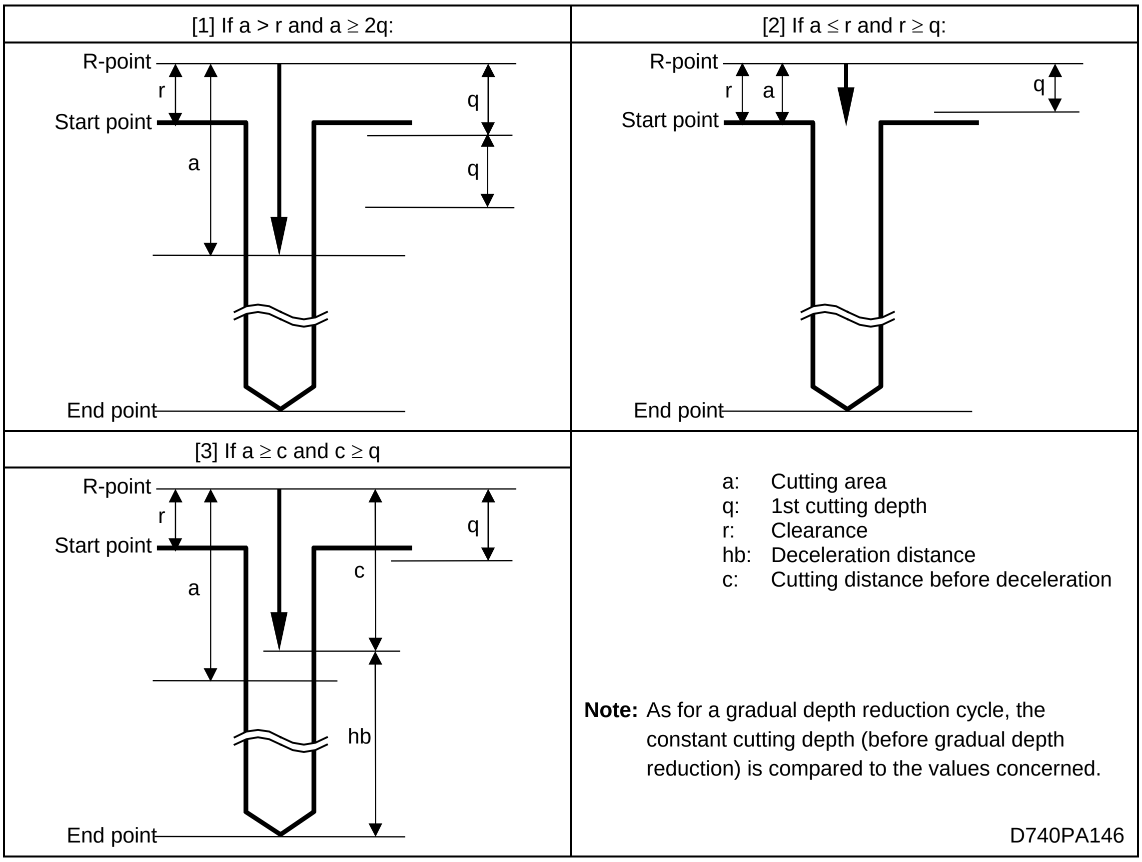


Fig. 3-4

Note 4: For very deep stop-hole or very deep through-hole drilling cycles, the feedrate or the surface speed can be changed in some cases within the feedrate updating distance from the hole bottom (hb specified under **PRE-DIA** of the tool sequence).

Machining pattern: Very deep stop-hole drilling cycle

When hb (feedrate updating distance from the hole bottom) is reached, machining by second cutting feed f_4 starts.

The surface speed is kept at the speed (S_1) set as **C-SP** in the tool sequence. (Fig. 3-5)

Let Q represent the feedrate updating rate (specified under **PRE-DEP** of the tool sequence). Then second cutting feed f_4 can be calculated from feed f_2 set as **FR** in the tool sequence, using the following equation.

$$f_4 = f_2 \times \frac{Q}{100}$$

If Q = 0, Q is taken to be 100. If Q > 200, alarm **402 ILLEGAL NUMBER INPUT** will be issued.

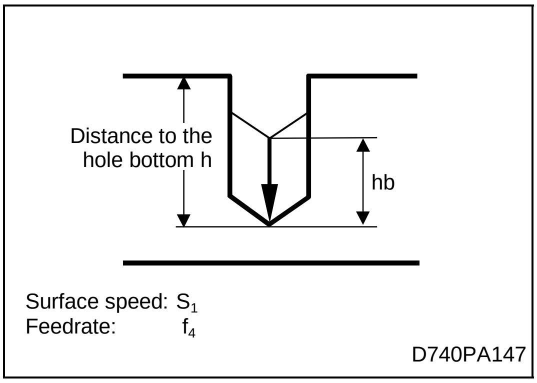


Fig. 3-5

Machining pattern: Very deep through-hole drilling cycle

When hb (feedrate updating distance from the hole bottom) is reached, machining by second surface speed S_2 and second cutting feed f_4 starts.

On the return path (G0/G1) after reaching hb, the tool operates at second surface speed S_2. After the tool has finished machining and returned to its initial point, its speed returns to first surface speed S_1.

Second surface speed S_2 is calculated from surface speed (S_2) set as **C-SP** in the tool sequence and the surface speed updating rate (set with the parameter **D59**), using the following equation.

$$S_2 = S_1 \times \frac{\mathbf{D59}}{100}$$

If **D59** = 0 or **D59** > 100, **D59** is taken to be 100.

Let Q represent the feedrate updating rate (specified under **PRE-DEP** of the tool sequence). Then, second cutting feed f_4 can be calculated from feed f_2 set for the tool sequence, using the following equation.

$$f_4 = f_2 \times \frac{Q}{100}$$

If Q = 0, Q is taken to be 100. If Q > 200, alarm **402 ILLEGAL NUMBER INPUT** will be issued.

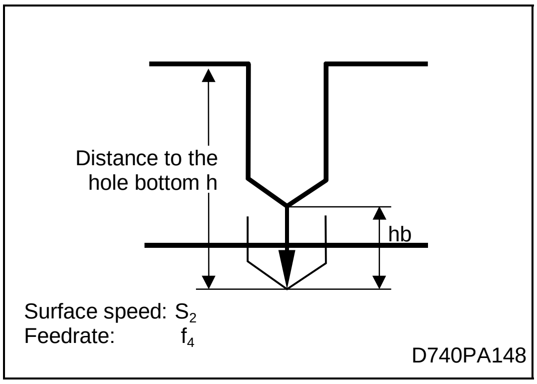


Fig. 3-6

If hb > h (**PRE-DEP** of the tool sequence), hb is taken to be equal to h.

<u>If the cutting area (a) and the feedrate updating distance from the hole bottom (hb) overlap each other:</u>

1) If a and hp overlap between the R-point and the start point:
Between the R-point and the start point, feedrate f_1 for the cutting area (a) is valid and surface speed S_1 set as **C-SP** in the tool sequence is used. Between the start point and the end point, however, feedrate f_4 specified for the feedrate updating distance (hb) is valid. (Fig. 3-7)

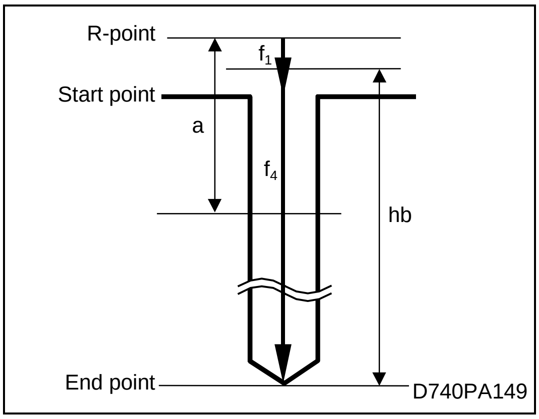


Fig. 3-7

2) If a and hb overlap between the start point and the end point:
Feedrate f_4 for the feedrate updating distance (hb) is valid and second surface speed S_2 is used. (Fig. 3-8)

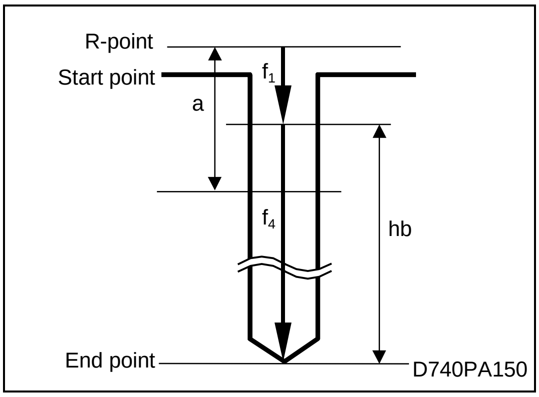


Fig. 3-8

Figures below show the relationship between feedrate updating distance from the hole bottom (hb) and each cutting depth (q)/last cutting depth (qn).

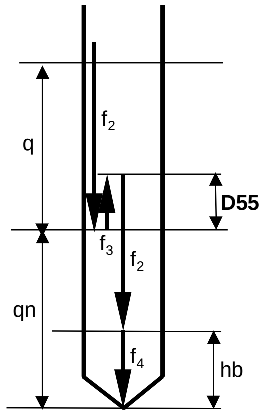
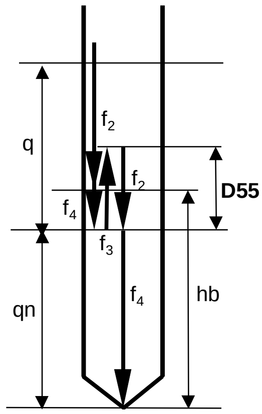
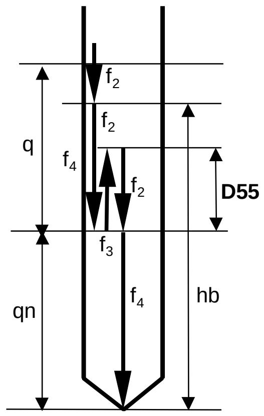
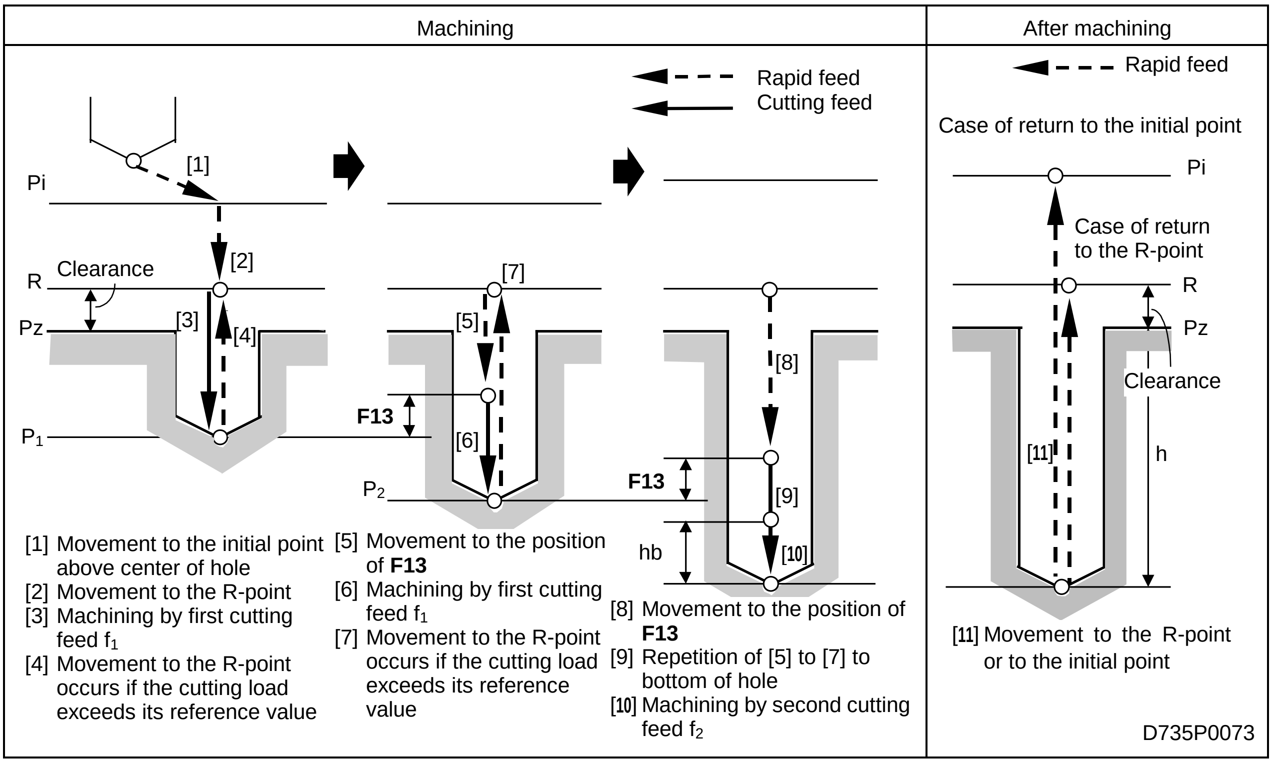
<table>
<tr><td>[1] If feedrate updating distance (hb) < last cutting depth (qn):</td><td>[2] If last cutting depth (qn) ≤ feedrate updating distance (hb) ≤ (qn + D55):</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>[3] If (qn + D55) < feedrate override distance (hb):</td><td></td></tr>
<tr><td>
</td><td>D740PA150</td></tr>
</table>

Fig. 3-9

Note 5: Hole bottom dwell [13] in the figure of tool path for “D. Very deep-hold drilling cycle (PECKING CYCLE 3)” is executed when the deceleration distance is zero, but not when it is more than zero (for very deep stop-hole or very deep through-hole drilling cycles or very deep stop-hole or very deep through-hole drilling cycles with a gradual depth reduction).

E. Auto-pecking cycle of the cutting load detection type (Option) (AUTOPECK CYCLE)

The cutting load torque of the drill is continually monitored during the auto-pecking cycle of the cutting load detection type. This pecking cycle will be performed only if the cutting load exceeds its reference value.



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

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

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG 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

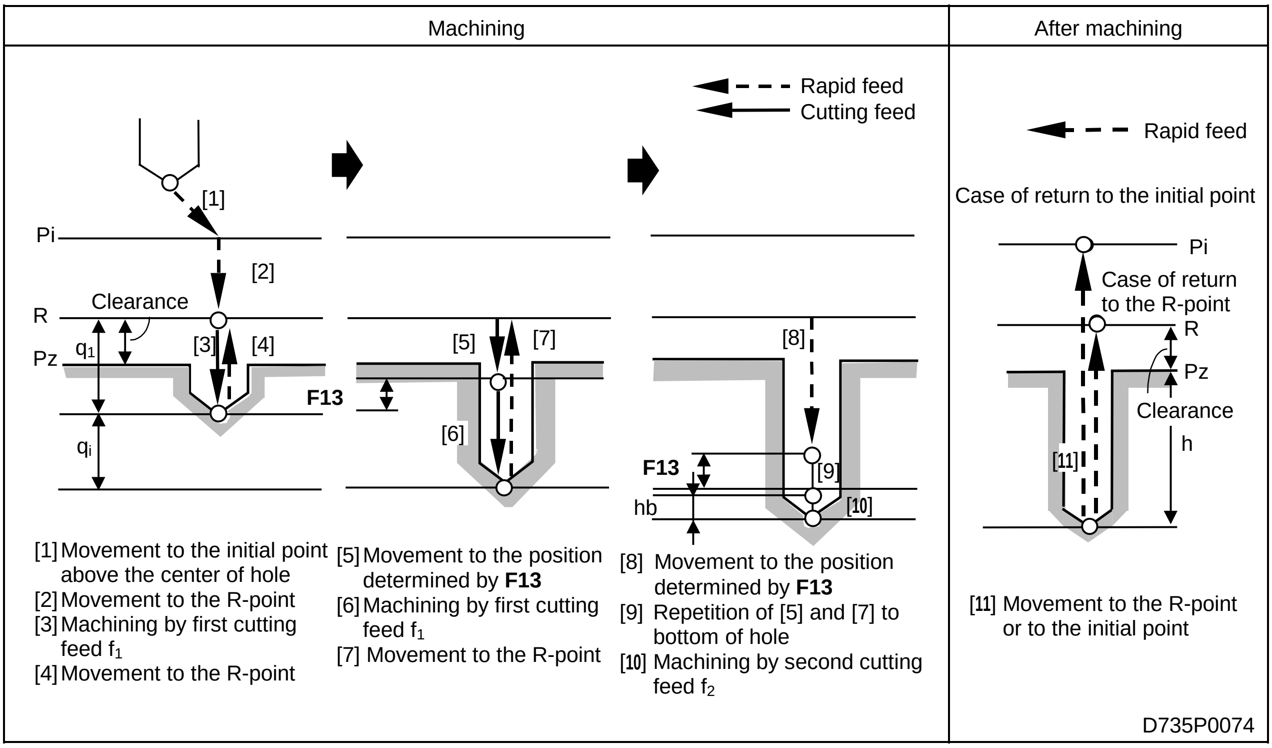
f_1: Feedrate (**FR**) to be set for the tool sequence

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

$f_2 = f_1 \times$ Feedrate updating rate

Note: Cutting load reference value (pecking threshold value) must be set using the DRILL MONITOR function of the **MACHIN. MONITOR** display mode.

F. Deep-hole drilling cycle for gradual depth reduction cycle (DECREME PECKING CYCLE 2)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

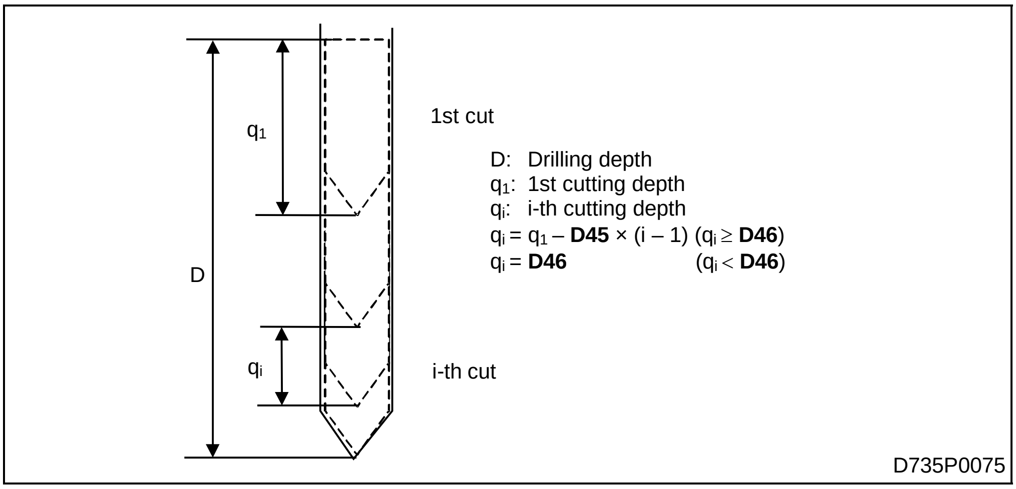
- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

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

h = Depth of machining hole + Tool correction

q_1: Cutting depth to be entered in the tool sequence data (first cutting depth)

q_i: i-th cutting depth
The i-th cutting depth q_i is calculated by the value of the **D45** parameter for drilling gradual reduction depth and of the **D46** parameter for minimum drilling depth as follows.



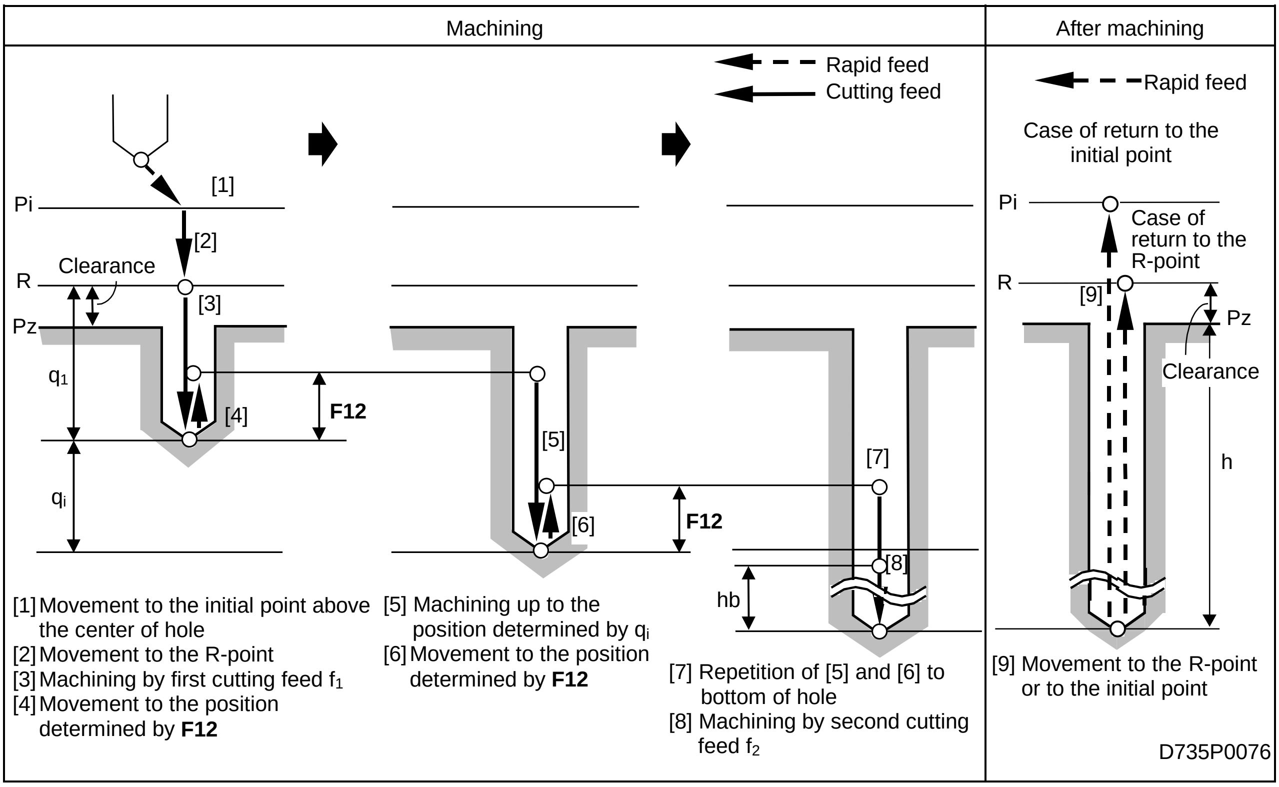
hb: Feedrate override distance from the hole bottom to be determined by the data **PRE-DIA** to be set for the tool sequence

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

G. High speed deep-hole drilling cycle for gradual depth reduction (DECREME PECKING CYCLE 1)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

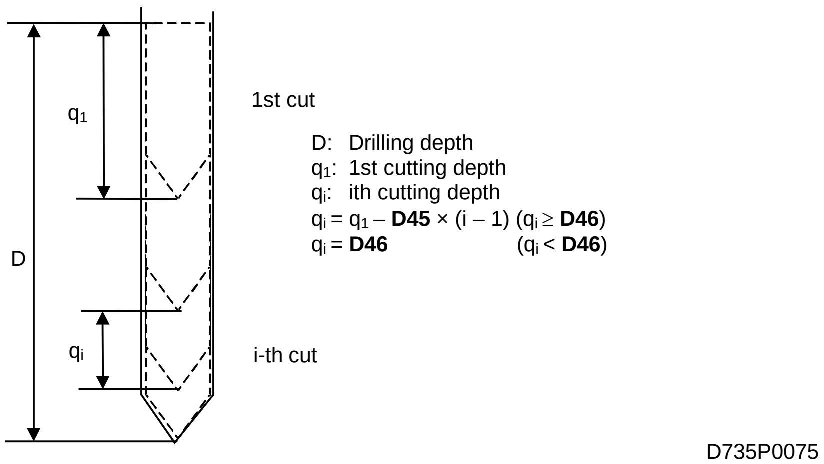
- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

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

h = Depth of machining hole + Tool correction

q_1: Cutting depth to be entered in the tool sequence data (first cutting depth)

q_i: i-th cutting depth
The i-th cutting depth q_1 is calculated by the value of the **D45** parameter for drilling gradual reduction depth and of the **D46** parameter for minimum drilling depth as follows.



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

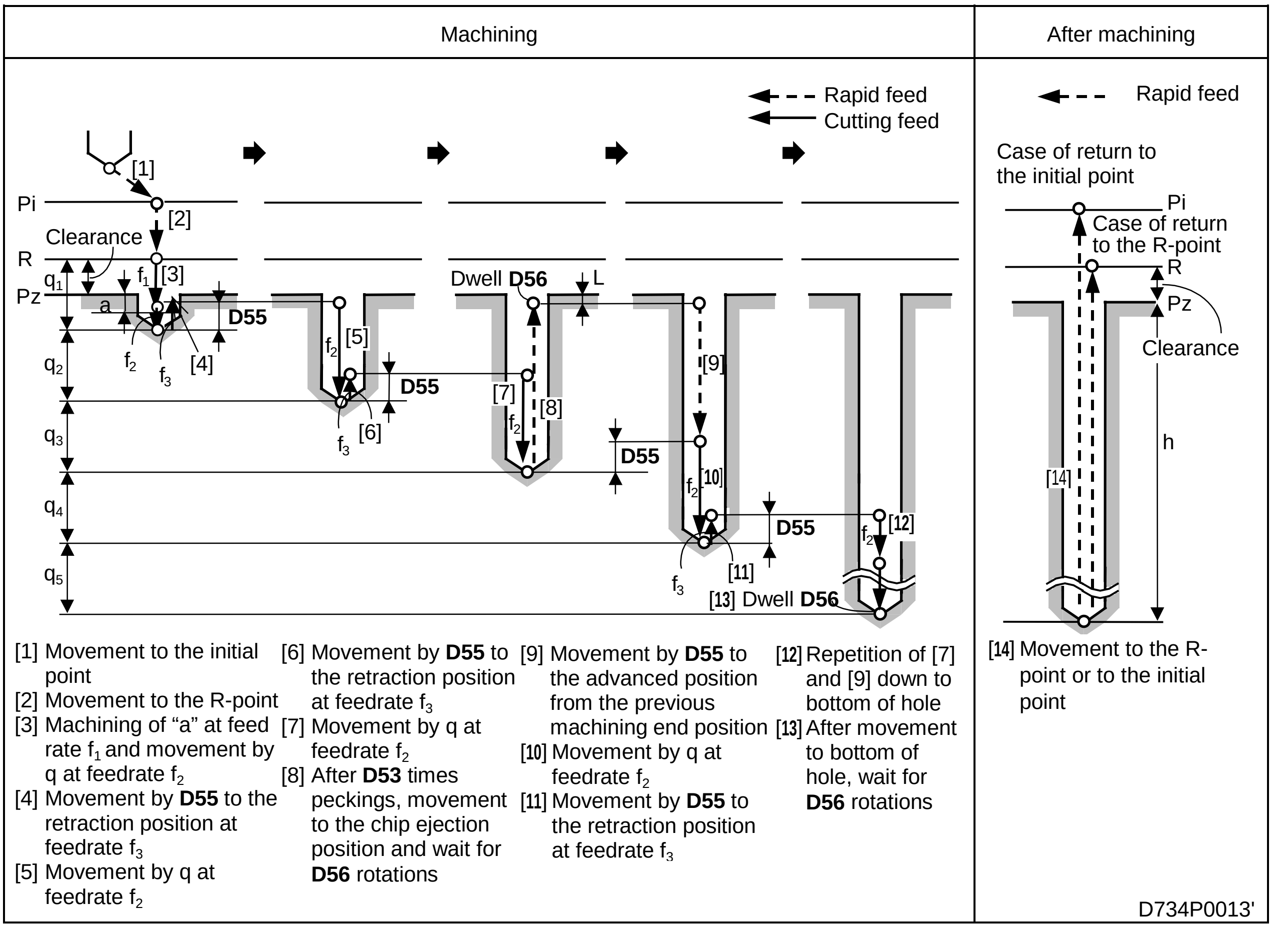
f_1: Feedrate (**FR**) to be set for the tool sequence

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 9999 mm/min or 999.9 inch/min for milimeter or inch specification respectively.

H. Very deep-hole drilling cycle for gradual depth reduction (DECREME PECKING CYCLE 3)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1** or **D42**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG COMP.** (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

a: Cutting area

Machining pattern: Very deep hole drilling cycle with a gradual depth reduction
a = K + R
Where K is **LENG COMP.** (tool correction) on the **TOOL DATA** display and R is the clearance.

Machining pattern: Very deep stop-hole or very deep through-hole drilling cycle with a gradual depth reduction

$a = D \times \frac{\mathbf{D58}}{100}$

Where D is the tool diameter and **D58** (parameter) is the feedrate updating distance rate at the start of cutting.

If **D58** > 300, **D58** is taken to be 100. If a < R, a is taken to be equal to R.

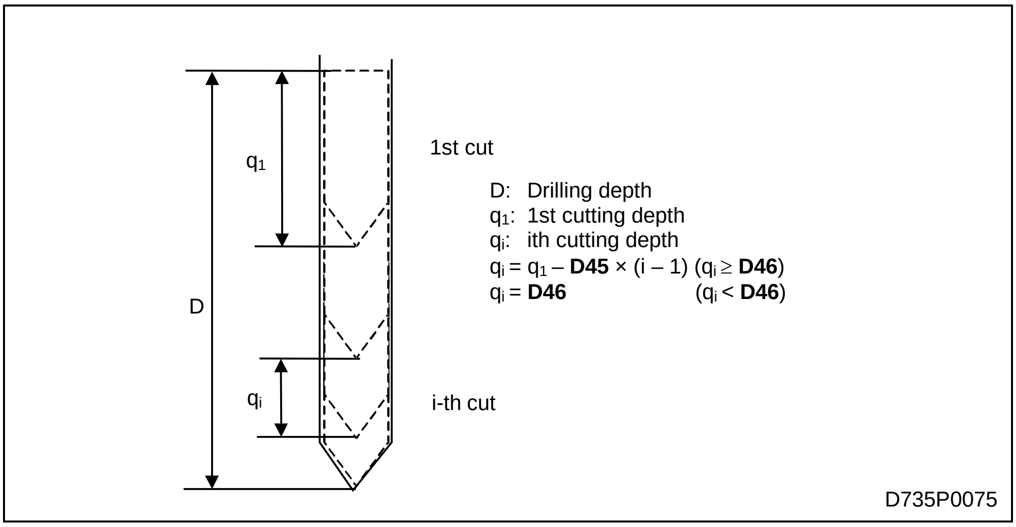
Alarm **748 CANNOT MAKE T-PATH (CHK DEPTH)** will be issued under some conditions. For details, see Fig. 3-4 in Note 3 of "D. Very deep-hold drilling cycle (PECKING CYCLE 3)."

q_1: Cutting depth (**DEPTH**) to be entered in the tool sequence data

q_i: i-th cutting depth

The i-th cutting depth q_1 is calculated by the value of the **D45** parameter for drilling gradual reduction depth and of the **D46** parameter for minimum drilling depth as follows.

Note: If the setting of parameter **D46** is 0 (zero), the minimum allowable cutting depth is 1 mm (or in inch units, 0.04 inches).



f_1: Feedrate (infeed rate) obtained by multiplying "f_2" by the "reduction ratio of the starting speed of cutting" specified in parameter **D54**
where, if **D54** = 0 or if **D54** > 100, then **D54** = 100.

$f_1 = f_2 \times \frac{\mathbf{D54}}{100}$

f_2: Feedrate (**FR**) to be set for the tool sequence

f_3: Pecking retraction speed (= setting of parameter **D57**)
where, if **D57** = 0, then **D57** = 1000.

L: Chip ejection distance calculated from the data **ACT-ϕ** (tool diameter: D) and data **LENG COMP.** (tool correction) on the **TOOL DATA** display

$L = \text{Data LENG COMP.} - \frac{D}{10}$ (D: Tool diameter)

Note 1: During the “n”th cutting operation, if (q × n) < **D55**, retraction through the **D55**-specified distance does not occur. During machining on the path [3], if the first cutting depth of “q” is greater than or equal to (Clearance at R-point + Cutting area “a”), machining at feedrate “f_1” will occur on the path [3] until (Clearance at R-point + Cutting area “a”) is reached, then retraction through the **D55**-specified distance from that position will occur on the path [4], and the workpiece will be cut to the next cutting position (next cutting depth) on the path [5].

Note 2: The feedrate on the path [8] is “G0 speed × **D52**/100”. (If the input value of **D52** is 0, then **D52** = 100.)

Note 3: For very deep stop-hole and very deep through-hole drilling cycles with a gradual depth reduction, the feedrate or the surface speed can be changed in some cases within the feedrate updating distance from the hole bottom (hb specified under the **PRE-DIA** in the tool sequence). For details, refer to Note 4 of “D. Very deep-hold drilling cycle (PECKING CYCLE 3).”

3. Chamfering cutter

Chamfering is classified into two types: Chamfering performed by the tool which only moves on the Z-axis (Cycle 1) and chamfering performed by the tool which moves on the X-, Y- and Z-axes (Cycle 2).

The cycle used is selected automatically.

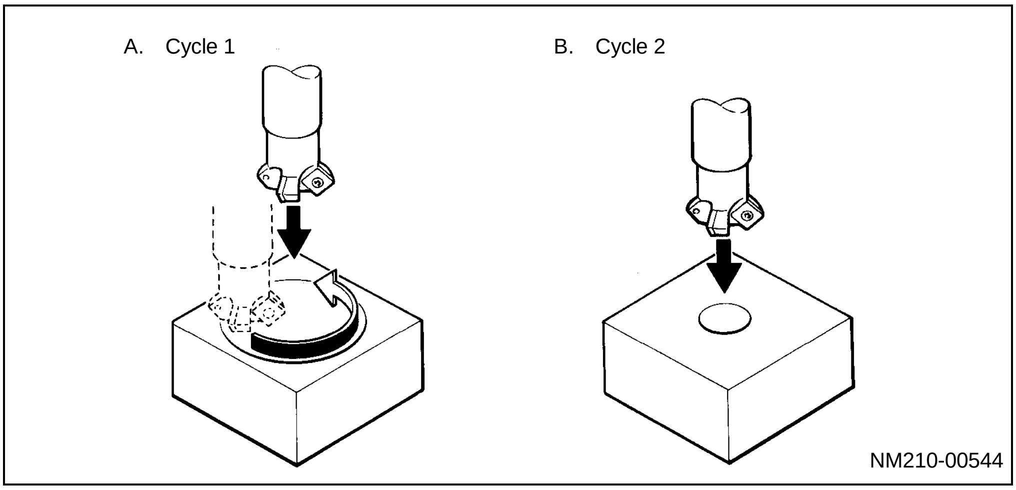


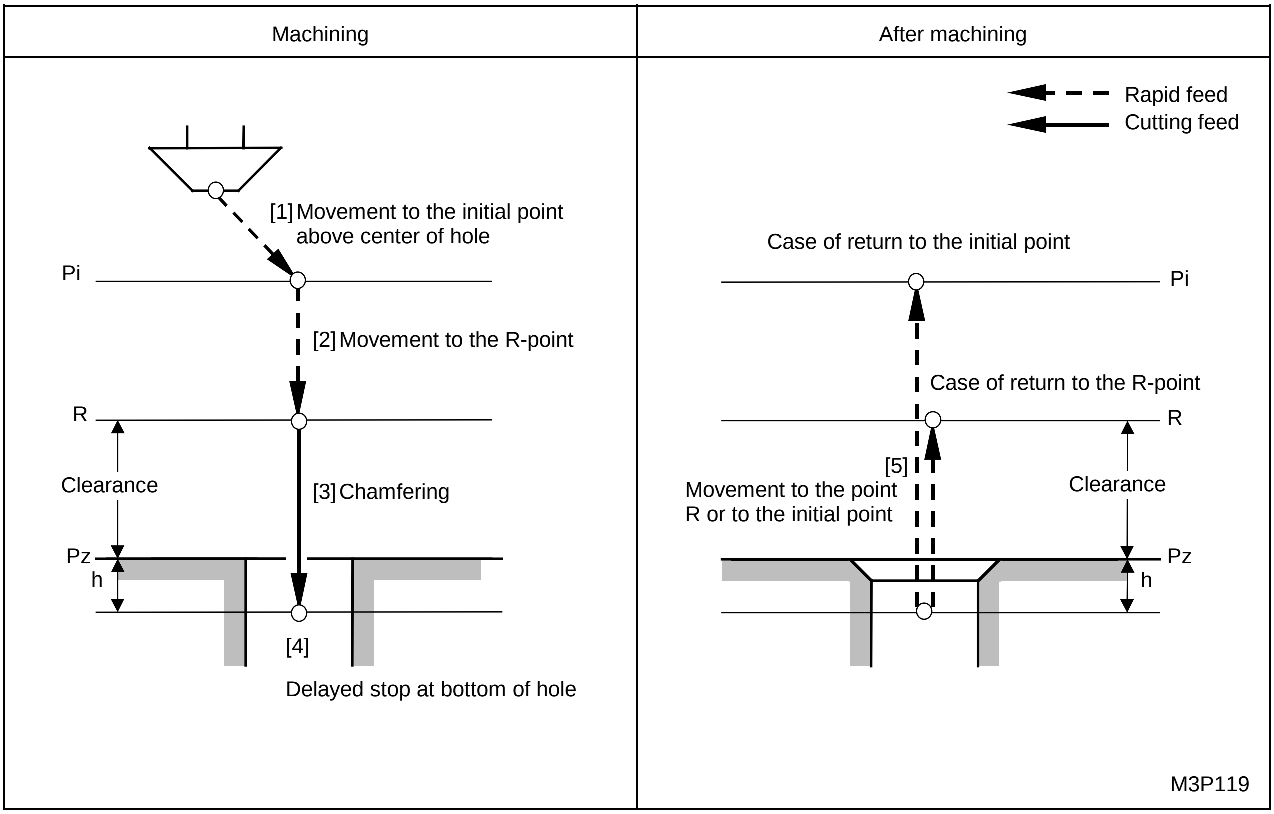
Fig. 3-10 Cycle 1 and cycle 2

Feedrates that are automatically determined vary according to the machining cycle selected. The feedrate in cycle 1 is the feedrate calculated by multiplying the automatically determined feed rate for cycle 2 by the setting of the parameter **D60** (%). The feed rate in cycle 1 is displayed yellow.

Parameter **D60**: Automatic setting ratio of axial cutting feed rate during chamfering in the point machining

The following shows the tool path of the chamfering cutter in each cycle.

A. Cycle 1



The bold codes represent the parameter addresses.

Pi: Initial point

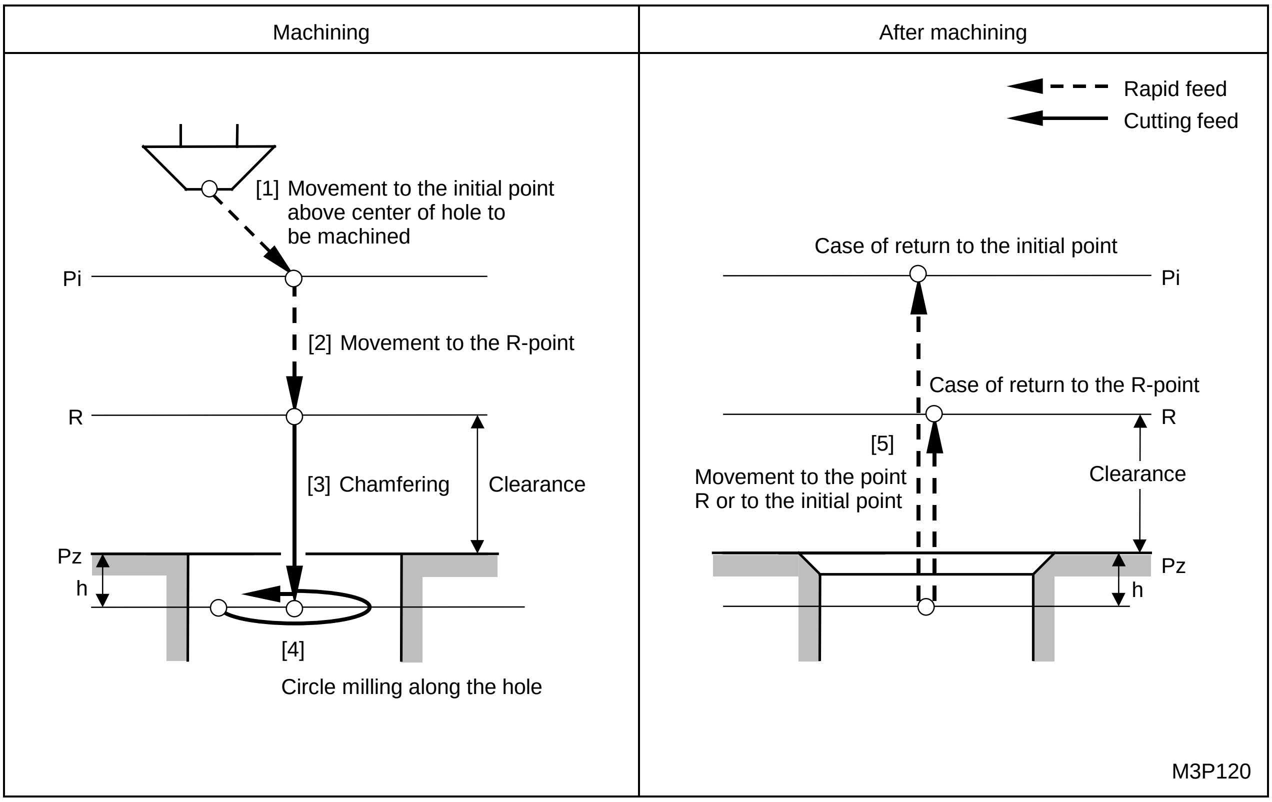
Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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 axial feed at bottom of hole is set by the parameter **D16**.

B. Cycle 2



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following condition is fulfilled, R before machining will be equaled to the parameter **D42**.
However, R after machining is always equaled to (safety clearance).
- 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 1: The time of delayed stop of the axial feed at bottom of hole is set by the parameter **D16**.

Note 2: For the circular milling, refer to the paragraph dealing with 4. End mill, C. Cycle 3.

4. End mill

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

TORNA.: 0 ....circular milling cycle
1 ....circular tornado milling cycle
2 ....precision rapid boring tornado 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 nominal diameter entered in the tool sequence.

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

1. For RGH CBOR and CBOR-TAP units
   - Diameter of machining hole = Nominal diameter (Cycle 1)
   - "Diameter of machining hole > Nominal diameter" and "Diameter of pre-hole > (Tool diameter + Safety clearance)" (Cycle 2)
   - "Diameter of machining hole > Nominal diameter" and "Diameter of pre-hole ≤ (Tool diameter + Safety clearance)" (Cycle 3)
2. For units other than those mentioned above
   - 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 **D23**.

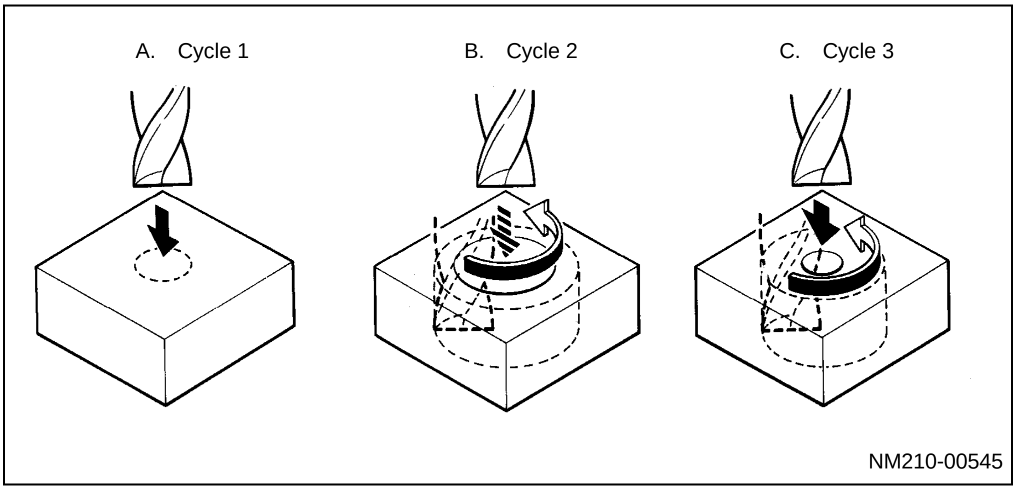
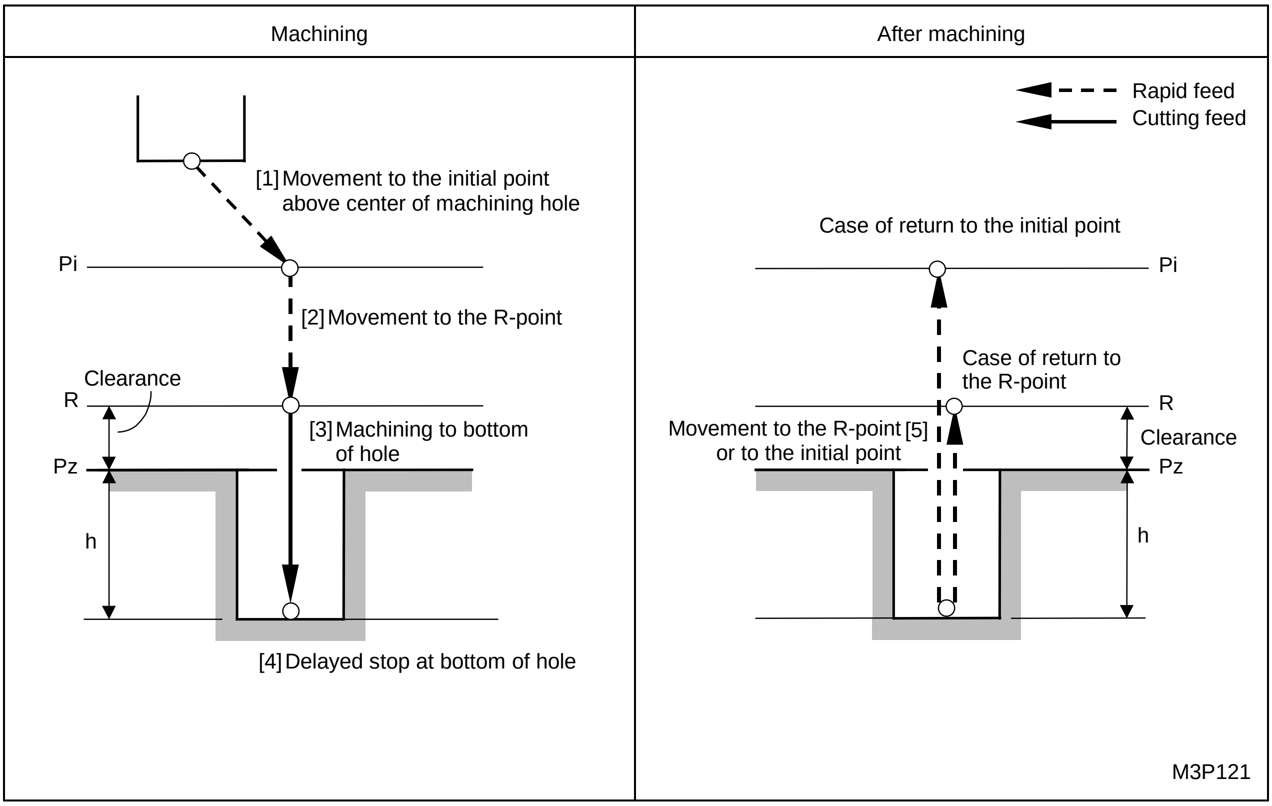


Fig. 3-11 Circular milling cycles 1, 2 and 3

The following shows the tool path of the end mill in each cycle.

A. Cycle 1



The bold codes represent the parameter addresses.

Pi: Initial point

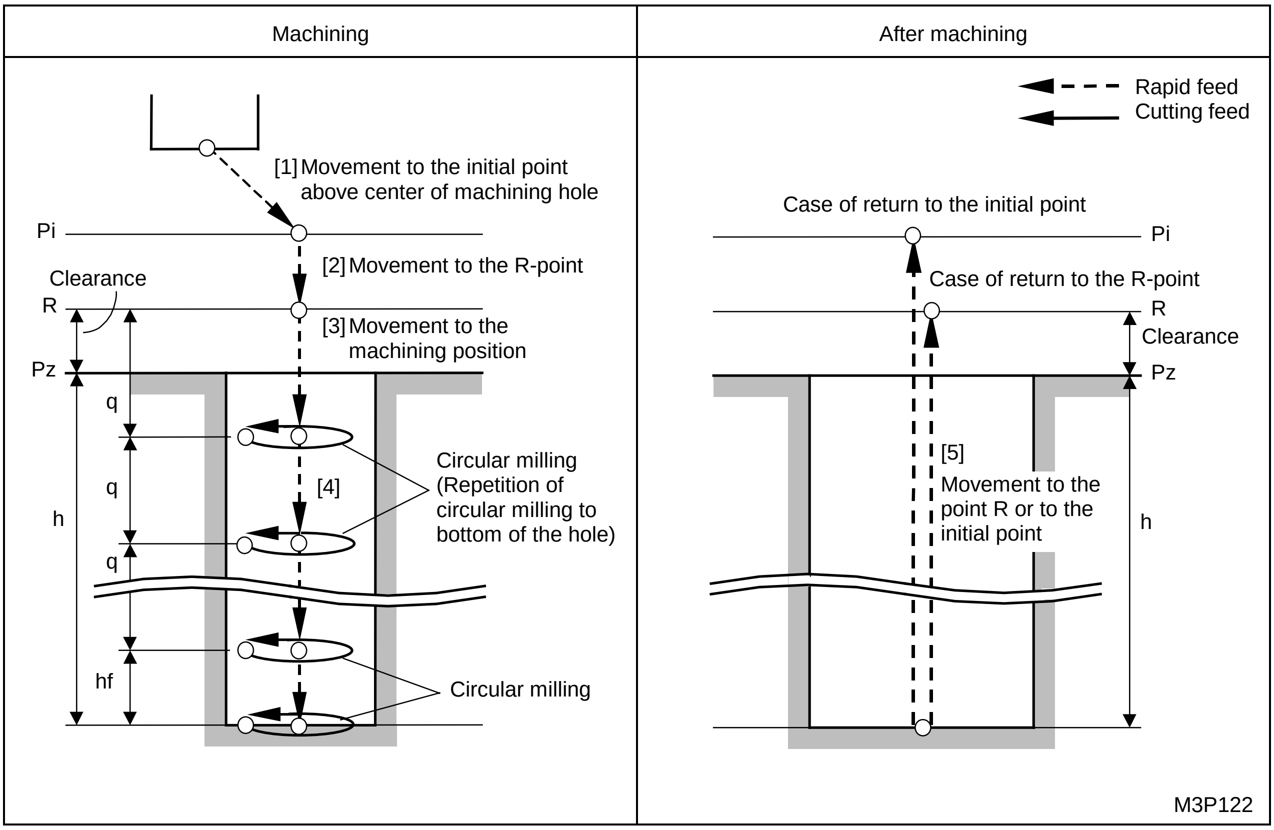
Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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

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

B. Cycle 2



The bold codes represent the parameter addresses.

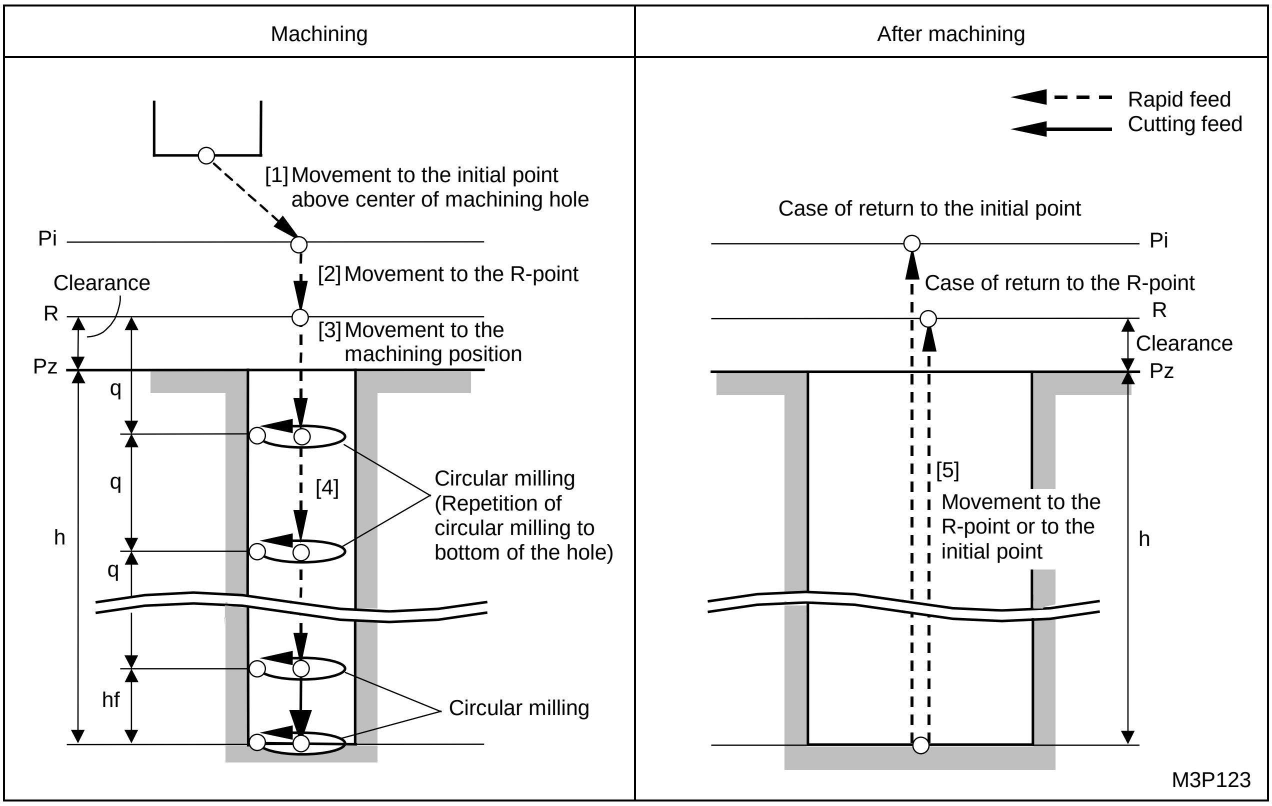
- Pi: Initial point
- Pz: Start point to be entered in the shape sequence
- R: Safety clearance above the point Pz
- h: Optimum distance to be automatically calculated by the data **HOLE-ϕ** and **HOLE-DEP** in the tool sequence and also the data **ANG** on the **TOOL FILE** display
- hf: Bottom finishing allowance to be determined by the data **RGH** entered in the tool sequence and also by the parameter **D21**
- q: Cutting depth in the axial direction per pass to be determined by:

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

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

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

C. Cycle 3



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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 in the axial direction per pass to be determined by:

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

(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.

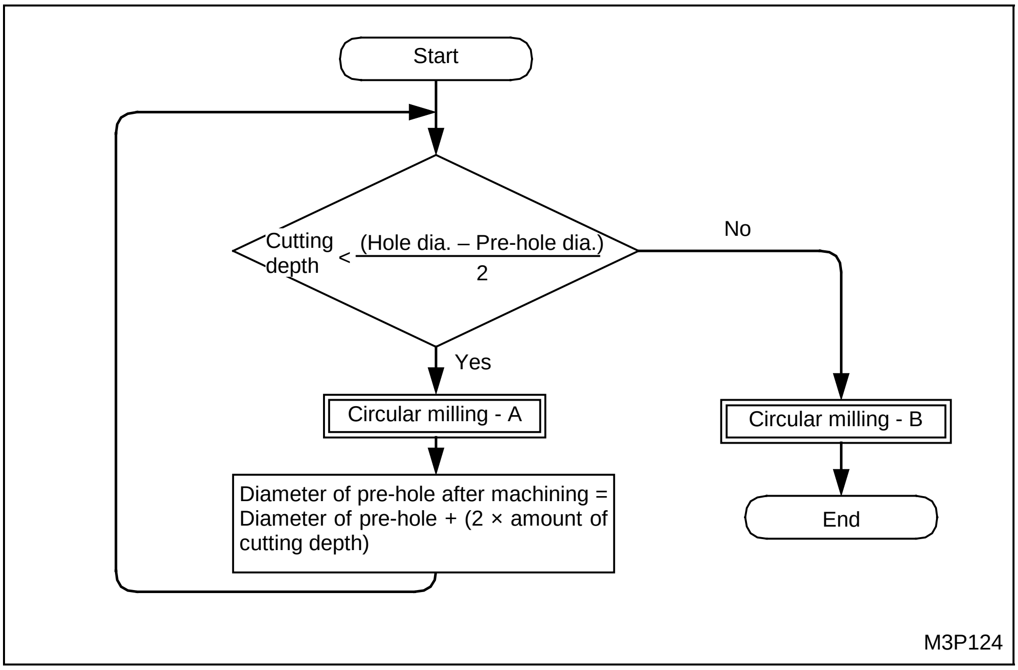


Fig. 3-12 Circular milling

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.

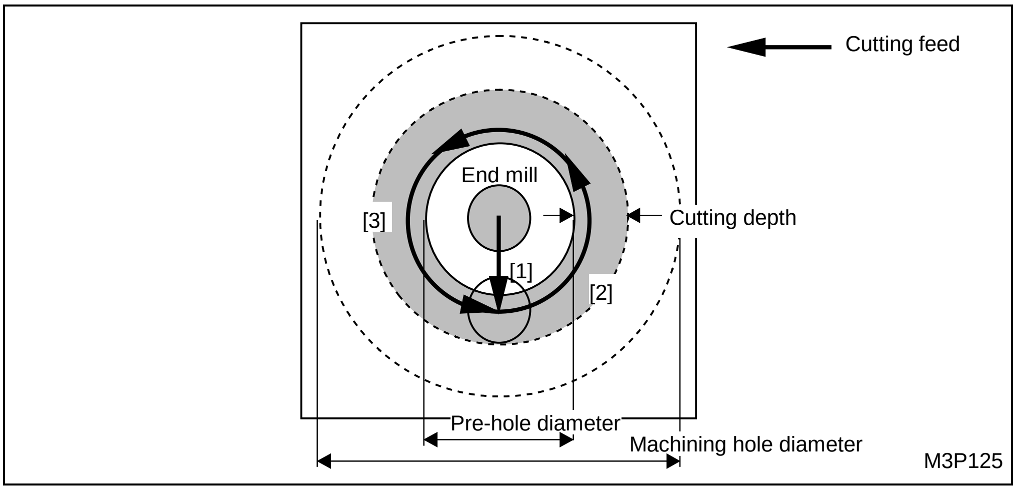


Fig. 3-13 Circular milling-A

- The cutting direction (CW or CCW) can be designated in the program.

Note: The cutting direction designated for the spindle No. 2 is opposite to that for the spindle No. 1.

- 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.

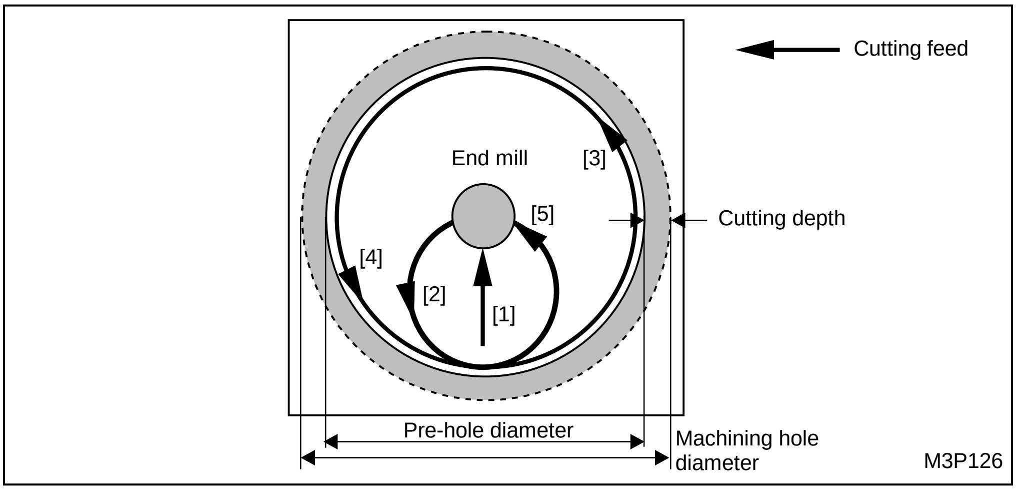


Fig. 3-14 Circular milling-B

- The movement is done in the order [1]→[2]→[3]→[4]→[5].
- The cutting direction is set to the left.
- The movement of [1] starts with the end point of the preceding circular milling-A.

Note: However, when bit 4 (bit 5 in the case of chamfering) of parameter **D91** is 1, the movement of [2] and [5] is done by the following shortened (rapid access) method.

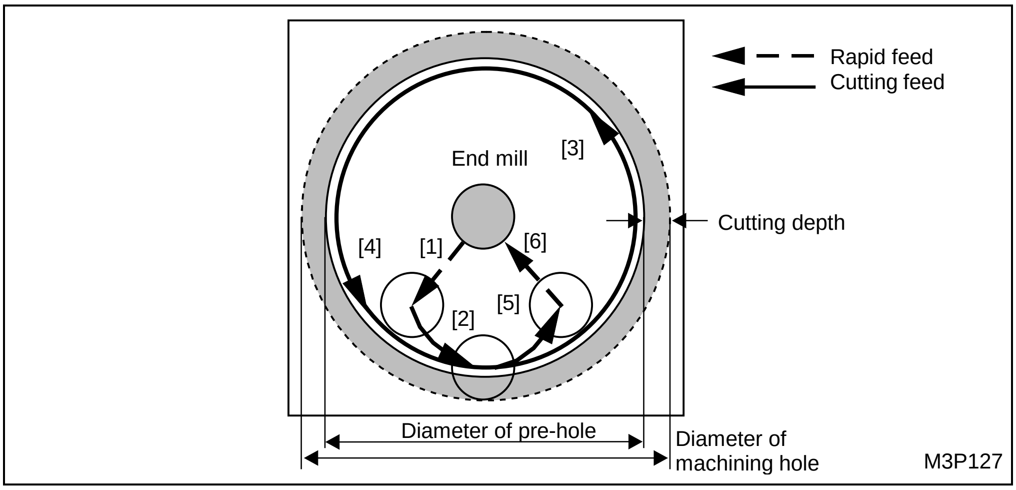
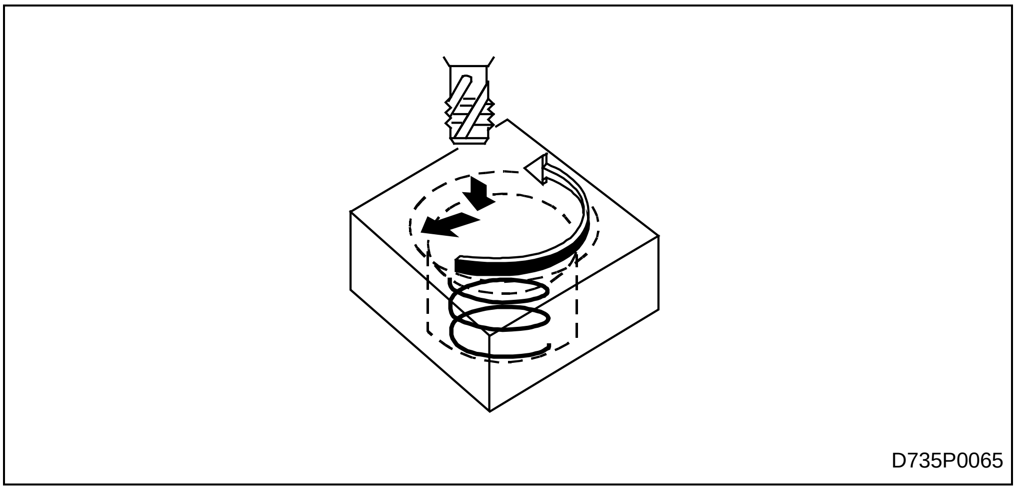
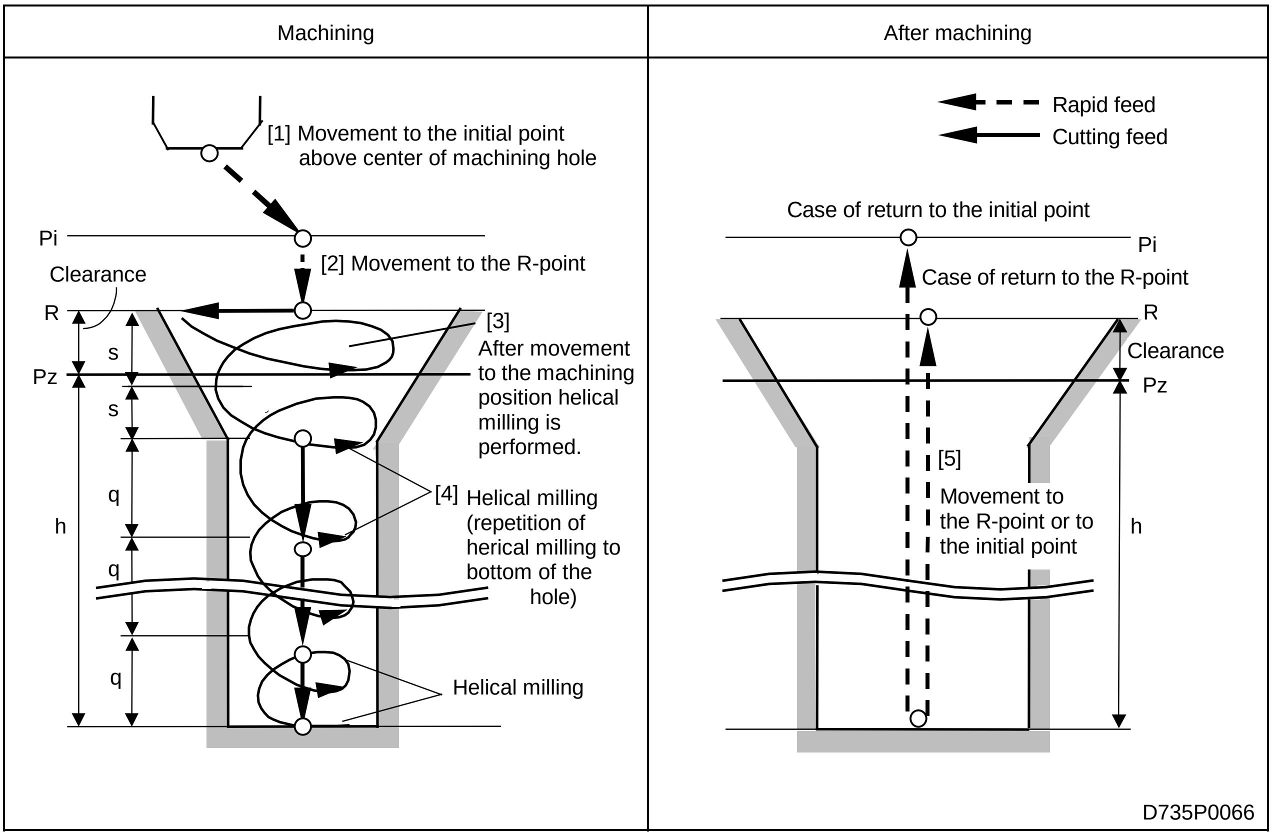


Fig. 3-15 Circular milling-B (case of shortening in chamfering)

- Case of shortening (rapid access) in chamfering is shown above
- The cutting direction (CW or CCW) can be designated in the program.
- The movement is done in the order [1]→[2]→[3]→[4]→[5]→[6].

<In case of circular tornado milling cycle>





The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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.

Circular milling

(1) With bottom finishing

The operation of the machine when it is programmed to perform bottom finishing operations is shown in Fig. 3-16.

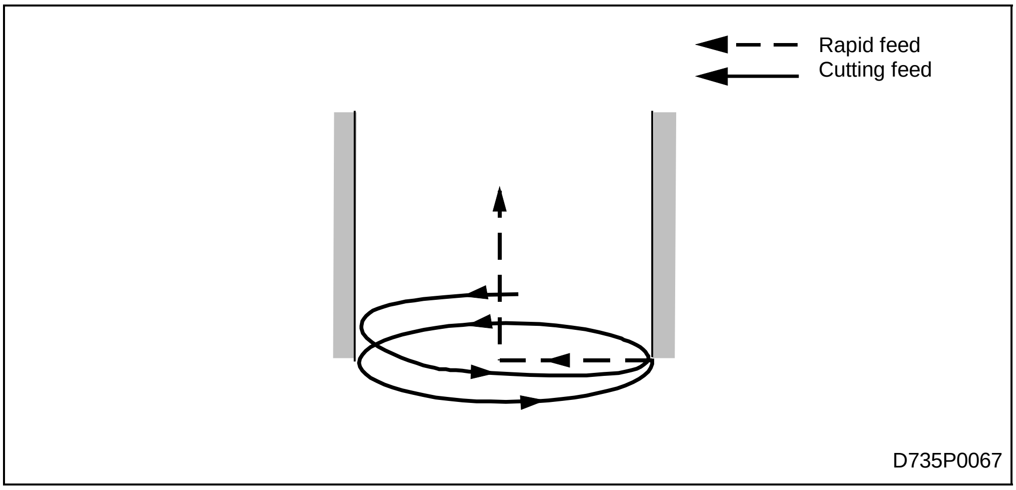


Fig. 3-16 Circular helical processing (with bottom finishing)

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 R-point in the axial direction.

(2) Without bottom finishing

The operation of the machine when it is not programmed to perform bottom finishing operations is shown in the Fig. 3-17.

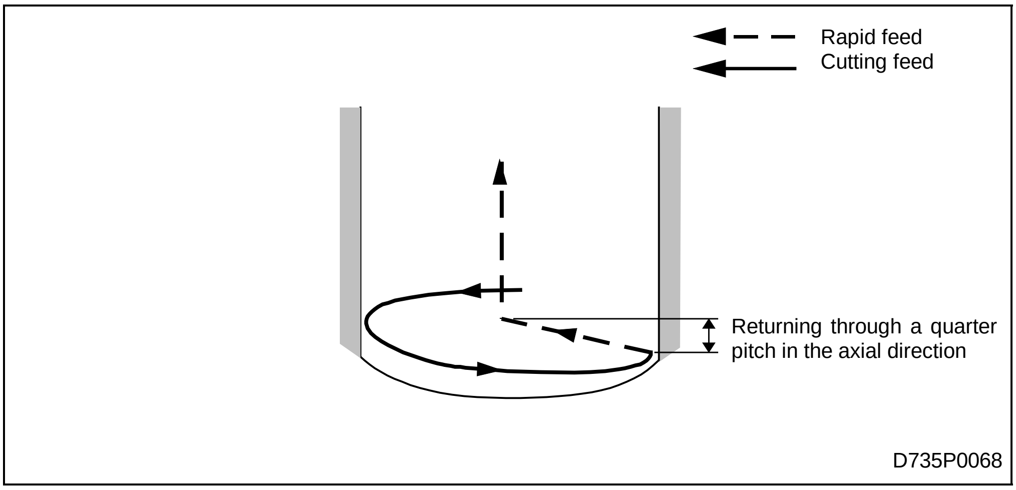
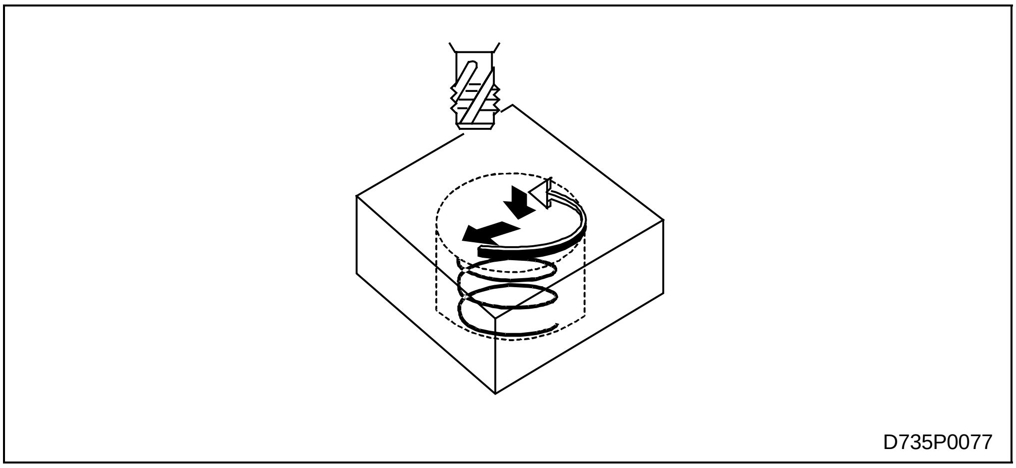


Fig. 3-17 Circular helical processing (without bottom finishing)

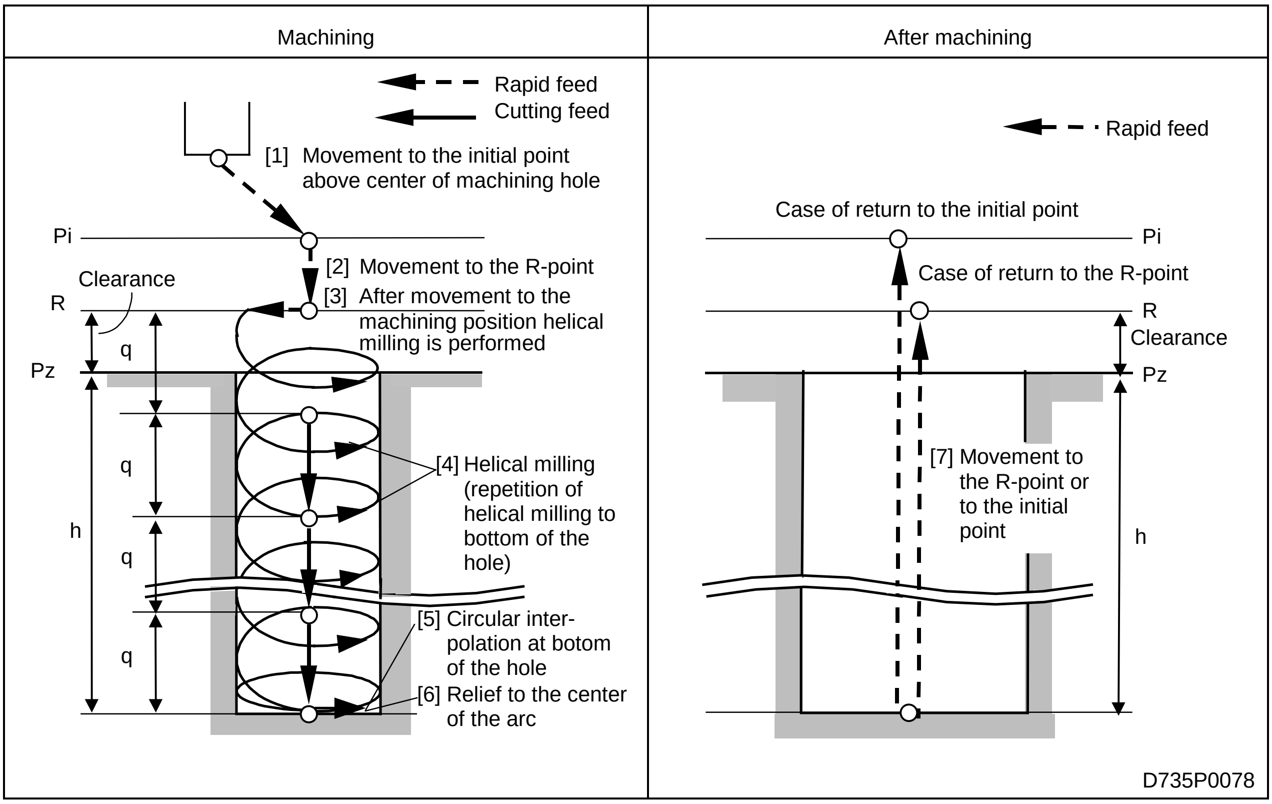
After helical interpolation down to the bottom of the hole, the tool moves to the center of the hole by returning through a quarter pitch in the axial direction and then moves in rapid feed rate to its initial point or to R-point in the axial direction.

The bottom of the hole does not undergo arc interpolation.

<In case of precision rapid boring tornado>



The tool path of end mill is as shown below.



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

h: **HOLE-DEP** in the tool sequence

q: Pitch 2 to be entered in the CIRC MIL unit.
The cutting depth on Z per pass “q” should not be greater than the data entered at **DEPTH** in **TOOL FILE** display.

Note 1: The precision rapid boring tornado cycle (setting 2 at item **TORNA.**) requires the shape correction option.

Note 2: The precision rapid boring tornado cycle (setting 2 at item **TORNA.**) is valid only on the G17 plane.

<u>Circular milling</u>

The movement of circular milling is as shown below.

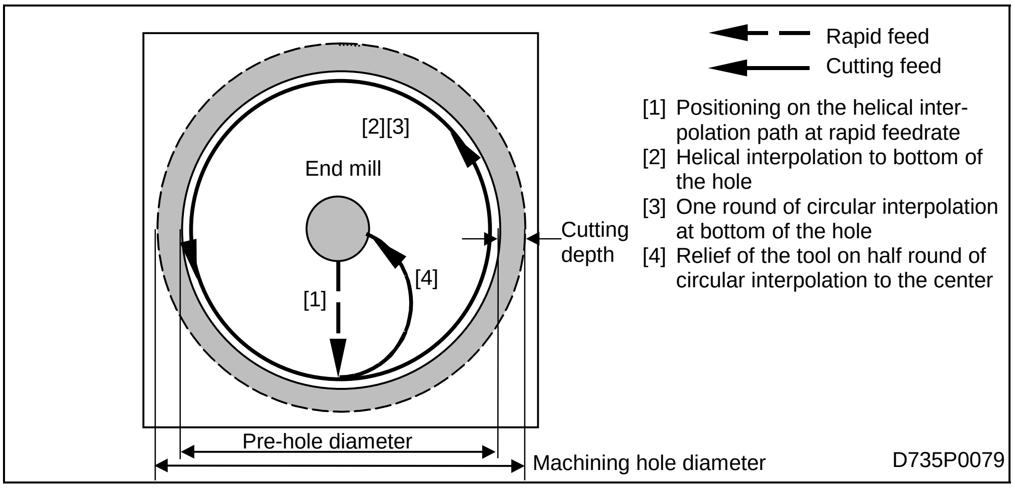


Fig. 3-18 Circular milling

- The movement is done in the order [1]→[2]→[3]→[4].
- The cutting direction (CW or CCW) can be designated in the program.
- The single block function is invalid during the sequence of [2]→[3]→[4].
  If the single block function is specified, the program will come to a single-block stop at the ending point of [4]. The feed hold function, however, is valid.

Note: However, when bit 4 of parameter **D91** is 1, the movement of [4] is done by the following shortened (rapid access) method.

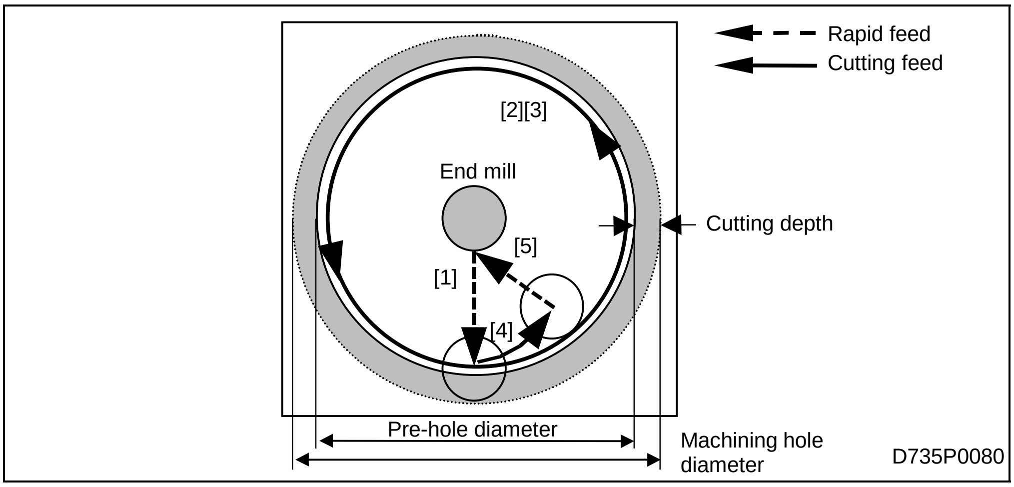
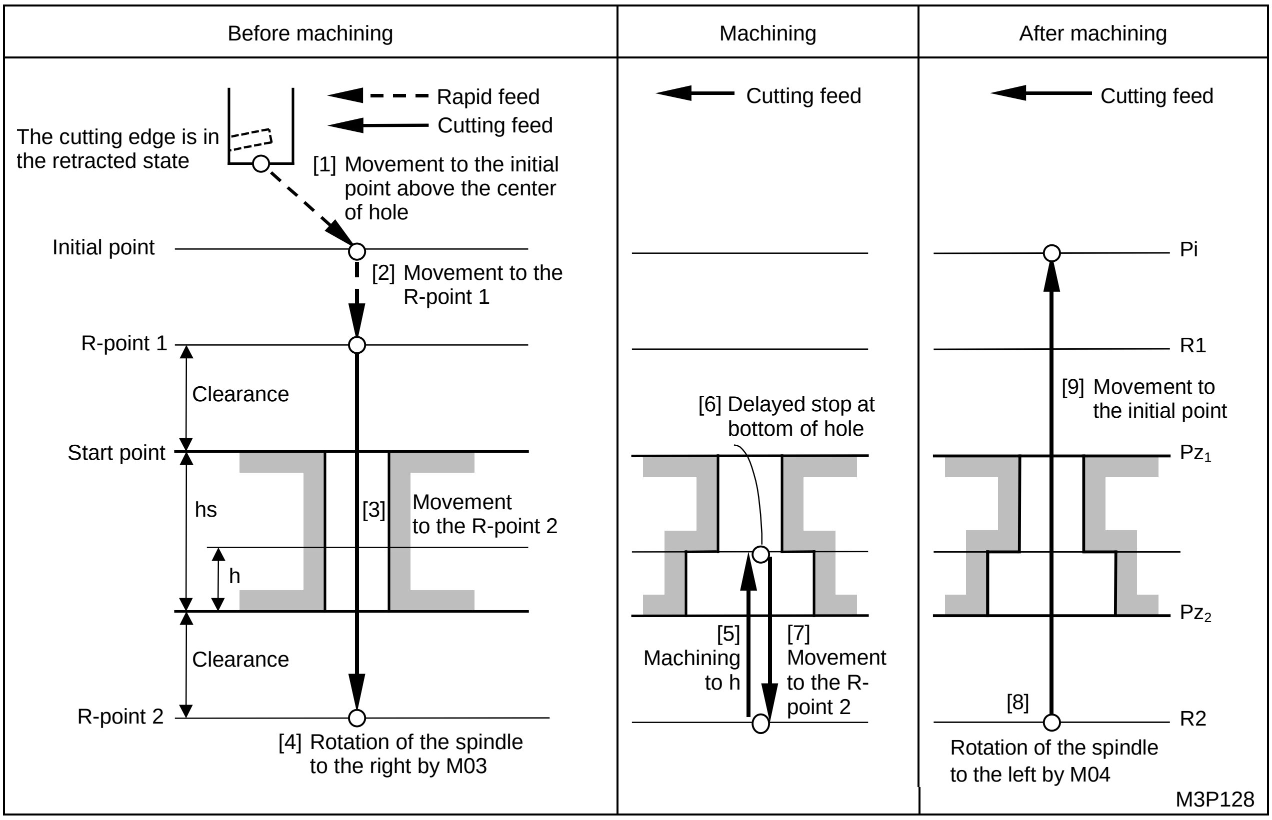


Fig. 3-19 Circular milling (case of shortening in chamfering)

- The cutting direction (CW or CCW) can be designated in the program.
- The movement is done in the order [1]→[2]→[3]→[4]→[5].

5. Back facing tool



The bold codes represent the parameter addresses.

Pi: Initial point

Pz_1: Start point to be entered in the shape sequence

Pz_2: Position at a distance of hs from Pz_1

R1, R2: Safety clearance above the points Pz_1, Pz_2 respectively

Note: As for R1 (in Step [2]), the setting of parameter **D1** becomes valid in case Bit 1 of parameter **D92** is set to "1."

R1-position after machining, however, is always determined by the (safety) clearance.

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

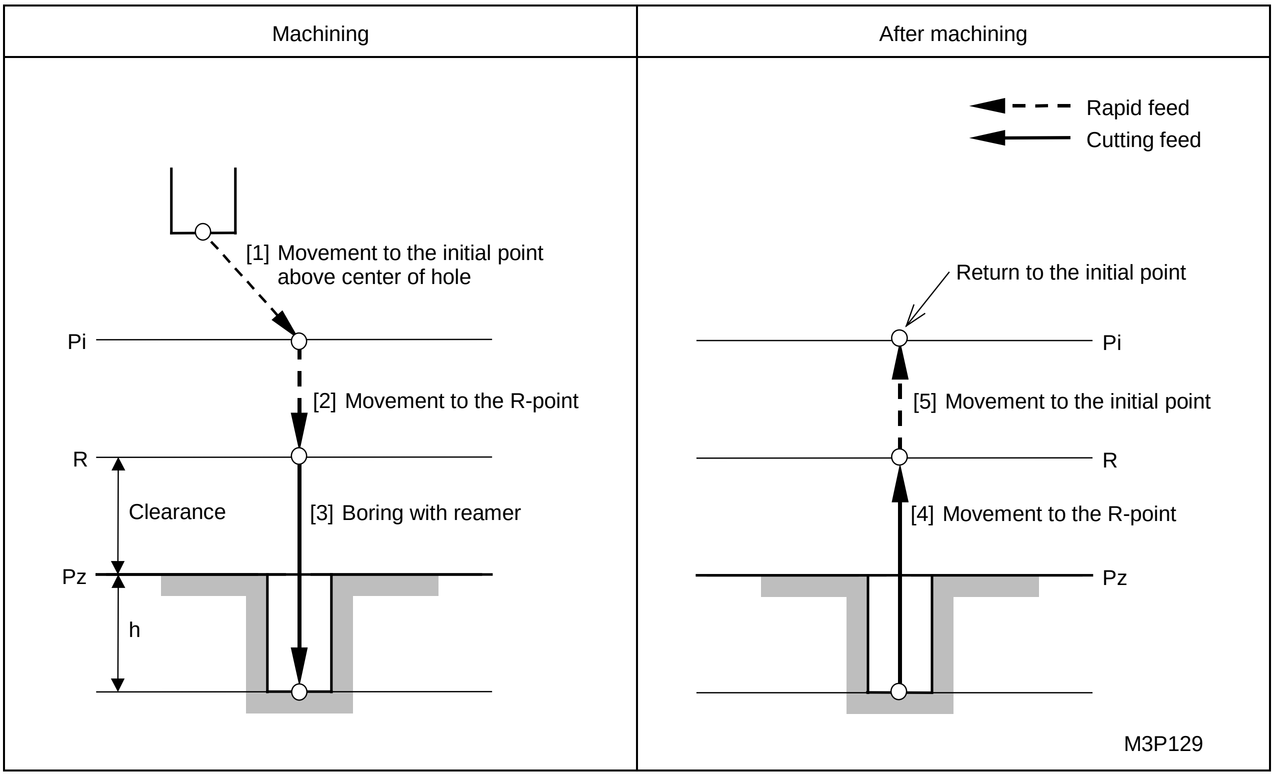
hs: Distance equal to the sum of the depth of pre-hole entered in the tool sequence and the tool data **LENG COMP.** (tool correction) on the **TOOL DATA** display

Note 1: The time of delayed stop of the feed in the axial direction 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 M03 entered in the tool sequence, whilst the rotation to the left is performed by M04.

6. Reamer



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1**.
However, R after machining is always equaled to the (safety) clearance.

- 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 **LENG 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.

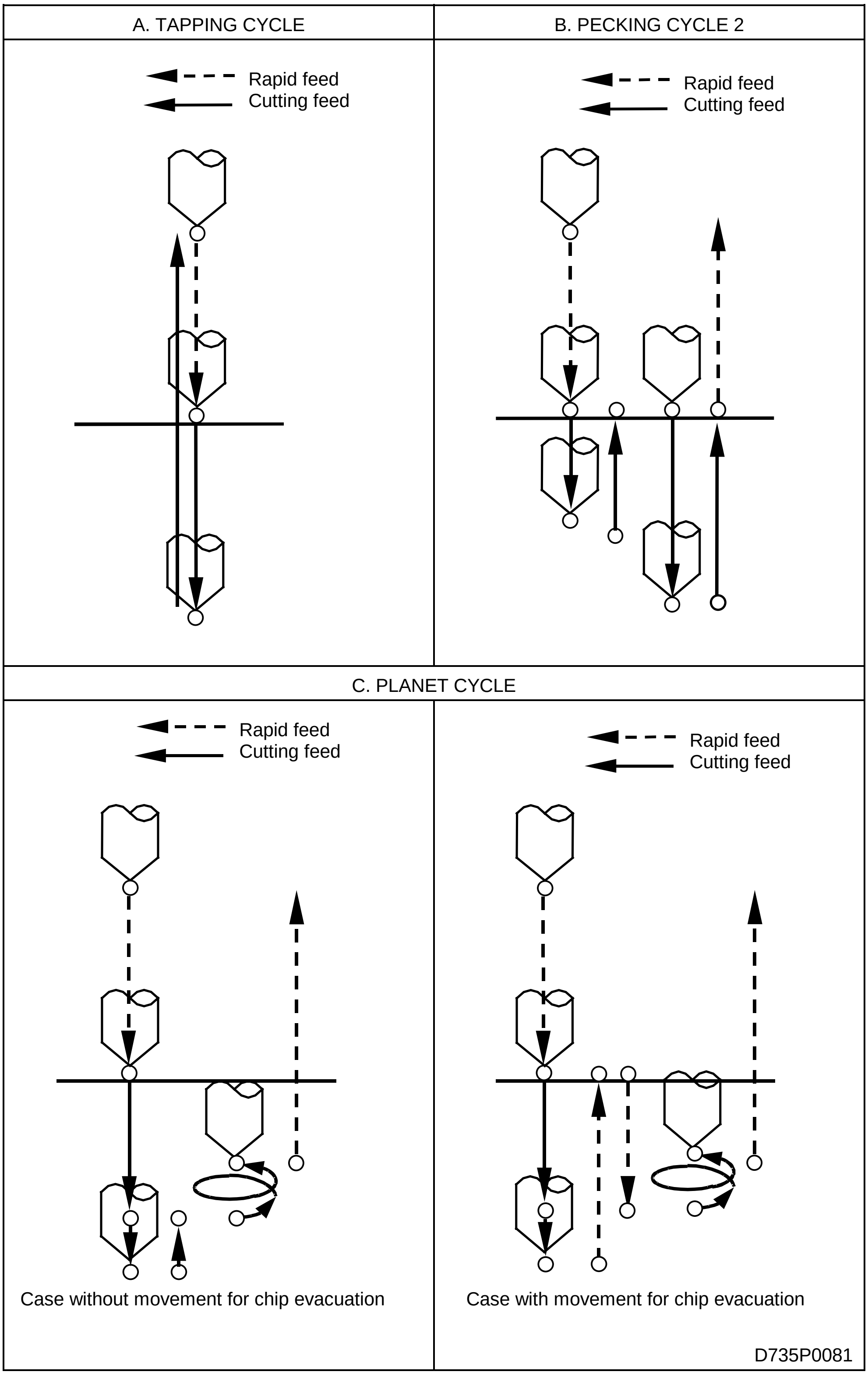
When **[CUT G01]** menu key is pressed .... Parameter **D18**

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

When the value is entered in the item **DEPTH** .... Entered value (/min)

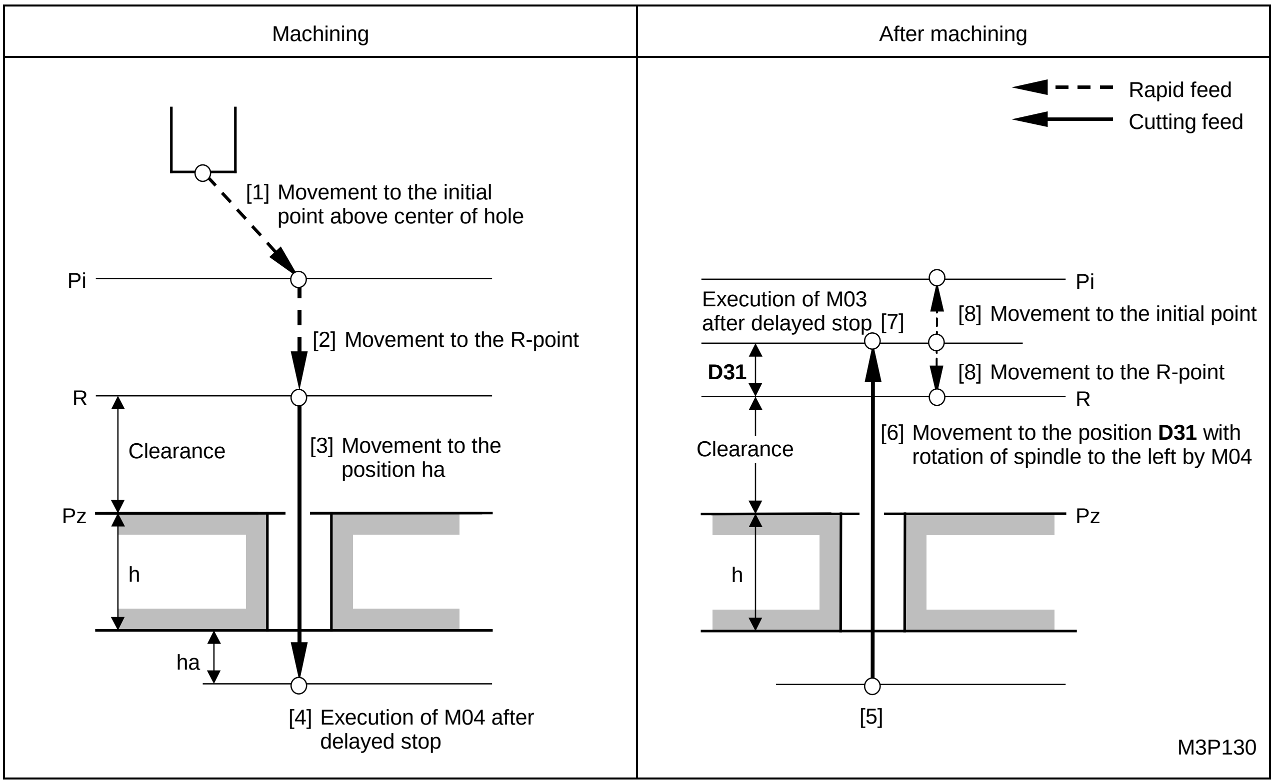
7. Tap

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



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

A. Tapping cycle (TAPPING CYCLE)



The bold codes represent the parameter addresses.

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1**.
However, R after machining is always equaled to the (safety) clearance.
- Case where the bit 3 of parameter **D92** is 1.
- Case where the respective tool sequence contains a chamfering cutter as pre-machining tool.

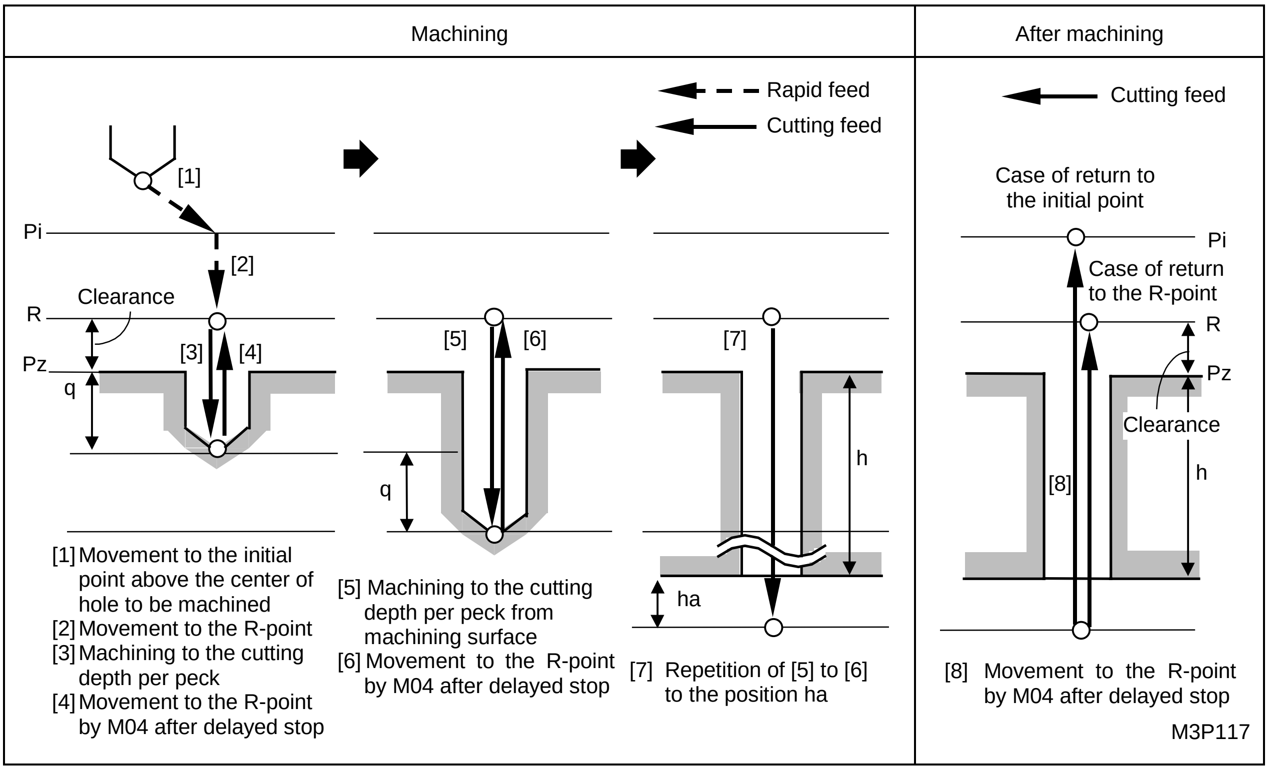
h: Hole depth to be calculated by the data **HOLE-DEP** entered in the tool sequence and also the data **LENG COMP.** (tool correction) on the **TOOL DATA** display
h = Depth of machining hole + Tool correction

ha: Distance to be determined by (*A* – **D32**) × *Pt*
A ....**D30** when using metric and unified screws, **D43** when using pipe screws
Pt ....Pitch entered in the machining unit

Note 1: The entry of 1 in the bit 0 to 2 of parameter **D91** causes the following delayed stop.
Bit 0 .... Delayed stop before execution of M04 at bottom of hole [4]
Bit 1 .... Delayed stop after execution of M04 at bottom of hole [5]
Bit 2 .... Delayed stop before execution of M03 after return [7]
Moreover, the delayed stop is entered in the item **RGH** in the tool sequence. If CYCLE FIX is selected, it will be determined by the parameter **D22**.

Note 2: When M04 is entered in the tool sequence, inversed tapping will be executed.

B. Deep hole drilling cycle (PECKING CYCLE 2)



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: When the following two conditions are fulfilled, R before machining will be equaled to the parameter **D1**.
However, R after machining is always equaled to the (safety) clearance.

- 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 and also the data **LENG COMP.** (tool correction) on the **TOOL DATA** display
h = Depth of machining hole + Tool correction

ha: Distance to be determined by (*A* – **D32**) × *Pt*
A .... **D30** when using metric and unified screws, **D43** when using pipe screws
Pt .... Pitch entered in the machining unit

q: Cutting depth (**DEPTH**) to be entered in the tool sequence data

Note 1: The entry of 1 in the bit 0 to 2 of parameter **D91** causes the following delayed stop.
Bit 0 .... Delayed stop before execution of M04 at bottom of hole [4]
Bit 1 .... Delayed stop after execution of M04 at bottom of hole [5]
Bit 2 .... Delayed stop before execution of M03 after return [7]
Moreover, the delayed stop is entered in the item **RGH** in the tool sequence. If CYCLE FIX is selected, it will be determined by the parameter **D22**.

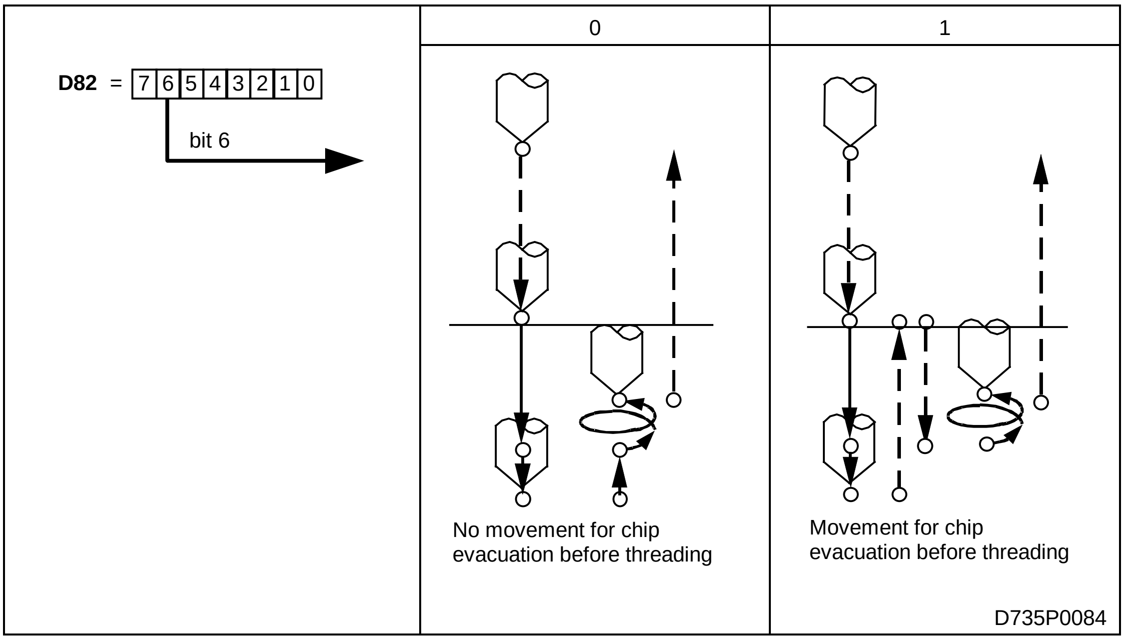
Note 2: When M04 is entered in the tool sequence, inversed tapping will be executed.

C. Planetary tapping (PLANET CYCLE)

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

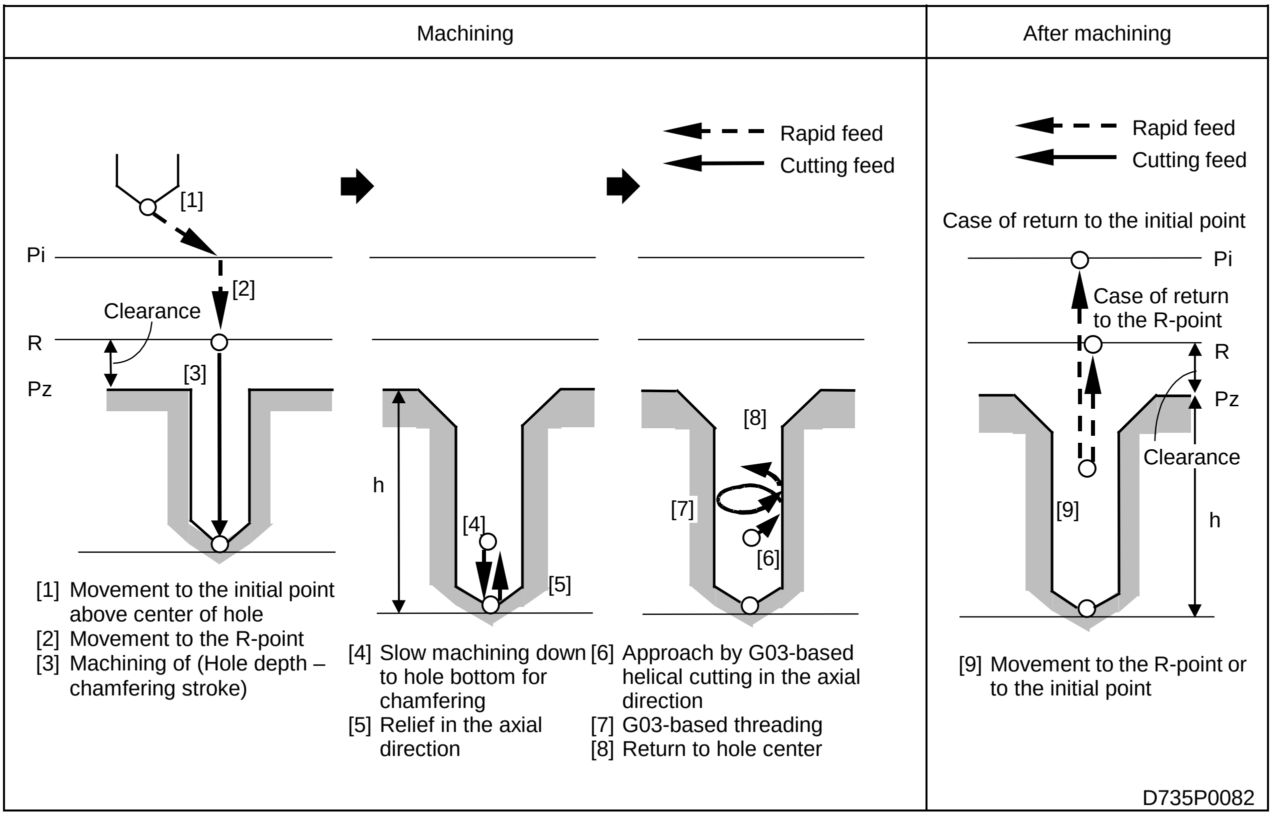
A machining pattern is selected by the parameter **D92**.

- Set either 0 or 1 in the bit 6.



The following shows the tool path in each cycle.

Case without movement for chip evacuation



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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

Note 1: The inversed tapping cannot be executed.

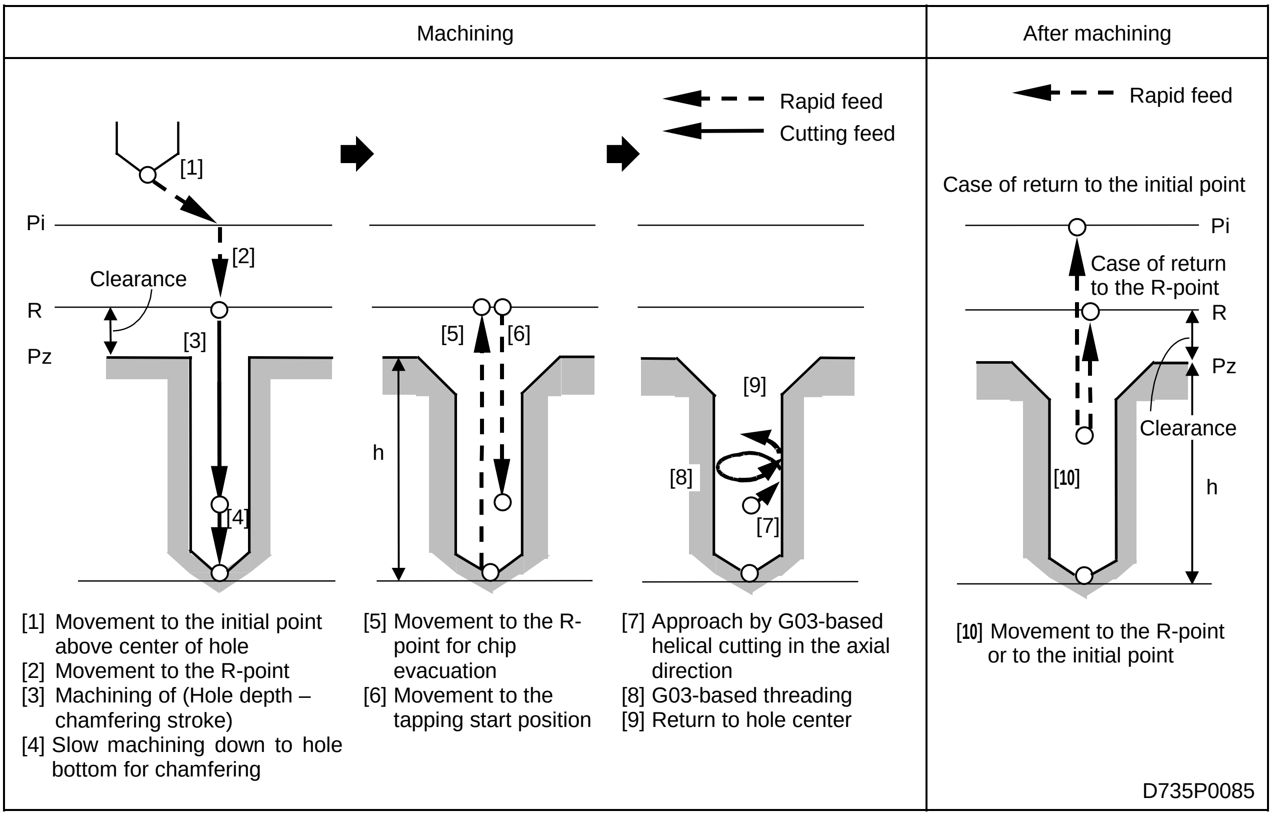
Note 2: The feed rate for chamfering on tool path [4] is calculated as follows:
Chamfering feed = Pre-hole machining feed × Chamfering feed override (parameter **D48**)/100

Note 3: The amount of return from hole bottom on tool path [5] is calculated as follows:
Amount of return = Tapping pitch × Number of thread (parameter **D49**)/10

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

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

Case with movement for chip evacuation



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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

Note 1: The 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 **D48**)/100

Note 3: The distance from hole bottom to the tapping start position on tool path [6] is calculated as follows:
Distance from hole bottom to the tapping start position =
Tapping pitch × Number of thread (parameter **D49**)/10

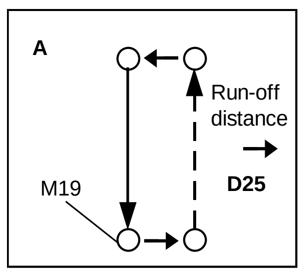
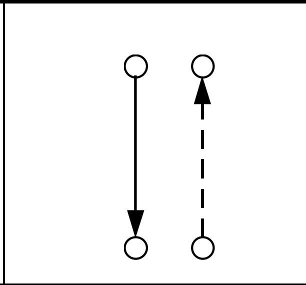
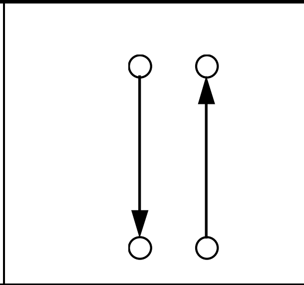
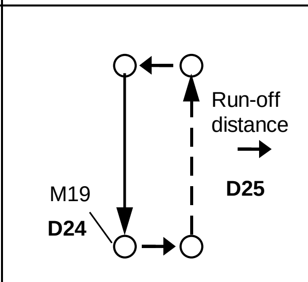
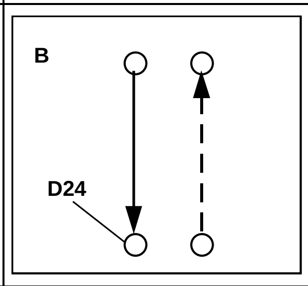
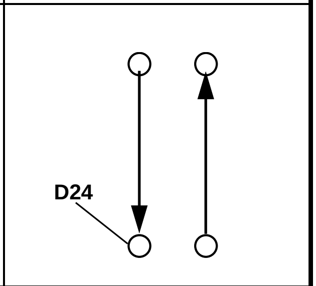
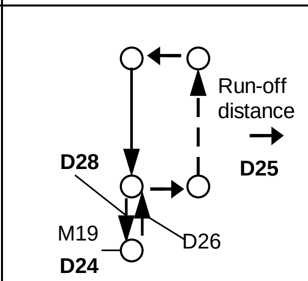
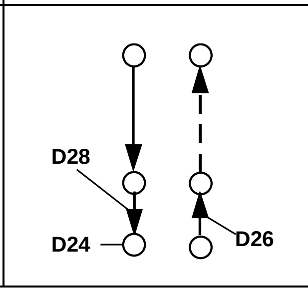
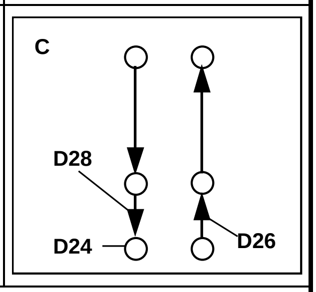
Note 4: The tool diameter in the tool data is modified for tapping diameter correction (fine-adjustment).

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

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, M19, **D24**, **D25**, **D26** and **D28** shown on the figure above, are explained.

M19: M-code to stop the spindle in the pre-determined position (Orientation of spindle)

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.

D25: Parameter to determine the run-off distance on the X-Y plane. (Refer to Fig. 3-20.)
The spindle is oriented at the bottom of the hole. The tool moves to the initial point or to R-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.

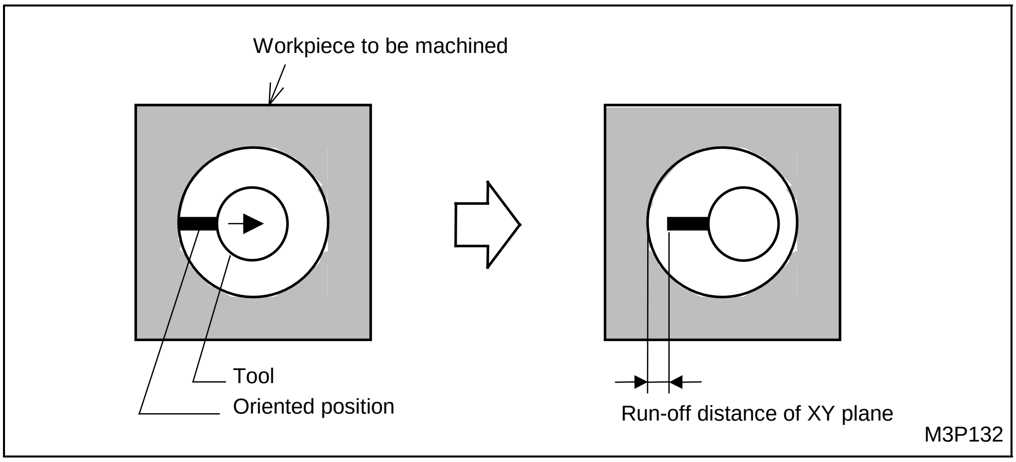


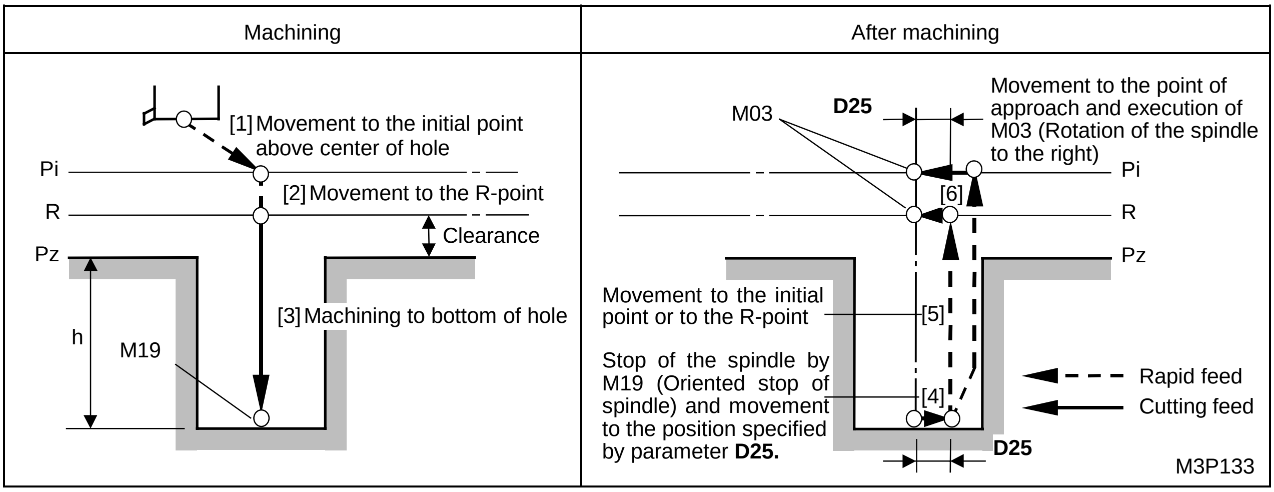
Fig. 3-20 Run-off distance on the X-Y plane

D26: Parameter to determine the run-off distance on the Z-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: Start point to be entered in the shape sequence

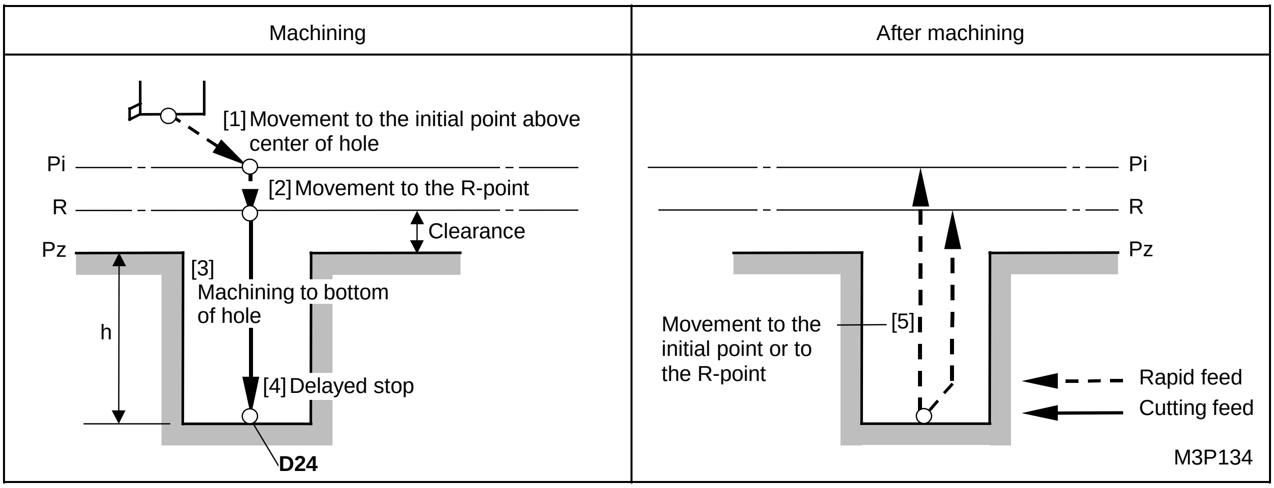
R: Safety clearance above the point Pz

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

Note 1: Direction of run-off distance on the XY plane (bit 3 and bit 4 of parameter **I14**)

Note 2: When M04 is entered in the tool sequence, the spindle rotates to the right.

B. Cycle 2 with roughness 1



The bold codes represent the parameter addresses.

Pi: Initial point

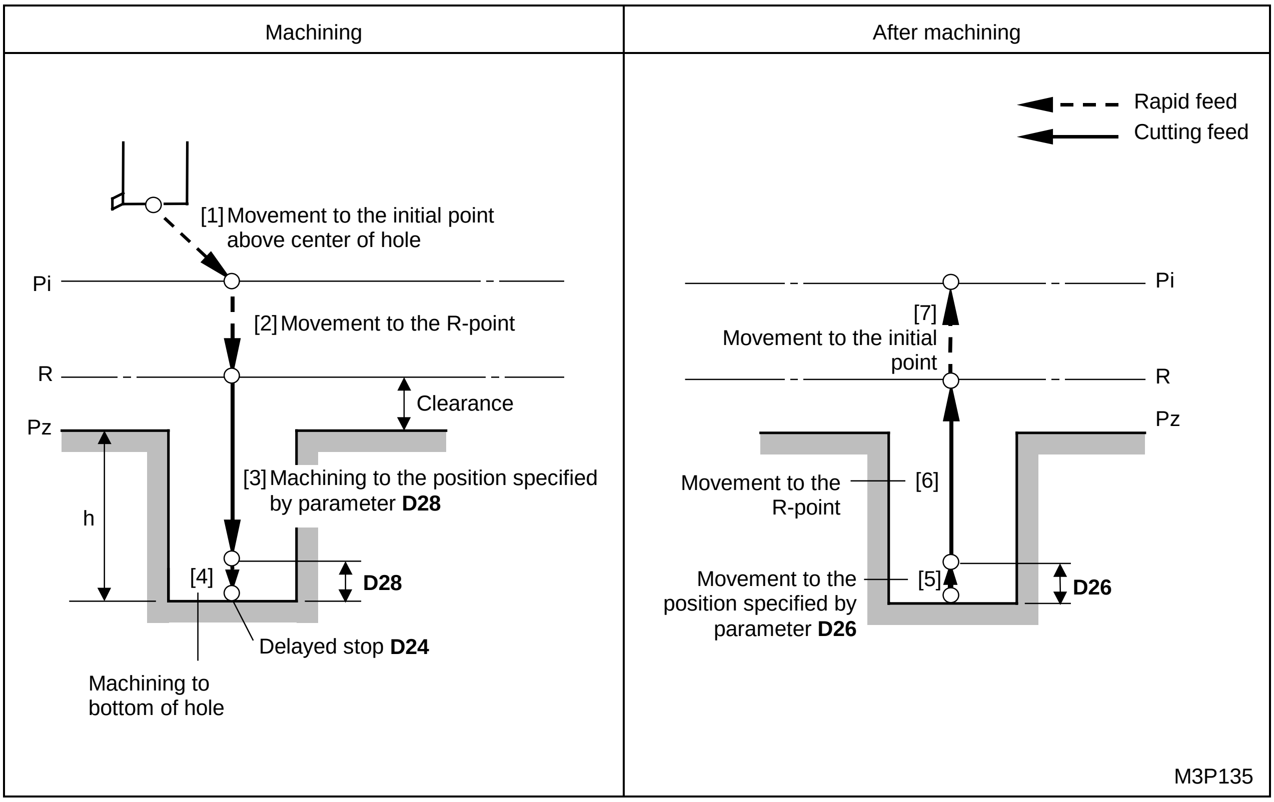
Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

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

Note: The delayed stop time of the axial feed at the bottom of the 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: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

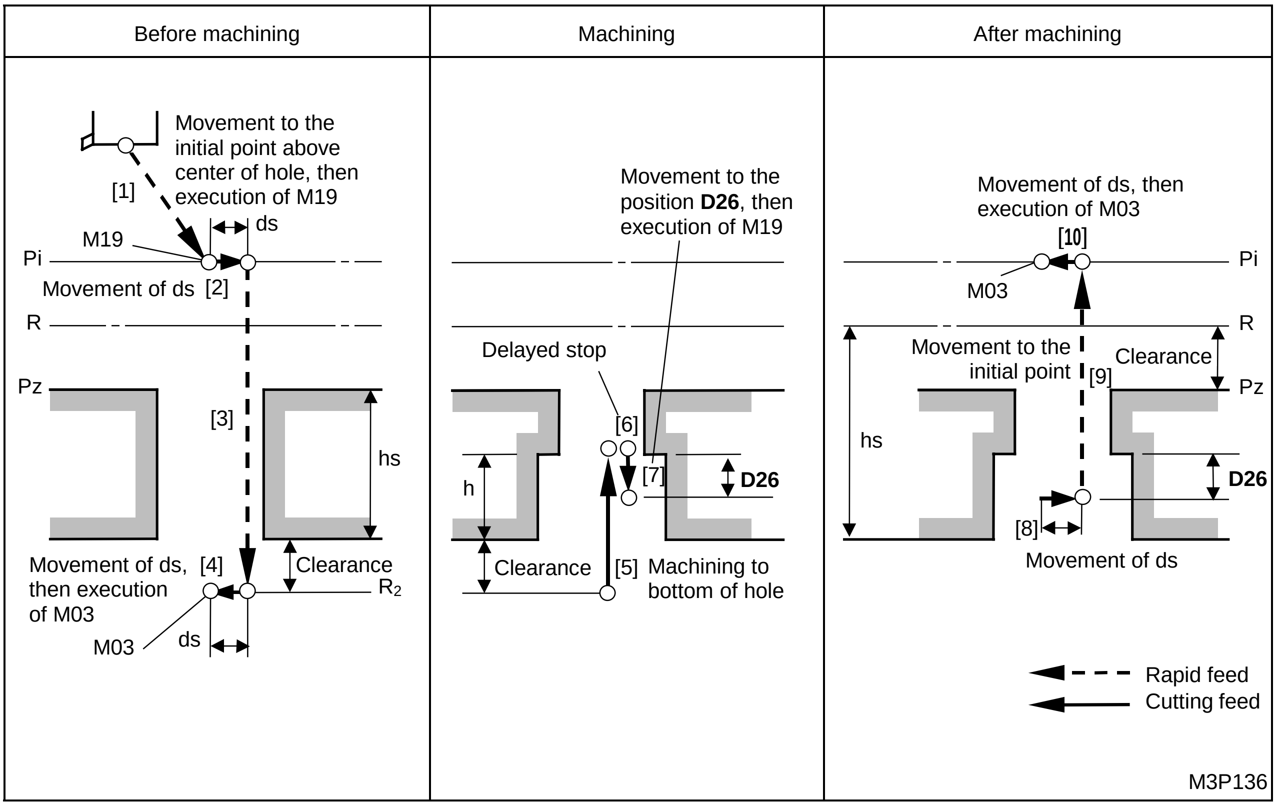
h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the **data LENG COMP.** (tool correction) in the **TOOL DATA** display

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 axial 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: Start point to be entered in the shape sequence

R, R_2: Safety clearance in the axial direction

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

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

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

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

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

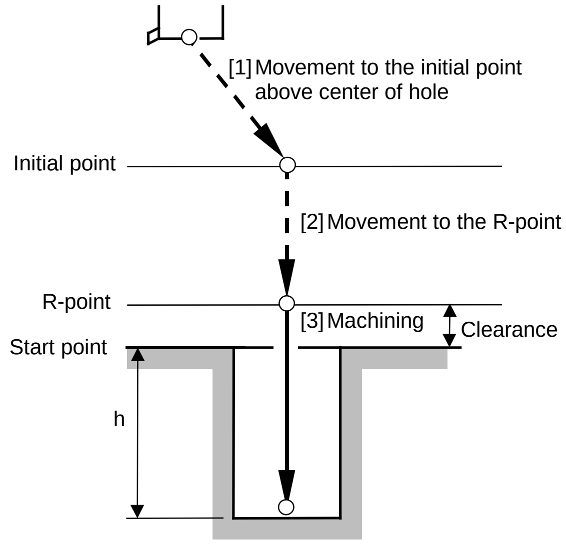
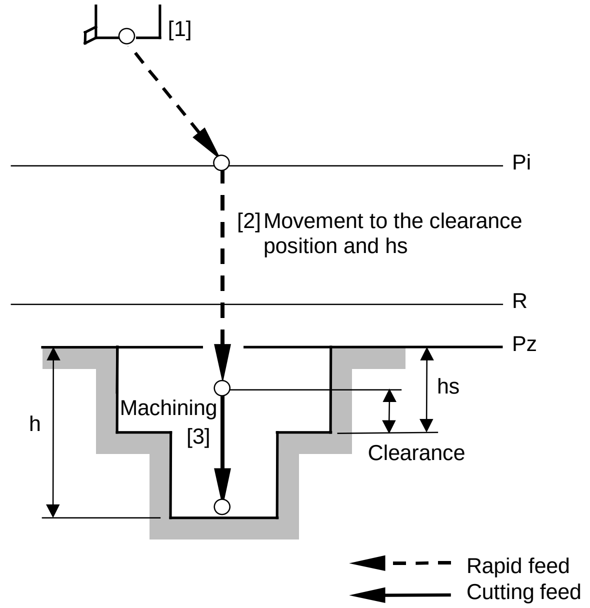
D33 .... Movement on the XY plane entered in the parameter

Note 1: The direction of movement [2] and [7] are determined by the data set in bit 3 and bit 4 of parameter **I14**, respectively. The direction of movement [4] and [10] is opposite to that of [2].

Note 2: The delayed stop time of the axial feed is set by the parameter **D40**.

<u>General precautions concerning the path of the boring tool</u>

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

Case of non-stepping boring	Case of stepped boring
	

M3P137

Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance in the axial direction

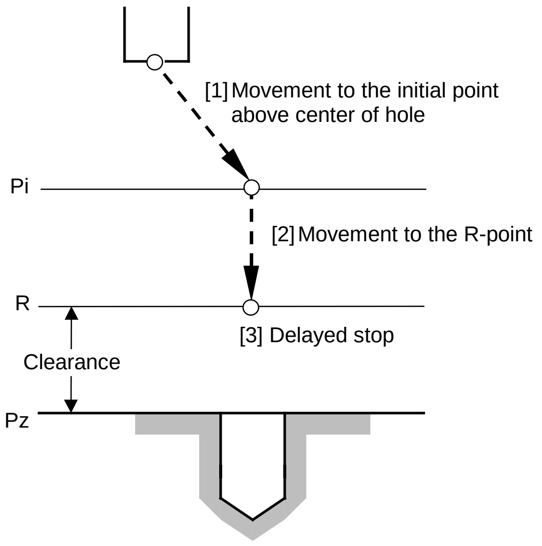
h: Distance equal to the sum of the depth of hole (**HOLE-DEP**) entered in the tool sequence and the data **LENG COMP.** (tool correction) in the **TOOL DATA** display

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

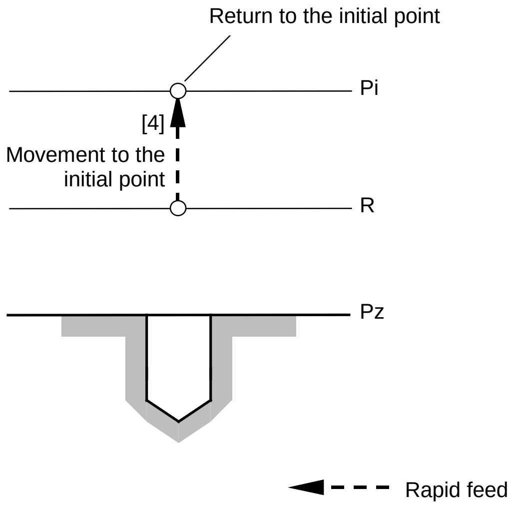
Note: Cutting start point is moved from R-point to a distance specified in hs (depth of pre-hole).

10. Chip vacuuming tool (option)

Movement 1



Movement 2



Pi: Initial point

Pz: Start point to be entered in the shape sequence

R: Safety clearance above the point Pz

Note: The delayed stop time of the axial feed is set by the parameter **D29**.

3-5-8 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 (**MODE**) specified for the unit.

○: Selection possible, ×: Selection impossible

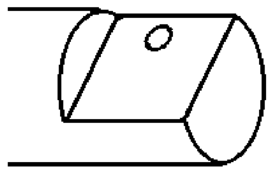
Shape	Mode									
	ZC	XC	$\boxed{\text{XC}}$	/C	$\boxed{\text{/ C}}$	ZY	XY	$\boxed{\text{XY}}$	/Y	$\boxed{\text{/ Y}}$
PT	○	○	○	○	○	○	○	○	○	○
ARC	○	○	○	○	○	○	○	○	○	○
LIN	×	×	×	×	×	○	○	○	○	○
SQR	×	×	×	×	×	○	○	○	○	○
GRD	×	×	×	×	×	○	○	○	○	○

Refer to the section 3-4-1 “Planes to be machined and machining methods“ for the detail of the modes.

2. Entry of shape sequence data

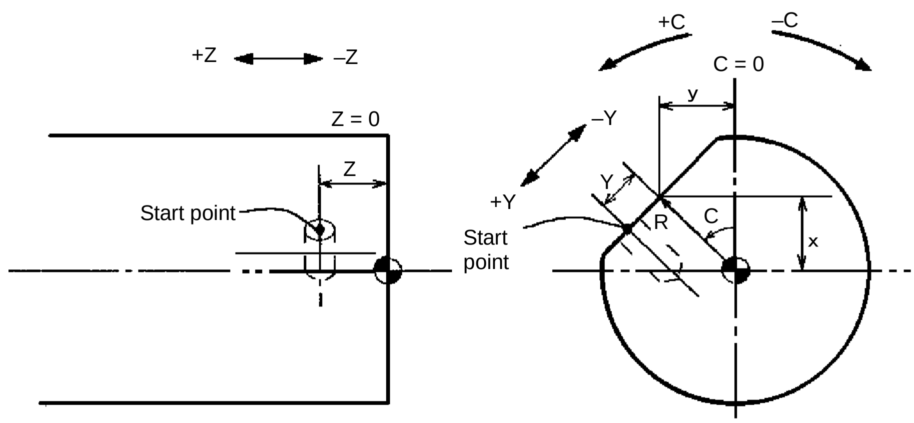
A. When the selected mode in the unit is ZC

1. Point (**PT**)

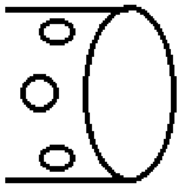


D734P0014

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

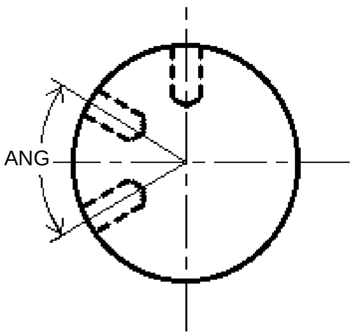
Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z [4] SPT-Y	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the 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	PTN	SPT-R/x	SPT-C/y	SPT-Z	SPT-Y	NUM.	ANG	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] ANG	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, $\boxed{\text{XC}}$, /C or $\boxed{\text{/C}}$

1. Point (**PT**)

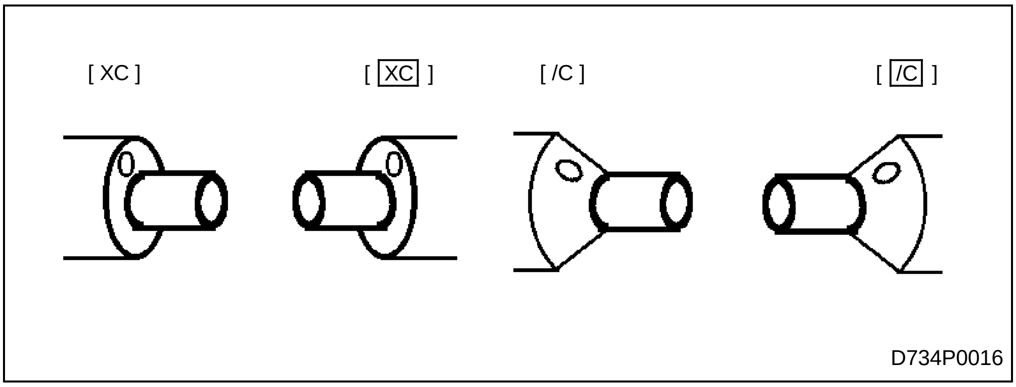
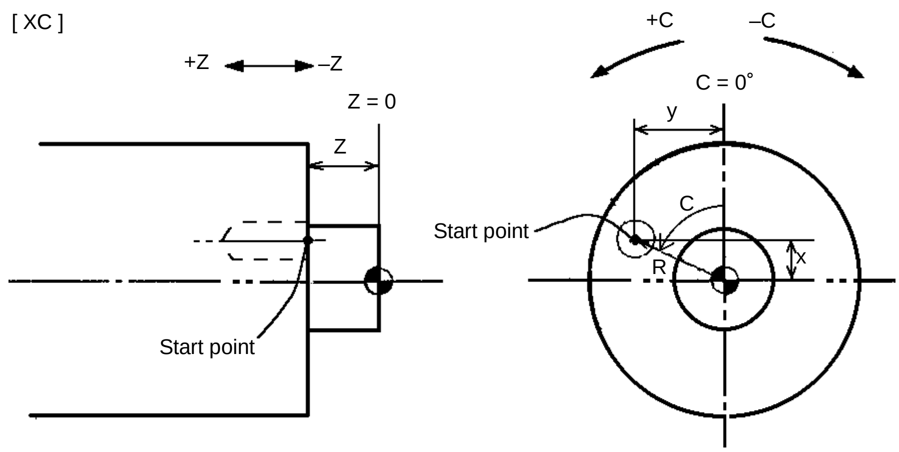
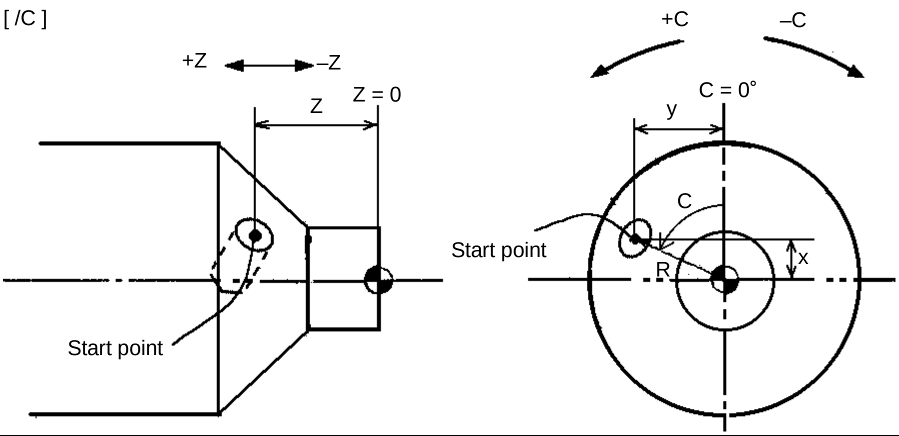


FIG	PTN	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANG	Q	R
1	PT	[1]	[2]	[3]	◆	◆	◆	[4]

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. [XC]  [/C] 
[4] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)

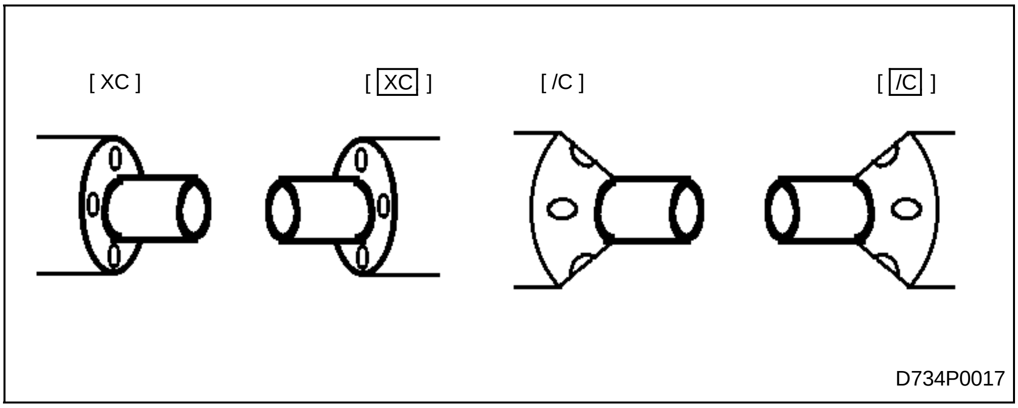
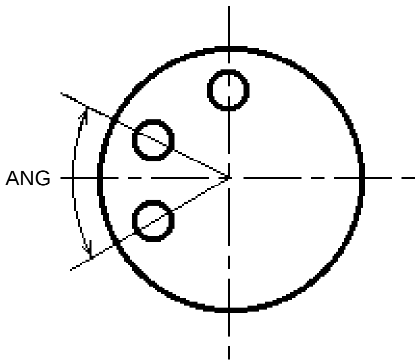
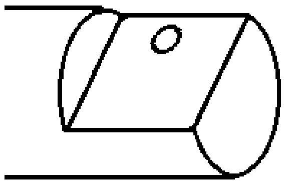


FIG	PTN	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANG	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	[6]	[7]

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] ANG	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

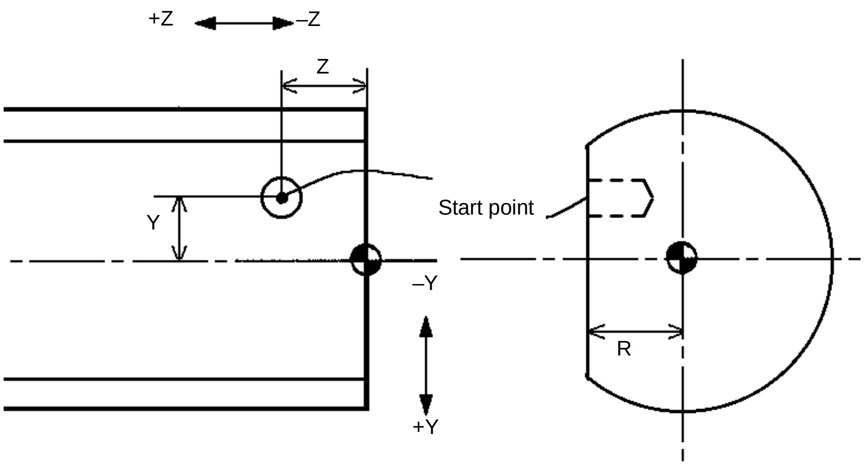
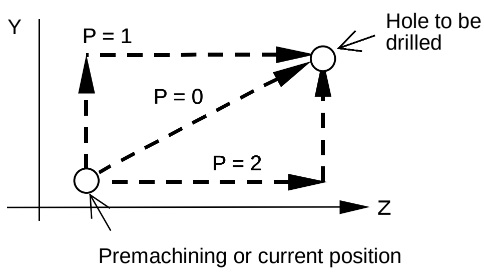
C. When the selected mode in the unit is ZY

1. Point (**PT**)

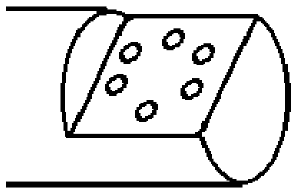


D734P0014

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

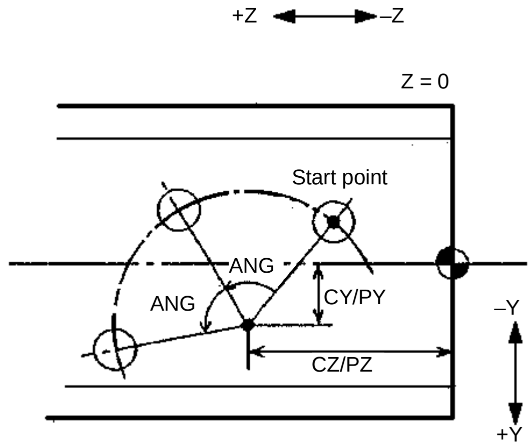
Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be drilled. 
[4] P	Specify the tool path. 
[5] R	Specify the position to which the tool returns after machining. 0: Initial point 1: Reference point

2. Arc (**ARC**)

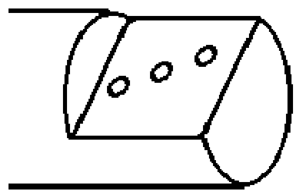


D734P0018

FIG	PTN	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANG	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] ANG	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

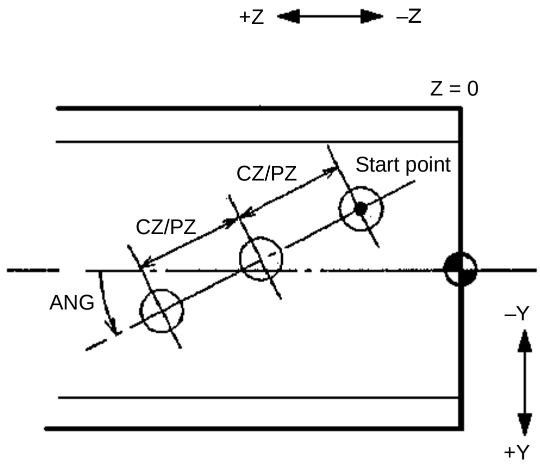
3. Line (**LIN**)



D734P0019

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

<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>[1] SPT-Z
[2] SPT-Y
[3] SPT-R</td><td>Specify the start point of the hole to be machined.
(See “1 Point (PT)” for further details.)</td></tr>
<tr><td>[4] CZ/PZ</td><td>Specify the pitch between two adjacent holes in the line of holes.



</td></tr>
<tr><td>[5] M</td><td>Specify the number of holes to be drilled.</td></tr>
<tr><td>[6] ANG</td><td>Specify the angle formed by the line of holes and the Z-axis.
(See the figure of the item [4] CZ/PZ.)</td></tr>
<tr><td>[7] 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>[8] R</td><td>Specify the position to which the tool returns after machining.
0: Initial point
1: Reference point</td></tr>
</table>

4. Square (**SQR**)

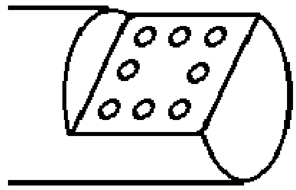
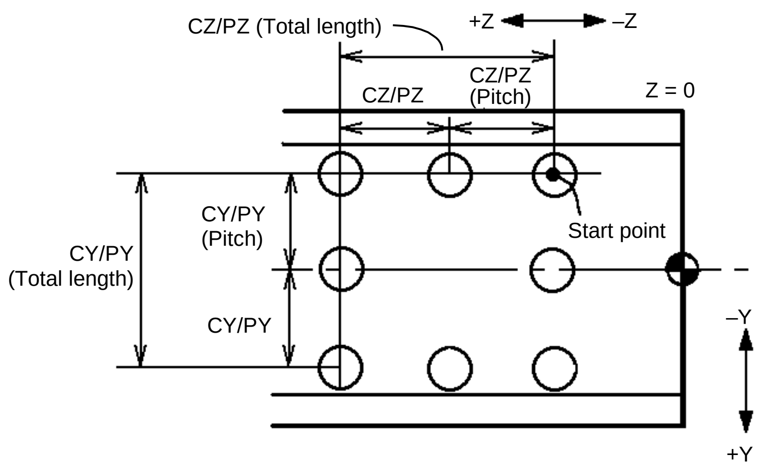
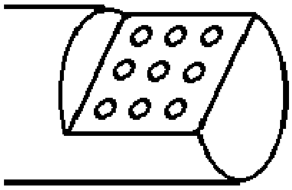

D734P0020

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

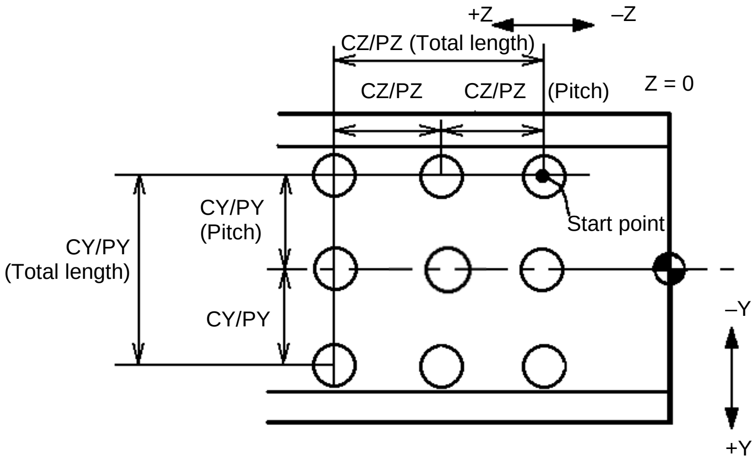
Cursor position	Description
[1] SPT-Z [2] SPT-Y [3] SPT-R	Specify the start point of the hole to be machined. (See “1 Point (PT)” for further details.)
[4] CZ/PZ	Specify the pitch between holes or the total length of the Z-axis. 
[5] CY/PY	Specify the pitch between holes or the total length of the Y-axis. (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**)



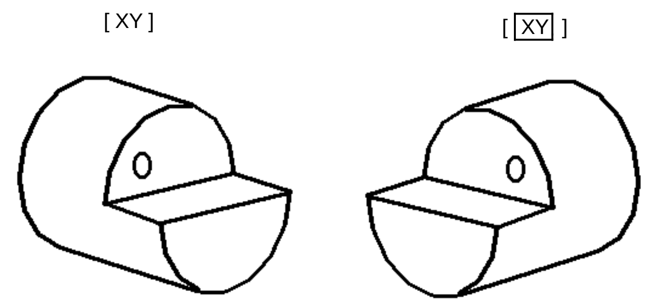
D734P0021

FIG	PTN	SPT-Z	SPT-Y	SPT-R	CZ/PZ	CY/PY	F	M	N	ANG	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

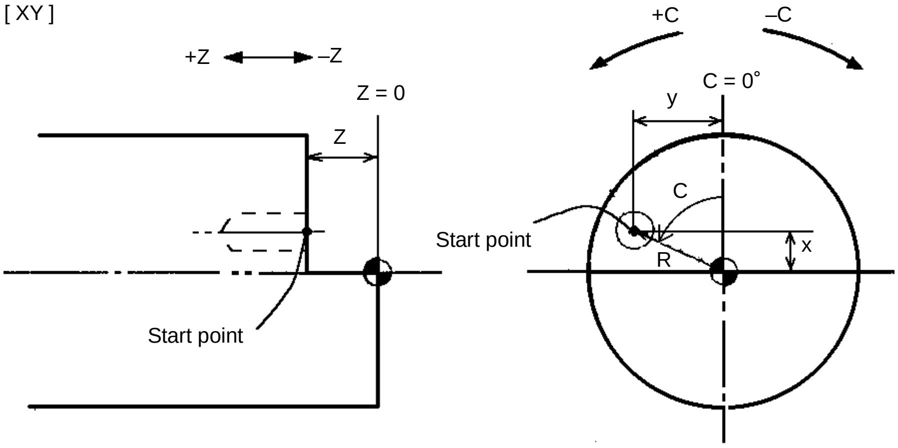
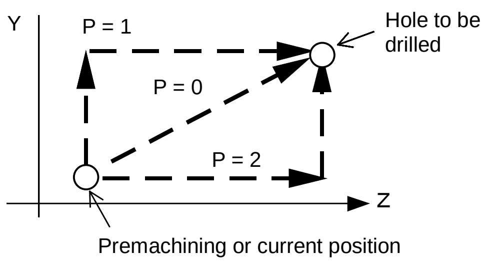
D. When the selected mode in the unit is XY or $\boxed{XY}$

1. Point (**PT**)

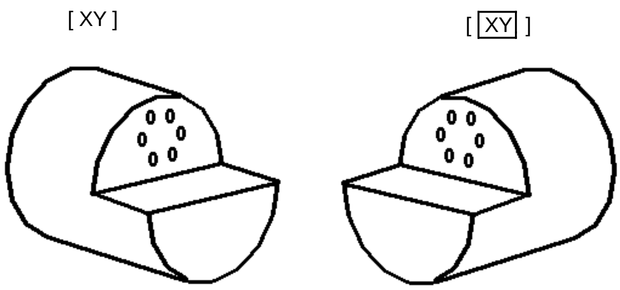


D734P0022

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

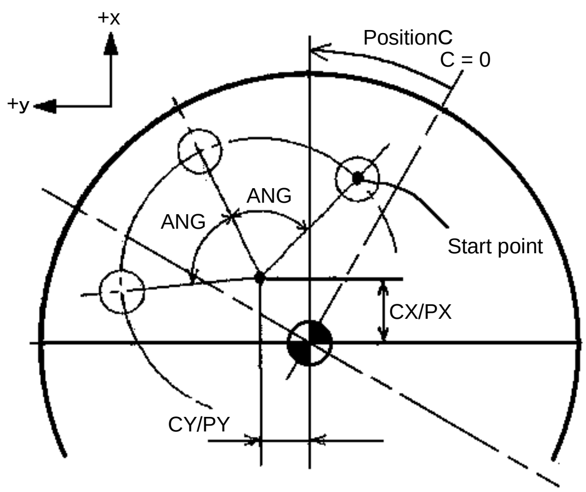
<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>[1] SPT-R/x
[2] SPT-C/y
[3] SPT-Z</td><td>Specify the start point of the hole to be machined.
- To set the start point in R-C coordinates, enter the radius and the angle as they are.
- To set the start point in x-y coordinates, change the [x-y INPUT] menu item to the reverse display mode before entering data.
[XY]

</td></tr>
<tr><td>[4] P</td><td>Specify the tool path.

</td></tr>
<tr><td>[5] R</td><td>Specify the position to which the tool returns after machining.
0: Initial point
1: Reference point</td></tr>
</table>

2. Arc (**ARC**)



D734P0023

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

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 [5] CY/PY	Specify the coordinate of the center of the arc. 
[6] M	Specify the number of holes to be drilled.
[7] ANG	Specify the angle between two adjacent holes. (See the figure of the item [4] CX/ PX, [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**)

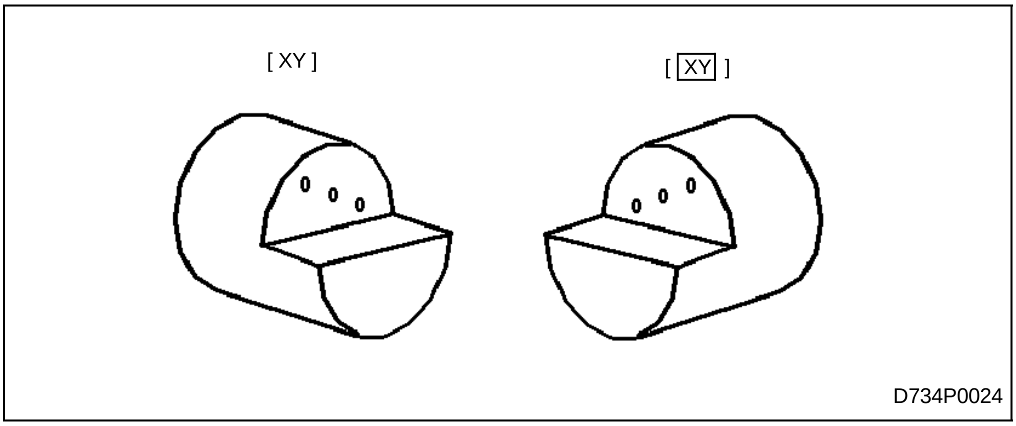
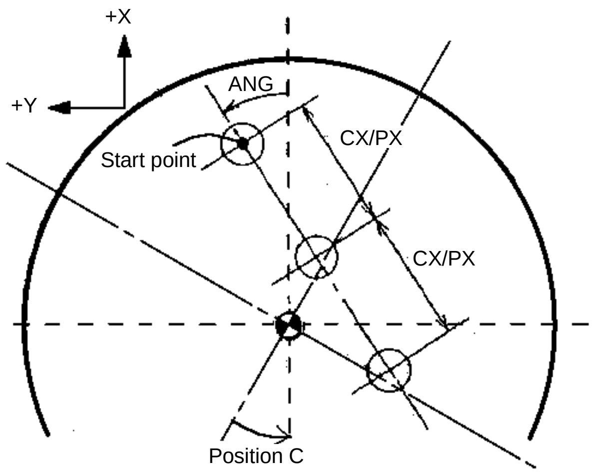


FIG	PTN	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANG	P	Q	R
1	LIN	[1]	[2]	[3]	[4]	◆	◆	[5]	◆	[6]	◆	[7]	[8]

<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</td><td>Specify the pitch between two adjacent holes in the line of holes.

</td></tr>
<tr><td>[5] M</td><td>Specify the number of holes to be drilled.</td></tr>
<tr><td>[6] ANG</td><td>Specify the angle formed by the line of holes and the Z-axis. (See the figure of the item [4] CX/ PX.)</td></tr>
<tr><td>[7] 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>[8] R</td><td>Specify the position to which the tool returns after machining.
0: Initial point
1: Reference point</td></tr>
</table>

4. Square (**SQR**)

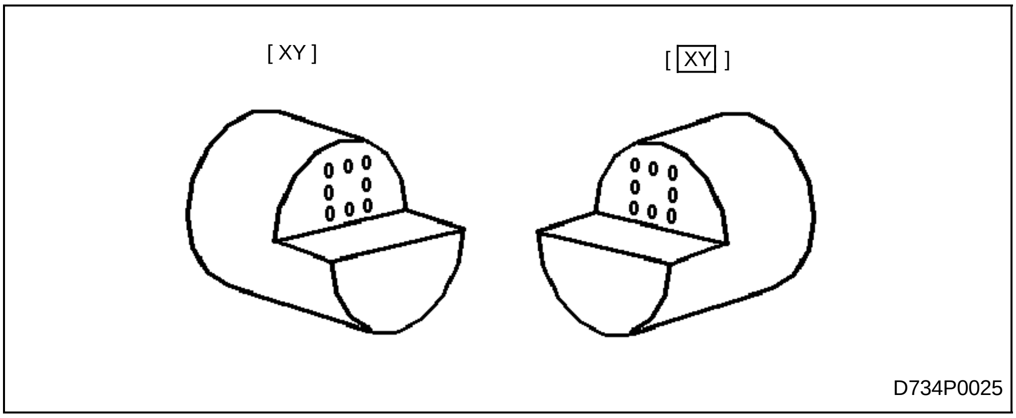
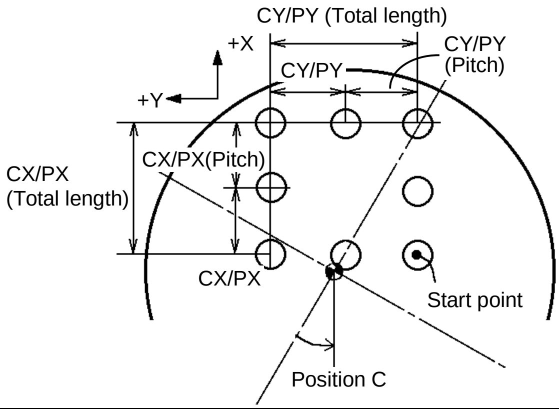
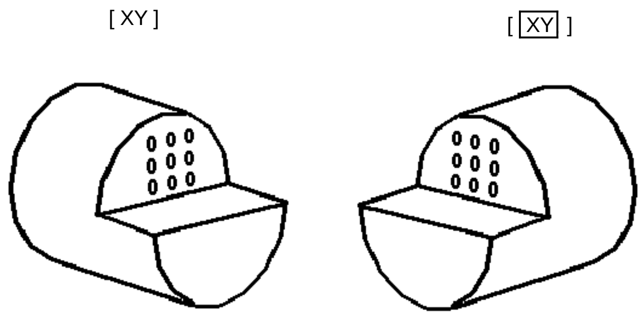


FIG	PTN	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANG	P	Q	R
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	◆	[9]	[10]	[11]

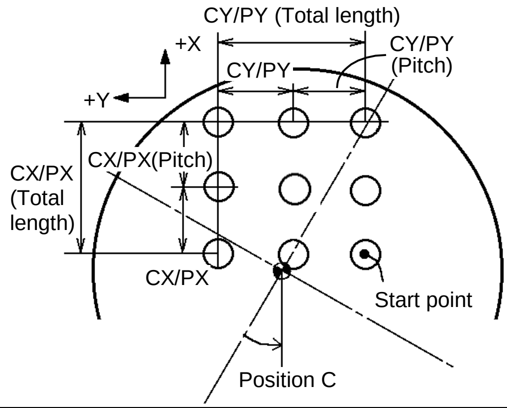
Cursor position	Description
[1] SPT-R/x [2] SPT-C/y [3] SPT-Z	Specify the start point of the hole to be machined. - To set the start point in R-C coordinates, enter the radius and the angle as they are. - To set the start point in x-y coordinates, change the **[x-y INPUT]** menu item to the reverse display 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	PTN	SPT-R/x	SPT-C/y	SPT-Z	CX/PX	CY/PY	F	M	N	ANG	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**)

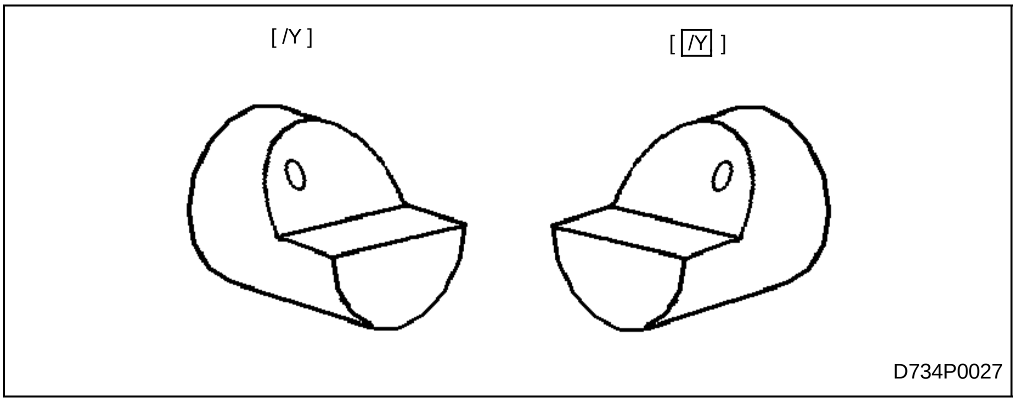
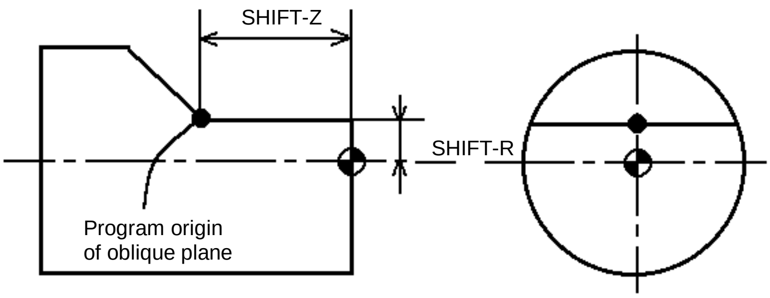
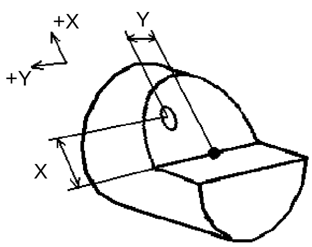
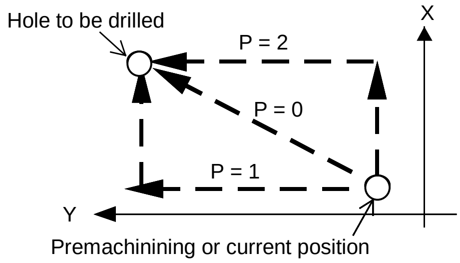


FIG	PTN	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANG	P	Q	R
1	PT	[1]	[2]	[3]	[4]	◆	◆	◆	◆	◆	◆	[5]	◆	[6]

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**)

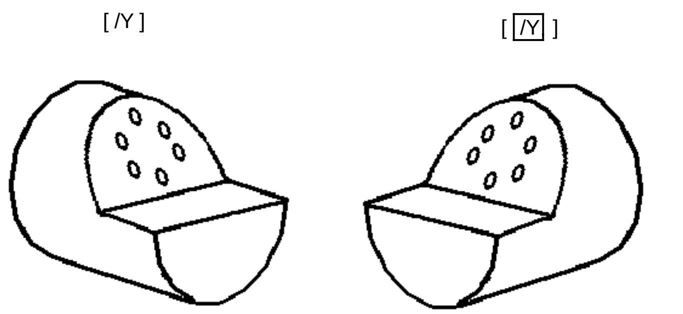
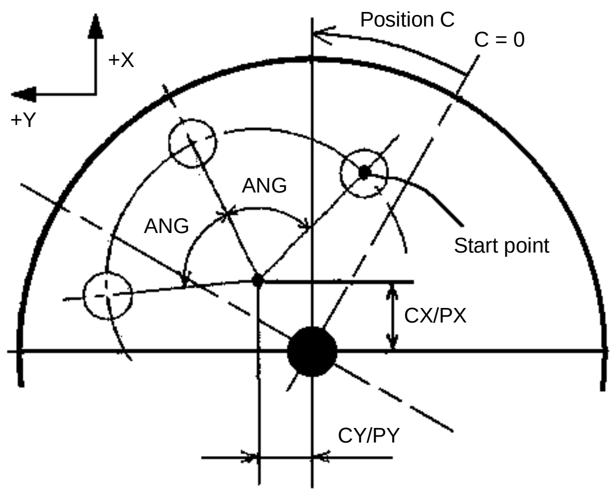


FIG	PTN	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANG	P	Q	R
1	ARC	[1]	[2]	[3]	[4]	[5]	[6]	◆	[7]	◆	[8]	◆	[9]	[10]

<table>
<tr><th>Cursor position</th><th>Description</th></tr>
<tr><td>[1] SHIFT-Z
[2] SHIFT-R</td><td>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.)</td></tr>
<tr><td>[3] SPT-X
[4] SPT-Y</td><td>Specify the start point of the hole to be machined.
(See “1 Point (PT)” for further details.)</td></tr>
<tr><td>[5] CX/PX
[6] CY/PY</td><td>Specify the central coordinates of the arc.

</td></tr>
<tr><td>[7] M</td><td>Specify the number of holes to be drilled.</td></tr>
<tr><td>[8] ANG</td><td>Specify the angle between two adjacent holes.
(See the figure of the item [5] CX/ PX, [6] CY/ PY.)</td></tr>
<tr><td>[9] 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>[10] 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**)

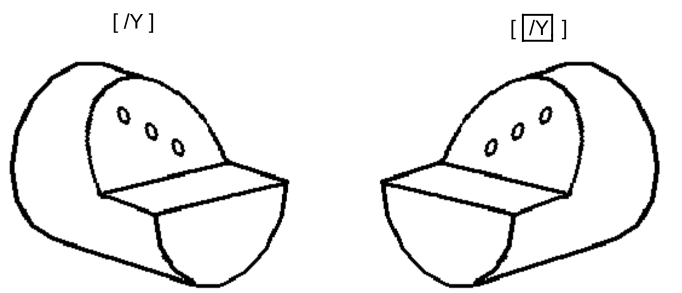
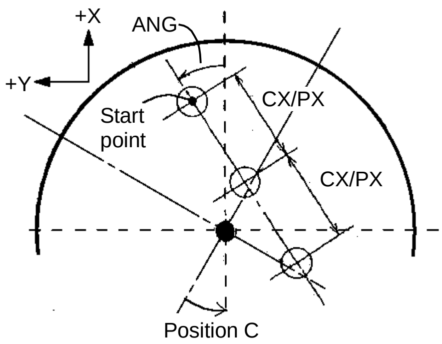
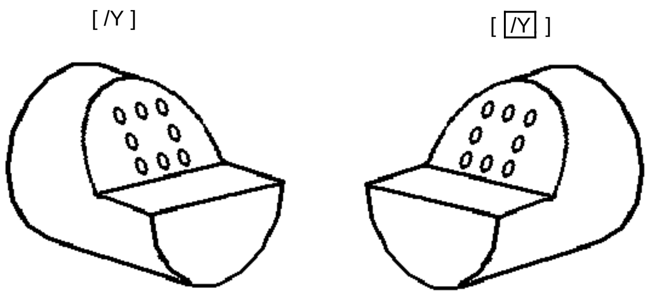


FIG	PTN	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANG	P	Q	R
1	LIN	[1]	[2]	[3]	[4]	[5]	◆	◆	[6]	◆	[7]	◆	[8]	[9]

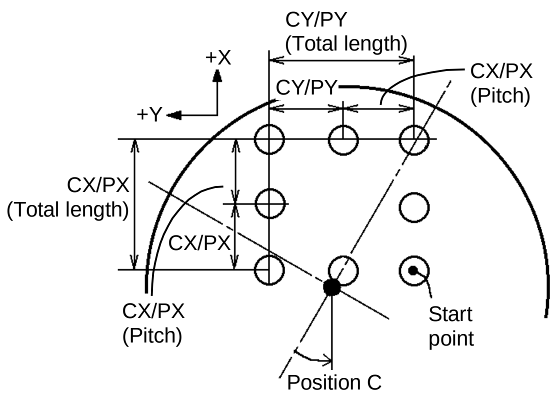
Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See “1 Point (PT)” for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See “1 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] ANG	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**)

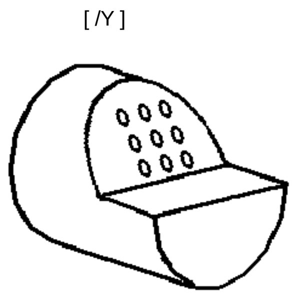


D734P0030

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

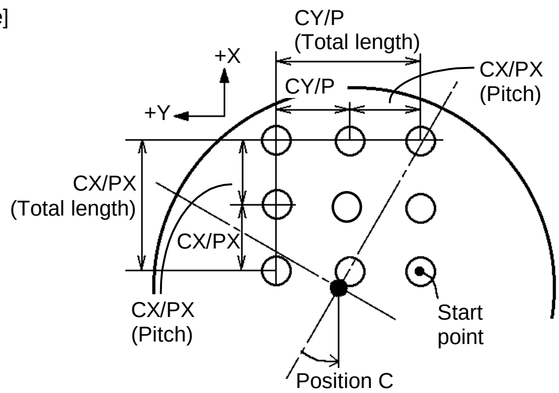
Cursor position	Description
[1] SHIFT-Z [2] SHIFT-R	Specify the value of shift from the work origin to the oblique plane (the distance to the program origin of the oblique plane). (See “1 Point (PT)” for further details.)
[3] SPT-X [4] SPT-Y	Specify the start point of the hole to be machined. (See “1 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**)



D734P0031

FIG	PTN	SHIFT-Z	SHIFT-R	SPT-X	SPT-Y	CX/PX	CY/PY	F	M	N	ANG	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

3-6 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.

3-6-1 Types of line machining units

As shown below 9 types of line machining units are available:

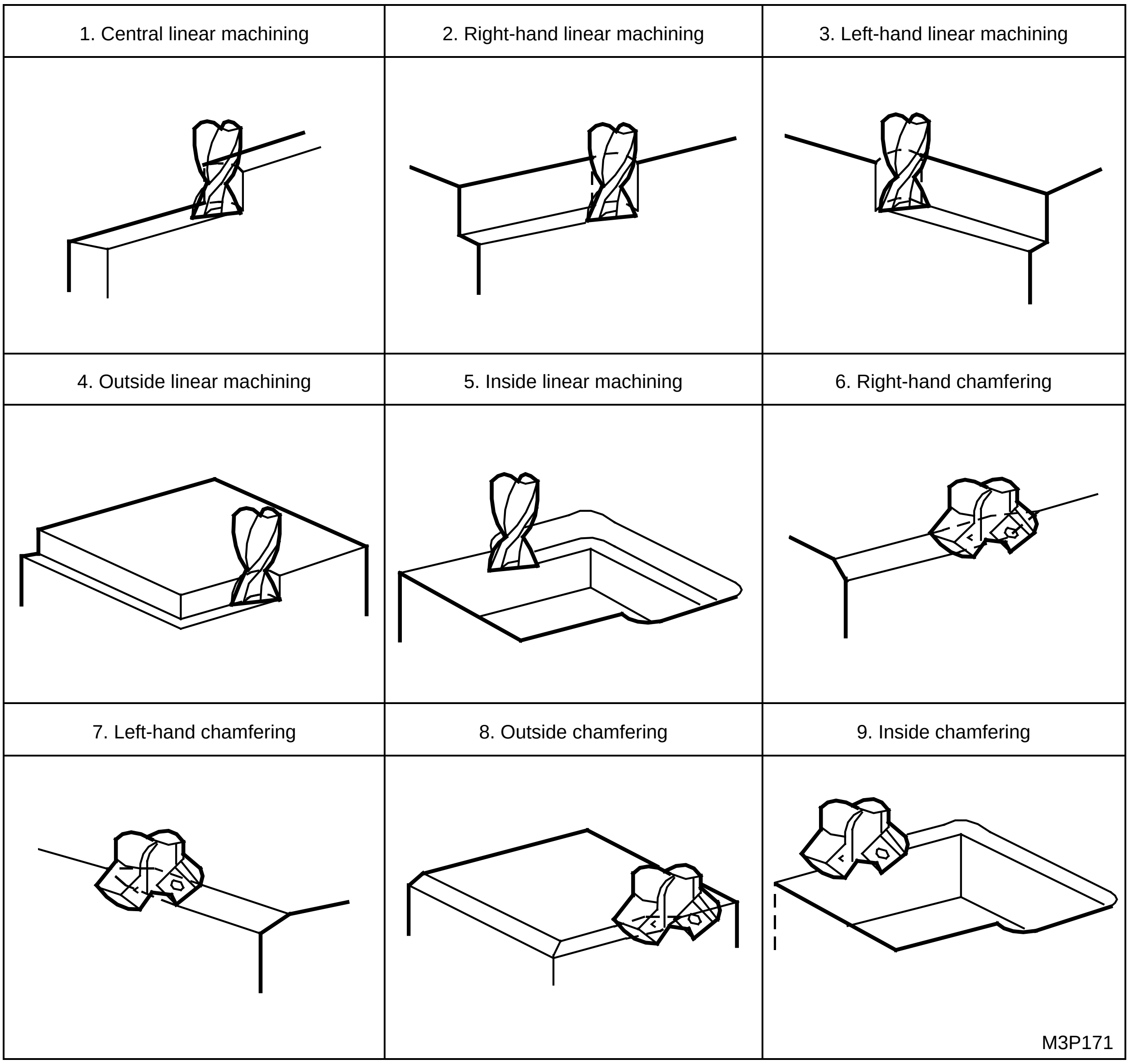


Fig. 3-21 Types of line machining units

3-6-2 Procedure for selecting line machining unit

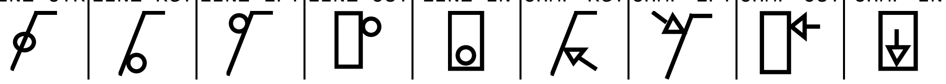
(1) Press the menu selector key (key located at the right of the menu keys) to display the following menu.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Presse the **[LINE MACH-ING]** menu key.

➔ The following line machining unit menu will be displayed.

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.

3-6-3 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.

A. Data setting

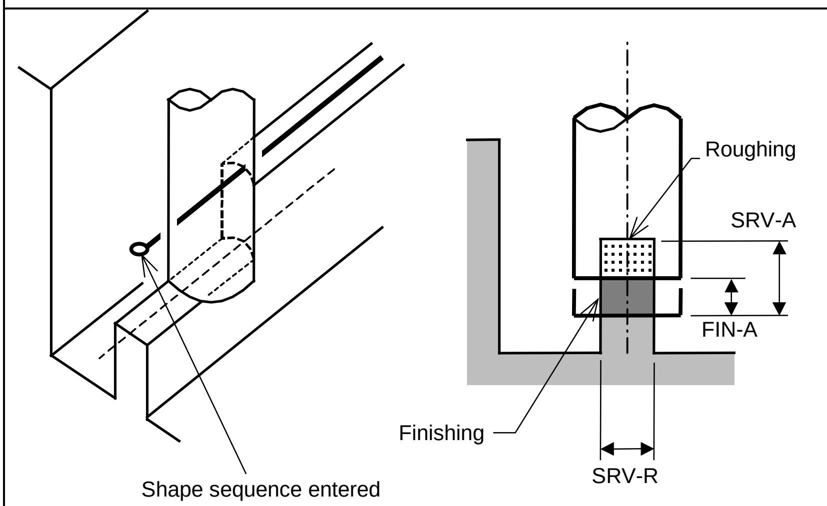
```
UNo.   UNIT    MODE  POS-B POS-C   SRV-A    SRV-R   RGH    FIN-A          START  END
 1   LINE CTR
SNo.    TOOL      NOM-ϕ  No.  #   APRCH-1  APRCH-2  TYPE   AFD    DEP-A   WID-R  C-SP   FR   M   M   M
 R1   END MILL                                       ◆                      ◆
 F2   END MILL                                       ◆              ◆       ◆
```

◆: Data are not necessary to be set here.

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

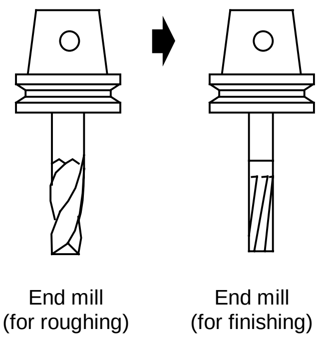
Remark 2: For the tool sequence data setting, refer to Subsection 3-6-4.

LINE CTR unit



D740PA121

Tool sequence



D740PA043

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.

B. Automatic tool development

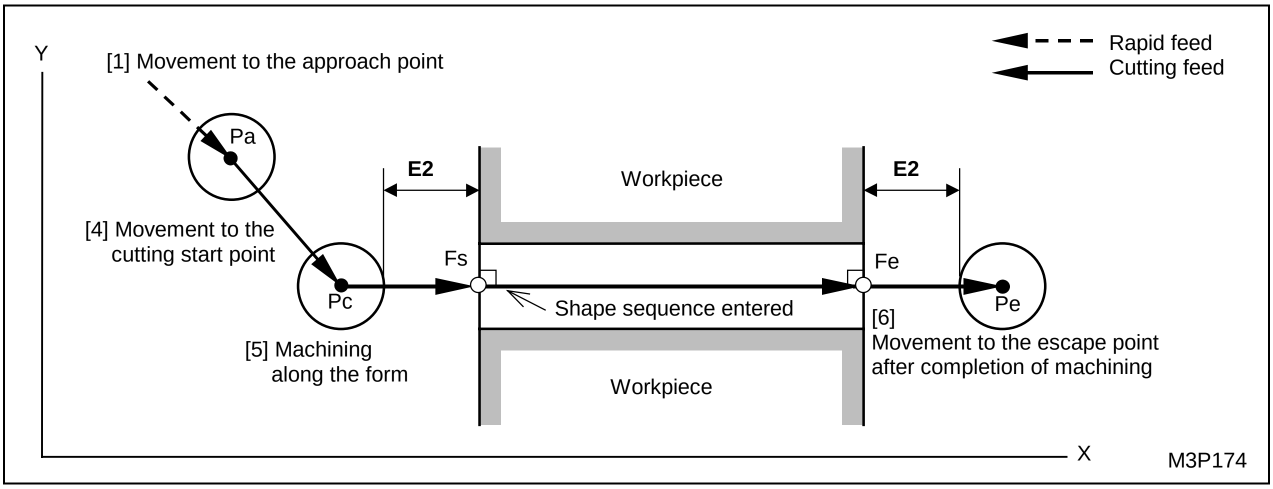
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A** and on **FIN-A**.

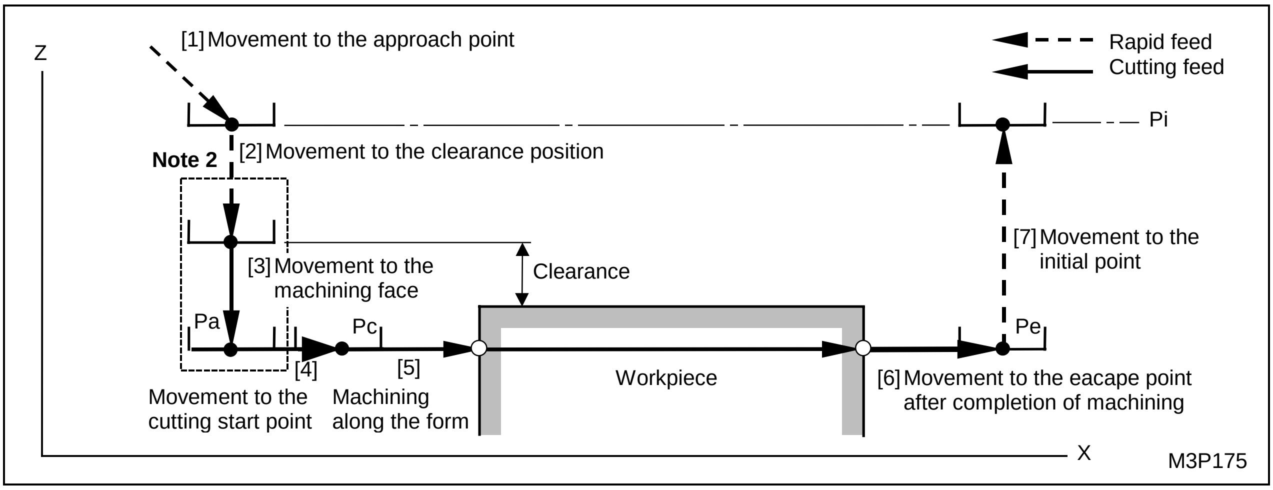
Machining	Pattern
R1 (Roughing)	FIN-A = 0: One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A: One tool is selected.

C. Tool path

X-Y-axis



X-Z-axis



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

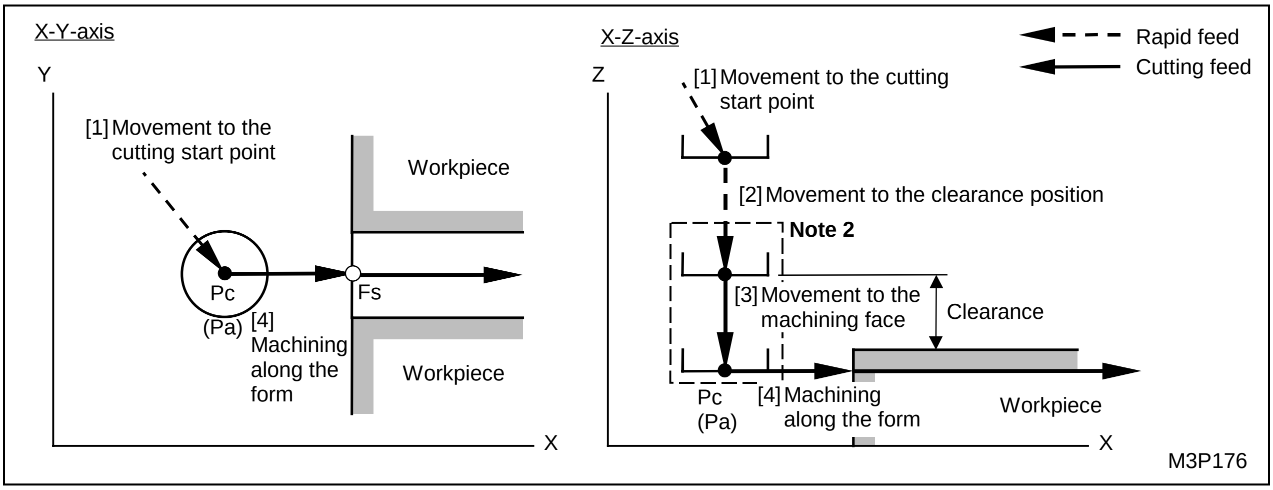
Pc: Cutting start point to be automatically established

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 items **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 items.



Note 2: See Subsection 3-6-6, “Precautions in line machining.”

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

D. Start point (START) and End point (END)

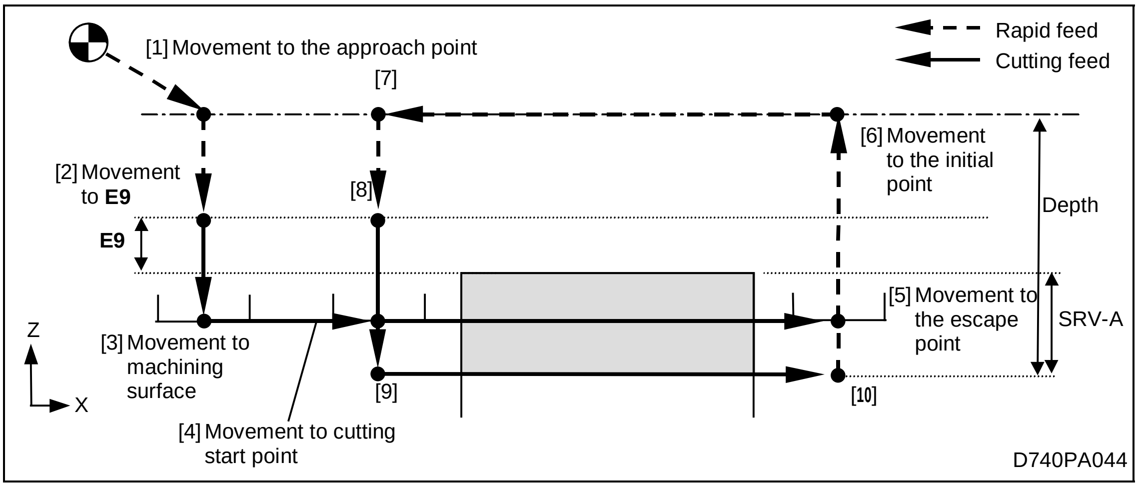
Excessive cutting that may occur during approach or retraction can be prevented by specifying wall attributes for the line machining start and end points. The term “walls” are defined as the surfaces perpendicular to the shape at both the start point and the end point.

Wall attributes can be specified for the following 5 units.

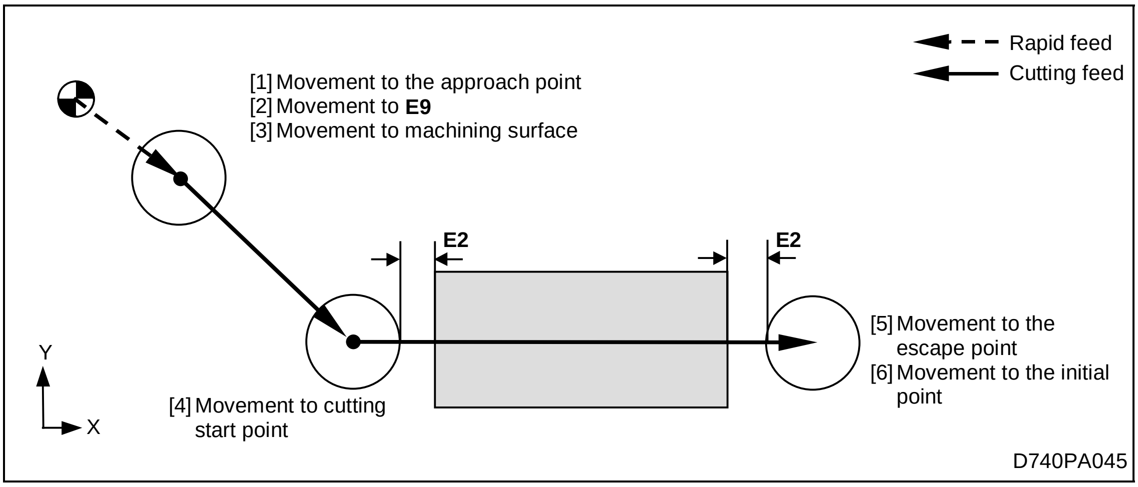
- LINE CTR
- LINE RGT
- LINE LFT
- CHMF RGT
- CHMT LFT

<When **OPEN** is set for **START** and **END**>

X-Z-axis

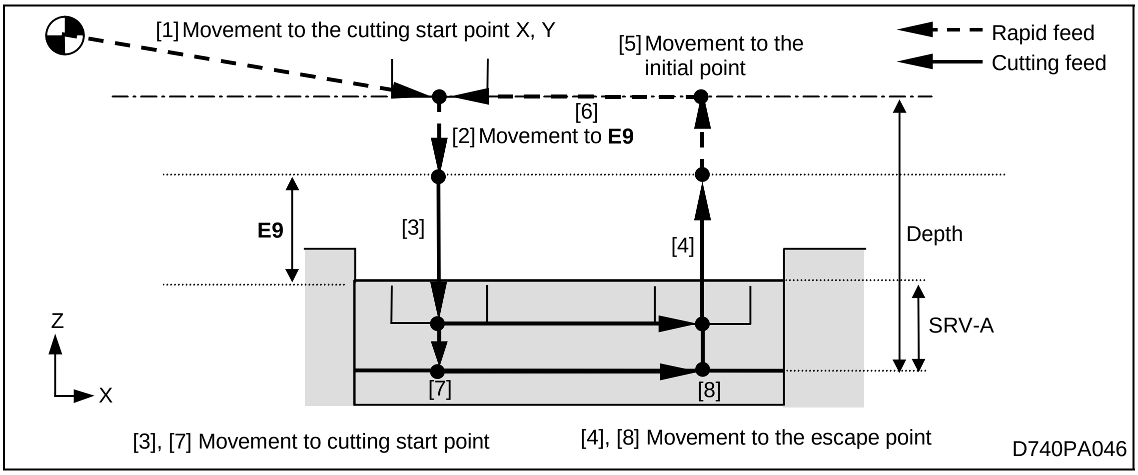


X-Y-axis

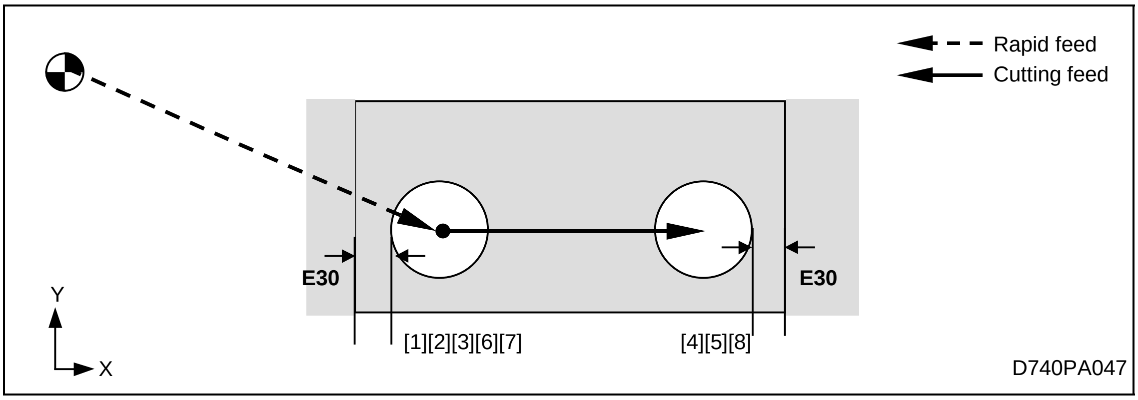


<When **CLOSED** is set for **START** and **END**>

X-Z-axis



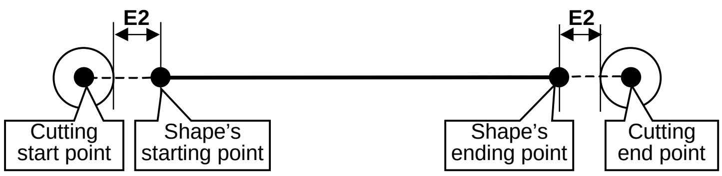
X-Y-axis



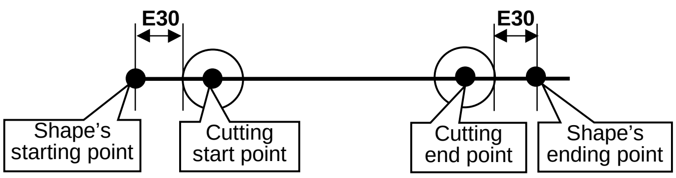
Note: The tool path is determined according to the setting of parameter **E104** bit 3.

- **E104** bit 3 = 0

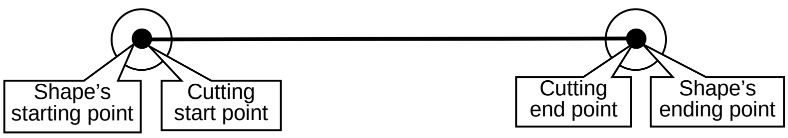
  Attribute: **OPEN**



  Attribute: **CLOSED**



- **E104** bit 3 = 1 (the same tool path as when MT Pro parameter **P3** bit 4 = 1)



  The starting (or ending) point of the shape and the starting (or ending) point of cutting agree, irrespective of the attributes.

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.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R	START	END	INTER-R	CHMF
1	LINE RGT												

SNo.	TOOL	NOM-φ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M	M	M
R1	END MILL						◆			◆					
F2	END MILL						◆		◆	◆					

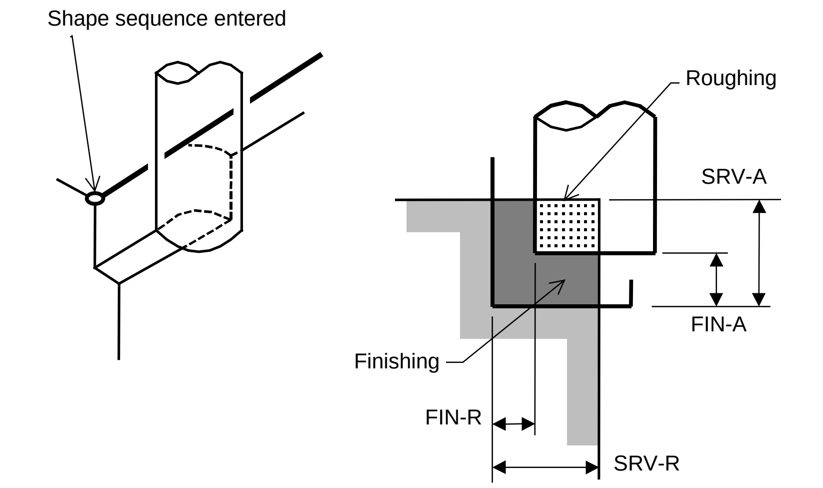
◆: Data are not necessary to be set here.

Remark 1: For data setting in **START** and **END**, refer to “1. Central linear machining unit (LINE CTR).”

Remark 2: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

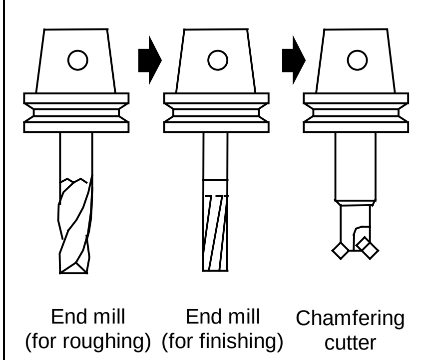
Remark 3: For the tool sequence data setting, refer to Subsection 3-6-4.

LINE RGT unit



D740PA123

Tool sequence



D740PA048

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.

B. Automatic tool development

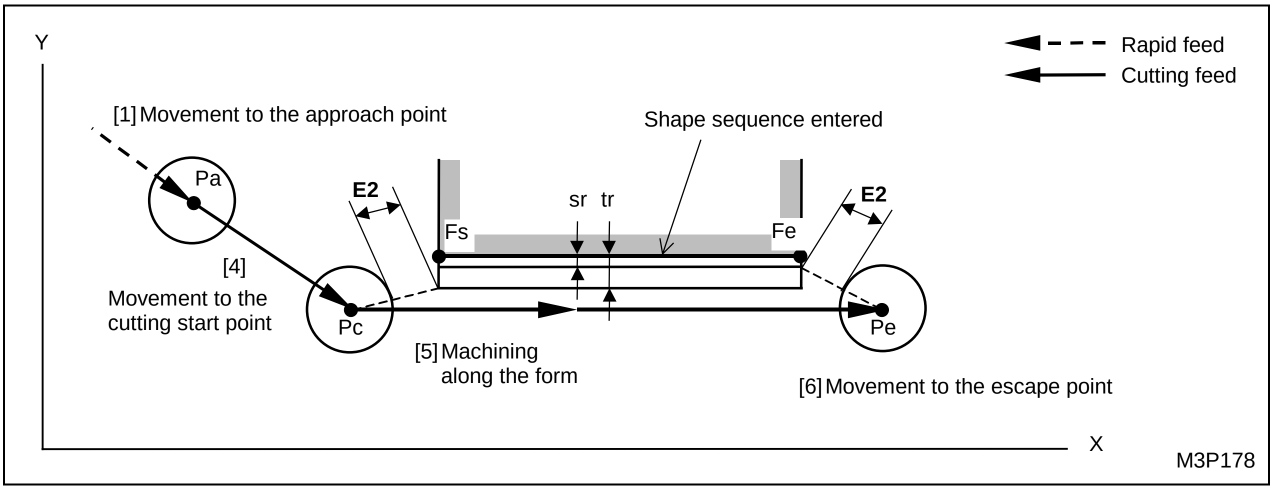
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to three tools are automatically developed though dependent upon the data **SRV-A**, **SRV-R**, **FIN-A** and **CHMF**.

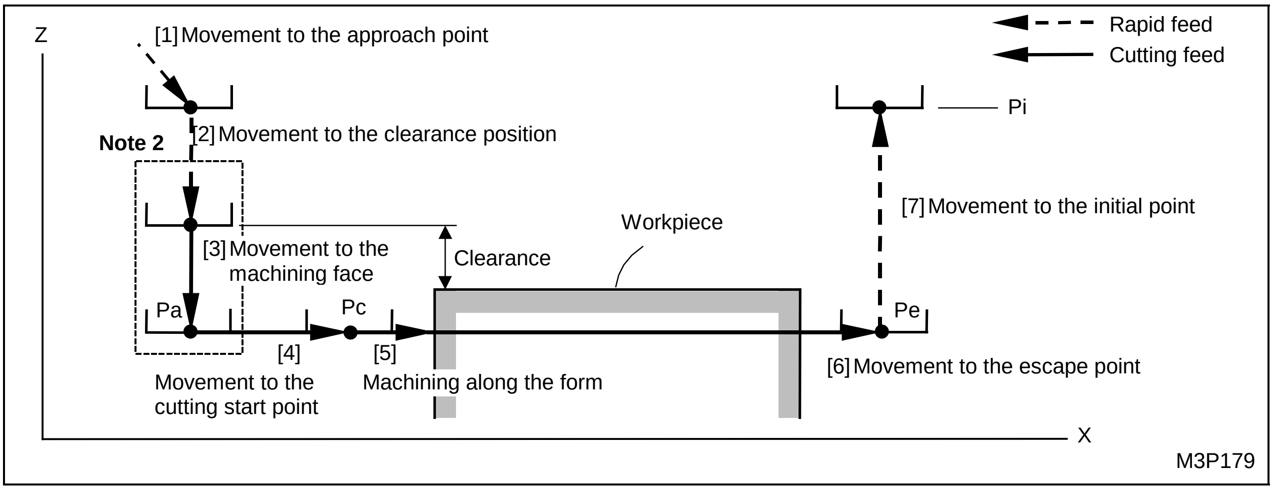
Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A or SRV-R ≤ FIN-R : One tool is selected.
(Chamfering)	CHMF≠ 0 : One tool is selected.

C. Tool path

X-Y-axis



X-Z-axis



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

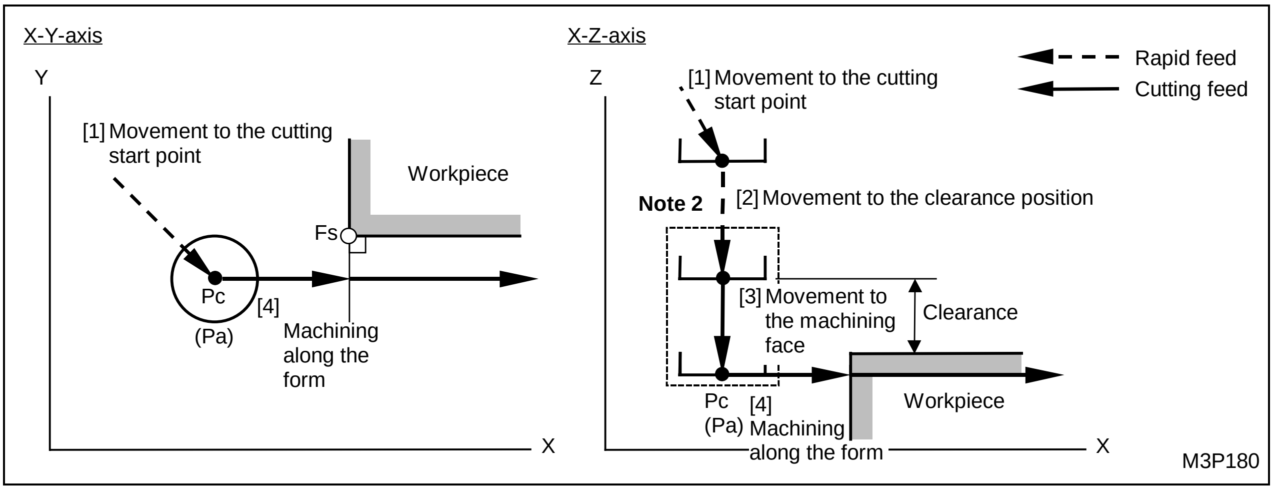
Fe: End point of form to be entered in the shape sequence

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 items **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 items.



Note 2: See Subsection 3-6-6 “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.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R	START	END	INTER-R	CHMF		
1	LINE LFT														
SNo.	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M	M	M
R1	END MILL						◆			◆					
F2	END MILL						◆		◆	◆					

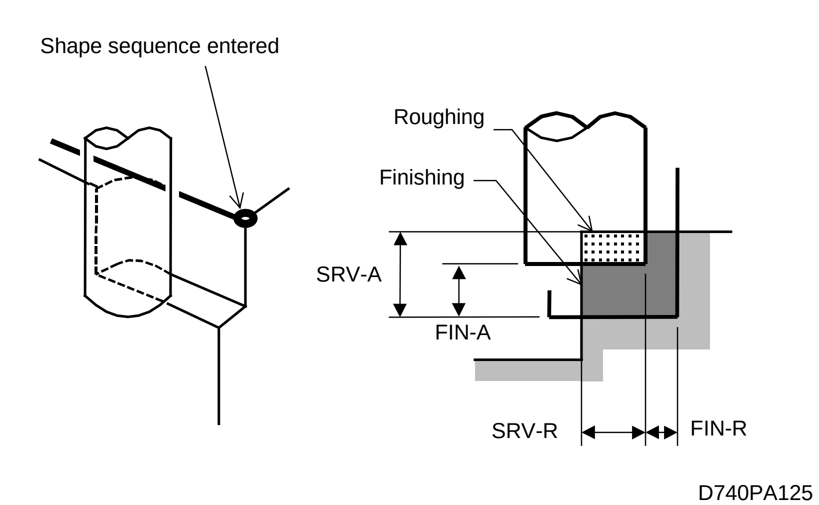
◆: Data are not necessary to be set here.

Remark 1: For data setting in **START** and **END**, refer to “1. Central linear machining unit (LINE CTR).”

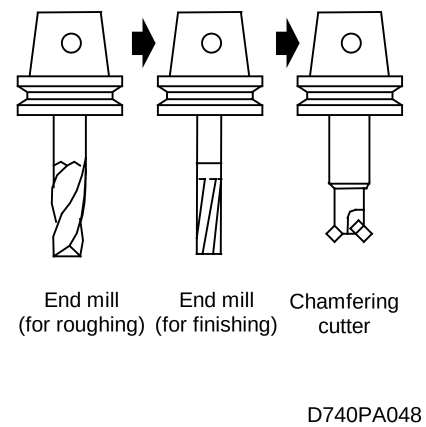
Remark 2: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 3: For the tool sequence data setting, refer to Subsection 3-6-4.

LINE LFT unit



Tool sequence



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.

B. Automatic tool development

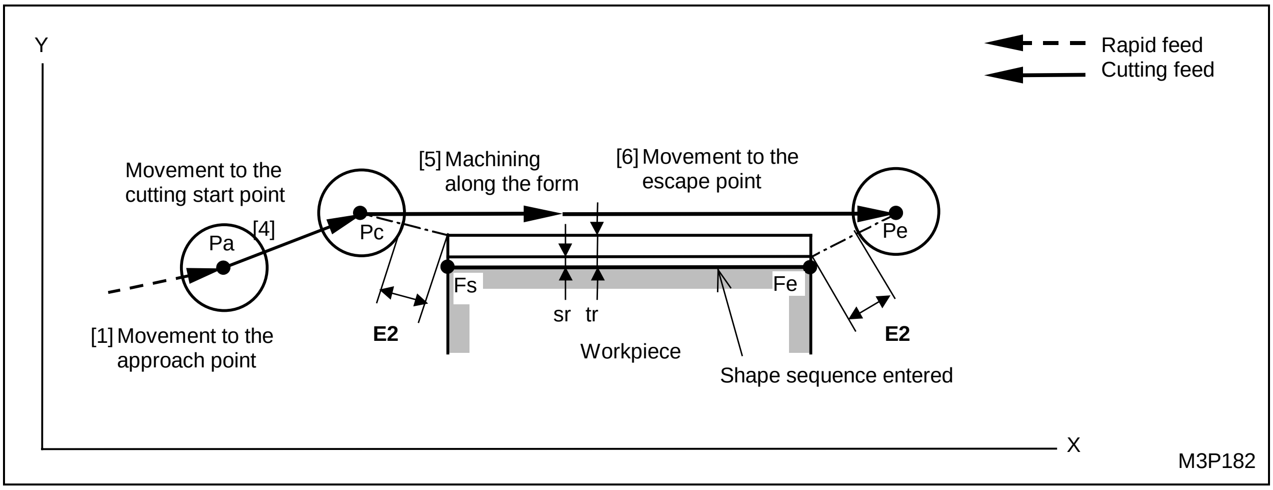
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to three tools are automatically developed though dependent upon the data **SRV-A**, **SRV-R**, **FIN-A** and **CHMF**.

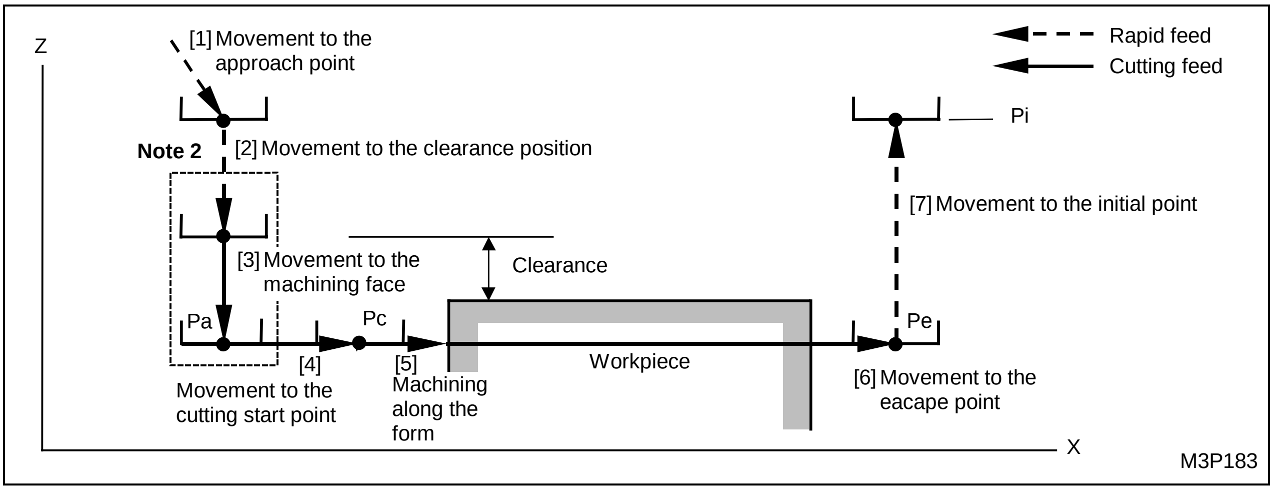
Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A or SRV-R ≤ FIN-R : One tool is selected.
(Chamfering)	CHMF≠ 0 : One tool is selected.

C. Tool path

X-Y-axis



X-Z-axis



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

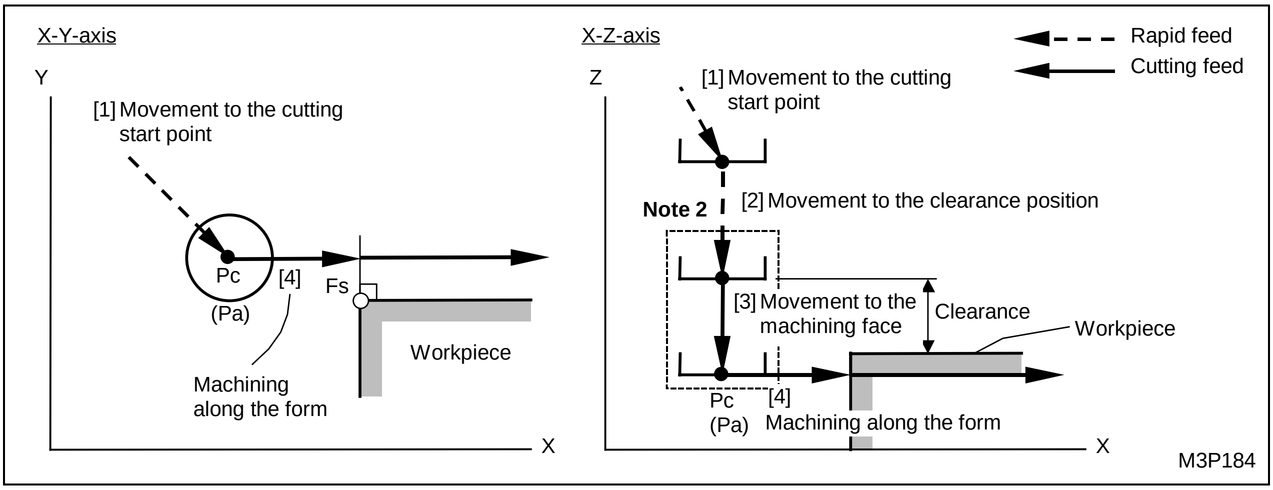
Fe: End point of form to be entered in the shape sequence

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 items **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 items.



Note 2: See Subsection 3-6-6, “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.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R	INTER-R	CHMF
1	LINE OUT										

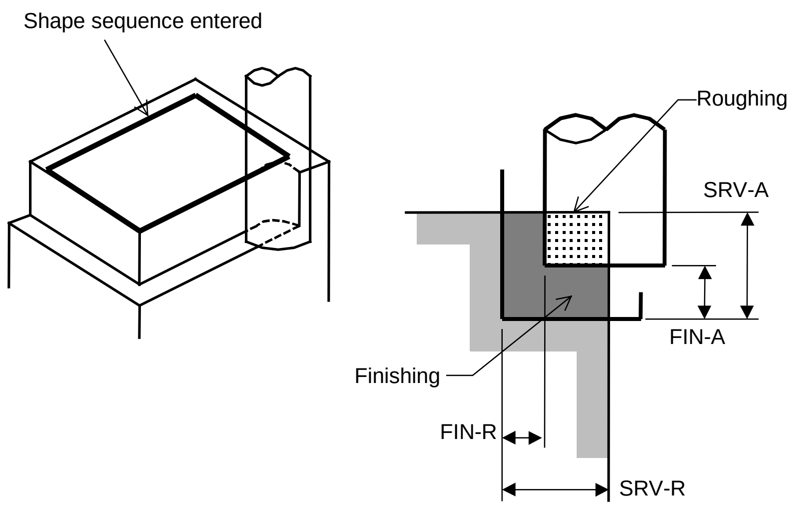
SNo.	TOOL	NOM-φ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M	M	M
R1	END MILL						◆			◆					
F2	END MILL						◆		◆	◆					

◆: Data are not necessary to be set here.

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

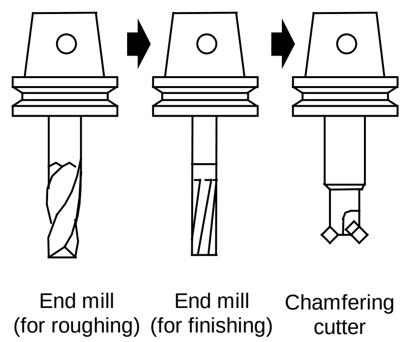
Remark 2: For the tool sequence data setting, refer to Subsection 3-6-4.

LINE OUT unit



D740PA126

Tool sequence



D740PA048

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.

B. Automatic tool development

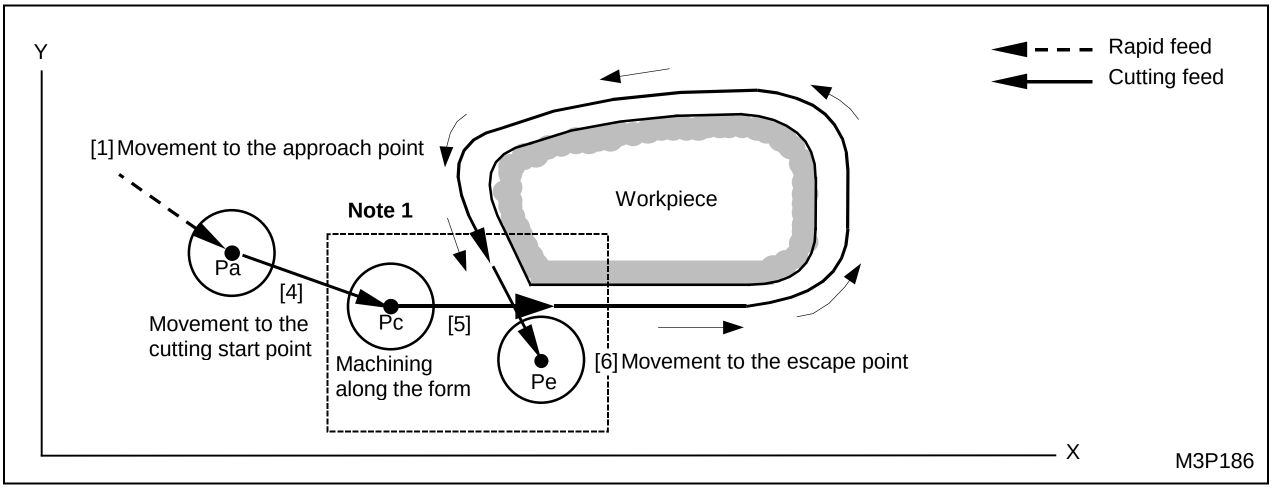
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to three tools are automatically developed though dependent upon the data **SRV-A**, **SRV-R**, **FIN-A** and **CHMF**.

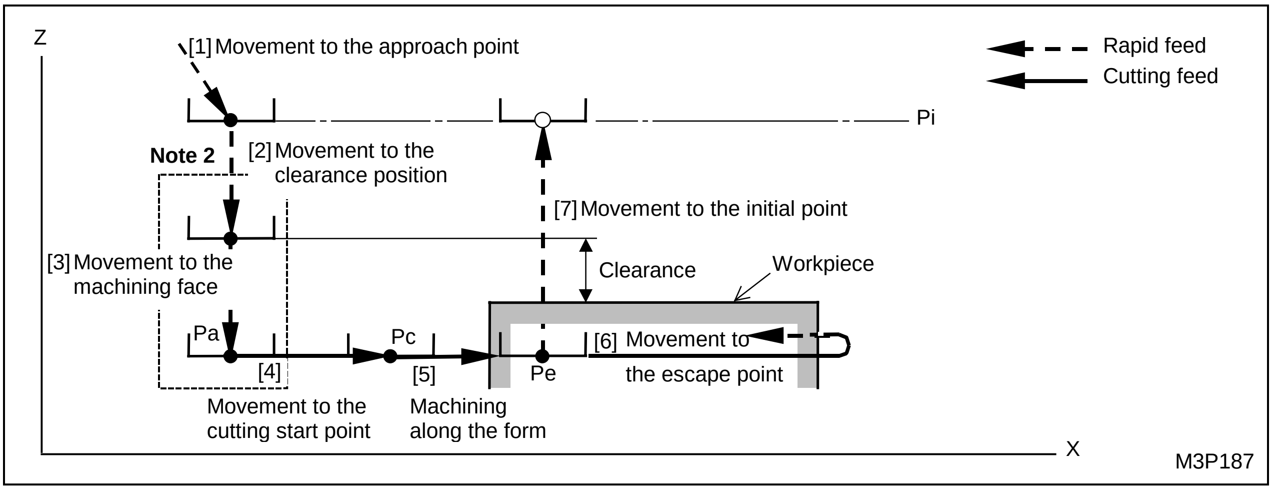
Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A or SRV-R ≤ FIN-R : One tool is selected.
(Chamfering)	CHMF≠ 0 : One tool is selected.

C. Tool path

X-Y-axis



X-Z-axis



Pi: Initial point

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

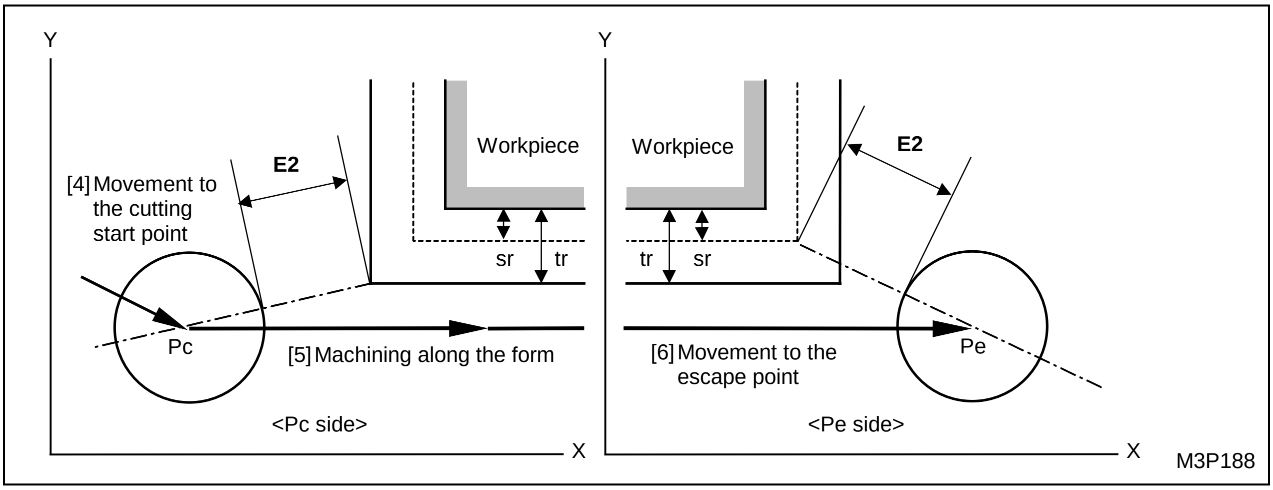
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

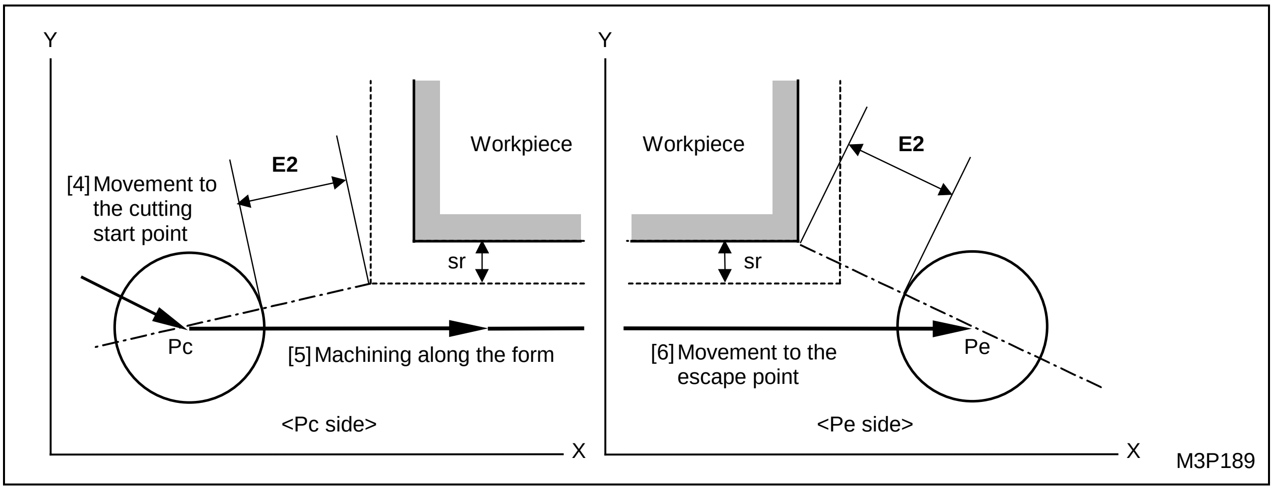
Note 1: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form

- In case of roughing



- In case of finishing



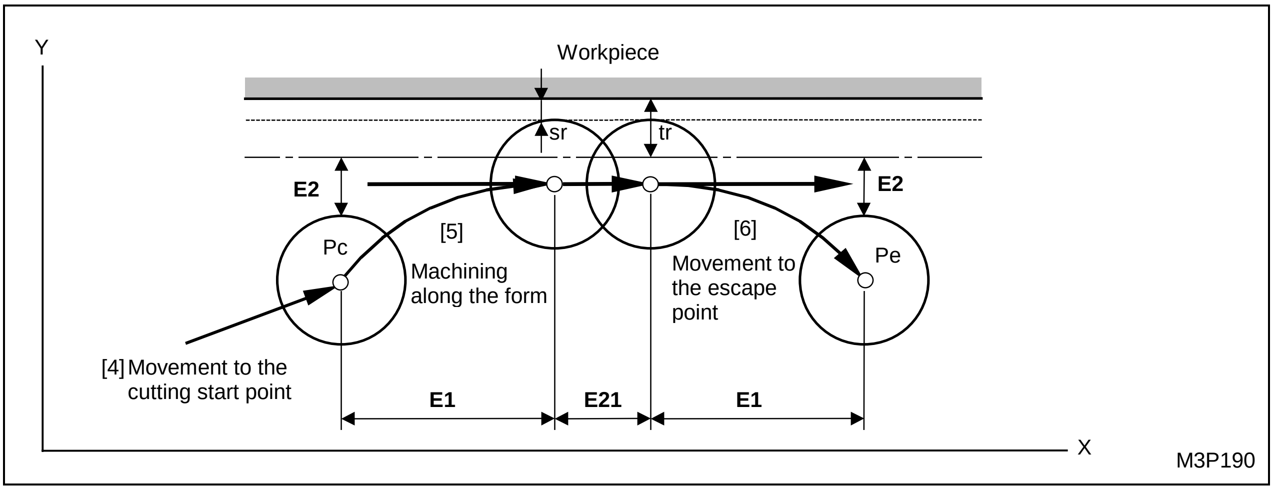
The bold codes represent parameter addresses.

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

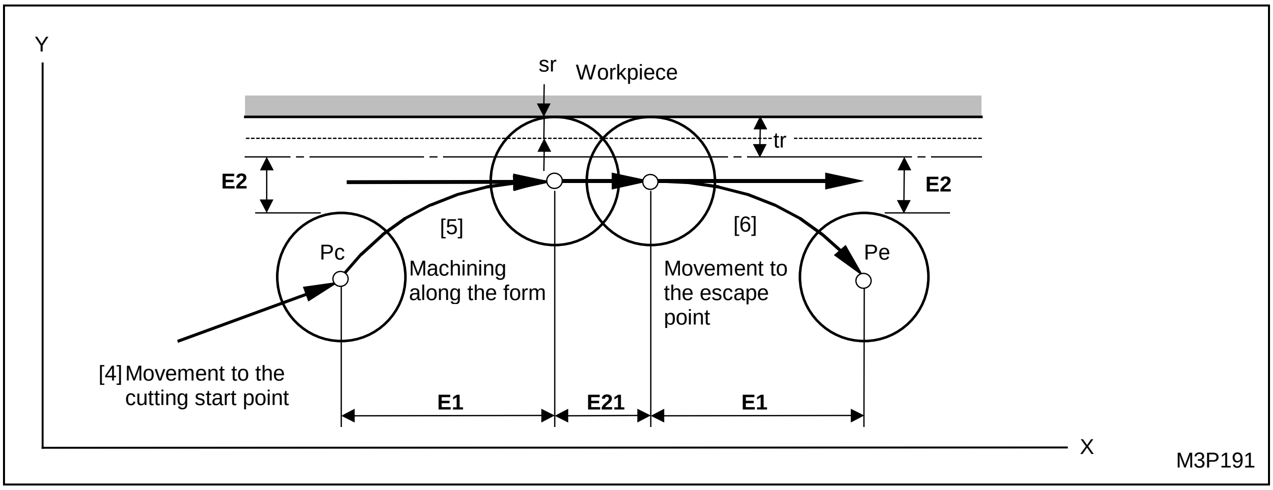
sr: Radial finishing allowance determined by the data **FIN-R** in the machining unit

<u>When cutting begins near the non-convex form</u>

- In case of roughing



- In case of finishing



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 3-6-6, “Precautions in line machining.”

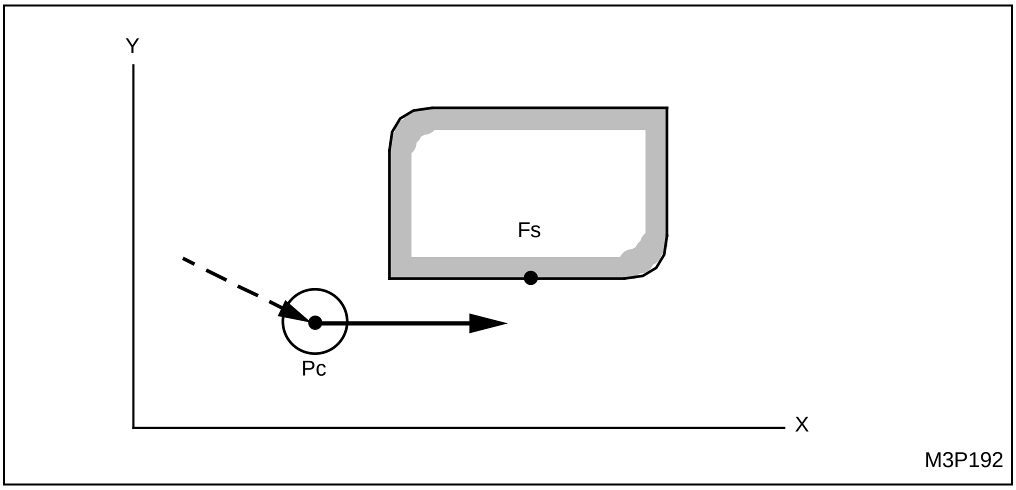
Note 3: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

Note 4: According to the position of the approach point entered in the tool sequence and to a machining shape entered in the shape sequence, a cutting start point and a cutting method vary as follows:

* The description below is entirely given, with the cutting direction taken CCW (counterclockwise).

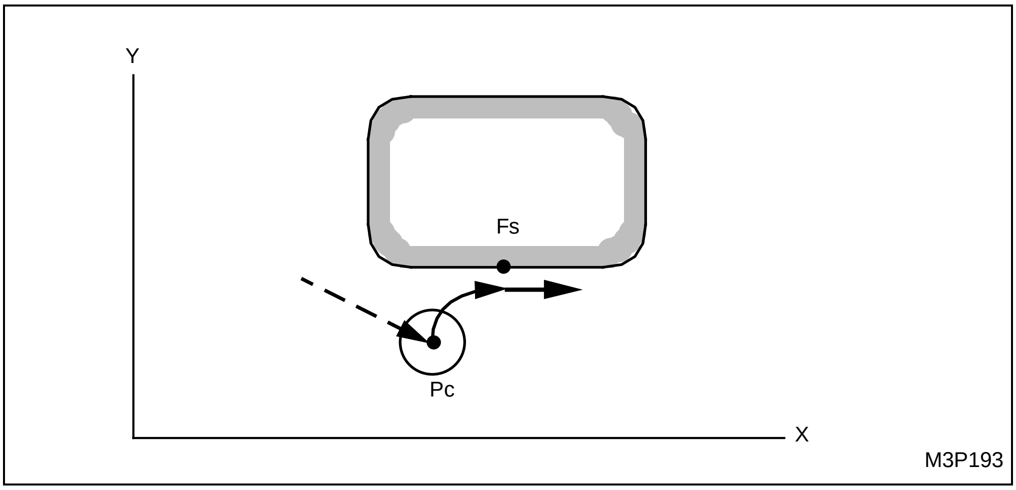
<u>When **?** is displayed in the items **APRCH-1**, **-2**</u>

- Form having a convex point:



Cutting is started from the convex point nearest the start point (Fs) entered in the shape sequence.

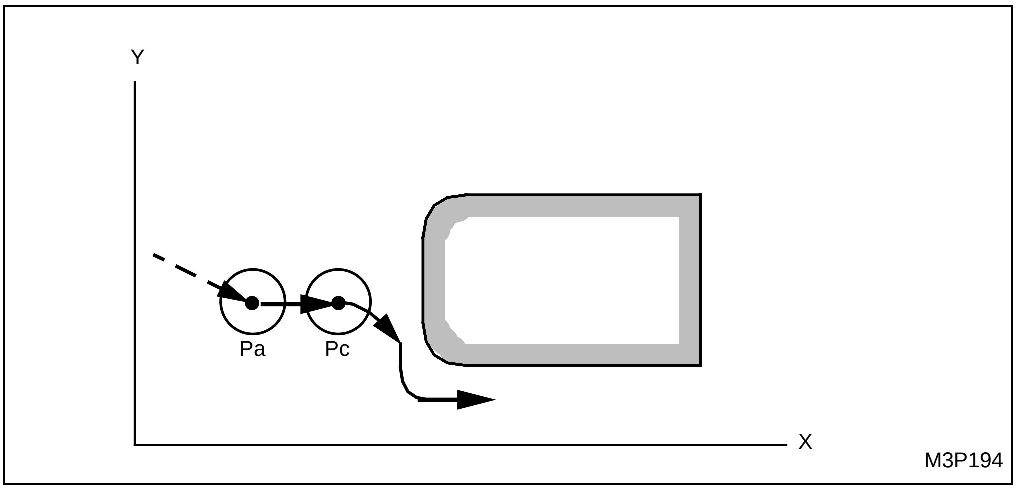
- Form having no convex point:



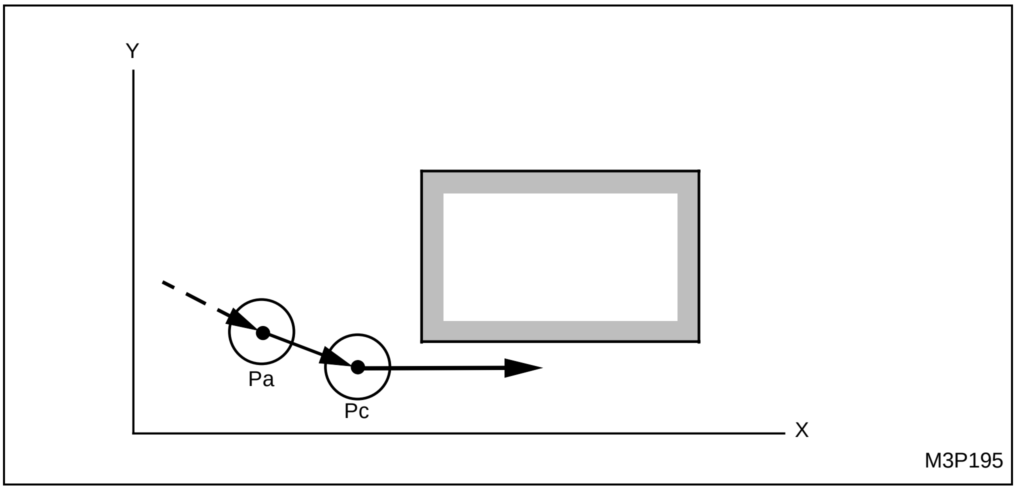
Cutting is started from the start point (Fs) entered in the shape sequence.

<u>When the data is entered in the items **APRCH-1**, **-2**</u>

- If there is not any convex point near the approach point:



- If there is a convex point near the approach point:



Pc: Cutting start point to be automatically established

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

Pa: Approach point to be determined using the numeric keys
When **?** is displayed by pressing the **[AUTO SET]** menu key, the coordinates of the cutting start point will be entered automatically.

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.

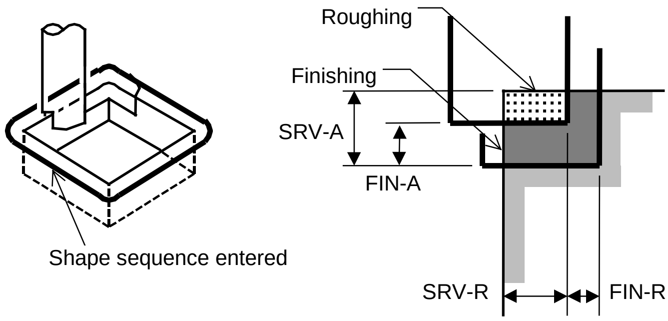
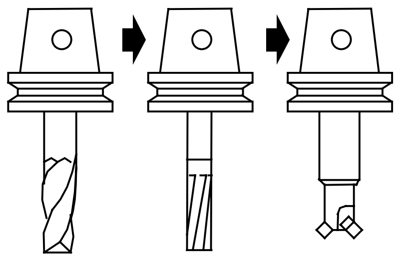
A. Data setting

```
UNo.   UNIT    MODE  POS-B POS-C   SRV-A    SRV-R   RGH    FIN-A     FIN-R          INTER-R   CHMF
 1    LINE IN
SNo.    TOOL      NOM-ϕ   No.  #   APRCH-1  APRCH-2  TYPE   AFD    DEP-A   WID-R  C-SP   FR   M   M   M
 R1   END MILL                                        ◆                       ◆
 F2   END MILL                                        ◆              ◆        ◆
```

◆: Data are not necessary to be set here.

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

Remark 2: For the tool sequence data setting, refer to Subsection 3-6-4.

LINE IN unit	Tool sequence
 D740PA127	 D740PA048

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.

B. Automatic tool development

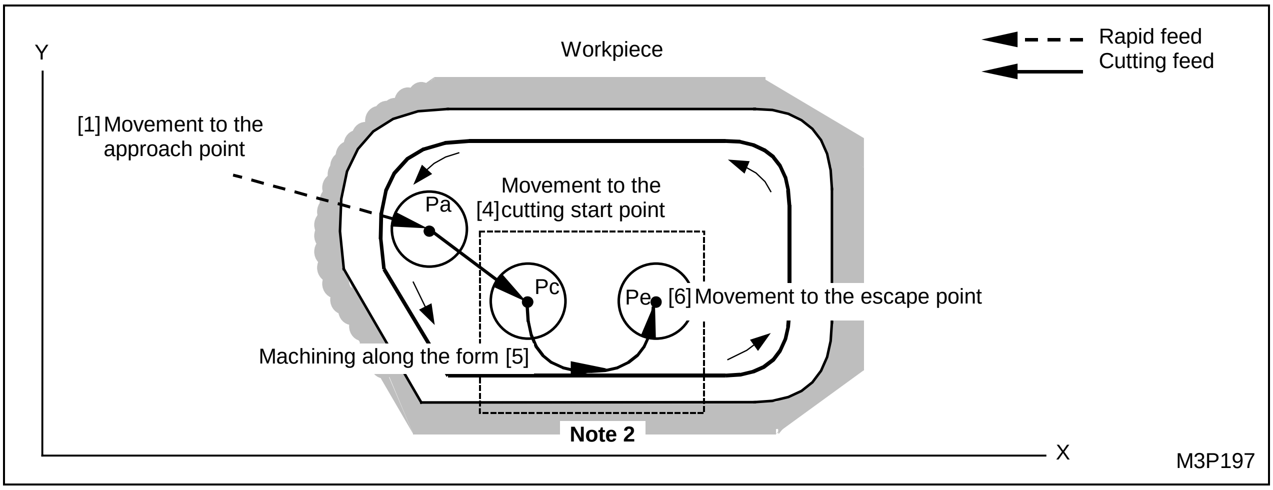
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to three tools are automatically developed though dependent upon the data **SRV-A**, **SRV-R**, **FIN-A** and **CHMF**.

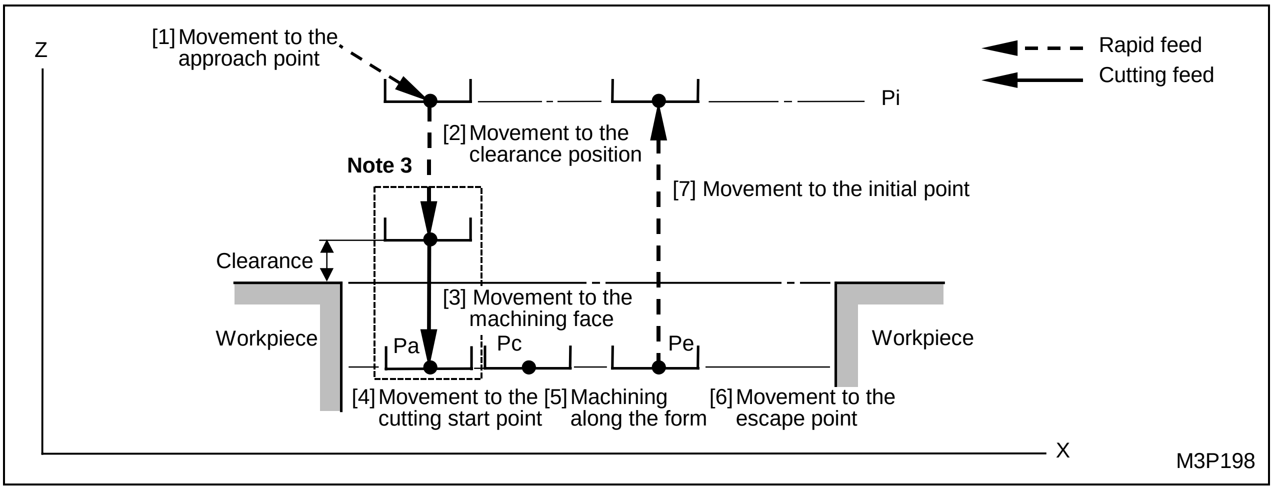
Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A or SRV-R ≤ FIN-R : One tool is selected.
(Chamfering)	CHMF≠ 0 : One tool is selected.

C. Tool path

X-Y-axis



X-Z-axis



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

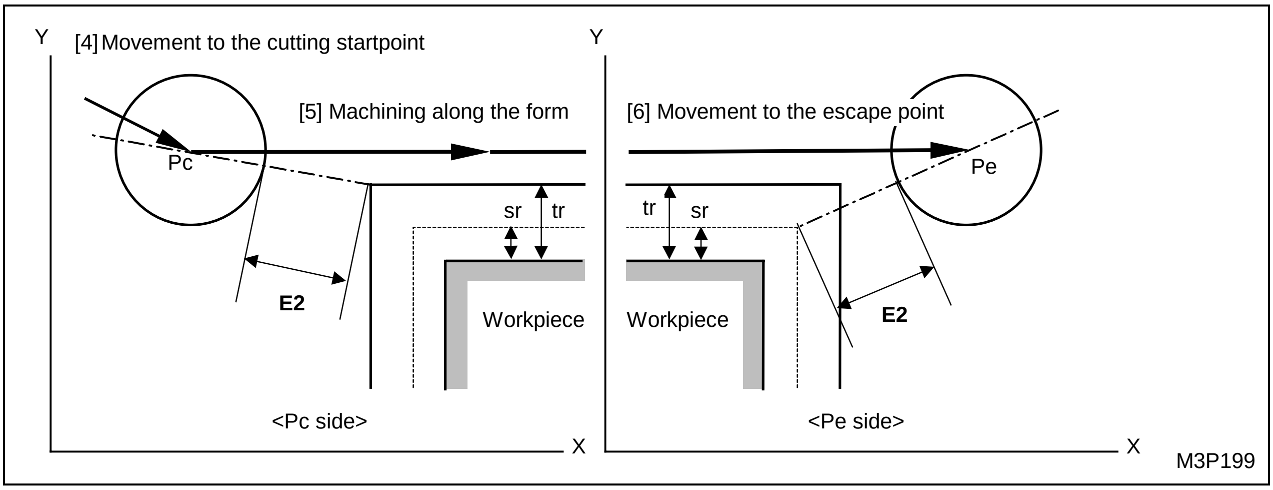
Pe: Escape point to be automatcially established

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

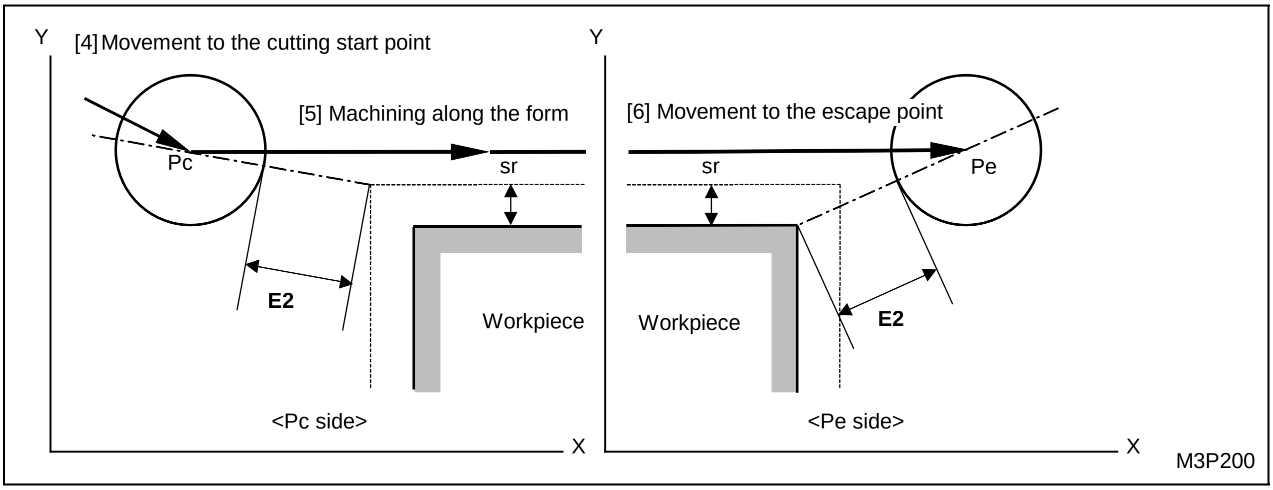
When the cutting begins near the convex form

- In case of roughing



M3P199

- In case of finishing



M3P200

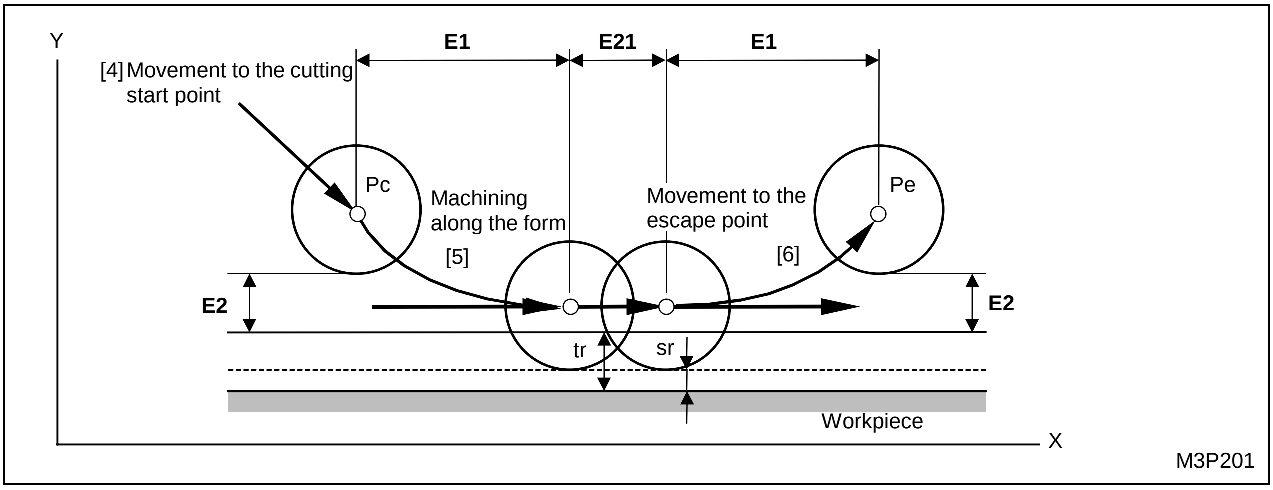
The bold codes represent parameter addresses.

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

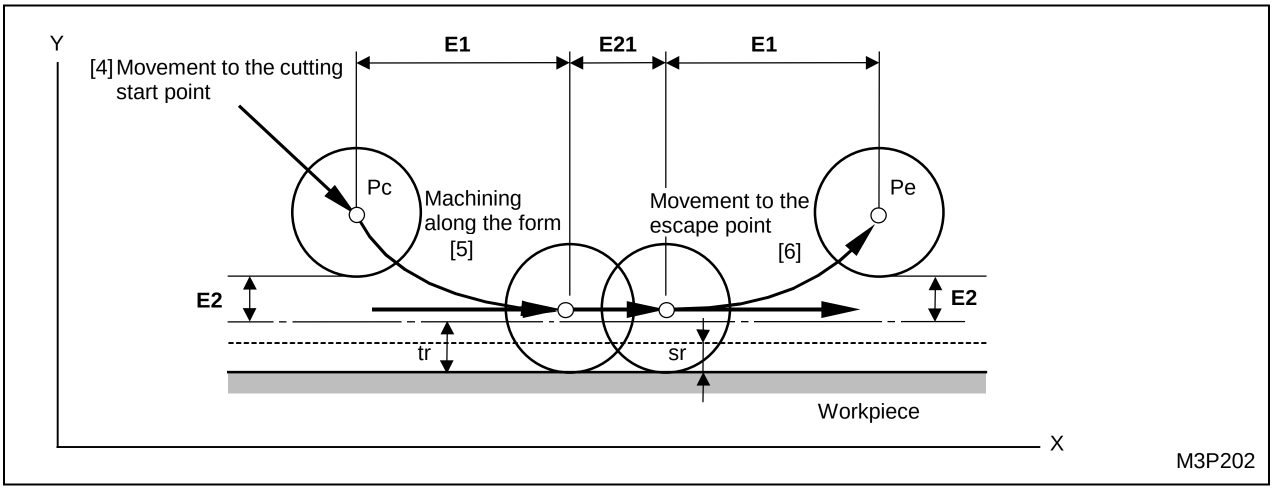
sr: Radial finishing allowance determined by the data **FIN-R** in the machining unit

When the cutting begins near the non-convex form

- In case of roughing



- In case of finishing



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

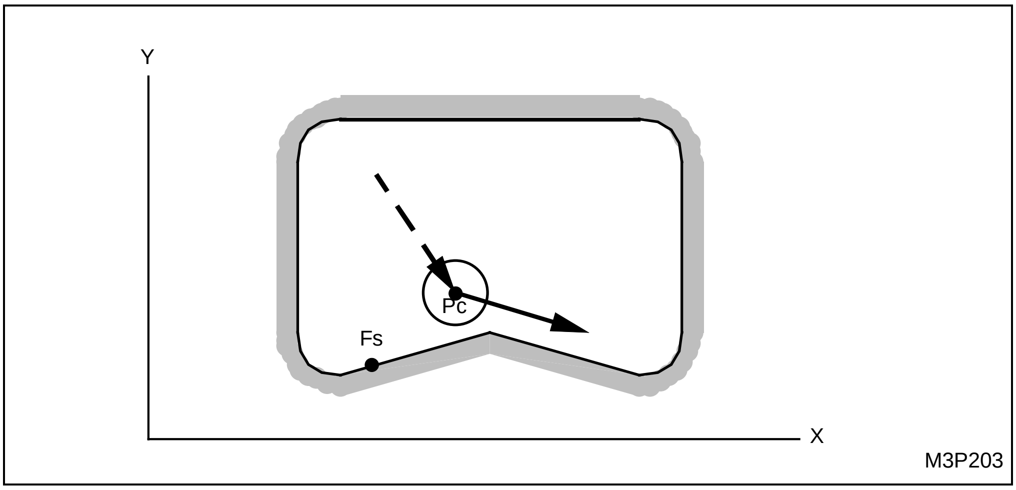
Note 3: See Subsection 3-6-6, “Precautions in line machining.”

Note 4: According to the position of the approach point entered in the tool sequence and to a machining shape entered in the shape sequence, a cutting start point and a cutting method vary as follows:

* The description below is entirely given, with the cutting direction taken CCW (counterclockwise).

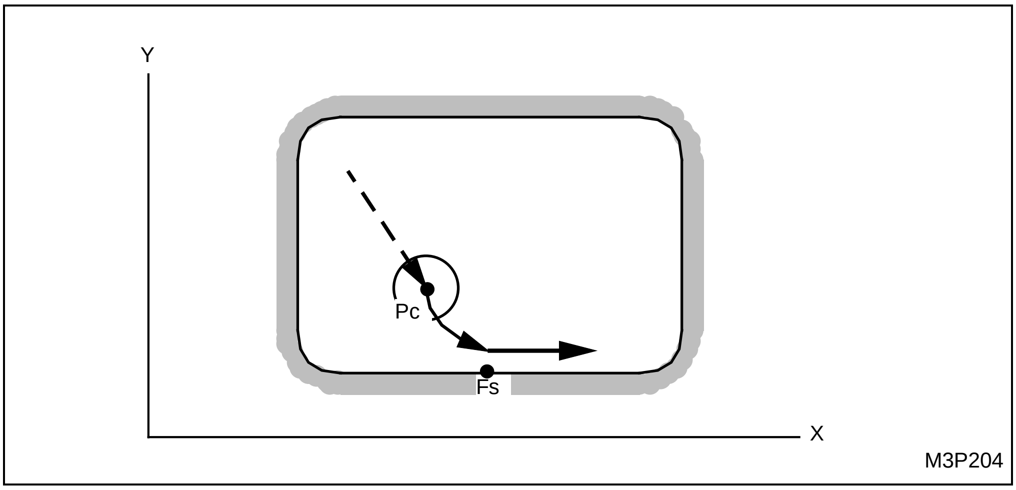
When **?** is displayed in the items **APRCH-1**, **-2**

- Form having a convex point:



Cutting is started from the convex point nearest the start point (Fs) entered in the shape sequence.

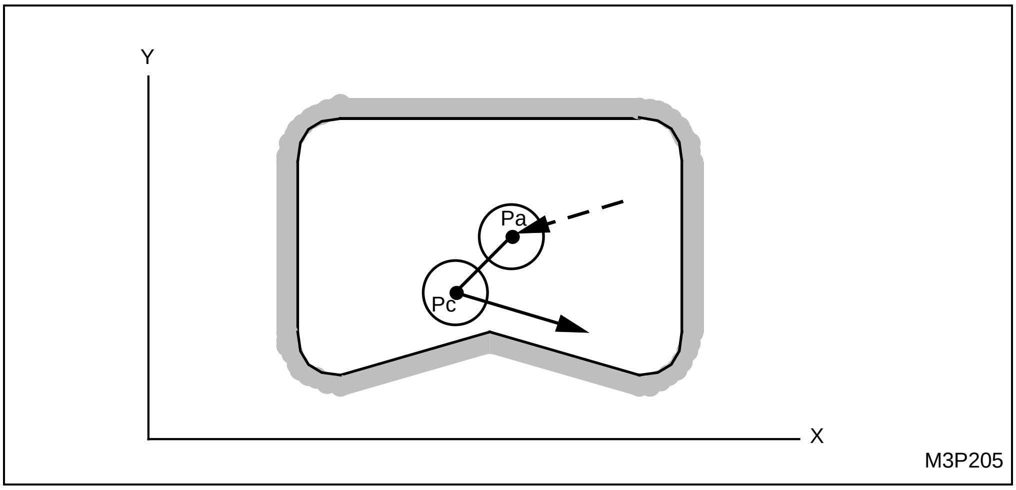
- Form having no convex point:



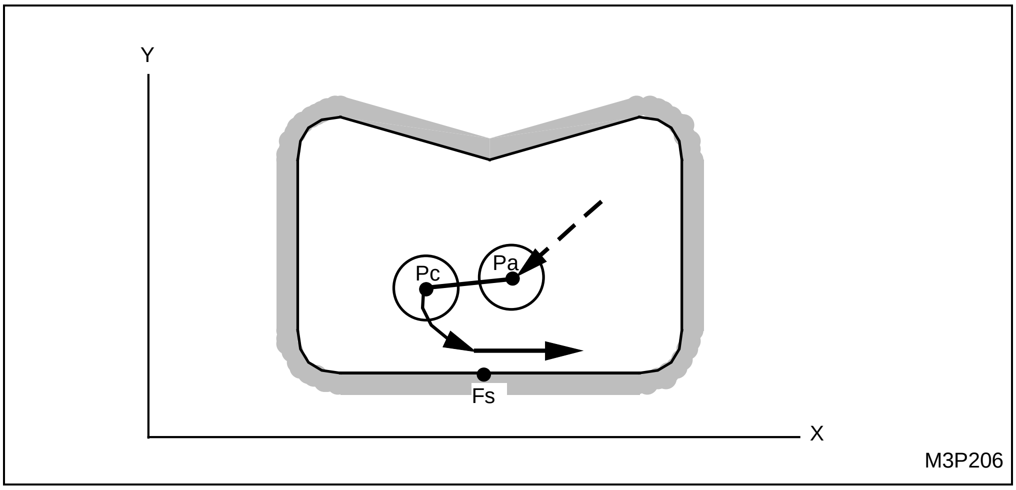
Cutting is started from the start point (Fs) entered in the shape sequence.

When the data is entered in the items **APRCH-1**, **-2**

- If there is a convex point near the approach point:



- If there is not any convex point near the approach point:



Pc: Cutting start point to be automatically established

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

Pa: Approach point to be determined using the numeric keys.
When **?** is displayed by pressing the **[AUTO SET]** menu key, the coordinates of the cutting start point will be entered automatically.

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.

A. Data setting

```
UNo.   UNIT    MODE  POS-B POS-C INTER-AX INTER-R  CHMF                 START  END
 1    CHMF RGT
SNo.    TOOL      NOM-ϕ  No.  #   APRCH-1  APRCH-2  TYPE   AFD    DEP-A   WID-R  C-SP   FR   M   M   M
 1    CHAMFER                                        ◆             ◆       ◆
```

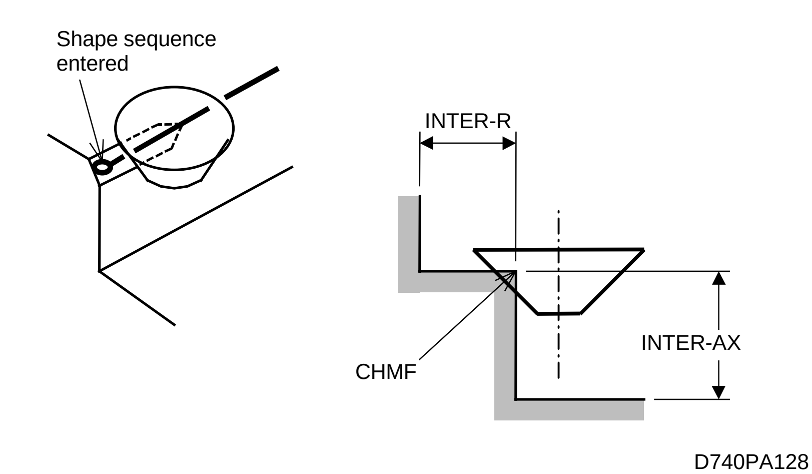
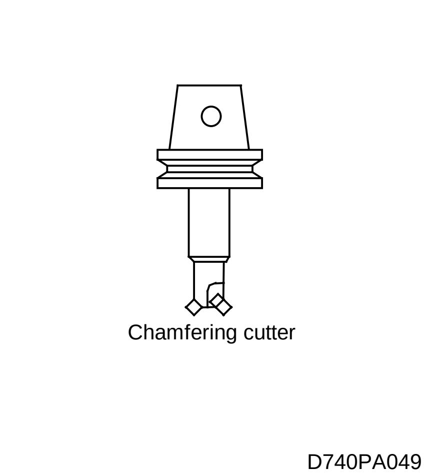
◆: Data are not necessary to be set here.

Remark 1: For data setting in **START** and **END**, refer to “1. Central linear machining unit (LINE CTR).”

Remark 2: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 3: For the tool sequence data setting, refer to Subsection 3-6-4.

Note: If a centering drill is used, a nose angle of 90 degrees is set for machining.

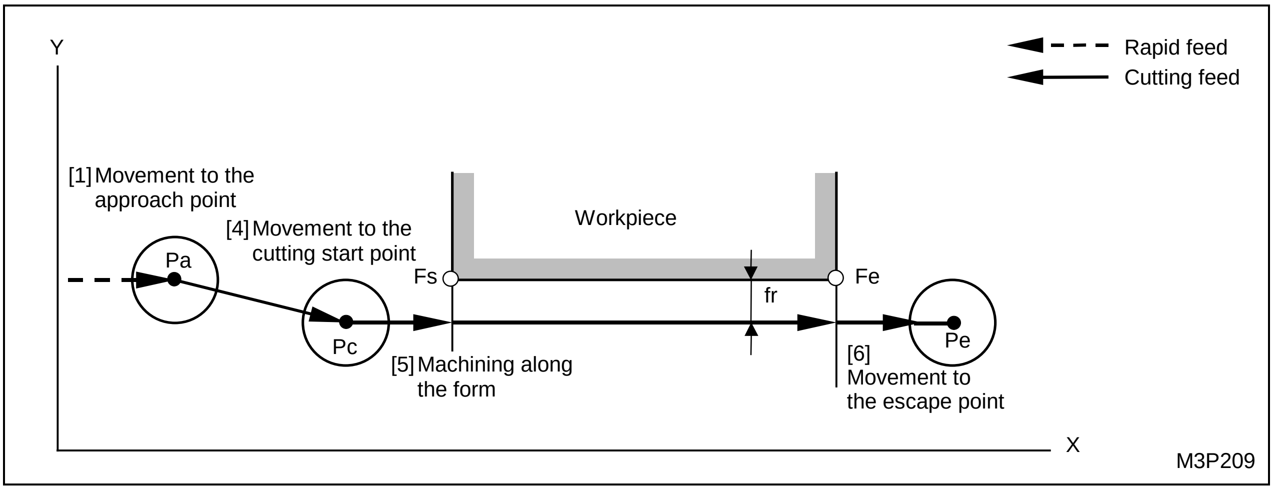
CHMF RGT unit	Tool sequence
	

B. Automatic tool development

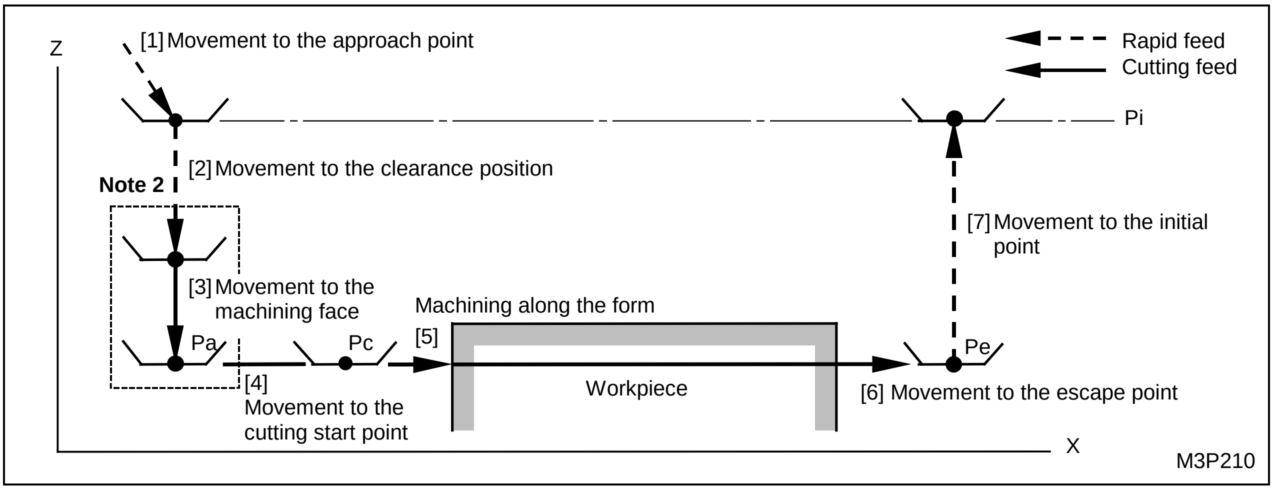
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

C. Tool path

X-Y-axis



X-Z-axis



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

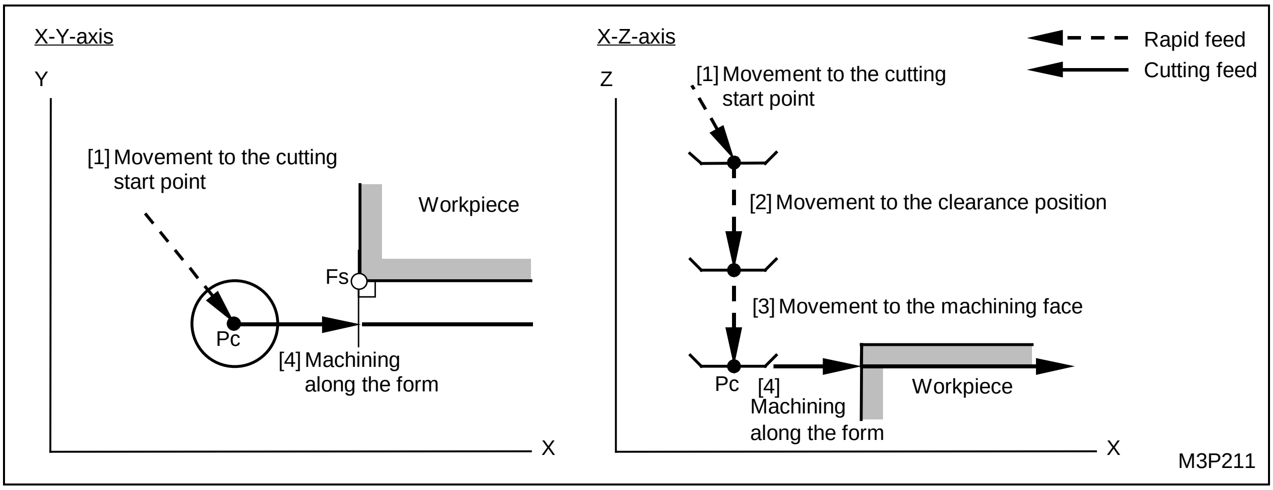
Fs: Start point of form to be entered in the shape sequence

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 items **APRCH-1** and **-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, a coordinate of the cutting start point is entered automatically in the items.



Note 2: See Subsection 3-6-6, “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.

A. Data setting

```
UNo.   UNIT    MODE  POS-B POS-C INTER-AX  INTER-R  CHMF                START    END
 1   CHMF LFT
SNo.    TOOL      NOM-ϕ  No.  #  APRCH-1  APRCH-2  TYPE   AFD    DEP-A   WID-R  C-SP   FR   M   M   M
 1   CHAMFER                                        ◆              ◆        ◆
```

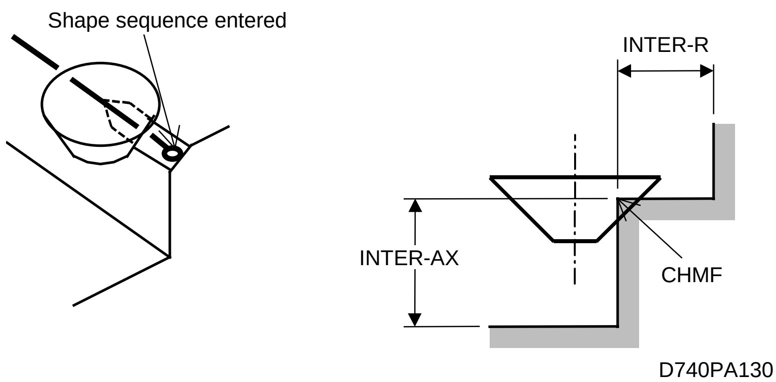
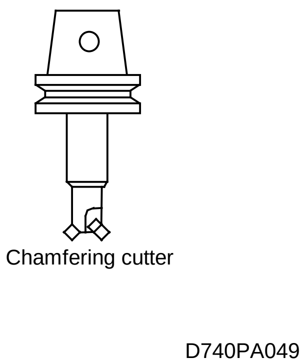
◆: Data are not necessary to be set here.

Remark 1: For data setting in **START** and **END**, refer to "1. Central linear machining unit (LINE CTR)."

Remark 2: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 3: For the tool sequence data setting, refer to Subsection 3-6-4.

Note: If a centering drill is used, a nose angle of 90 degrees is set for machining.

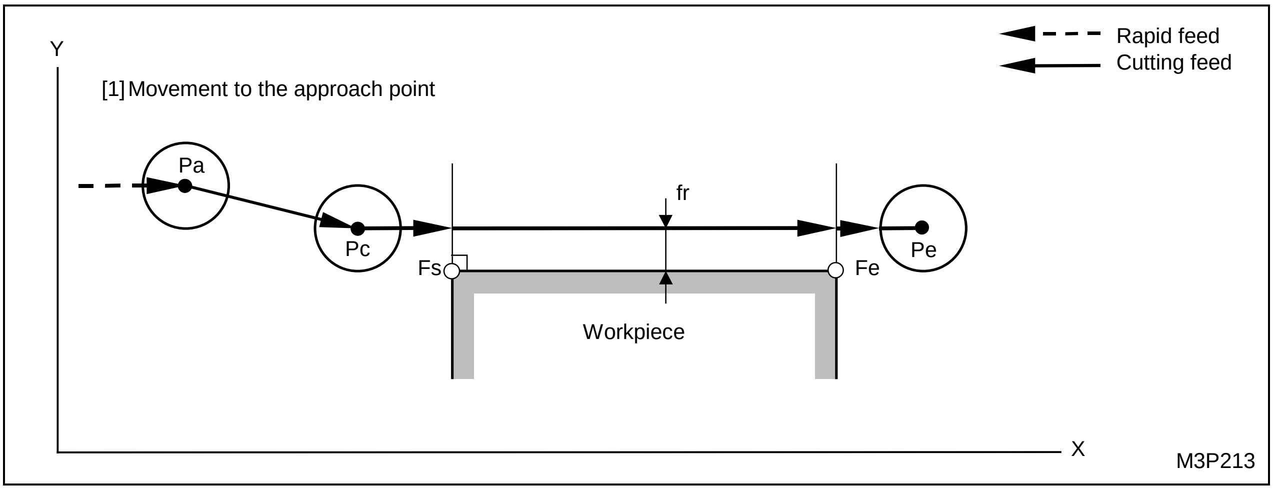
CHMT LFT unit	Tool sequence
	

B. Automatic tool development

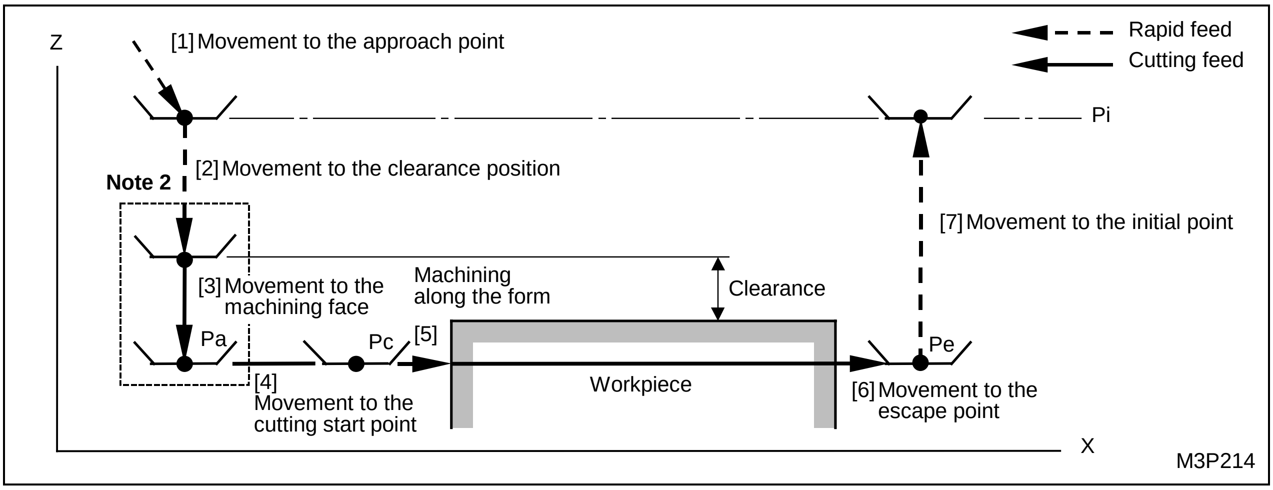
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

C. Tool path

X-Y-axis



X-Z-axis



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

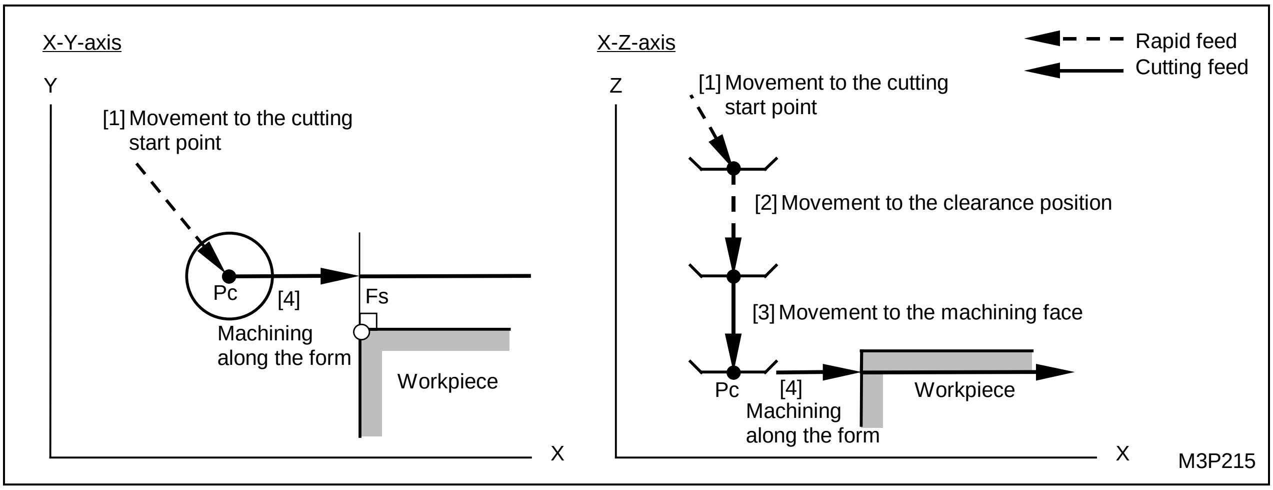
Fs: Start point of form to be entered in the shape sequence

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 items **APRCH-1** and **-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, a coordinate of the cutting start point is entered automatically in the items.



Note 2: See Subsection 3-6-6, “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.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	INTER-AX	INTER-R	CHMF
1	CHMF OUT						

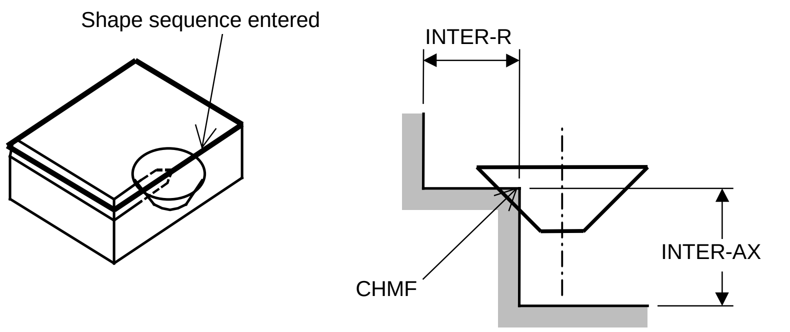
SNo.	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M	M	M
1	CHAMFER								◆	◆					

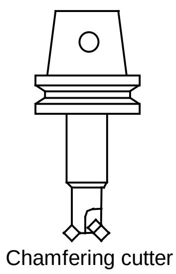
◆: Data are not necessary to be set here.

Remark 1: In this unit, chamfering tools are automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 2: For the tool sequence data setting, refer to Subsection 3-6-4.

Note: If a centering drill is used, a nose angle of 90 degrees is set for machining.

<table>
<tr><th>CHMF OUT unit</th><th>Tool sequence</th></tr>
<tr><td>


D740PA131</td><td>


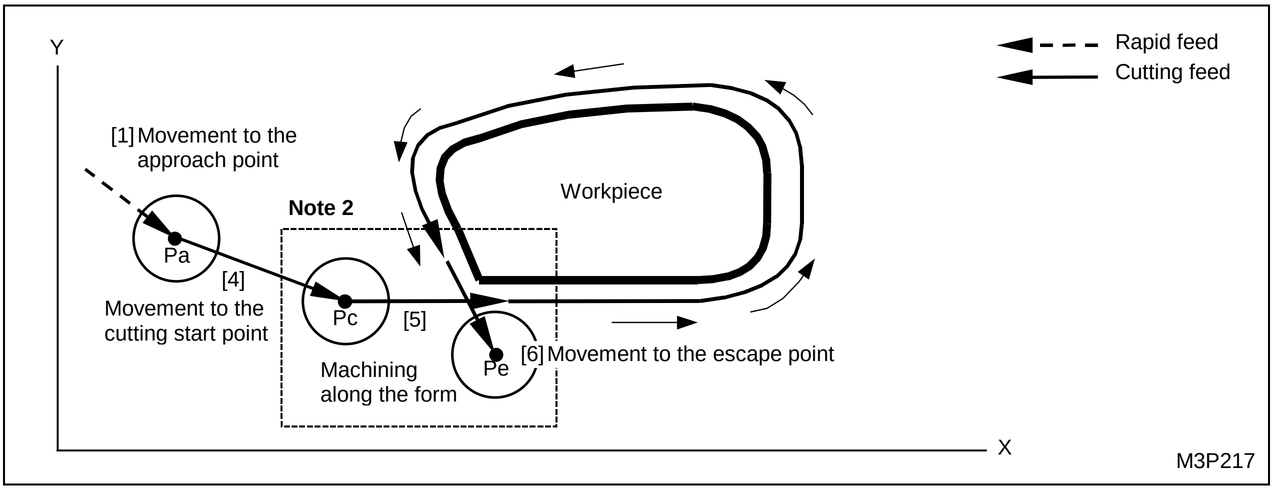
D740PA049</td></tr>
</table>

B. Automatic tool development

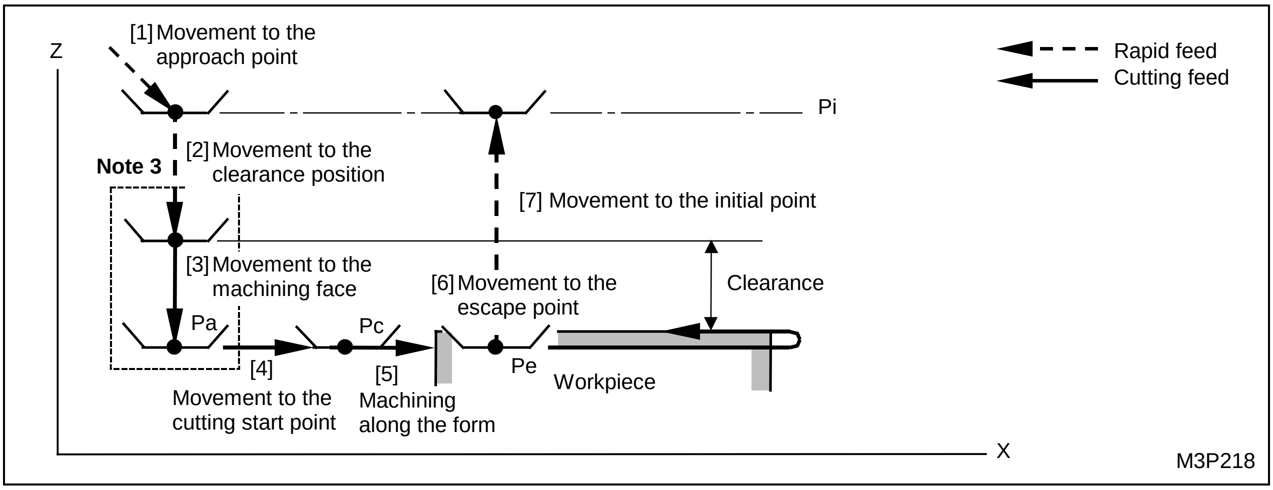
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

C. Tool path

X-Y-axis



X-Z-axis



Pi: Initial point

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

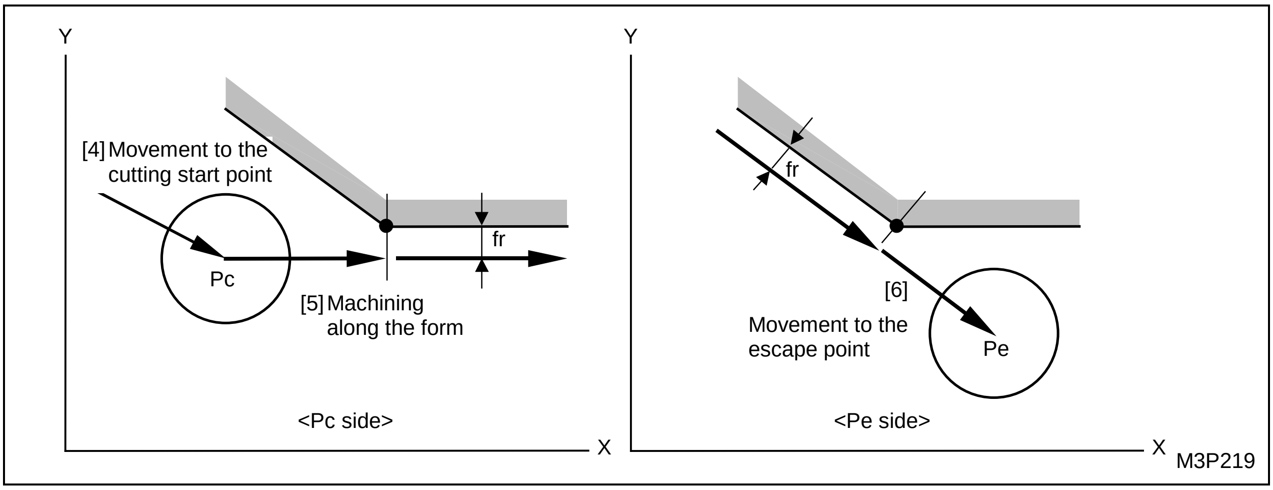
Pc: Cutting start point to be automatically established

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.

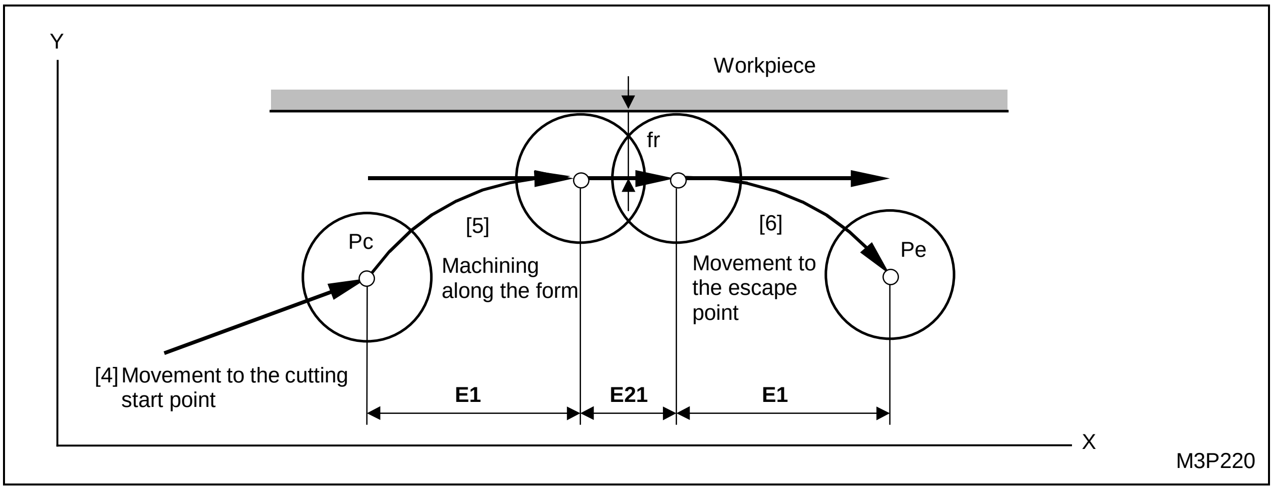
Note 2: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form



M3P219

When the cutting begins near the non-convex form



M3P220

The bold codes represent parameter addresses.

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

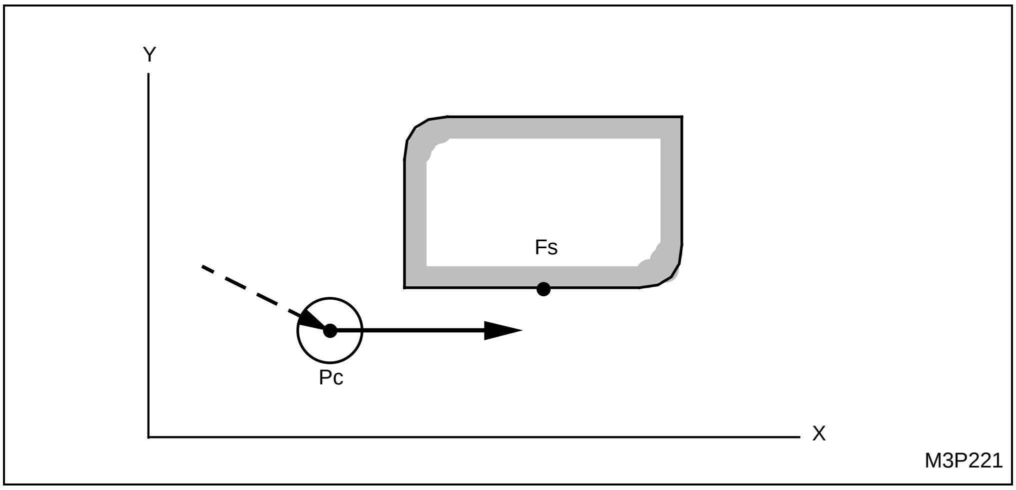
Note 3: See Subsection 3-6-6, "Precautions in line machining."

Note 4: According to the position of the approach point entered in the tool sequence and to a machining form entered in the shape sequence, a cutting start point and a cutting method vary as follows:

* The description below is entirely given, with the cutting direction taken CCW (counterclockwise).

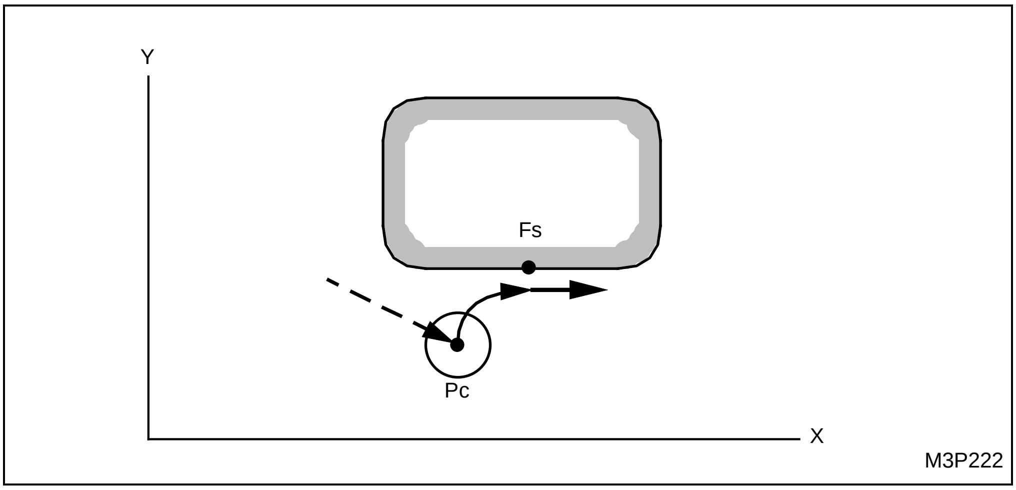
When **?** is displayed in the items **APRCH-1**, **-2**

- Form having a convex point:



Cutting is started from the convex point nearest the start point (Fs) entered in the shape sequence.

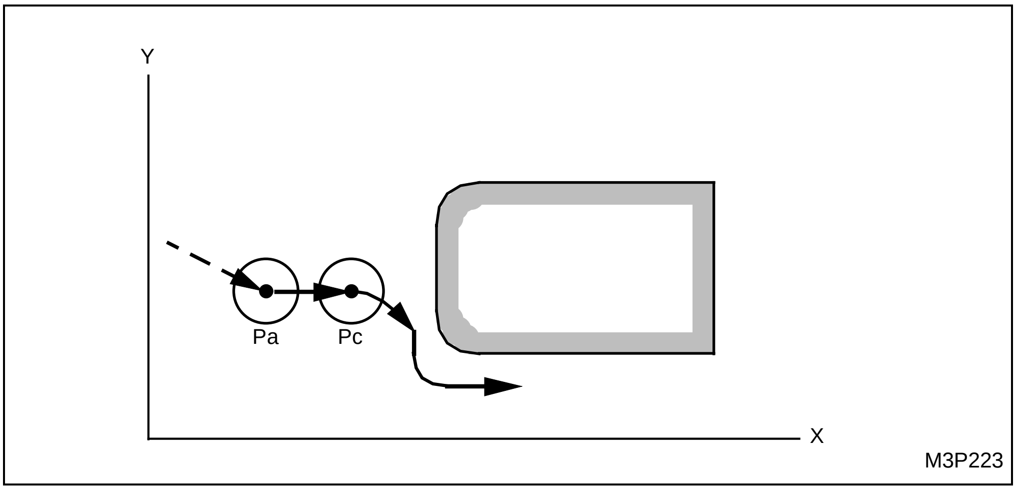
- Form having no convex point:



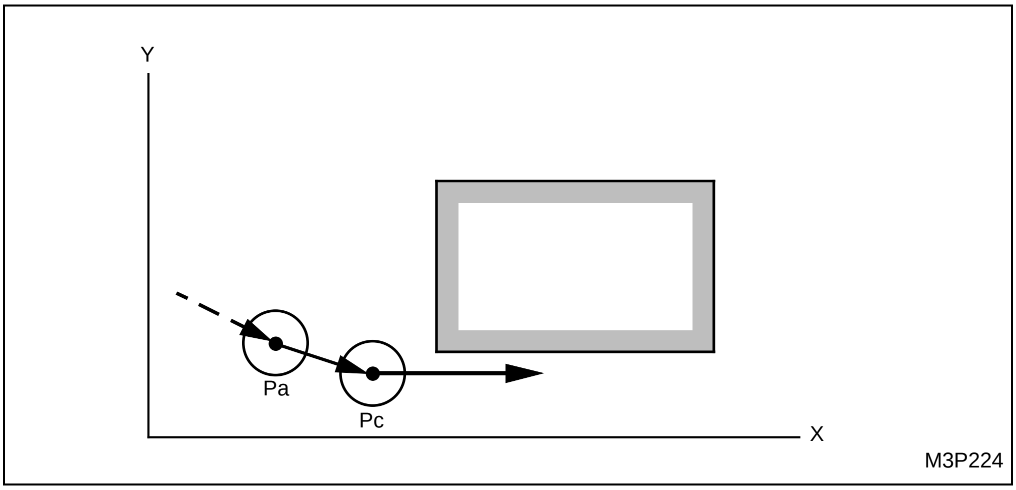
Cutting is started form the start point (Fs) entered in the shape sequence.

<u>When the data is entered in the items **APRCH-1**, **-2**</u>

- If there is not any convex point near the approach point.



- If there is a convex point near the approach point.



Pa: Approach point to be determined using the numeric keys
If **?** is displayed by pressing the **[AUTO SET]** menu key, the coordinates of cutting start point will be entered automatically.

Pc: Cutting start point to be automatically established

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

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.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	INTER-AX	INTER-R	CHMF
1	CHMF IN						

SNo.	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M	M	M
1	CHAMFER								◆	◆					

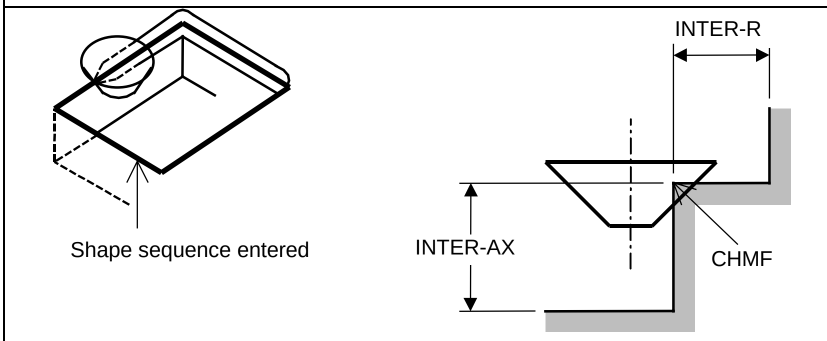
◆: Data are not necessary to be set here.

Remark 1: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 2: For the tool sequence data setting, refer to Subsection 3-6-4.

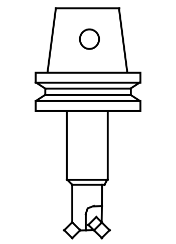
Note: If a centering drill is used, a nose angle of 90 degrees is set for machining.

CHMF IN unit



D740PA132

Tool sequence



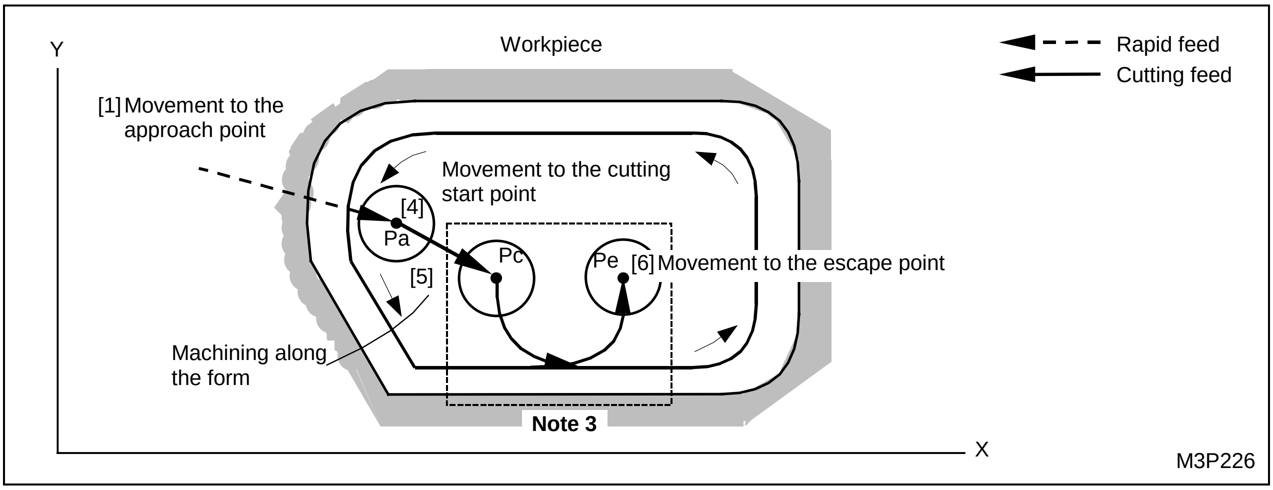
D740PA049

B. Automatic tool development

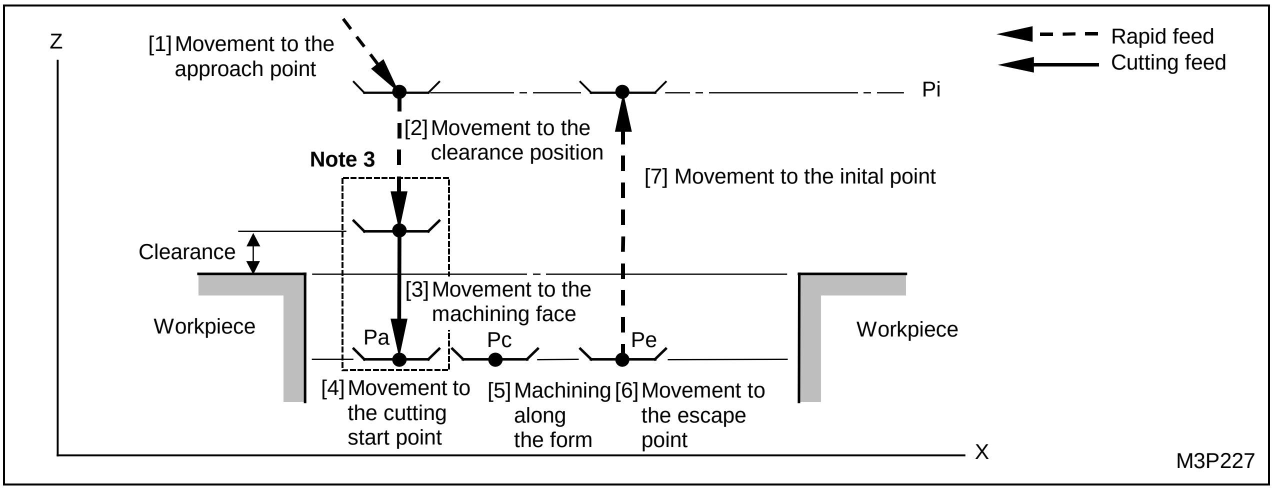
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

C. Tool path

X-Y-axis



X-Z-axis



Pi: Initial point

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

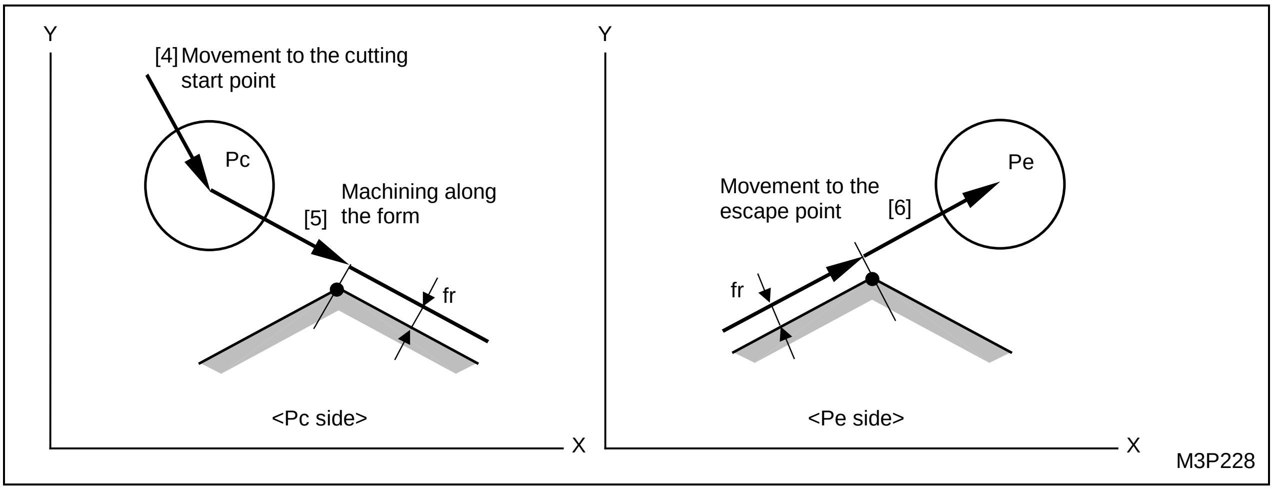
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

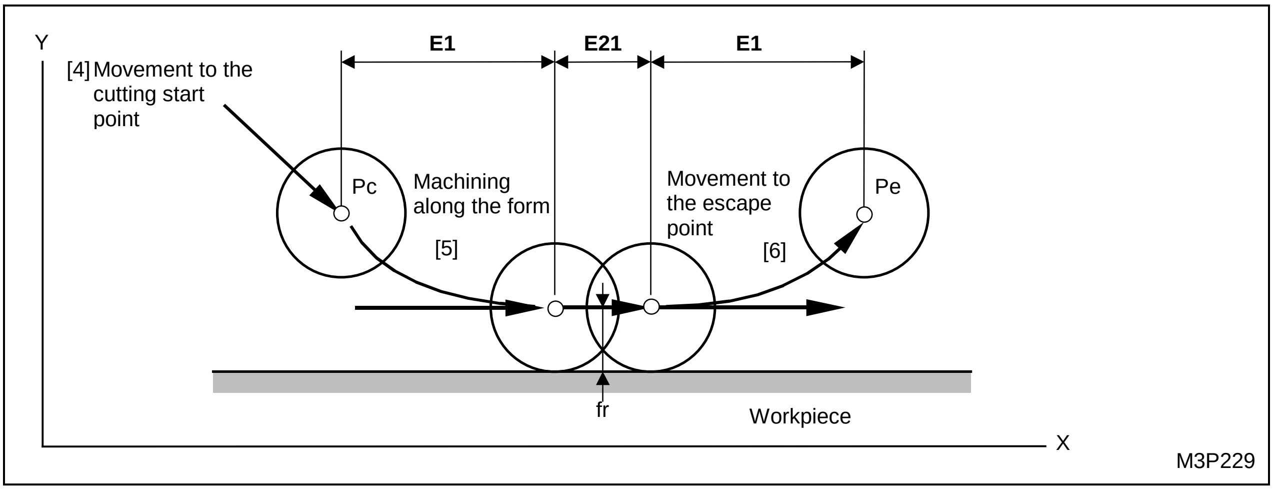
Note 1: The feedrate on tool path [3] is dependent upon the **AFD** (axial feed) in the tool sequence.

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

When the cutting begins near the convex form



When the cutting begins near the non-convex form



The bold codes represent parameter addresses.

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

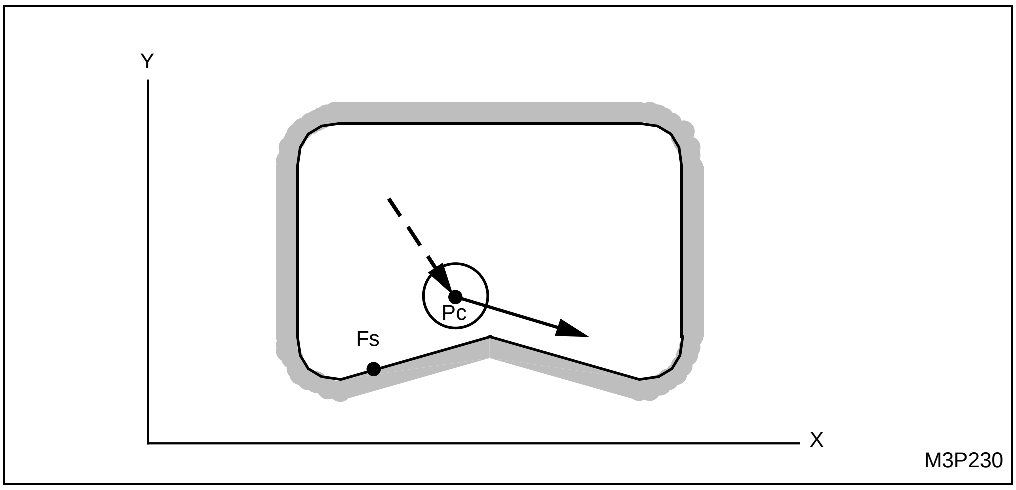
Note 3: See Subsection 3-6-6, "Precautions in line machining."

Note 4: According to the position of the approach point entered in the tool sequence and to a machining form entered in the shape sequence, a cutting start point and a cutting method vary as follows:

* The description below is entirely given, with the cutting direction taken CCW (counterclockwise).

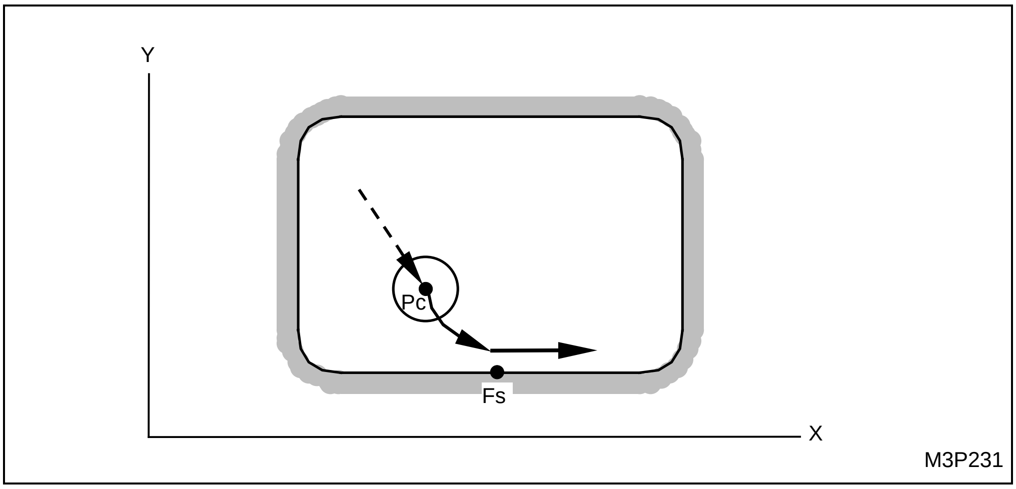
When **?** is displayed in the items **APRCH-1**, **-2**

- Form having a convex point:



Cutting is started from the convex point nearest the start point (Fs) entered in the shape sequence.

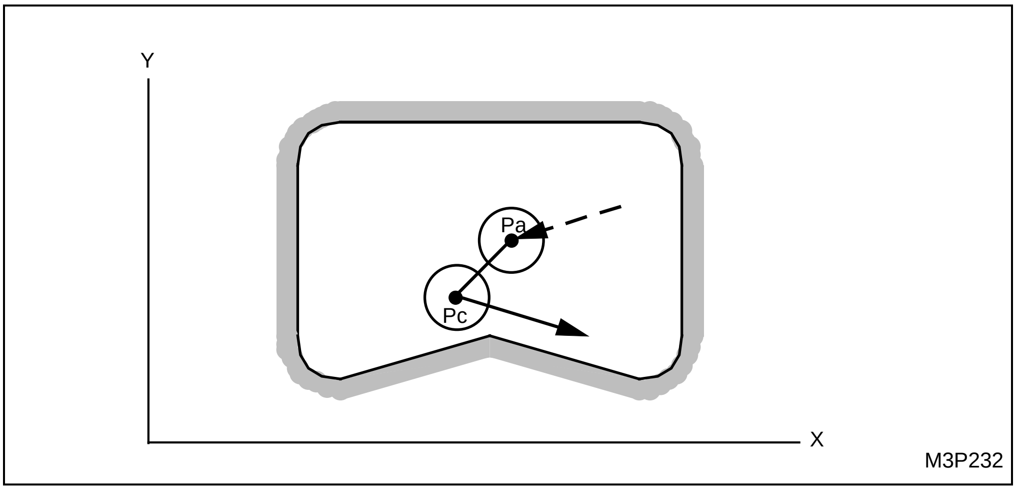
- Form having no convex point:



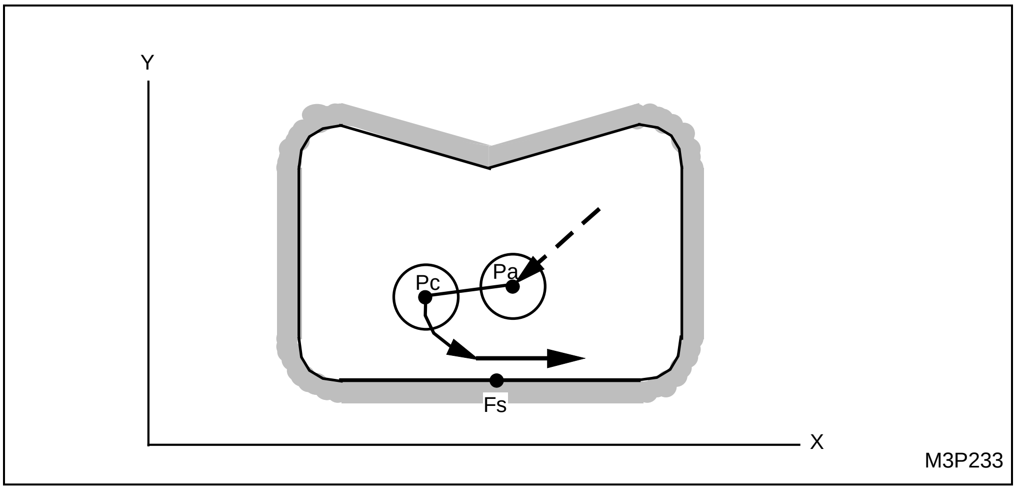
Cutting is started form the start point (Fs) entered in the shape sequence.

When the data is entered in the items **APRCH-1**, **-2**

- If there is a convex point near the approach point.



- If there is not any convex point near the approach point.



Pa: Approach point to be determined using the numeric keys
When **?** is displayed by pressing the **[AUTO SET]** menu key, the coordinates of cutting start point will be entered automatically.

Pc: Cutting start point to be automatically established

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

3-6-4 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.

<u>Tool sequence data</u>

	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	WID-R	C-SP	FR	M M M
Item	(1)	(2)(3)(4)	(5)	(6)	(7)	(7)	(8)	(9)	(10)	◆	(11)	(12)	(13)

◆: Not necessary to be set here.

For setting of each data item refer to 1 to 13 below.

1. TOOL (Tool designation)

The name of a tool can be changed by the use of menu keys.
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, a chamfering cutter and a centering drill are selectable.

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	CENTER DRILL					

2. NOM-ϕ (Nominal diameter of tool)

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. NOM-ϕ (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	HEAVY TOOL	>>>

To slowly change a heavy tool in the ATC mode, select a heavy tool identification code.

With the **[HEAVY TOOL]** menu key pressed, the display will change over to the menu for heavy tool identification code. Then select a code from the menu to identify those tools which have an identical nominal diameter.

4. NOM-ϕ (Turret selection)

For the machine with the lower turret, select the turret in which the tool to be used is mounted. The following menu is displayed (if **[SET UPPER TURRET]** is selected, the column will remain blank, and if **[SET LOWER TURRET]** is selected, "▼" will be displayed). See Chapter 5, LOWER-TURRET CONTROL FUNCTIONS, for further details:

SET UPPER TURRET	SET LOWER TURRET								

5. **No. (Priority No.)**

Assign priority levels in the order of machining. The following menu is displayed. A press of a menu key displays the menu item in reverse mode, allowing a priority number to be assigned.

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN			PRI.No. ALL ERAS	SUB PROG PROC END	
	(a)		(b)	(c)			(d)	(e)	

The function of menu item (a) to (e) is described below:

Menu item	Function
(a)	Select to conduct subsequent-machining.
(b)	Select to change the priority number for the tool within the particular process. If the cursor is present at a blank space, assign a new number in a usual manner. Entry of an existing priority number displays alarm **420 SAME DATA EXISTS**.
(c)	Select to assign a priority number to the tool to be used repeatedly in the particular process. Alarm **420 SAME DATA EXISTS** will be displayed if the assigned priority number has already been set on any other unit line.
(d)	Selection of this item displays message **ALL ERASE (PROC:0, PROG:1)?**. Setting 0 will erase the priority numbers preassigned to the tool to be used repeatedly in the process. Setting 1 will erase the priority numbers preassigned to the tool to be used repeatedly in the program.
(e)	Select to terminate the process with the subprogram unit.

For details see Chapter 4, “PRIORITY FUNCTION FOR THE SAME TOOL.”

6. **# (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. For details see Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”.

LOWER TURRET POS.1	LOWER TURRET POS.2								

7. **APRCH-1, APRCH-2 (Coordinates of the approach point)**

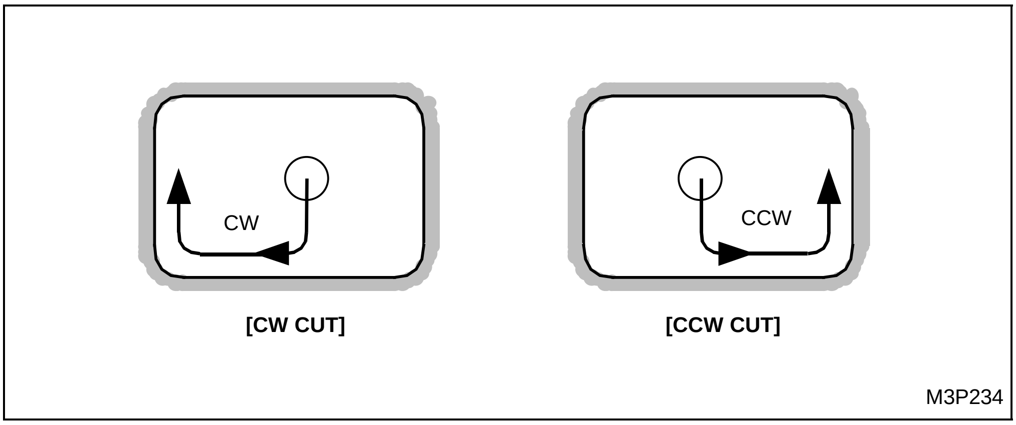
Enter coordinates of the position at which a tool is to cut in axially.

Pressing the **[AUTO SET]** menu key sets a question mark (**?**). After the tool path check is performed, **?** will automatically change over to the coordinates of a cutting start point. (Refer to tool path for each unit.)

8. TYPE (Machining method)

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.

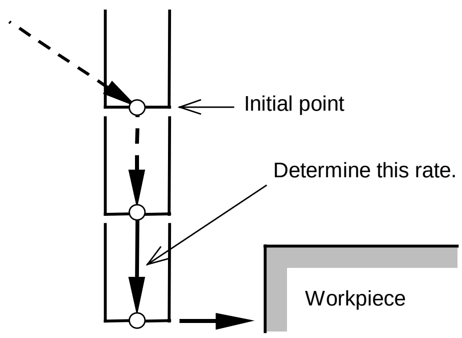
CW CUT	CCW CUT								



9. AFD (Axis feedrate)

Enter the feedrate in 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 × $\frac{\mathbf{E17}}{10}$
Numeric value (α)	Feed × α

M3P235

10. DEP-A (Cutting stroke)

In roughing, a maximum axial cutting stroke in one cycle is entered. With the **[AUTO SET]** menu key 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 3-6-6, “Precautions in line machining.”)

11. C-SP (Surface speed)

To auto-set a surface 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 - W. MAT./T. MAT** display.

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

HSS AUTO	CARBIDE AUTO								

Data can also be set using the numeric keys.

12. FR (Feedrate)

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

13. M (M-code)

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 three M-codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu.

3-6-5 Shape sequence data of the line machining unit

The data setting items 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 data entry procedure in Subsection 3-7-7.

3-6-6 Precautions in line machining

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

- Cutting start position in the axial direction
- Type of routing through approach points
- Type of escape in the axial direction after machining

For each factor refer to A, B and C below.

[Basic tool path]

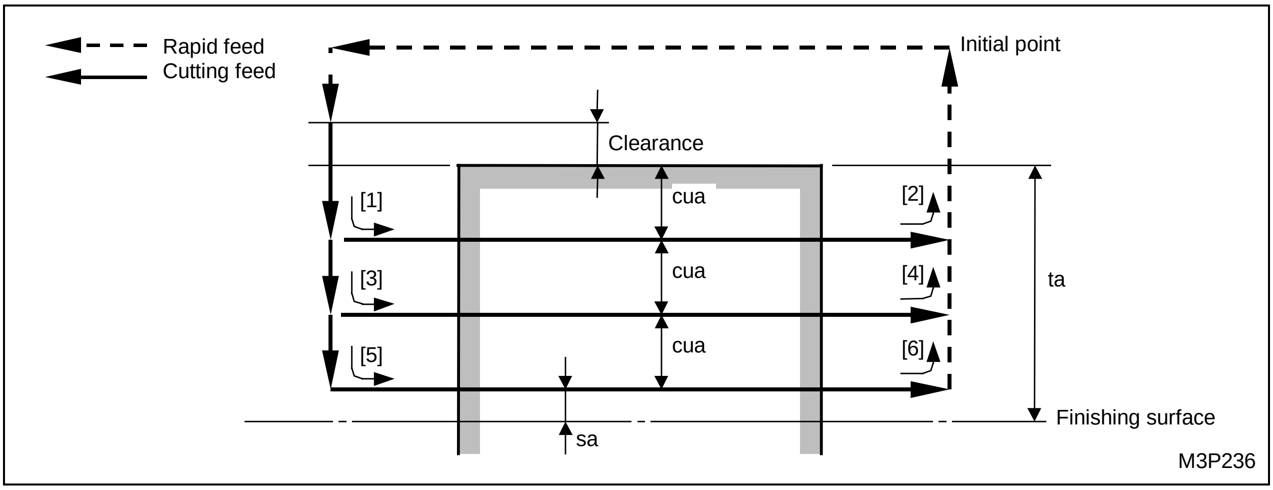


Fig. 3-22 Basic tool path

cua: Cutting depth in the axial direction per pass (Axial cutting depth **DEP-A** to be entered in the tool sequence)

Calculation of cua:

$$\boxed{\text{cua} = \frac{\text{ta} - \text{sa}}{\text{n}}}$$

$$\text{n} = \frac{\text{ta} - \text{sa}}{\text{cua}}$$

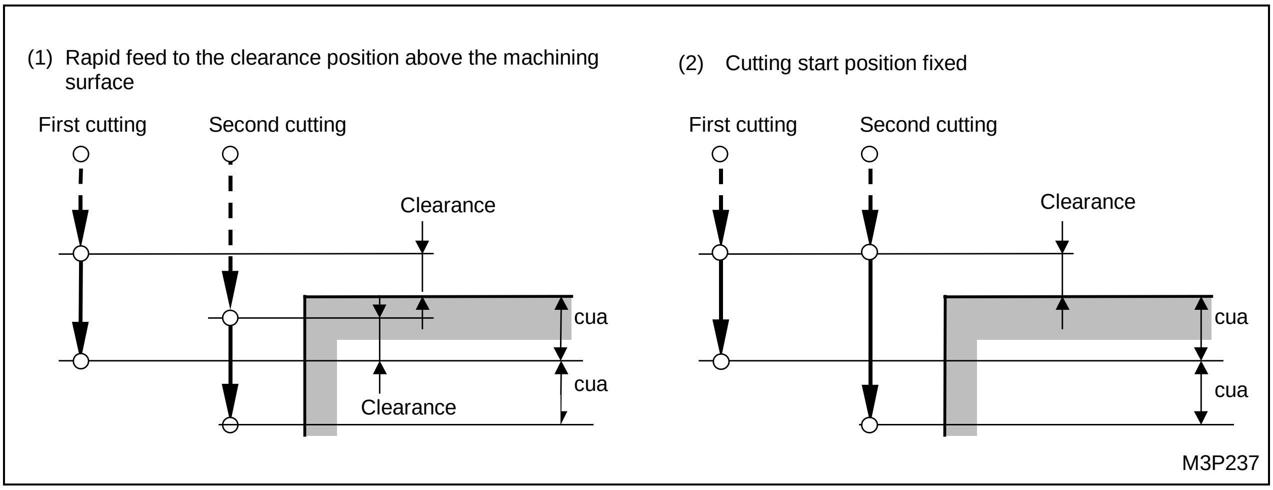
ta: Axial cutting allowance **SRV-A** to be entered in the machining unit

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

n: Number of passes in the axial direction (Integer obtained by rounding up the decimal fraction)

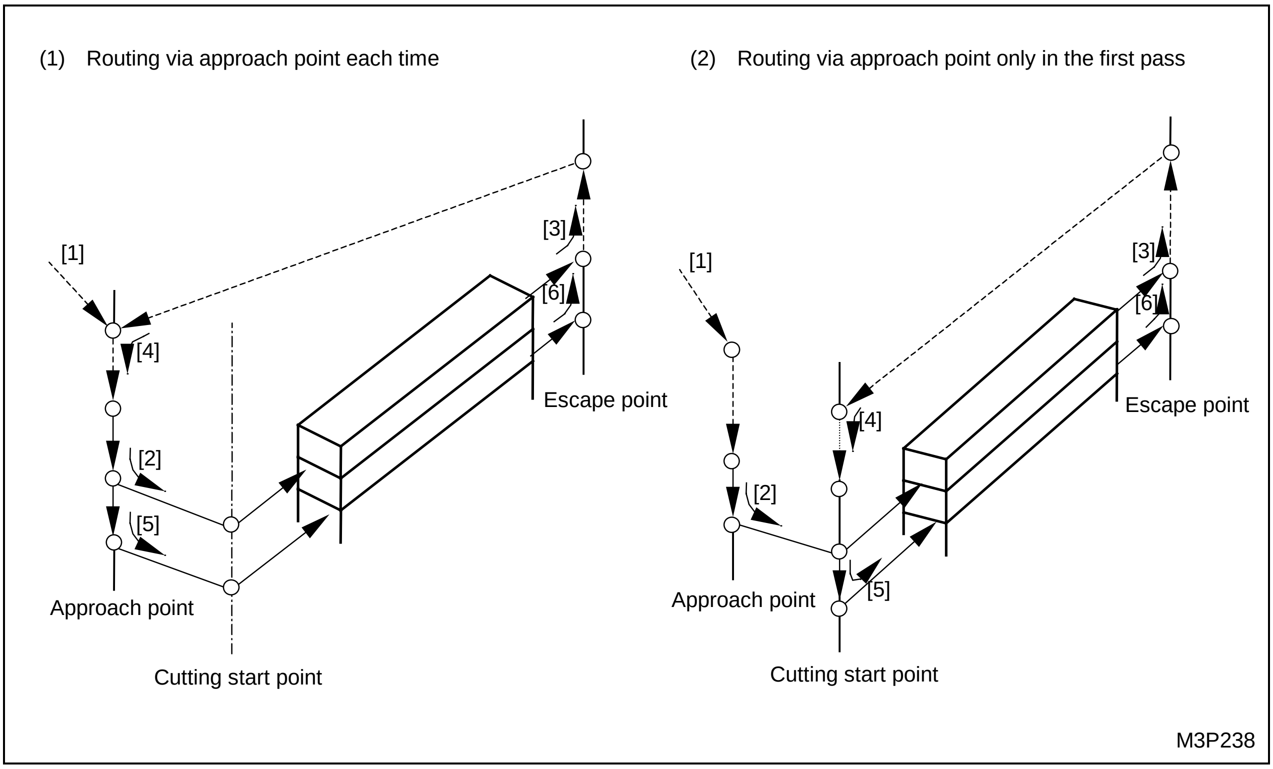
A. Cutting start position in the axial direction

Select one of the following two types:



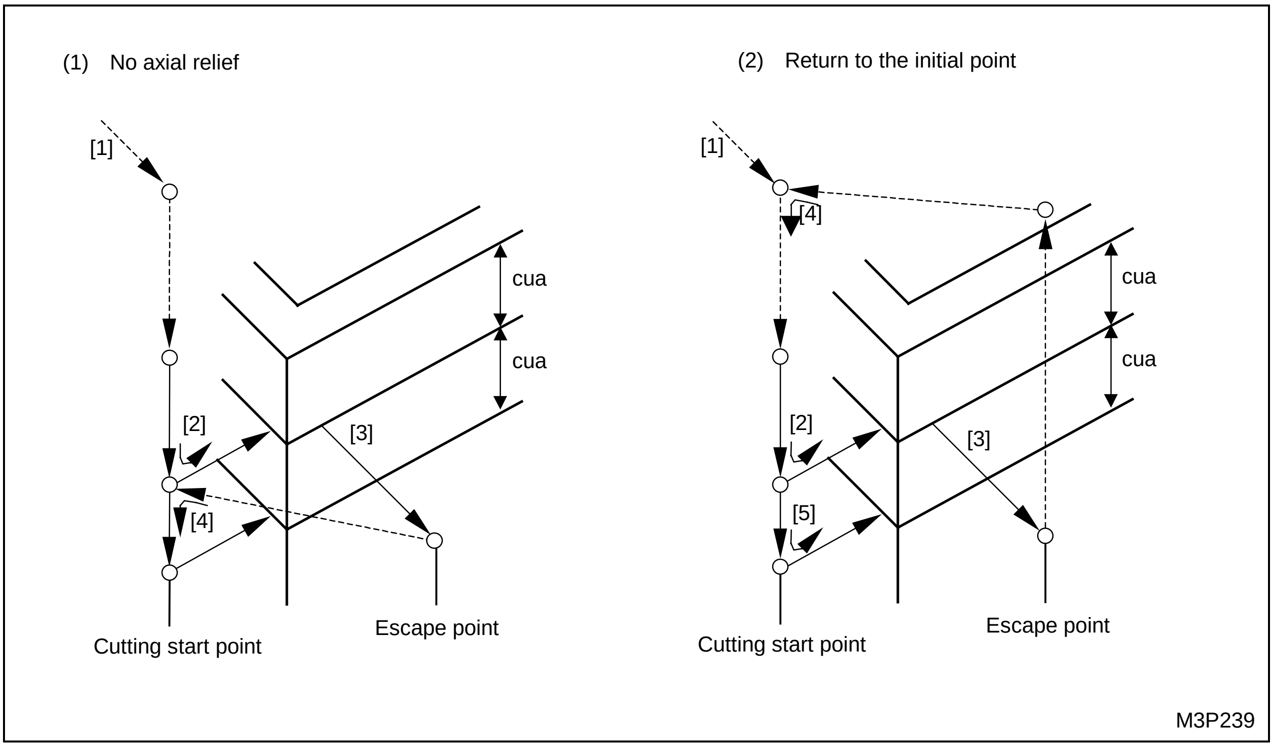
B. Type of routing via approach point

Select one of the following two types:



C. Type of escape in the axial direction after machining

Select one of the following two types:



Tool path setting parameter

Parameter **E95**

For A: bit 4 = 0: Cutting start position fixed -- (2)
1: Rapid feed to the clearance position 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 clearance) from the second cutting when the following conditions are satisfied:

- Bit 6 of parameter **E95** is set to “1”, and
- The unit concerned is LINE 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)

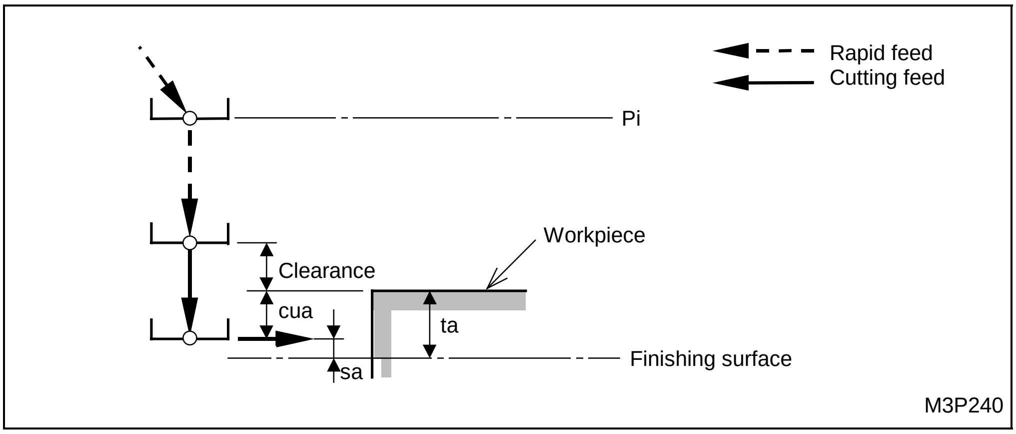
For C: bit 3 = 0: Return to the initial point -- (2)
1: No escape in the axial direction -- (1)

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.

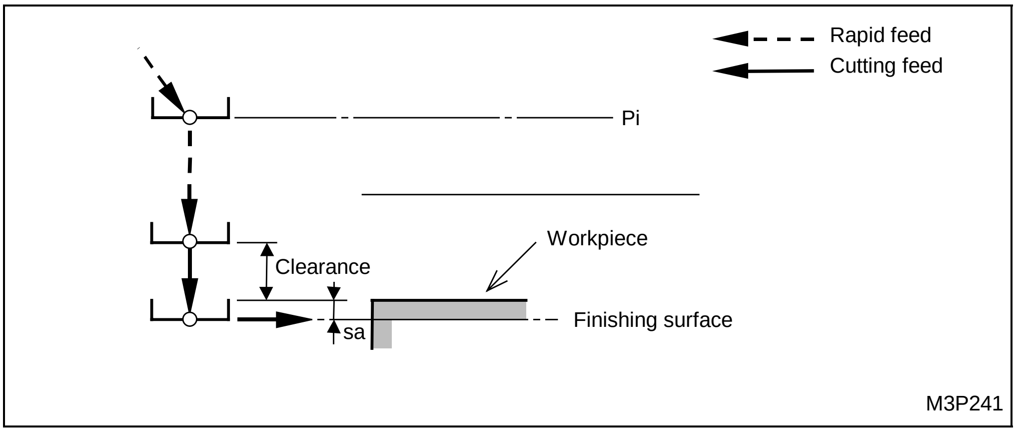
Note 2: The tool path shown at basic tool path above is selected automatically for machining units that are not subject to the selection of the parameter **E95.**

2. Detail tool path of an axial cut-in

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pi: Initial point

cua: Axial cutting depth **DET-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 starting allowance of axial cutting, specified by the (safety) 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 starting allowance of cutting in radial direction, specified by parameter **E2**, 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.

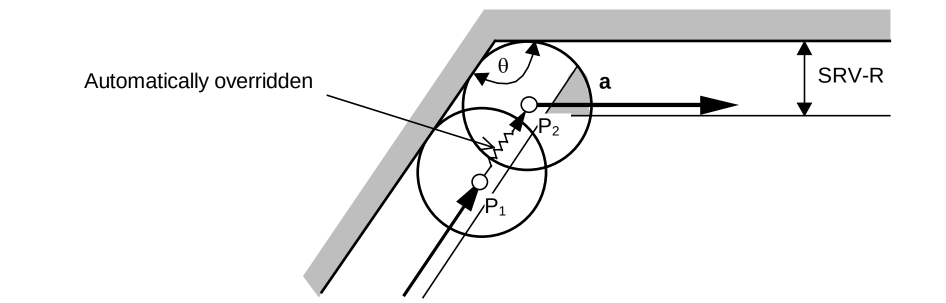
3. Other precaution on tool path

If shape data, tool data or parameter are modified after the automatic determination of coordinates of approach point (displayed in yellow), the approach point will not be located on the same cutting start point and the tool path will also be modified.

3-6-7 Automatic corner override

In line and face machining, cutting an inside corner will require a larger allowance to be cut, resulting in an increased load of cutting. The automatic corner override is to automatically override a feedrate at the allowance increased portions to reduce the cutting load.

1. Operating conditions

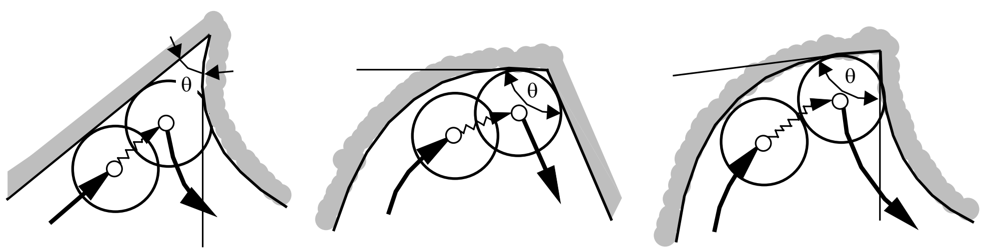


M3P242

Cutting an inside corner will increase a cutting allowance by area **a** while moving the tool from P_1 to P_2 in the illustration. In this span, the feedrate is automatically overriden.

This override, however, will be valid only when all of the following requirements (A, B, C) are satisfied:

A. Inside corner angle θ is equal to or less than the value entered in the parameter **E25** (with θ ≤ **E25**).

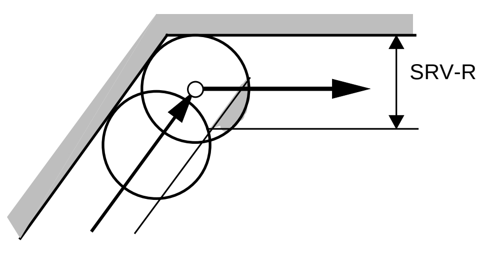


M3P243

B. A radial cutting allowance is equal to or less than the value entered in the parameter **E23**

(SRV-R ≤ tool diameter × **E23**/100)

The load scarcely varies when **SRV-R** is near to the tool diameter.

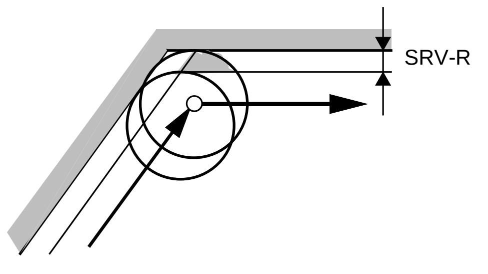


M3P244

C. A radial cutting allowance is equal to or less than the value entered in the parameter **E24**

(SRV-R ≤ tool diameter × **E24**/100)

When **SRV-R** is small, the load varies scarcely.



M3P245

2. **Valid machining**

The corner override is valid in roughing for each of the right-hand linear machining, left-hand linear machining, outside linear machining, inside linear machining, end milling-step, pocket milling, pocket milling-mountain and pocket milling-valley.

3. **Override rate**

An override rate on the programmed cutting feedrate should be entered in the parameter **E22**. With the parameter set at 0, the automatic corner override function is invalid.

3-7 Face Machining Units

Face machining units are used to enter the data relating to the procedures for machining an area and to the form to be machined. Available in each 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.

3-7-1 Types of face machining units

As shown below 7 types of face machining units are available:

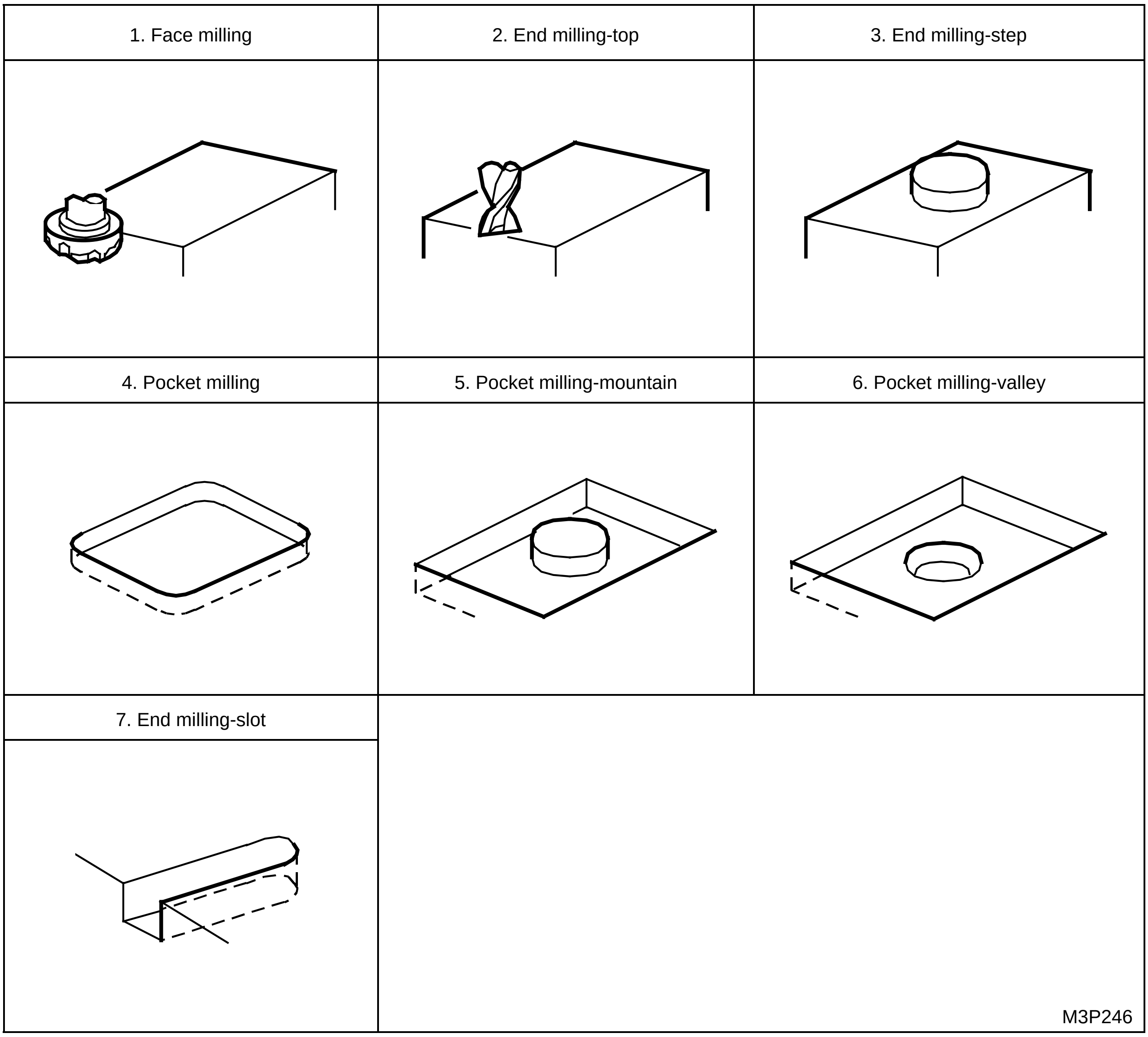


Fig. 3-23 Types of face machining unit

3-7-2 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[FACE MACH-ING]** menu key.

➔ The following menu is displayed.

FCE MILL	TOP EMIL	STEP	POCKET	PCKT MT	PCKT VLY	SLOT			

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

3-7-3 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.

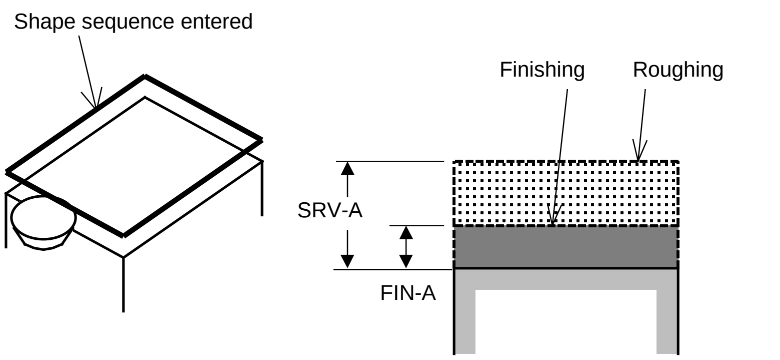
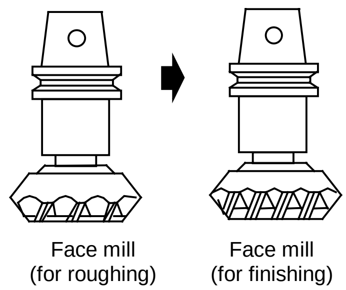
A. Data setting

```
UNo.   UNIT     MODE   POS-B   POS-C    SRV-A        BTM          FIN-A
 1    FCE MILL
SNo.   TOOL   NOM-ϕ  No.   #  APRCH-1  APRCH-2 TYPE  AFD     DEP-A   WID-R   C-SP    FR   M  M  M
 R1   FCE MILL                                        ◆
 F2   FCE MILL                                        ◆        ◆
```

◆: Data are not necessary to be set here.

Remark 1: In this unit, face mills are automatically developed.

Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

FCE MILL unit	Tool sequence
 D740PA133	 D740PA050

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. Automatic tool development

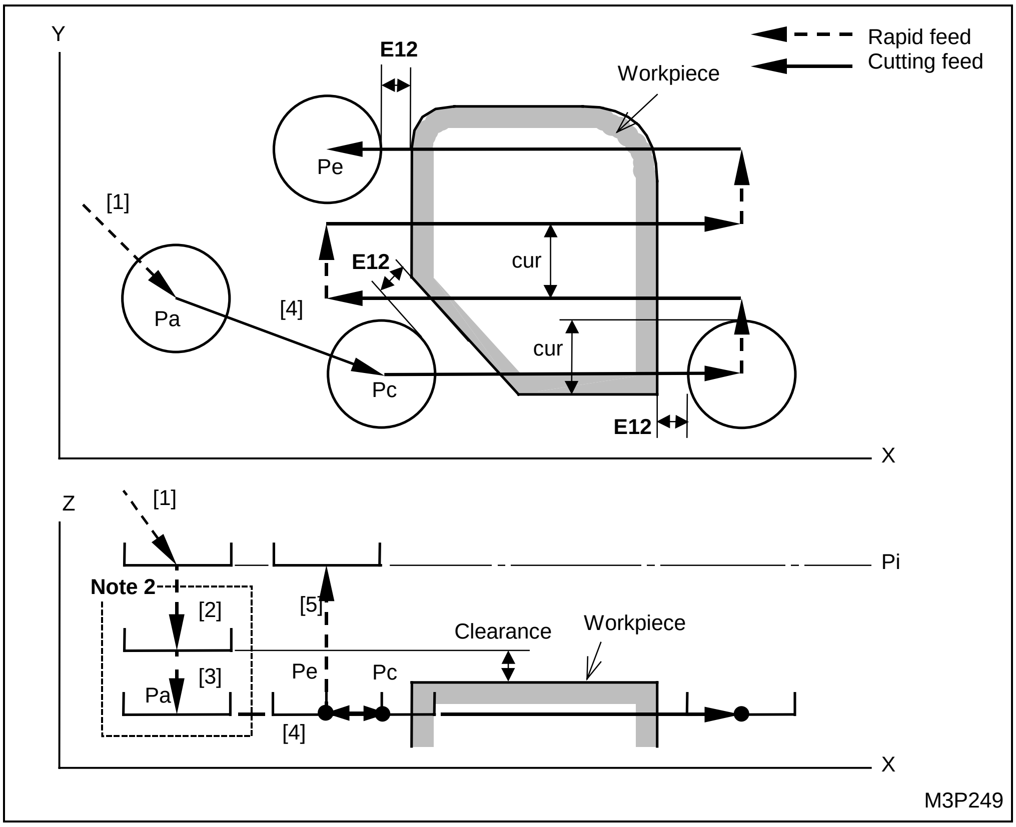
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A** and on **FIN-A**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.

C. Tool path

When **[X BI-DIR]** is selected for the item **TYPE** in the tool sequence



The bold codes represent parameter addresses.

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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

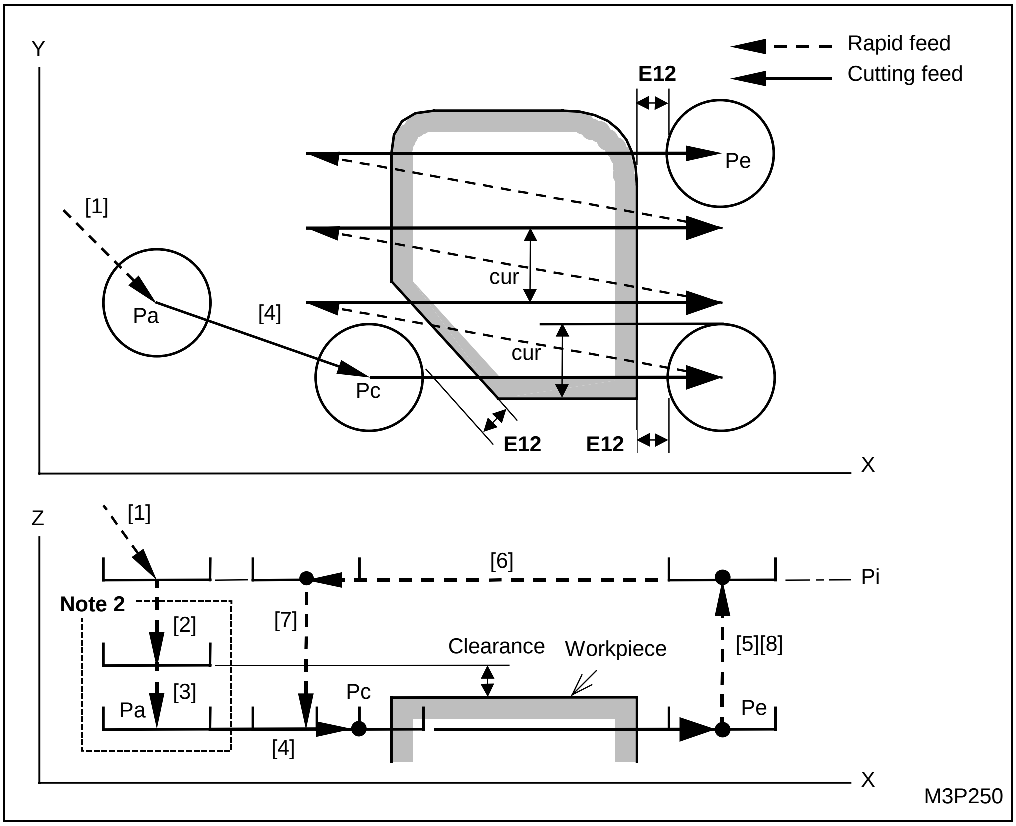
[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves at a rapid feedrate to the face to be machined.

[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

When **[X UNI-DIR]** is selected for the item **TYPE** in the tool sequence



The bold codes represent parameter addresses.

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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

[2] The tool moves at a rapid feedrate to the clearance position.

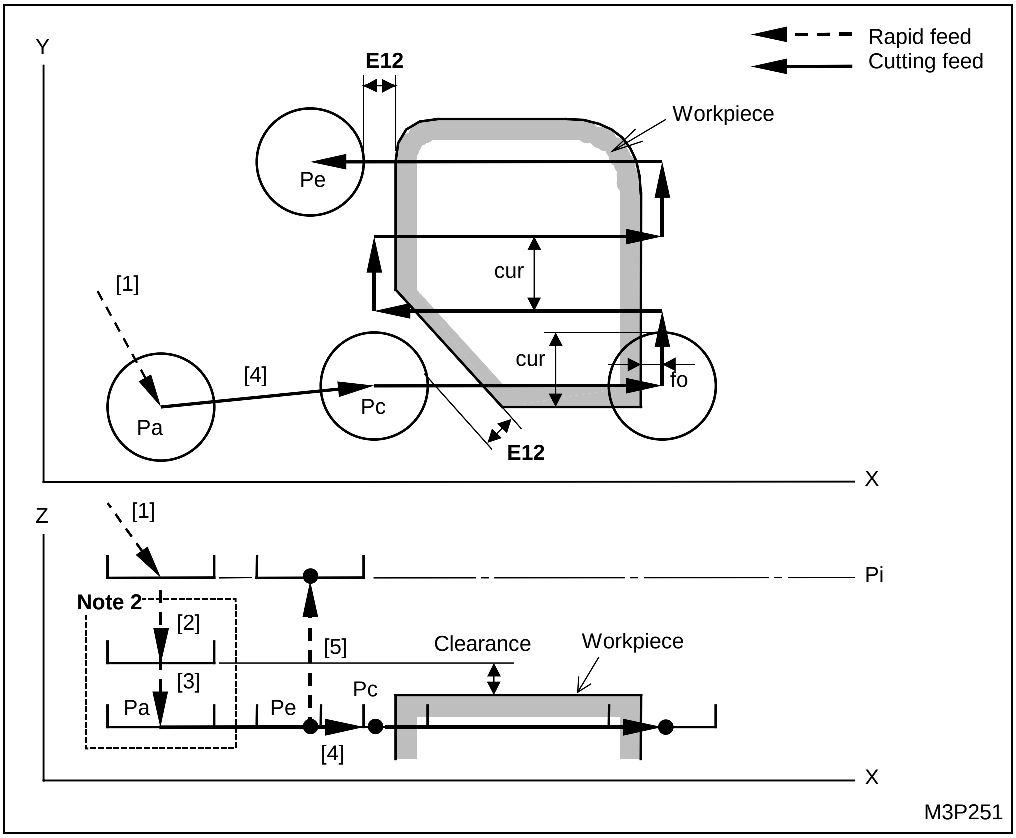
[3] The tool moves at a rapid feedrate to the face to be machined.

[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5], [6] and [7] Upon completion of machining in one direction, the tool moves at a rapid feedrate to initial point and to a subsequent cutting start point.

[8] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

When **[X BI-DIR SHORT]** is selected for the item **TYPE** in the tool sequence



The bold codes represent parameter addresses.

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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

fo: Form-offsetting clearance

$$fo = \text{tool diameter} \times \frac{\mathbf{E15}}{10}$$

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

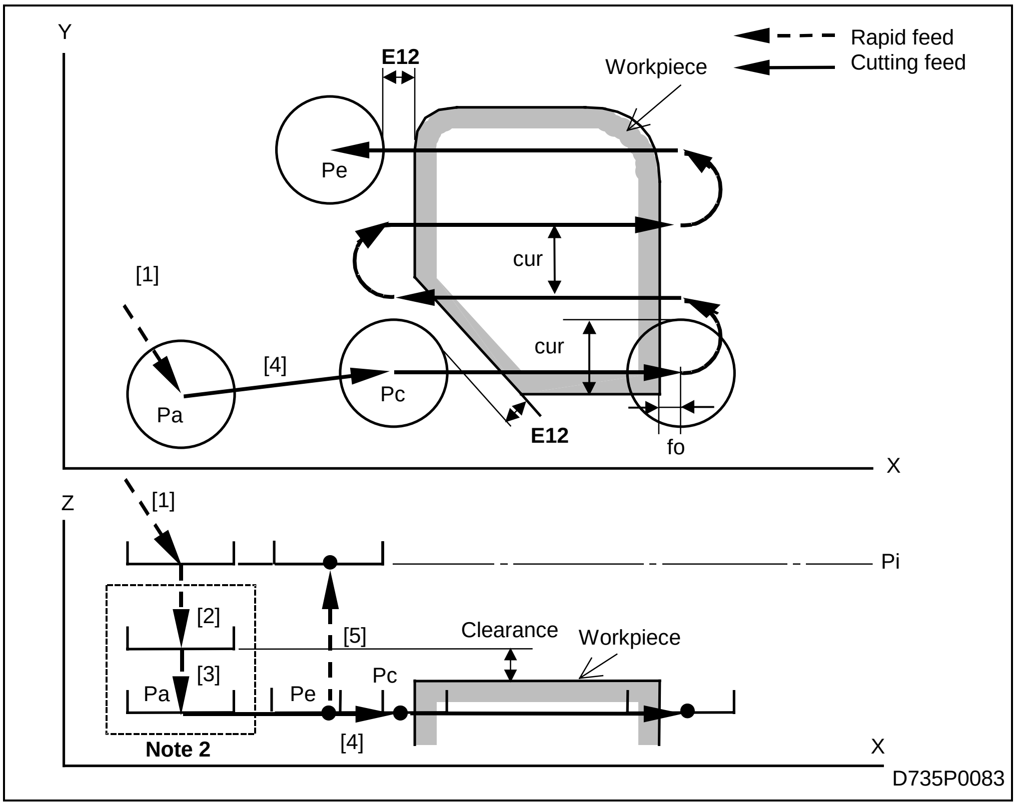
[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves at a rapid feedrate to the face to be machined.

[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

When **[X BI-DIR ARCSHORT]** is selected for the item **TYPE** in the tool sequence



The bold codes represent parameter addresses.

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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

fo: Form-offsetting clearance

$$fo = \text{tool diameter} \times \frac{\mathbf{E15}}{10}$$

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves at a rapid feedrate to the face to be machined.

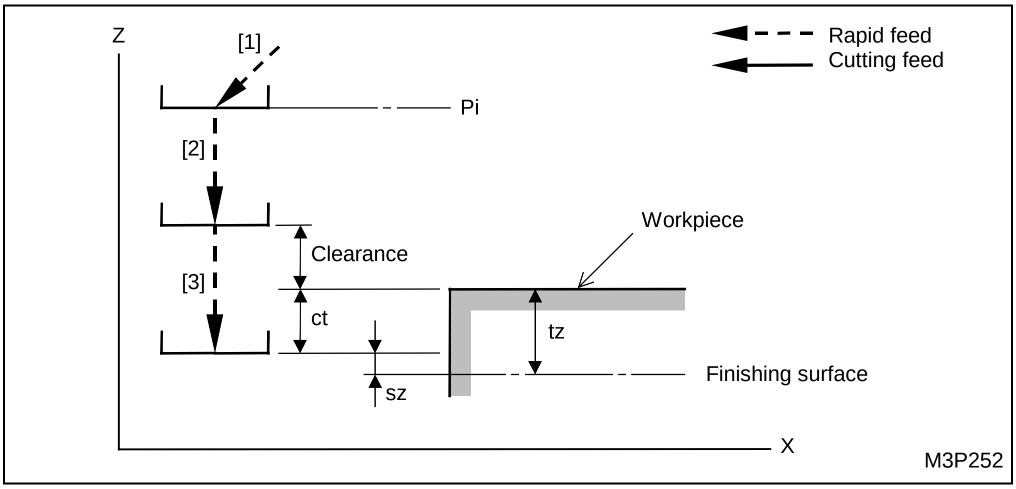
[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

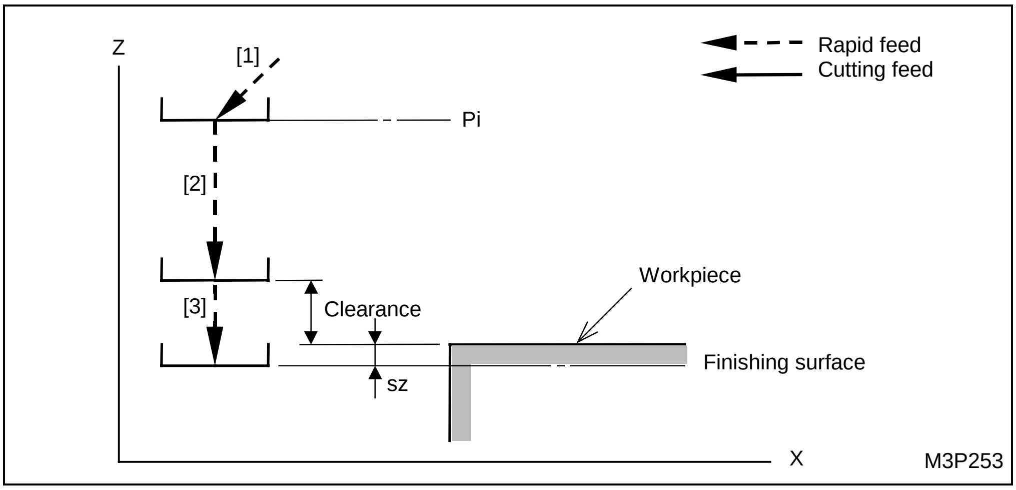
Note 1: When **?** is displayed in the items **APRCH-1**, **-2** by pressing the **[AUTO SET]** menu key, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate of cutting start point will be entered in these items.

Note 2: Detail of the Z-axial tool path.

- Roughing



- Finishing



Pi: Initial point

ct: Z-axial cutting stroke to be determined by the data **DEP-A** in the tool sequence

tz: Z-axial cutting allowance to be determined by the data **SRV-A** in a machining unit

sz: Z-axial finishing allowance **FIN-A** in a machining unit

Note 3: See Subsection 3-7-5, “Precautions in face machining.”

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

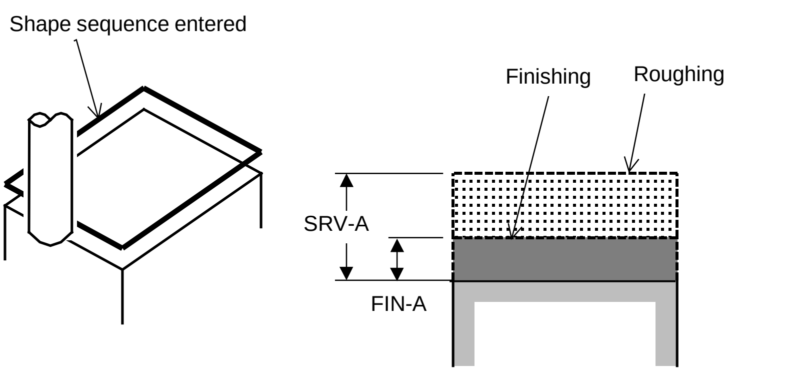
```
UNo.     UNIT     MODE   POS-B   POS-C    SRV-A          BTM           FIN-A
 1    TOP EMIL
SNo.    TOOL   NOM-ϕ  No.    #  APRCH-1  APRCH-2 TYPE  AFD      DEP-A   WID-R   C-SP    FR    M   M   M
 R1   END MILL
 F2   END MILL                                                 ◆
```

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills are automatically developed.

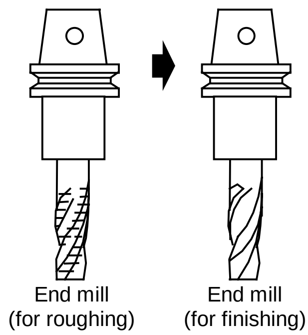
Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

TOP EMIL unit



D740PA135

Tool sequence



D740PA051

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. Automatic tool development

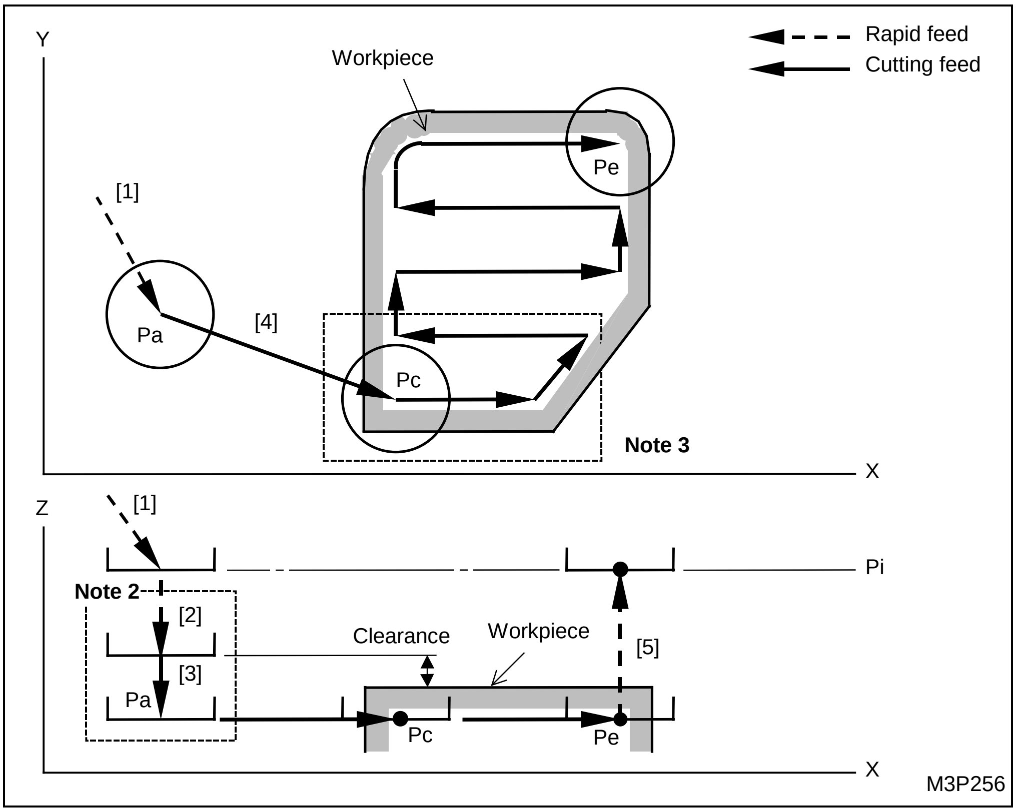
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A** and on **FIN-A**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.

C. Tool path

<u>When **[X BI-DIR]** is selected for the item **TYPE** in the tool sequence</u>



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

Pi: Initial point

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

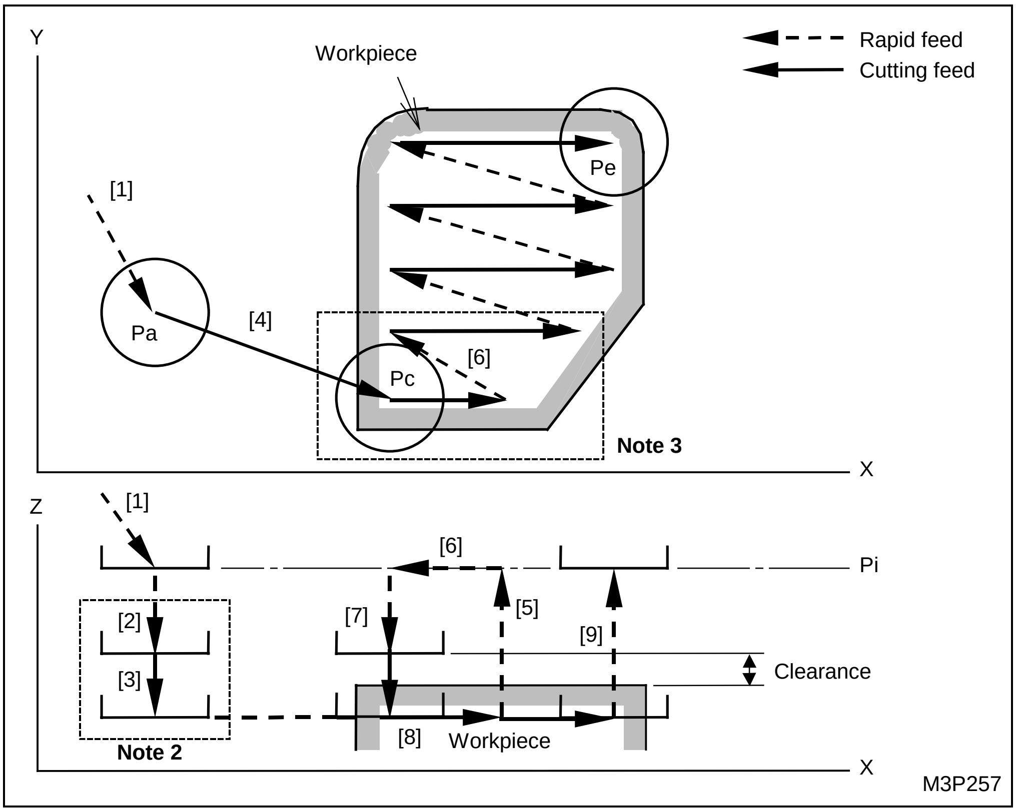
[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

When **[X UNI-DIR]** is selected for the item **TYPE** in the tool sequence



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

Pi: Initial point

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point.

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon **AFD** in the tool sequence.)

[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

[5], [6] and [7] Upon completion of machining in one direction, the tool moves at a rapid feedrate to initial point.
Then, it moves at a rapid feedrate to the subsequent cutting start point specified by the clearance above the next cutting start point.

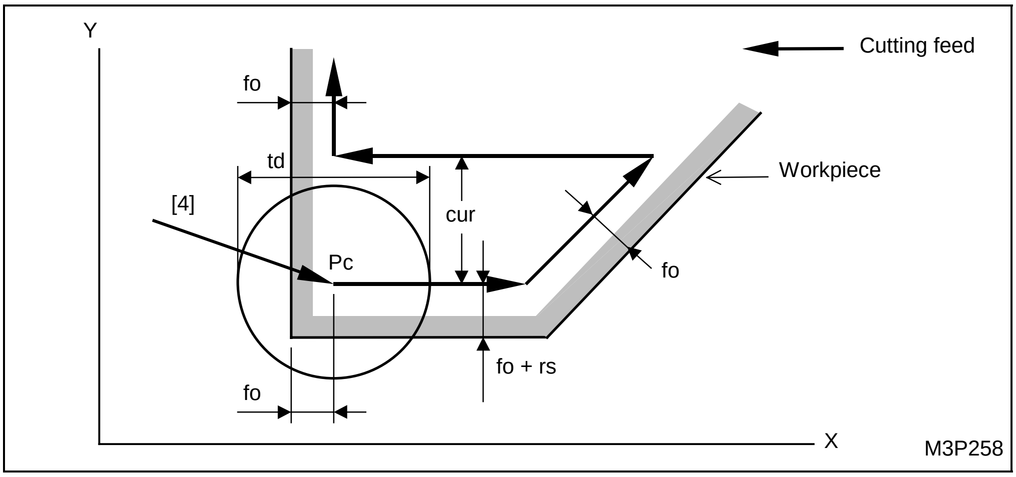
[8] The tool moves at a cutting feedrate to the face to be machined and starts machining.

[9] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

Note 1: When **?** is displayed in the items **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 coordinates of cutting start point will be entered in these items.

Note 2: See Subsection 3-7-5 “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:

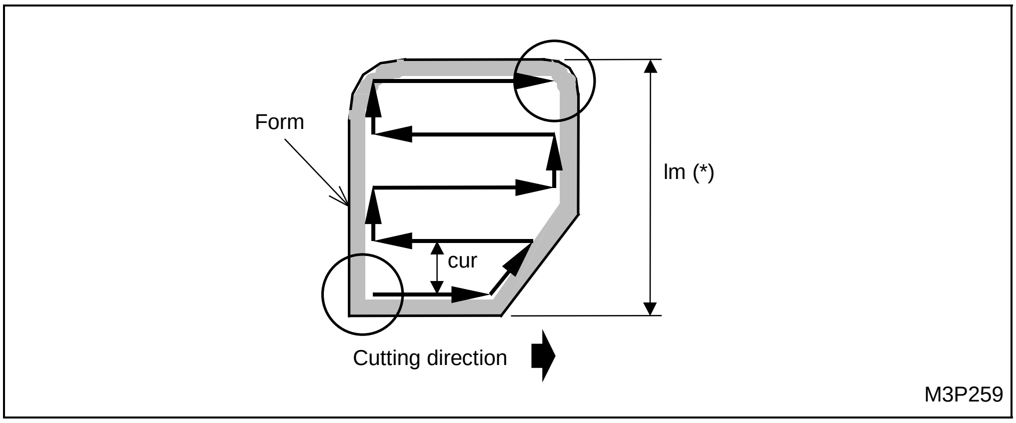
$$cur = \frac{\ell v}{n}$$

$$\ell v = \ell m\ (*) - 2 \times (fo + rs)$$

$$n = \frac{\ell v}{cr}$$

cr: Radial cutting depth (**WID-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. End milling-step unit (STEP)

This unit is selected to machine a workpiece flatly on the surface by the use of an end mill, with a relief left behind.

A. Data setting

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R
1	STEP								

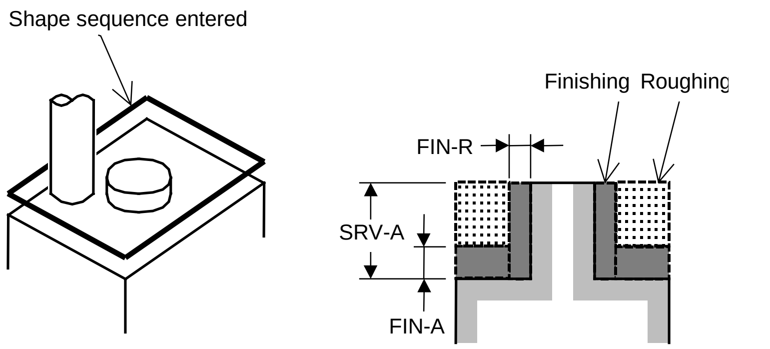
SNo.	TOOL	NOM-φ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	TYPE	PK-DEP	DEP-A	WID-R	C-SP	FR	M	M	M
R1	END MILL									◆							
F2	END MILL									◆	◆						

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills are automatically developed.

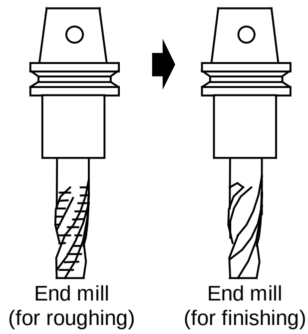
Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

STEP unit



D740PA137

Tool sequence



D740PA051

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.

B. Automatic tool development

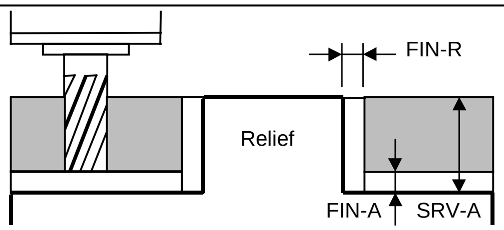
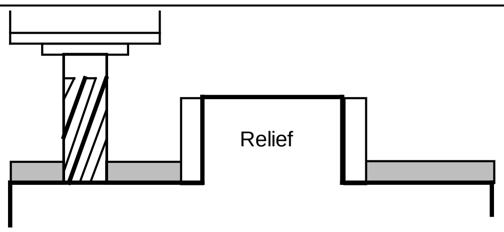
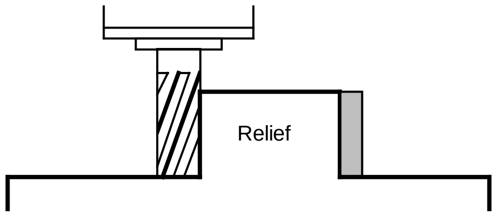
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A**, **FIN-A** and **FIN-R**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.

C. Machining sequence

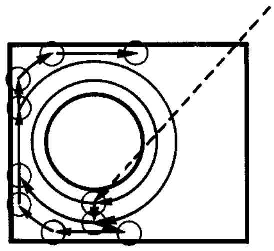
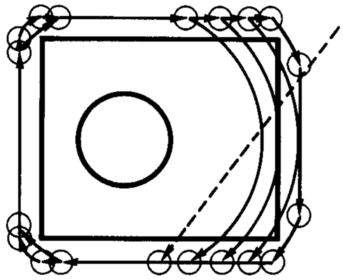
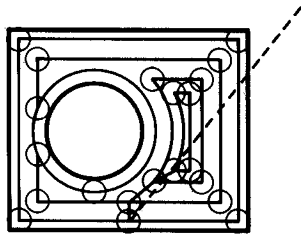
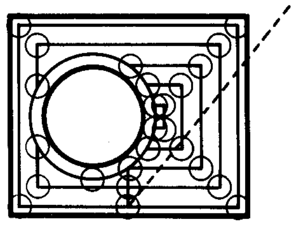
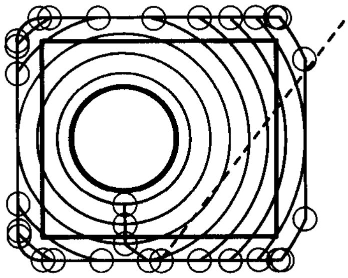
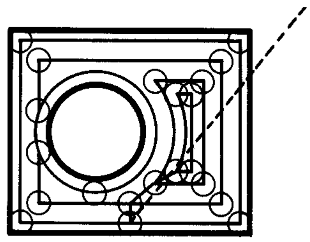
End milling-step is performed in the following order.

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	 M3P261	Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.

D. Machining pattern

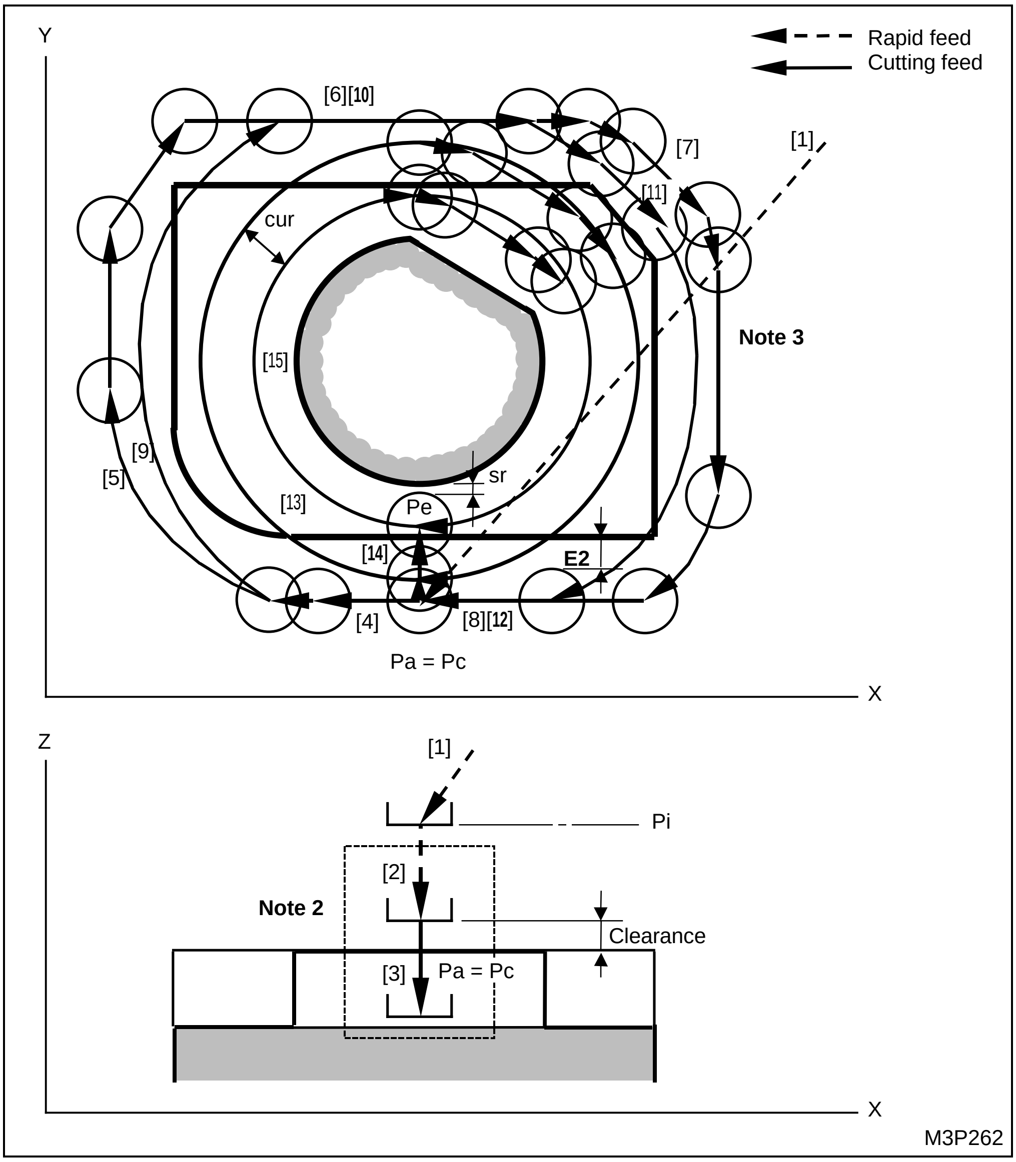
For roughing or bottom finishing, a machining pattern is selected by the parameter **E91**.

- Either 0 or 1 in the related bit accordingly. For the details of the parameter **E91**, refer to the separate Parameter List/Alarm List/M-Code List.

E91 = 7 6 5 4 3 2 1 0	0	1
bit 0	 Machining is performed from inside to outside.	 Machining is performed from outside to inside.
bit 1	 Cutting direction reversing type: With the cutting direction reserved inside and outside are machined. After that, the rest is machined.	 Fixed cutting direction type: Machining is performed in an identical direction while turning along an inside form.
bit 7 NM210-00546	 In case of machining from the outside, the tool moves inwards along an inside form.	 To machine from the outside, the tool moves inwards along an outside form.

E. Tool path

Roughing or bottom finishing



The bold codes represent parameter addresses.

Pa, Pc: Approach point to be determined by the data **APRCH-1**, **-2** to be entered in the tool sequence and cutting start point (In the illustration above, the cutting start point is the approach point.)

Pe: Escape point automatically established

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

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

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

Note 3: When a tool moves over a tool path distant by the value entered in the parameter **E2** from a machining form, the cutting feedrate is multiplied by the number entered in the parameter **E16**.

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point (Cutting start point). (See Note 1.)

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] - [15] The tool machines along an inside form while turning around a workpiece on the circumference. ([6] and [10] and [4], [8] and [12] have some portions pass through an identical path.)

Wall finishing

The tool is machining through a tool path identical with that for finishing in the LINE OUT unit.

F. Finishing

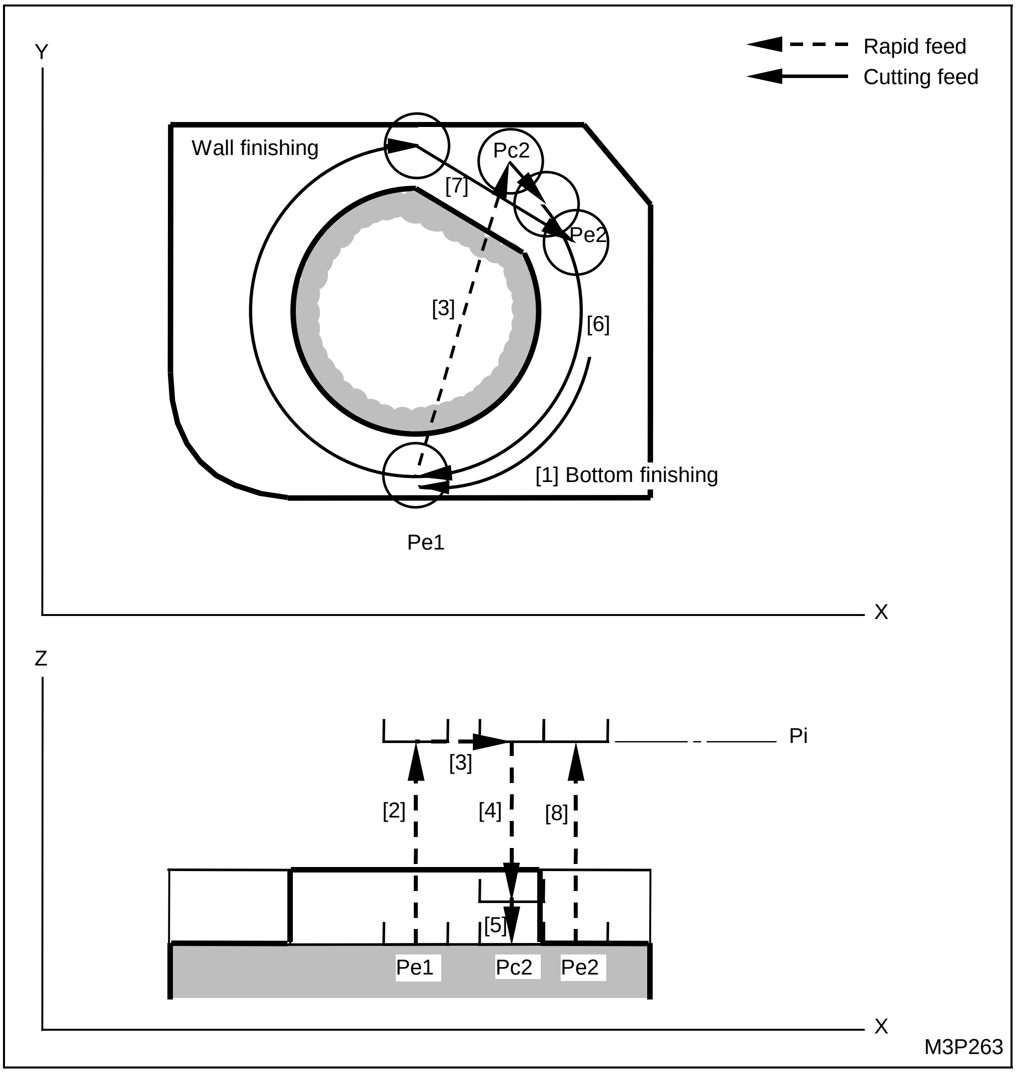
Finishing is performed, based on the entered data **FIN-A** and **FIN-R**.

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** in 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

Pe2: Escape point in wall finishing

Pi: Initial point

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

Note 2: When a tool moves over a tool path distant by the value entered in the parameter **E2** from a machining form, the cutting feedrate is multiplied by the number entered in the parameter **E16**.

4. 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

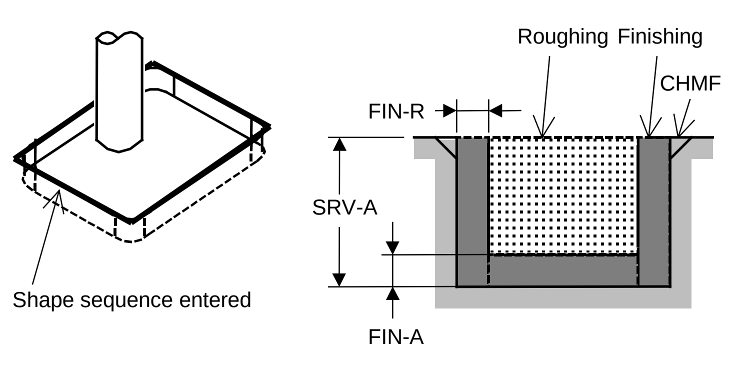
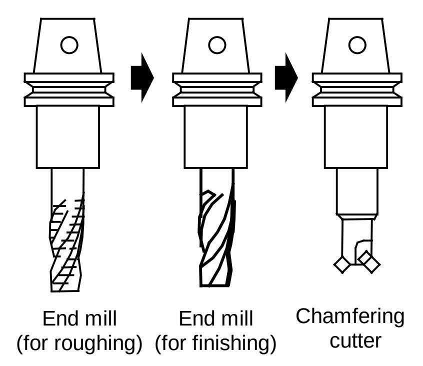
```
UNo.    UNIT     MODE   POS-B   POS-C    SRV-A          BTM    WAL    FIN-A    FIN-R   INTER-R  CHMF
  1    POCKET
SNo.    TOOL   NOM-ϕ No.  #  APRCH-1 APRCH-2 TYPE  AFD  TYPE PK-DEP DEP-A WID-R  C-SP  FR  M   M   M
 R1    END MILL                                              ◆
 F2    END MILL                                              ◆      ◆
```

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills and a chamfering cutter are automatically developed. Instead of the chamfering cutter, a centering drill can be used for chamfering.

Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

Note: If a centering drill is used, a nose angle of 90 degrees is set for machining.

POCKET unit	Tool sequence
 D740PA138	 D740PA052

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. Automatic tool development

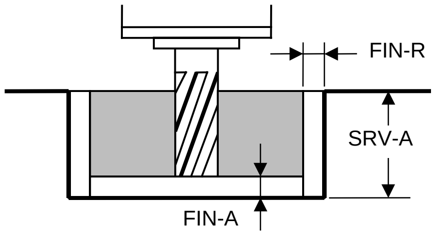
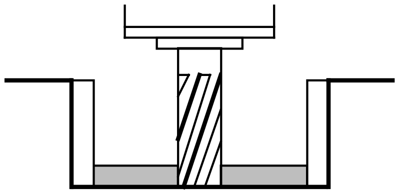
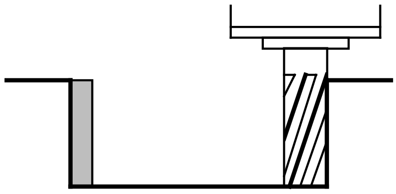
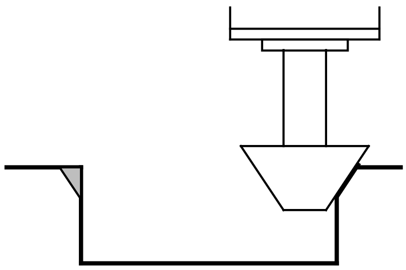
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to three tools are automatically developed, based on **SRV-A**, **FIN-A**, **FIN-R** and **CHMF**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.
(Chamfering)	CHMF ≠ 0 : One tool is selected.

C. Machining sequence

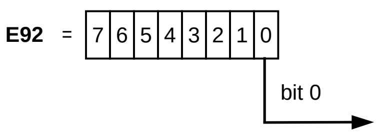
Pocket milling is performed in the following order.

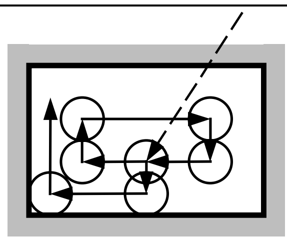
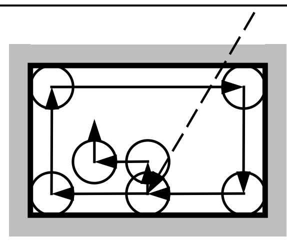
<table>
<tr><td colspan="2">Roughing</td><td>
</td><td>Machining is performed with the end mill developed in the tool sequence R1. With SRV-A = FIN-A, this machining is not performed.</td></tr>
<tr><td rowspan="2">Finishing</td><td>Bottom</td><td></td><td>Machining is performed with the end mill developed in the tool sequence F1. With FIN-A = 0, this machining is not performed.</td></tr>
<tr><td>Wall</td><td>
M3P265</td><td>Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.</td></tr>
<tr><td colspan="2">Chamfering</td><td></td><td>Machining is performed with the chamfering cutter or centering drill in the tool sequence. With CHMF = 0, this machining is not performed.</td></tr>
</table>

D. Machining pattern

For roughing or bottom finishing, a machining pattern is selected by the parameter **E92**.

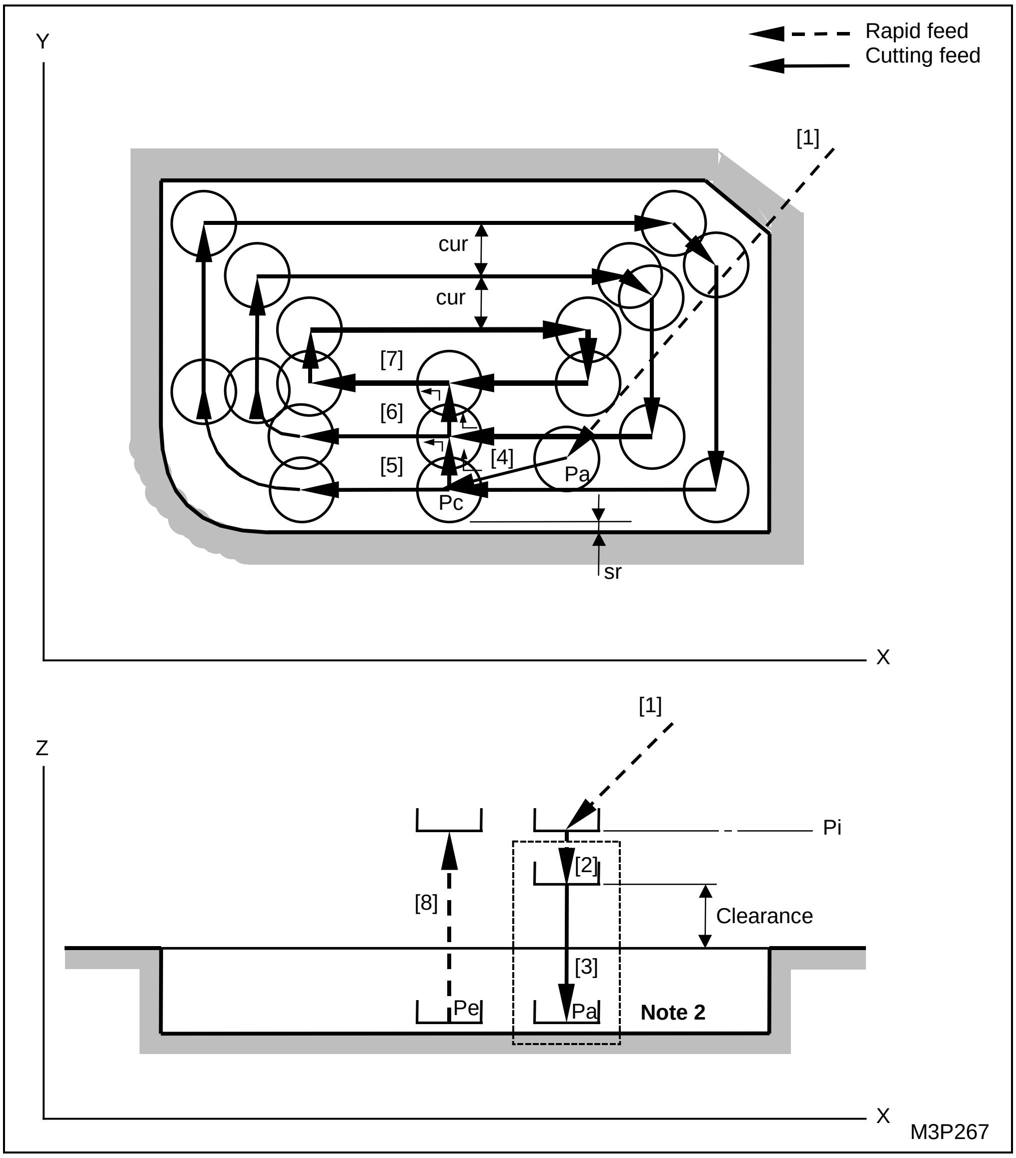
- Either 0 or 1 is entered in the related bit accordingly

<table>
<tr><td rowspan="2">

M3P266</td><td>0</td><td>1</td></tr>
<tr><td>
Machining is performed from inside to outside.</td><td>
Machining is performed from outside to inside.</td></tr>
</table>

E. Tool path

Machining from outside (roughing or bottom finishing)



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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

sr: Radial finishing allowance to be determined by the data in the machining unit

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point. (See Note 1.)

[2] The tool moves at a rapid feedrate to the clearance position.

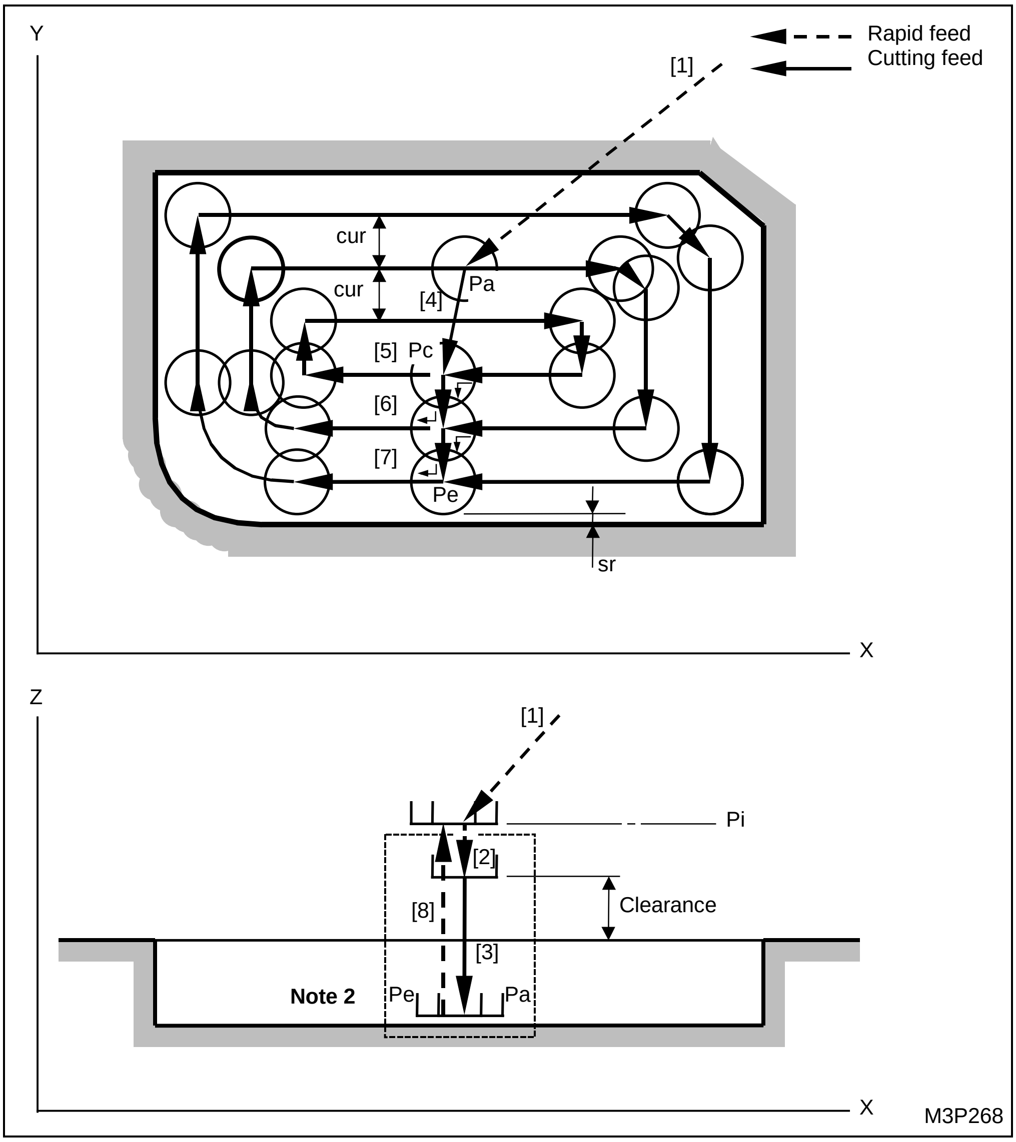
[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] The tool moves at a cutting feedrate to the cutting starting point.

[5], [6] and [7] The tool machines on an around by around basis inwards.

[8] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

Machining from inside (roughing or bottom finishing)



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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

sr: Radial finishing allowance to be determined by the data in the machining unit

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point. (See Note 1.)

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] The tool moves at a cutting feedrate to the cutting starting point.

[5], [6] and [7] The tool machines on an around by around basis outwards.

[8] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

<u>Wall finishing</u>

The tool is machining through a tool path identical with that for finishing in the LINE IN unit.

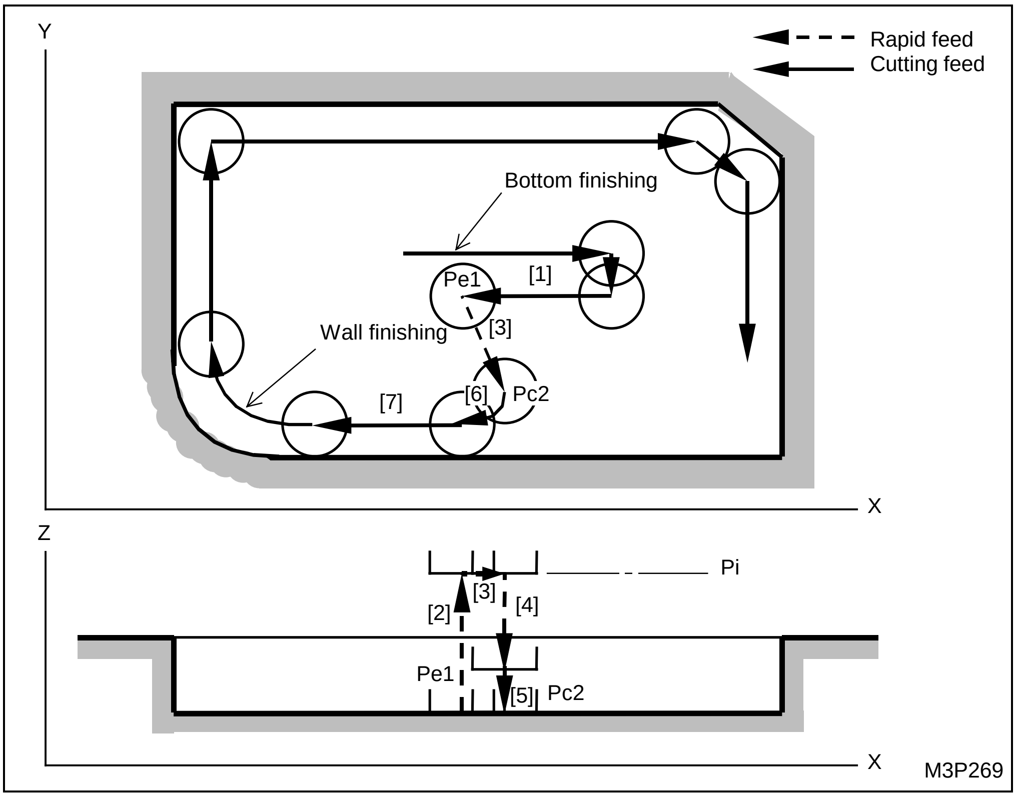
F. Finishing

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

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

5. Pocket milling-mountain unit (PCKT MT)

This unit is selected to carry out milling of a pocket form with relief left behind 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	PCKT MT								

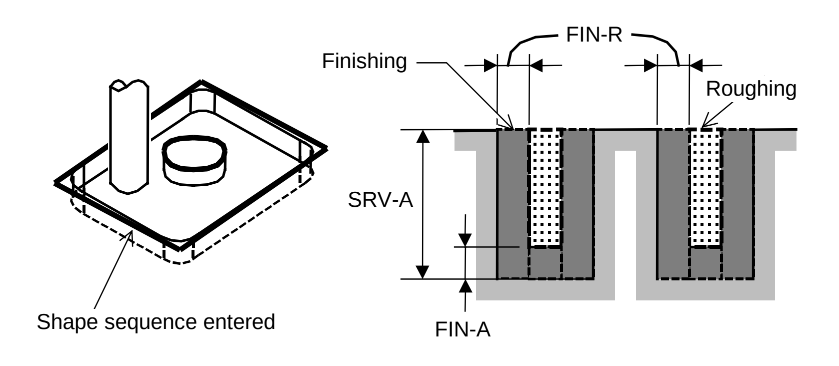
SNo.	TOOL	NOM-φ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	TYPE	PK-DEP	DEP-A	WID-R	C-SP	FR	M	M	M
R1	END MILL									◆							
F2	END MILL									◆	◆						

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills are automatically developed.

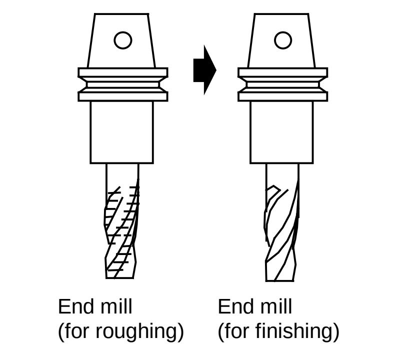
Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

PCKT MT unit



D740PA140

Tool sequence



D740PA051

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.

B. Automatic tool development

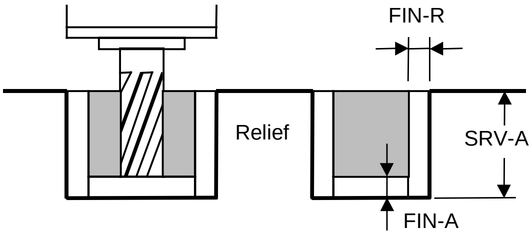
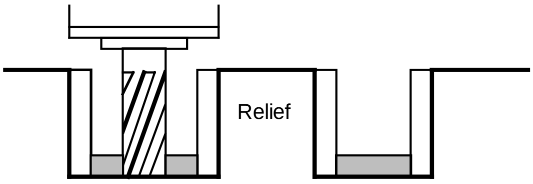
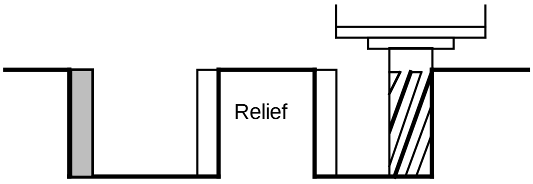
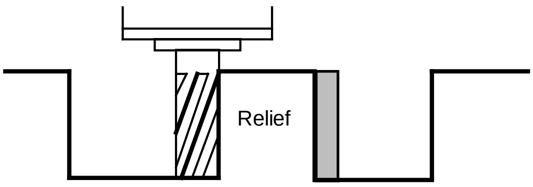
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A**, **FIN-A** and **FIN-R**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.

C. Machining sequence

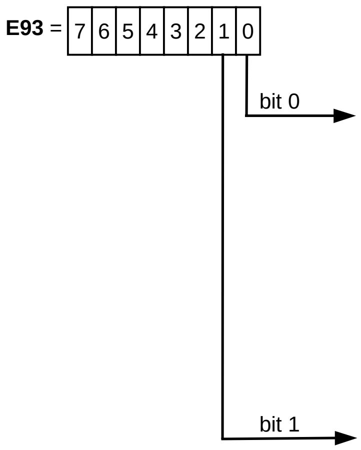
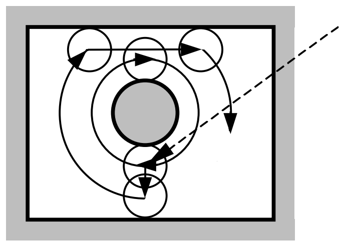
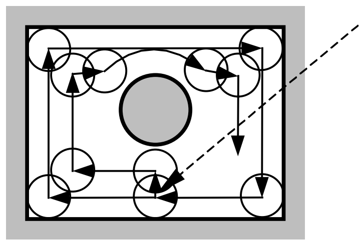
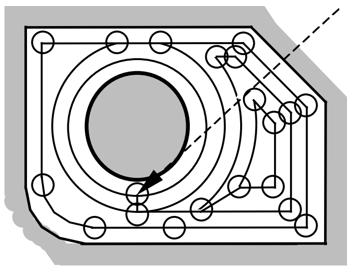
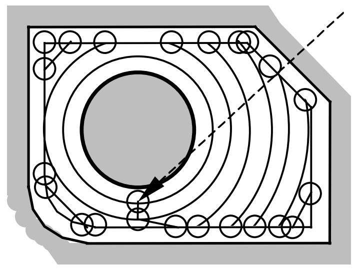
Pocket milling is performed in the following order.

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.
	Outer wall		Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.
	Inner wall	 M3P271	

D. Machining pattern

For roughing or bottom finishing, a machining pattern is selected by the parameter **E93**.

- Enter 0 or 1 in the related bit accordingly.

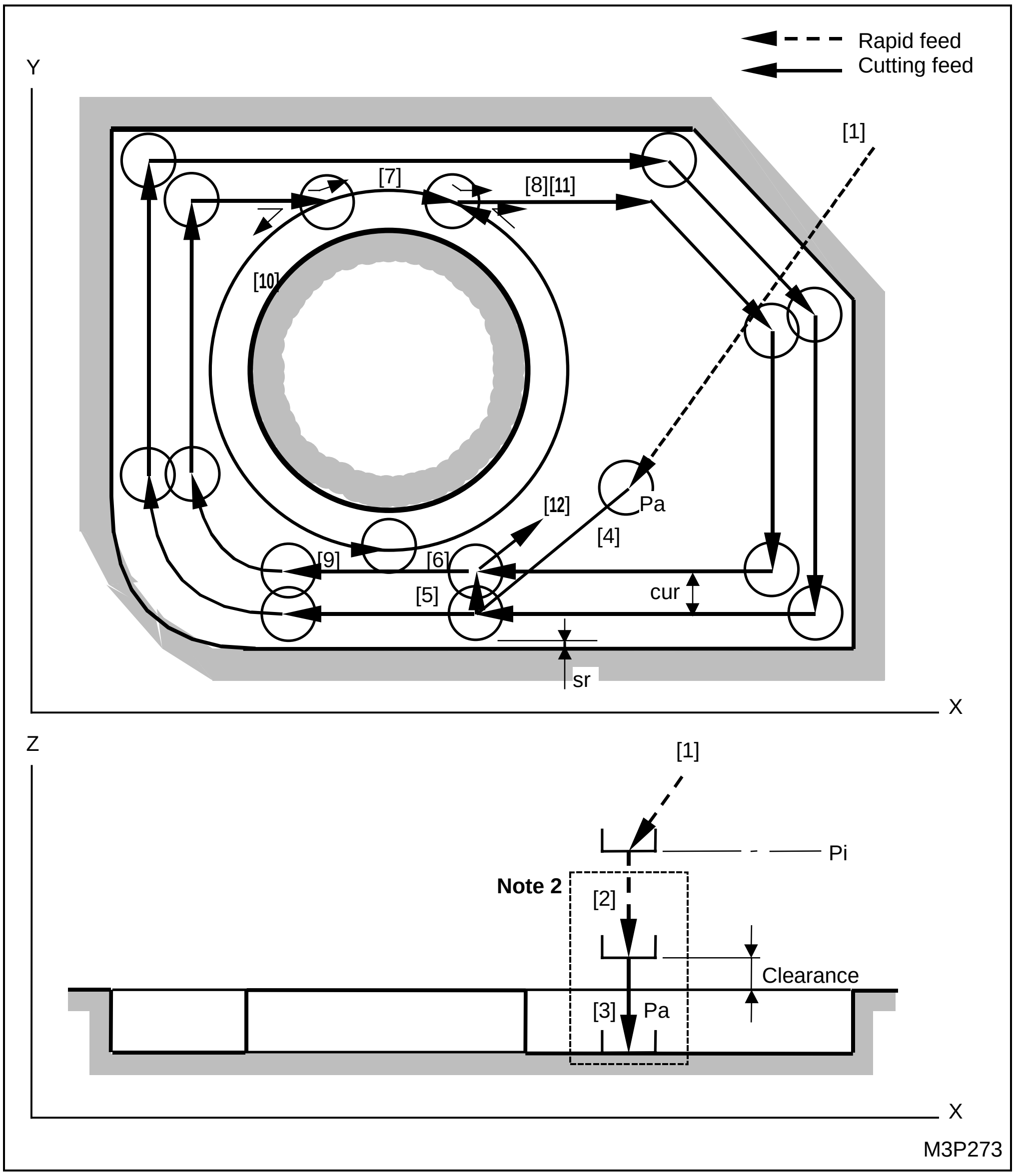
	0	1
bit 0	 Machining is performed from inside to outside.	 Machining is performed from outside to inside.
bit 1	 Cutting direction reversing type: With the cutting direction reversed, inside and outside are machined. After that, the rest is machined.	 Fixed cutting direction type: Machining is performed in an identical direction along an inside form.

M3P272

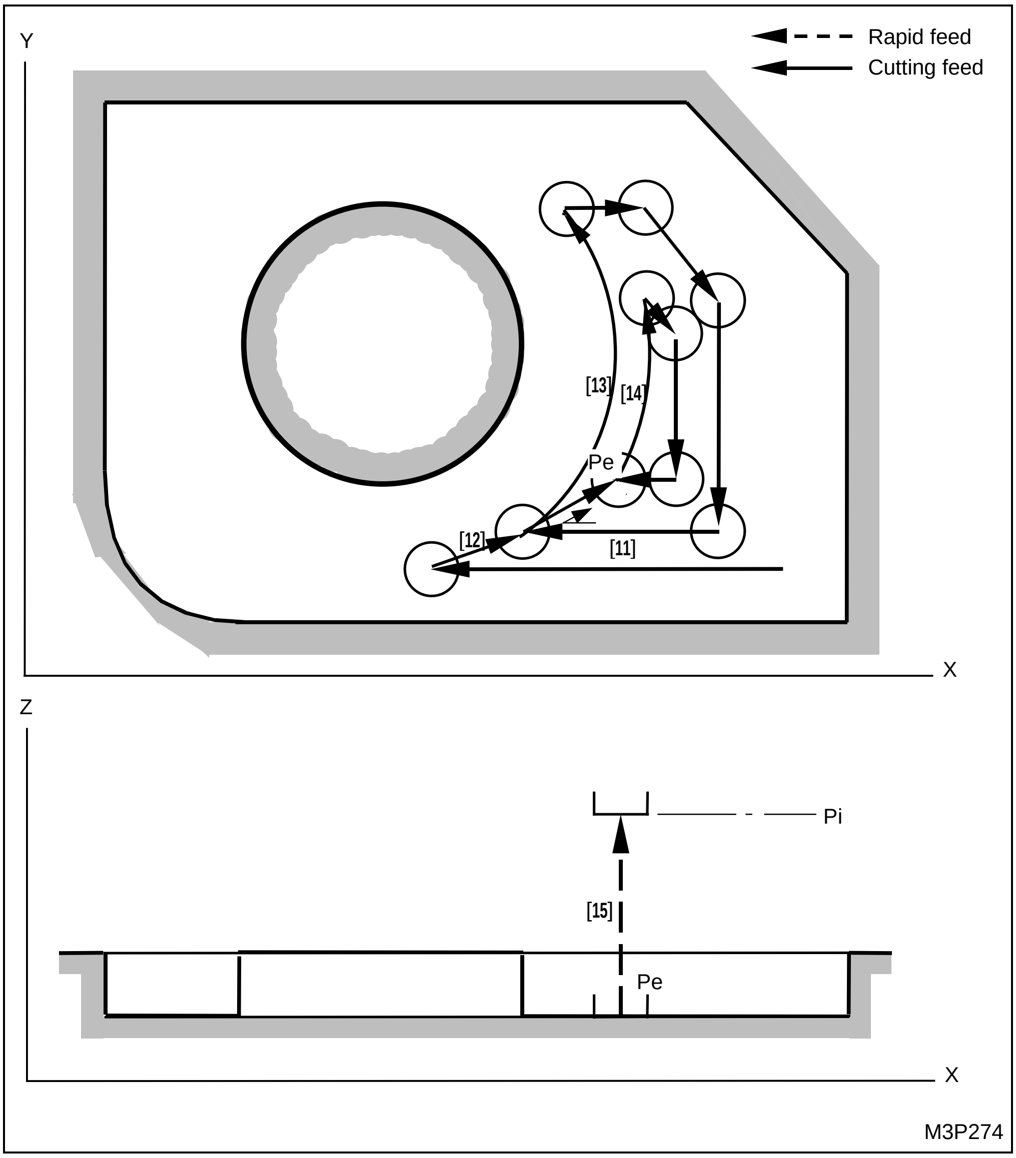
E. Tool path

Machining from outside (roughing or bottom finishing)

- Machining along outer and inner walls:



- Machining the rest:



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

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

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

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point. (See Note 1.)

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] The tool moves at a cutting feedrate to the cutting starting point.

[5] and [6] The tool machines along the outside form.

[7] Once it has interfered with the inside form, the tool machines along the inside form.

[8] Once it has got out of the inside form, the tool moves along the outside form.

[9] To machine along the inside form, the tool moves along the same path as that in [6].

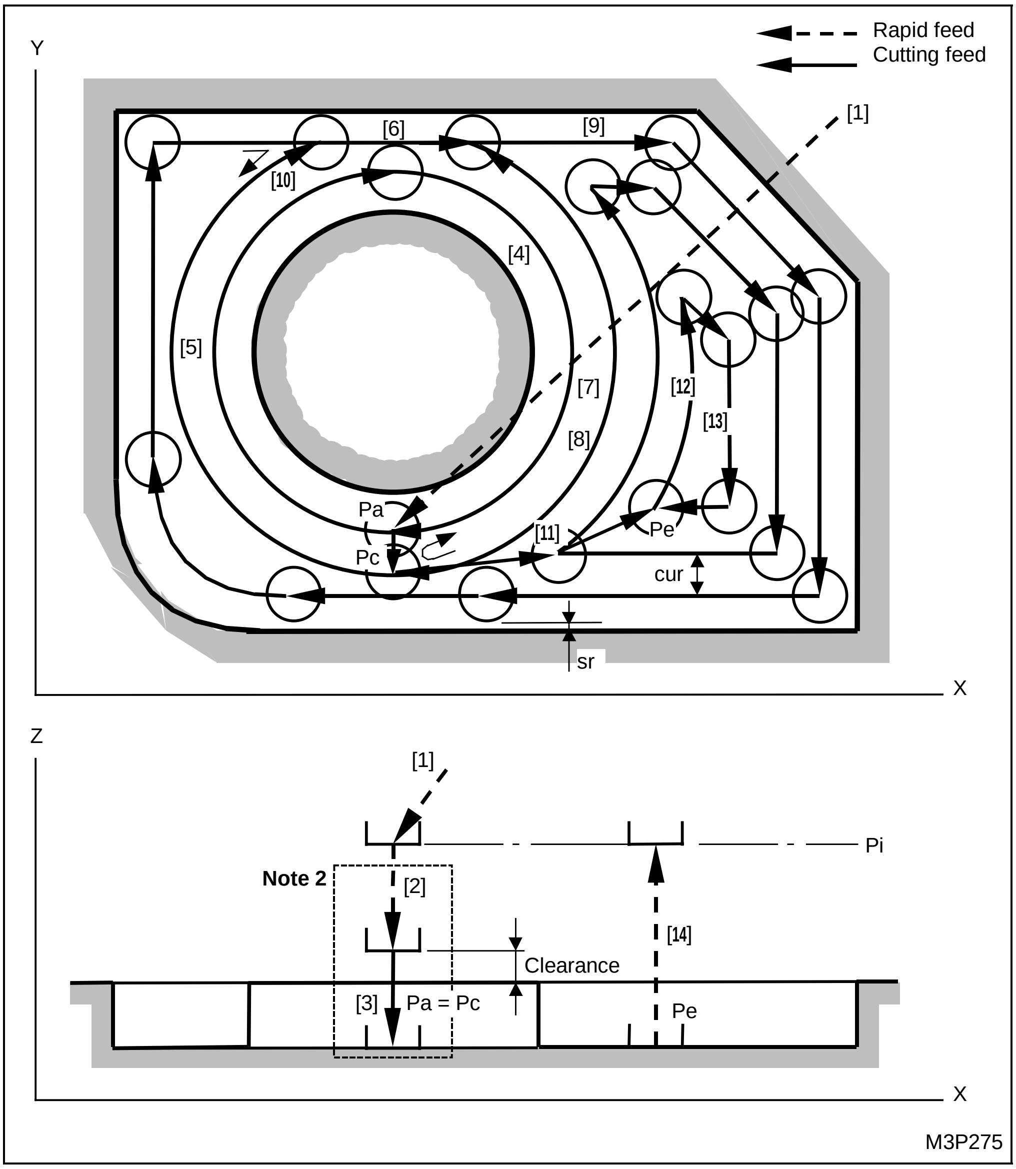
[10] Because of the path reversed, once it has interfered with the inside form, the tool machines along the inside form.

[11] To machine the rest, the tool moves along the same path as that in [8].

[12], [13] and [14] The rest is machined inwards on an around by around basis.

[15] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

Machining from inside (roughing or bottom finishing)



Pa, Pc: Approach point to be determined by the data **APRCH-1**, **-2** to be entered in the tool sequence and cutting start point. (In the illustration above, the cutting start point is the approach point.)

Pe: Escape point to be automatically established

<Route on which tool is to move>

In [7] and [8], and [5] and [10], the tool moves reversely on an identical path.

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

Outer wall finishing

The tool is machining through a tool path identical with that for finishing in the LINE IN unit.

Inner wall finishing

The tool is machining through a tool path identical with that for finishing in the LINE OUT unit.

F. Finishing

Finishing is performed, based on the entered data **FIN-A** and **FIN-R**.

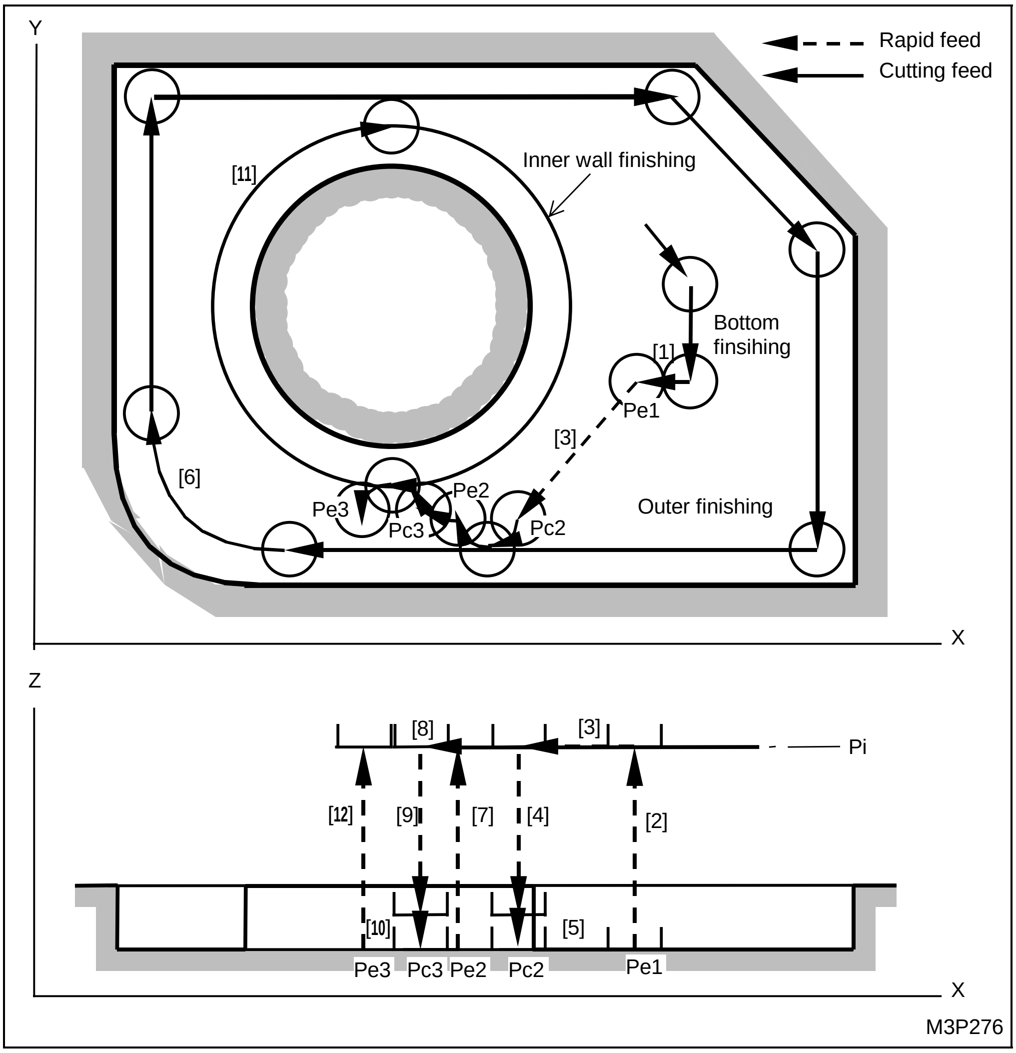
- Bottom finishing is performed, with 0 < **FIN-A**.
- Outer and inner walls are finished, with 0 < **FIN-R**.

<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 or from outer wall finishing to inner wall finishing, moreover, the tool moves at a rapid feedrate from the escape point to the cutting start point as illustrated below.

<To perform wall-finishing>

When wall-finishing is performed in finishing, outer wall and inner wall are finished in this order (outer wall finishing → inner wall finishing), irrespective of setting for parameter **E93** bit 0.



Pe1: Escape point in bottom finishing

Pc2: Cutting start point in outer wall finishing

Pe2: Escape point in outer wall finishing

Pc3: Cutting start point in inner wall finishing

Pe3: Escape point in inner wall finishing

Pi: Initial point

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

6. Pocket milling-valley unit (PCKT VLY)

This unit is selected to carry out pocket milling-valley by the use of an end mill.

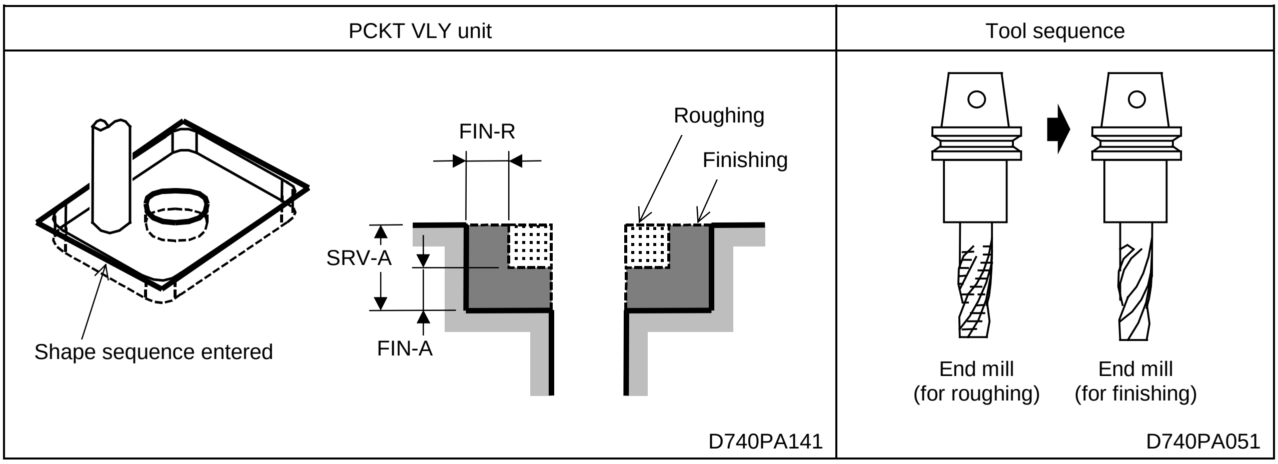
A. Data setting

```
UNo.    UNIT     MODE   POS-B  POS-C    SRV-A         BTM    WAL    FIN-A    FIN-R
 1    PCKT VLY
SNo.    TOOL    NOM-ϕ No.  #   APRCH-1  APRCH-2 TYPE  AFD  TYPE PK-DEP  DEP-A  WID-R C-SP  FR   M   M   M
 R1   END MILL                                                  ◆
 F2   END MILL                                                  ◆      ◆
```

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills are automatically developed.

Remark 2: For the tool sequence data setting, see Subsection 3-7-4.



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.

B. Automatic tool development

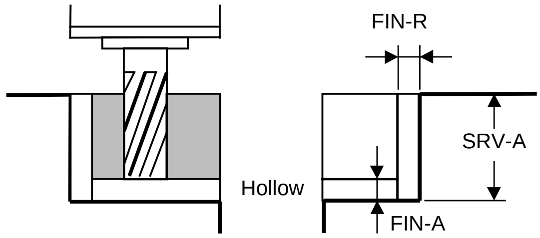
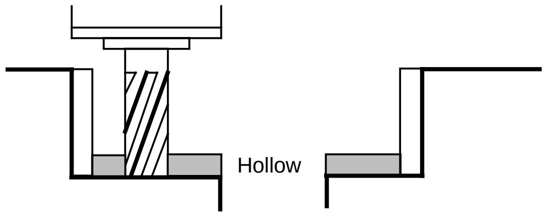
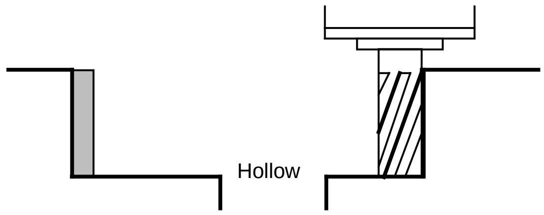
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

In the tool sequence, a maximum of up to two tools are automatically developed, based on **SRV-A**, **FIN-A** and **FIN-R**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A : One tool is selected.

C. Machining sequence

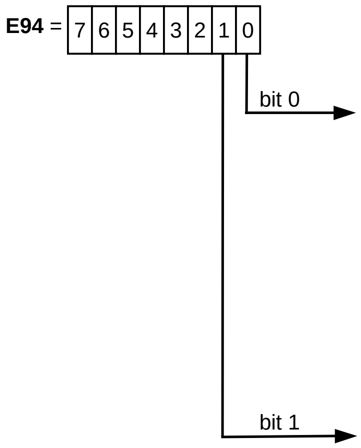
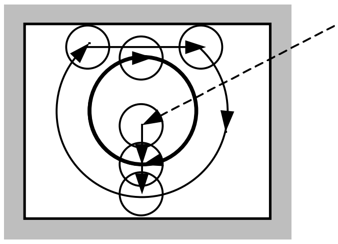
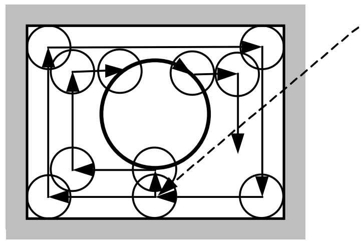
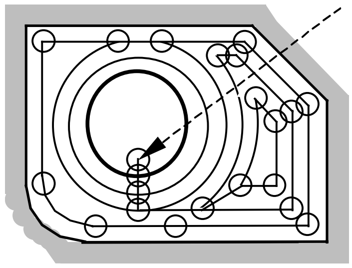
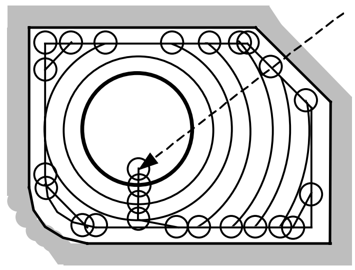
Pocket milling-valley is performed in the following order.

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	 M3P278	Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.

D. Machining pattern

For roughing or bottom finishing, a machining pattern is selected by the parameter **E94**.

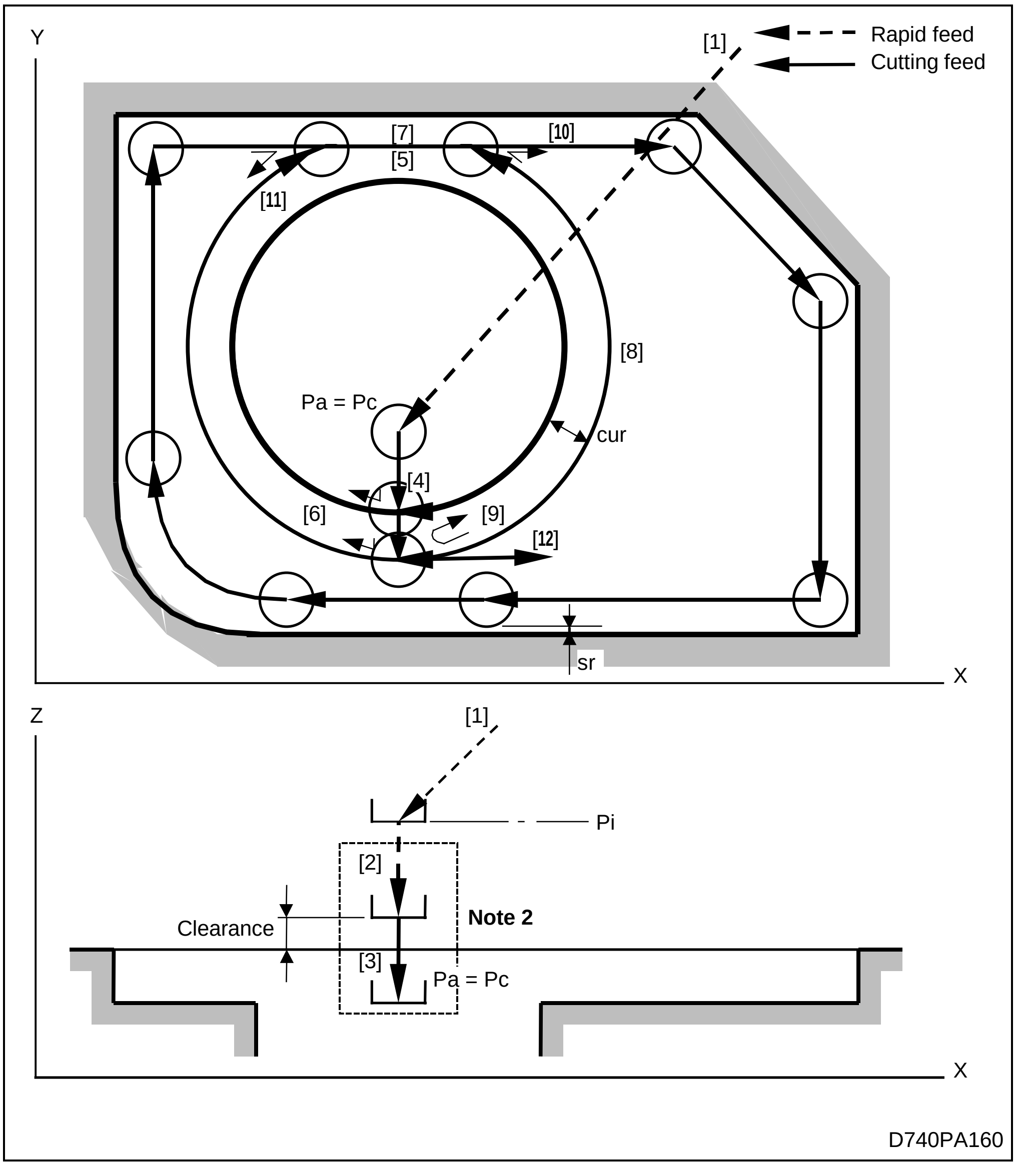
- Enter 0 or 1 in the related bit accordingly.

	0	1
	 Machining is performed from inside to outside.	 Machining is performed from outside to inside.
D740PA159	 Cutting direction reversing type: With the cutting direction reversed, inside and outside are machined. After that, the rest is machined.	 Fixed cutting direction type: Machining is performed in an identical direction along an inside form.

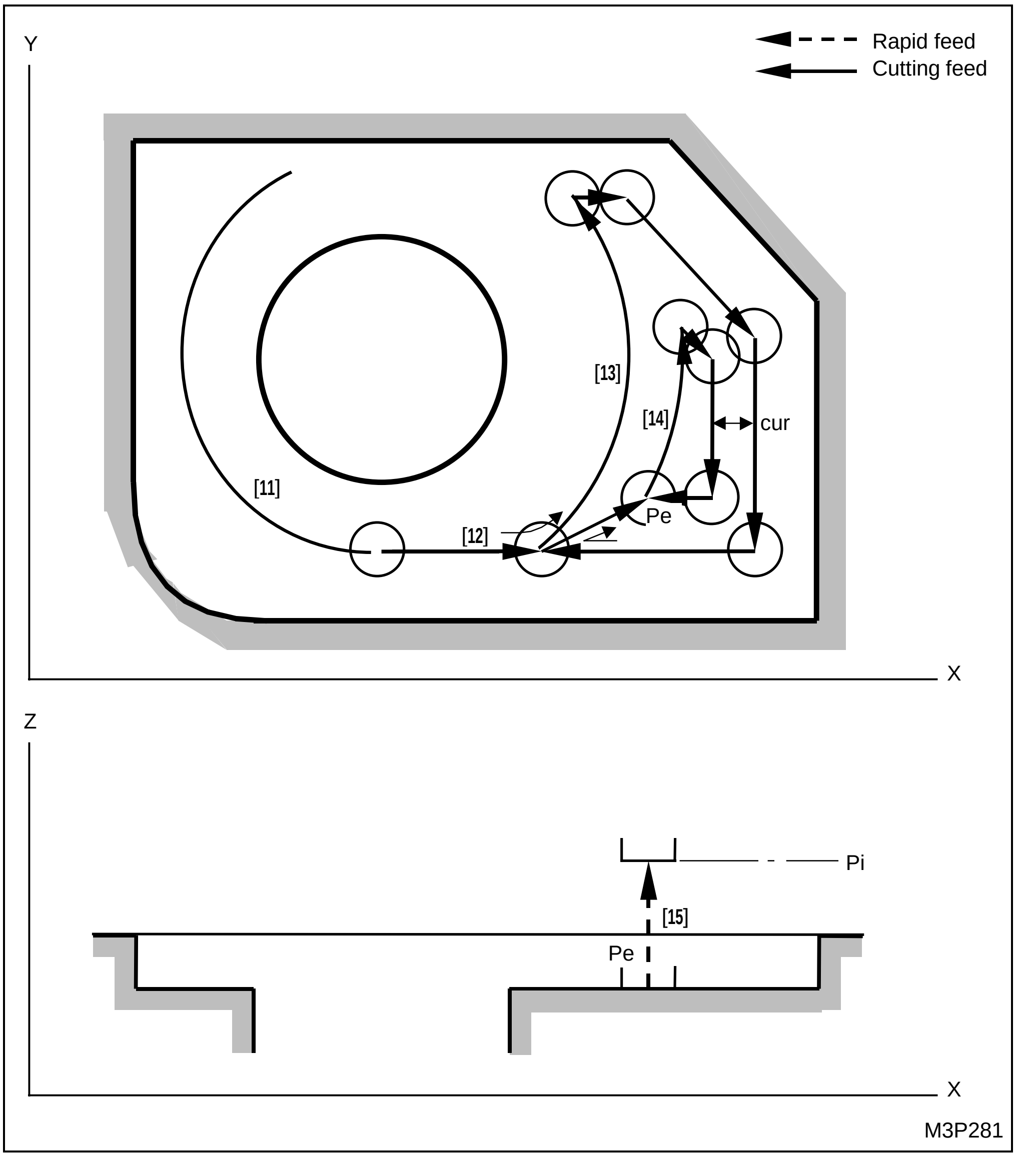
E. Tool path

<u>Machining from inside (roughing or bottom finishing)</u>

- Machining along an outer wall after expanding a valley along an inside form:



- Machining the rest:



Pa, Pc: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence and cutting start point (In the illustration above, the cutting start point is the approach point.)

Pe: Escape point to be automatically established

Pi: Initial point

cur: Radial cutting depth to be determined by the data **WID-R** in the tool sequence

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

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

<Route on which tool is to move>

[1] The tool moves at a rapid feedrate to approach point (cutting start point). (See Note 1.)

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data **AFD** in the tool sequence.)

[4] The tool moves to the cutting position in a workpiece.

[5] and [6] The tool expands a hole along the valley form.

[7] Once it has interfered with the wall of an outside form, the tool machines along the outside form.

[8] Once it has got out of the outside form, the tool expands the hole along the valley form.

[9] To machine along the outside form, the tool moves along the same path as that in [8].

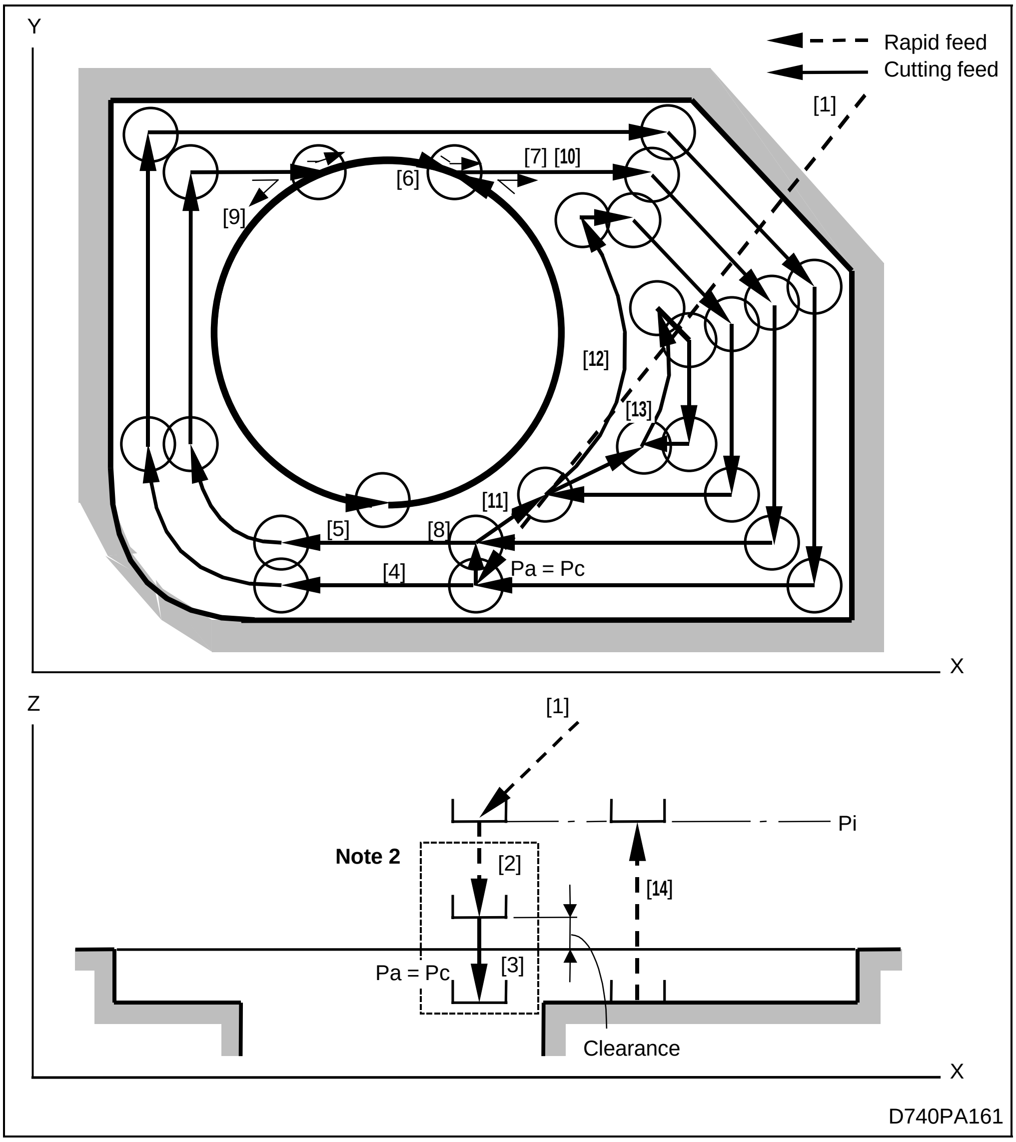
[10] Once it has interfered with the outside form, the tool machines along the outside form.

[11] To machine the rest, the tool moves along the same path as that in [6].

[12], [13] and [14] The rest is machined outwards on an around by around basis.

[15] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

Machining from outside (roughing or bottom finishing)



Pa, Pc: Approach point to be determined by the data **APRCH-1**, **-2** in the tool sequence. (In the illustration above, the cutting start point is the approach point.)

Pe: Escape point to be automatically established

<Route on which tool is to move>

Tool path [8] and [10] is the same as that of [5] and [7] respectively.

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

Note 2: See Subsection 3-7-5, “Precautions in face machining.”

Outer wall finishing

The tool is machining through a tool path identical with that for finishing in the LINE IN unit.

F. Finishing

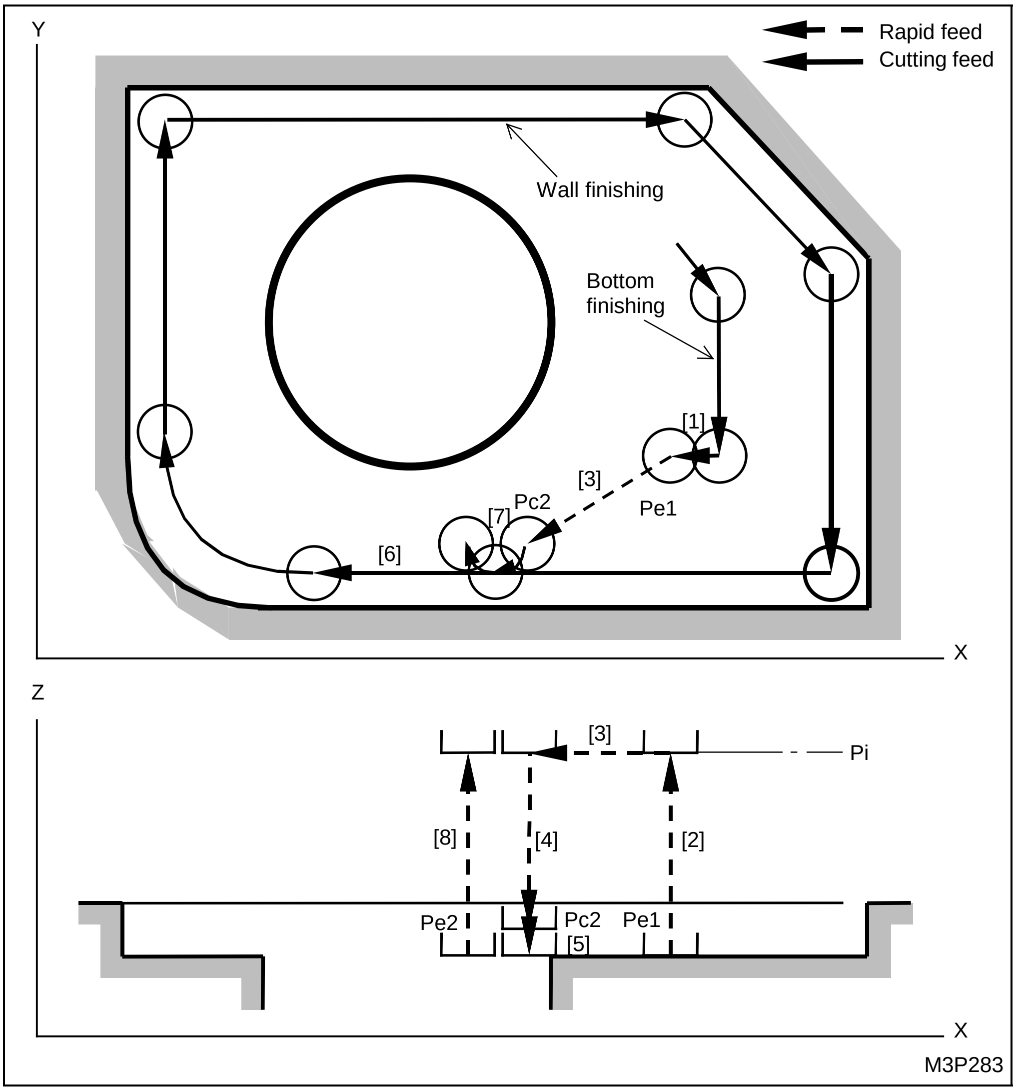
Finishing is performed, based on the entered data **FIN-A** and **FIN-R**.

Bottom finishing is performed, with 0 < **FIN-A**.

Wall finsihing 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

Pe2: Escape point in wall finishing

Pi: Initial point

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

7. End milling-slot unit (SLOT)

This unit is selected to carry out slot machining by the use of an end mill.

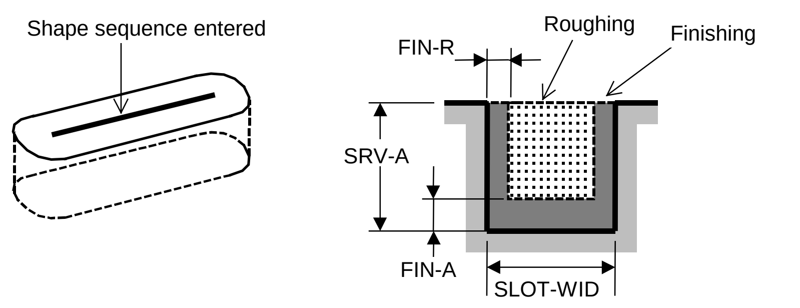
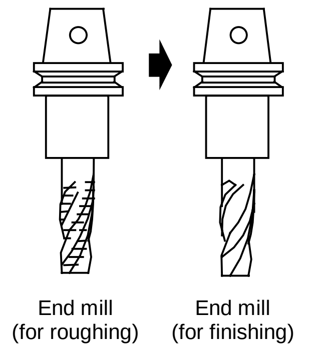
A. Data setting

```
UNo.    UNIT     MODE   POS-B   POS-C    SRV-A    SLOT-WID    BTM    WAL  FIN-A   FIN-R
 1    SLOT
SNo.    TOOL    NOM-ϕ No.   #   APRCH-1 APRCH-2 TYPE  AFD  TYPE PK-DEP  DEP-A  WID-R C-SP FR   M   M   M
 R1   END MILL                                                   ◆
 F2   END MILL                                                   ◆      ◆
```

◆: Data are not necessary to be set here.

Remark 1: In this unit, end mills are automatically developed.

Remark 2: For the tool sequence data setting, see Subsection 3-7-4.

SLOT unit	Tool sequence
 D740PA142	 D740PA051

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.

B. Automatic tool development

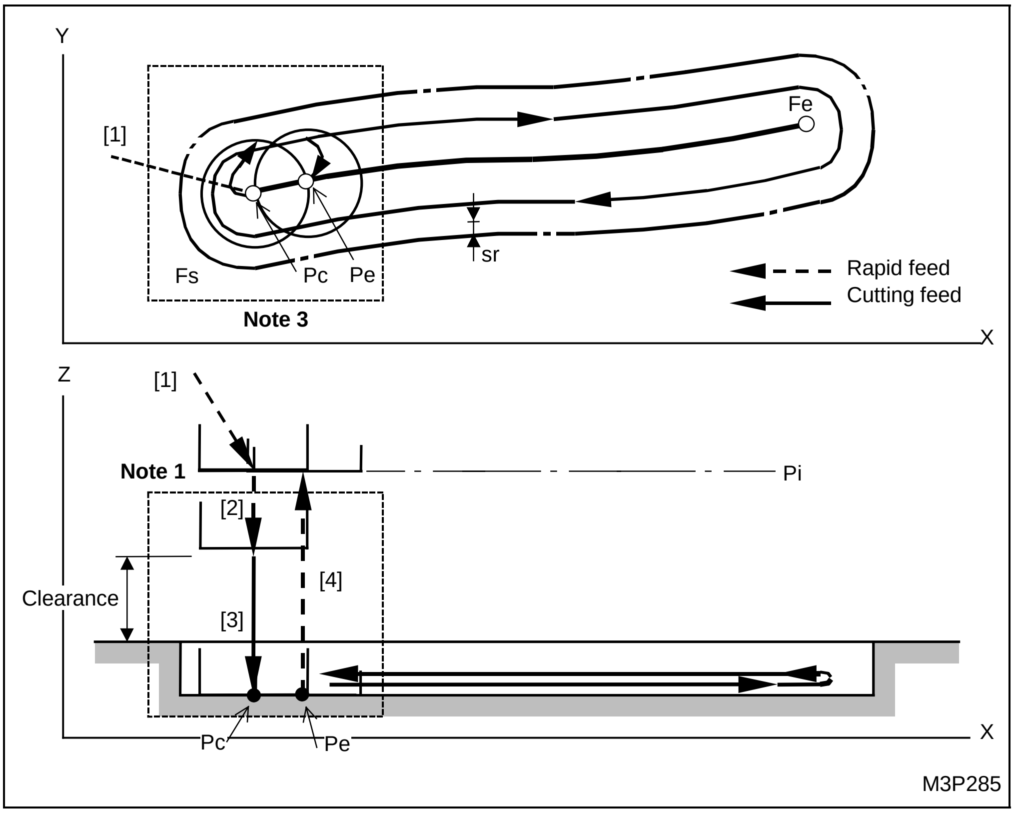
The tools are automatically developed according to different patterns on the basis of the data entered in the unit. The machining is executed on the basis of the tool sequence data and the unit data are not used for the machining. If the data developed are inappropriate for the machining, edit by modifying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

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**.

Machining	Pattern
R1 (Roughing)	FIN-A = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-A ≤ FIN-A or SLOT-WID ≤ (2 × FIN-R) : One tool is selected.

C. Tool path

With tool radius + FIN-R < $\frac{\text{SLOT-WID}}{2}$ < tool diameter + FIN-R



The bold codes represent parameter addresses.

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 automatically established

Pi: Initial point

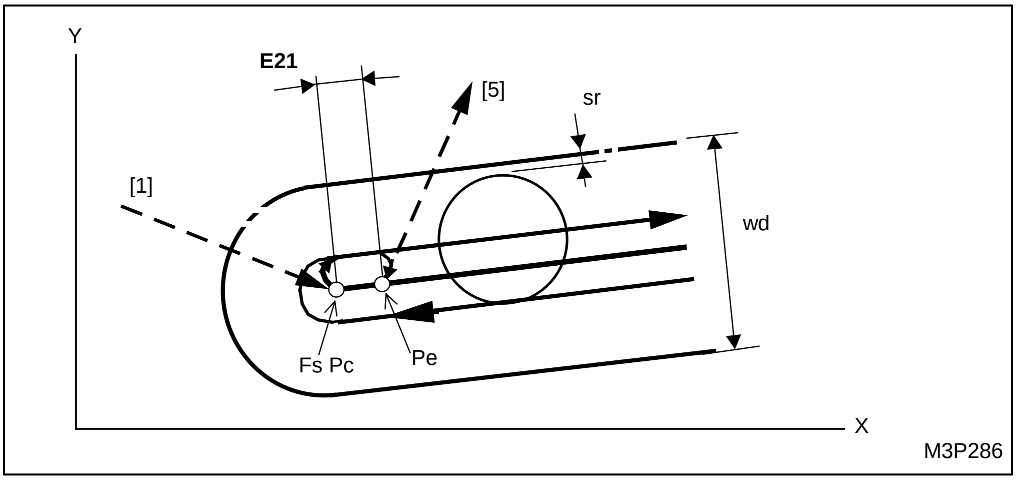
sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

Note 1: See Subsection 3-7-5, “Precautions in face machining.”

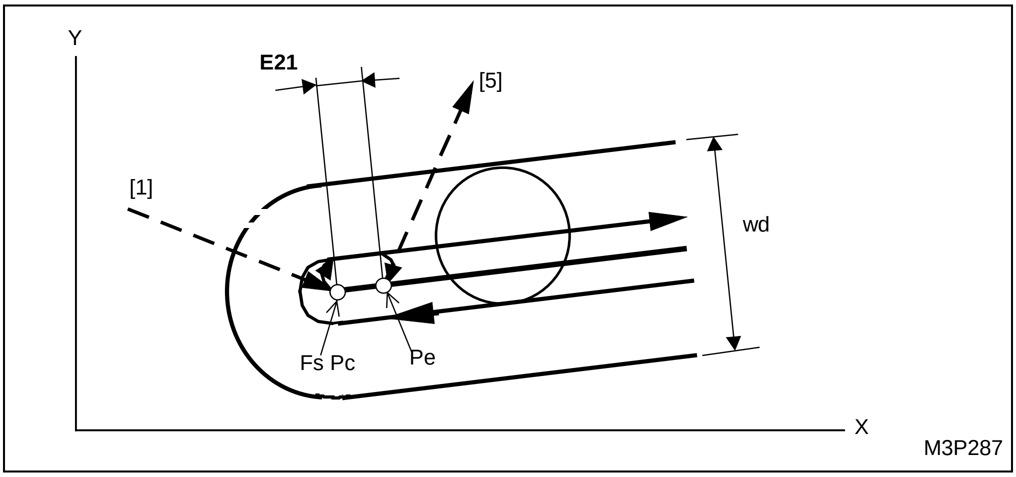
Note 2: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

Note 3: Detail of tool path near approach point and escape point

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pc: Cutting start point to be automatically established

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

Pe: Escape point to be automatically established

wd: Slot width (**SLOT-WID**) to be entered in a machining unit

sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

<Route on which tool is to move>

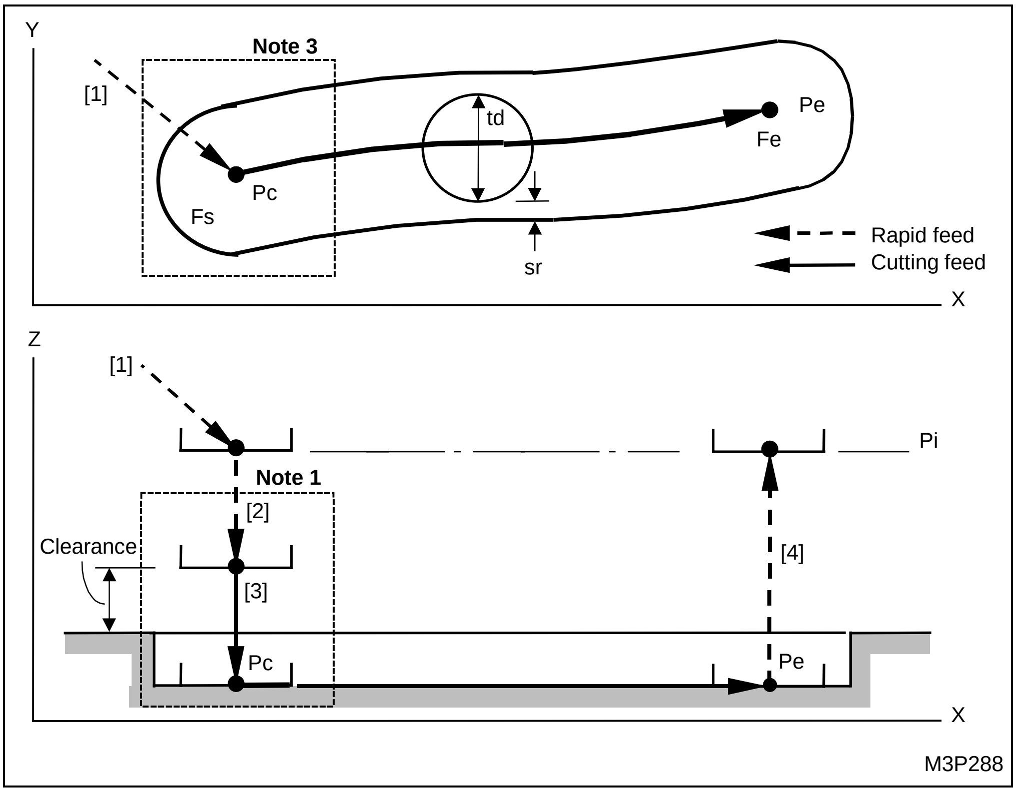
[1] The tool moves at a rapid feedrate to approach point (cutting start point).

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined and starts machining.

[4] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

$$\text{With tool radius + FIN-R} = \frac{\text{SLOT-WID}}{2}$$



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

Pi: Initial point

td: Tool diameter to be registered in the **TOOL DATA** display

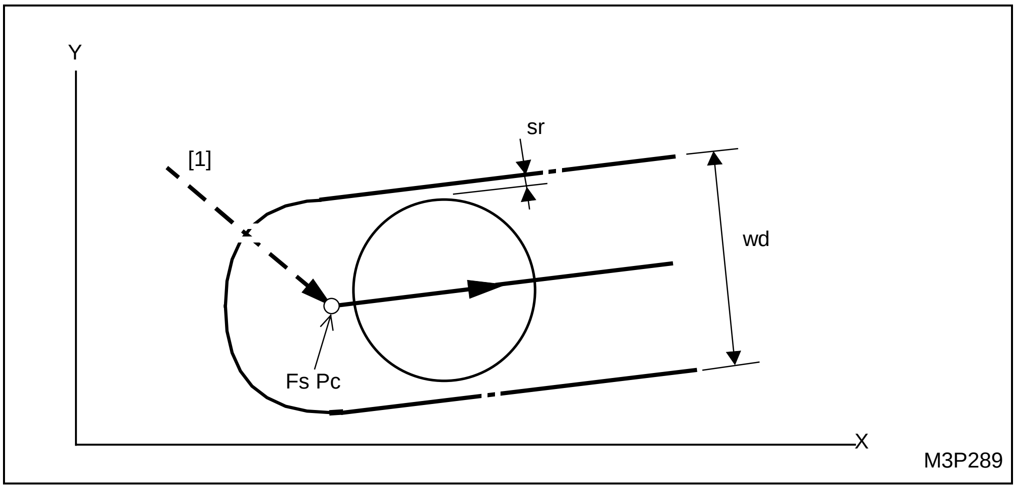
sr: Radial finishing allowance (**FIN-R**) to be entered in a machining unit

Note 1: See Subsection 3-7-5, “Precautions in face machining.”

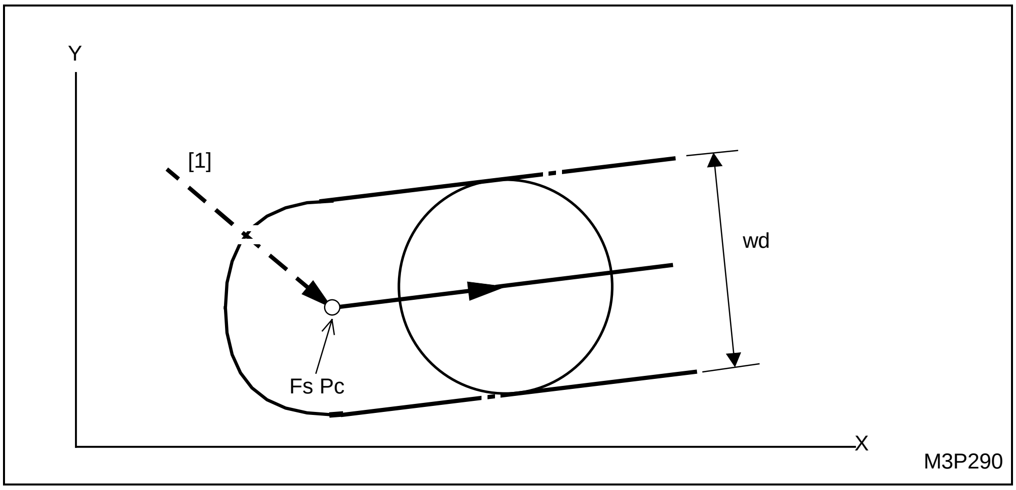
Note 2: The feedrate on tool path [3] is dependent upon the data **AFD** (axial feed) in the tool sequence.

Note 3: Detail of tool path near approach point

- Roughing



- Finishing



Pc: Cutting start point to be automatically established

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

wd: Slot width (**SLOT-WID**) to be entered in a machining unit

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 approach point (cutting start point).

[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined and starts machining.

[4] Upon completion of machining, the tool moves at a rapid feedrate to initial point.

3-7-4 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.

Tool sequence data

	TOOL	NOM-φ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	TYPE	PK-DEP	DEP-A	WID-R	C-SP	FR	M M M
Item	(1)	(2)(3)(4)	(5)	(6)	(7)	(7)	(8)	(9)	(10)	(11)	(12)	(13)	(14)	(15)	(16)

◆: Not necessary to be set here.

For setting of each data item refer to 1 to 16 below.

1. TOOL (Tool designation)

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

In the face milling unit, either one of the menu key **ENDMILL**, **FACEMILL** and **BALL ENDMILL** is selectable. In the pocket milling unit **ENDMILL**, **BALL ENDMILL**, **CHAMFER CUTTER** or **CENTER DRILL** can be selected. In other units, either the menu key **ENDMILL** or **BALL ENDMILL** is selectable.

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	CENTER DRILL					

2. NOM-φ (Nominal tool diameter)

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. NOM-φ (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	HEAVY TOOL	>>>

To slowly change a heavy tool in the ATC mode, select a heavy tool identification code.
Press the **[HEAVY TOOL]** menu key to change the menu to one for heavy tool identification code. Then select a code from the menu to identify those tools which have an identical nominal diameter.

4. NOM-φ (Turret selection)

For the machine with the lower turret, select the turret in which the tool to be used is mounted. The following menu is displayed (if **[SET UPPER TURRET]** is selected, the column will remain blank, and if **[SET LOWER TURRET]** is selected, "▼" will be displayed). See Chapter 5, LOWER-TURRET CONTROL FUNCTIONS, for further details:

SET UPPER TURRET	SET LOWER TURRET								

5. No. (Priority No.)

Assign priority levels in the order of machining. The following menu is displayed. A press of a menu key displays the menu item in reverse mode, allowing a priority number to be assigned.

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN			PRI.No. ALL ERAS	SUB PROG PROC END	
	(a)		(b)	(c)			(d)	(e)	

The function of menu item (a) to (e) is described below:

Menu item	Function
(a)	Select to conduct subsequent-machining.
(b)	Select to change the priority number for the tool within the particular process. If the cursor is present at a blank space, assign a new number in a usual manner. Entry of an existing priority number displays alarm **420 SAME DATA EXISTS**.
(c)	Select to assign a priority number to the tool to be used repeatedly in the particular process. Alarm **420 SAME DATA EXISTS** will be displayed if the assigned priority number has already been set on any other unit line.
(d)	Selection of this item displays message **ALL ERASE (PROC:0, PROG:1)?**. Setting 0 will erase the priority numbers preassigned to the tool to be used repeatedly in the process. Setting 1 will erase the priority numbers preassigned to the tool to be used repeatedly in the program.
(e)	Select to terminate the process with the subprogram unit.

For details see Chapter 4, “PRIORITY FUNCTION FOR THE SAME TOOL.”

6. # (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. For details see Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”.

LOWER TURRET POS.1	LOWER TURRET POS.2								

7. APRCH-1, APRCH-2 (Coordinates of the approach point)

Enter coordinates of the position at which a tool is to cut in axially.

Pressing the **[AUTO SET]** menu key sets a question mark (?). After the tool path check is performed, **?** will automatically change over to the coordinates of a cutting start point. (Refer to tool path for each unit.)

8. TYPE (Machining method)

Machining method differs according to the units as follows.

A. Face milling unit

A tool path pattern is selected out of three: BI-DIRECT., UNI-DIRECT., BI-DIRECT SHORT and BI-DIRECT ARCSHORT. In each pattern, moreover, it is possible to select whether machining is performed in parallel with the X-axis or the Y-axis.

X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR	X BI-DIR SHORT	Y BI-DIR SHORT	X BI-DIR ARCSHORT	Y BI-DIR ARCSHORT		

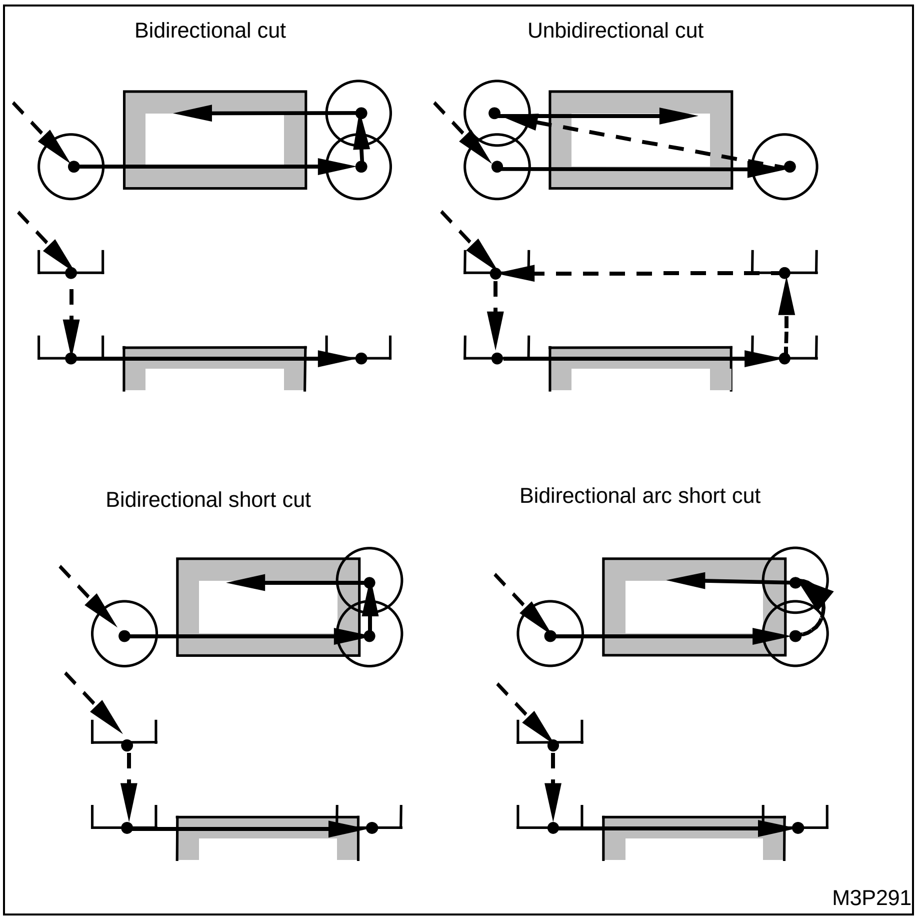
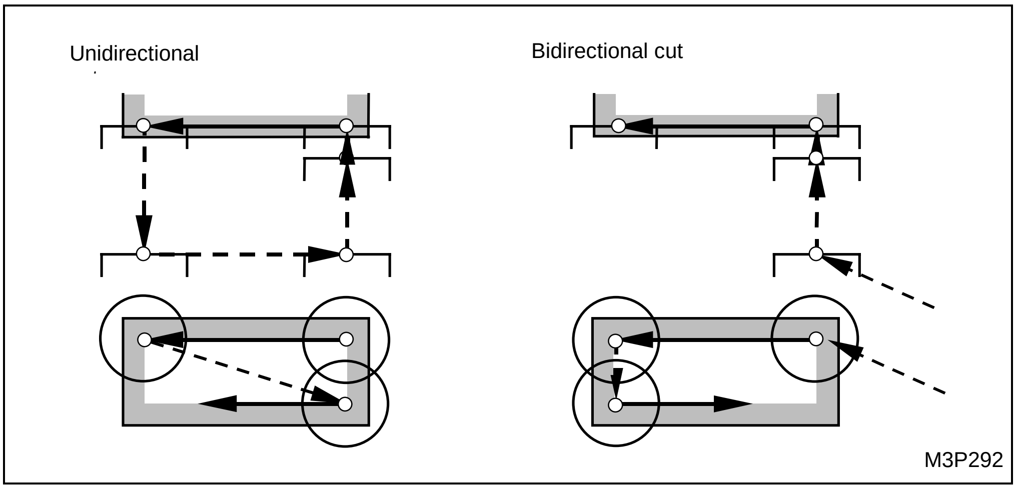


Fig. 3-24 Tool path patterns

B. End milling-top unit

A tool path pattern is selectable out of BI-DIRECT. or UNI-DIRECT. In each pattern, moreover, it is possible to select whether machining is performed in parallel with the X-axis or with the Y-axis.

X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR						

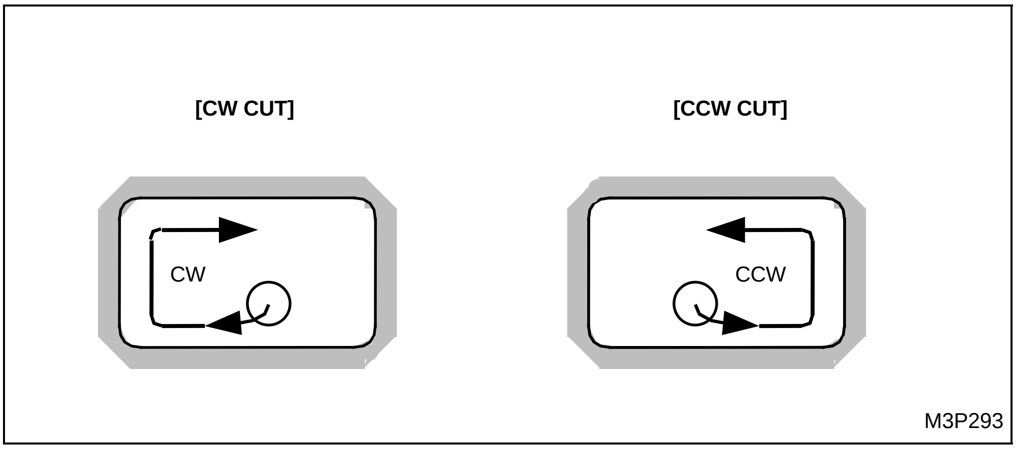


M3P292

C. Other units

A machining (turning) direction is selected by the use of menu keys.

CW CUT	CCW CUT								

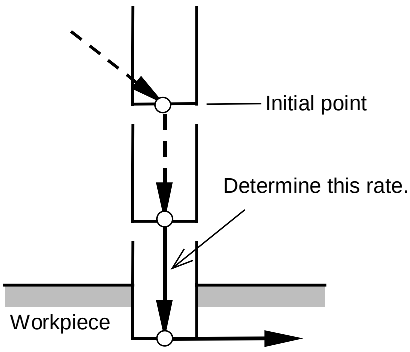


M3P293

9. AFD (Axial feedrate)

The radial 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 × $\frac{\mathbf{E17}}{10}$
Numeric value (α)	Feed × α

M3P294

10. TYPE

Select the type of operation during axial cutting.

STANDARD	TAPER	HELICAL	PECKING						

For details see Subsection 3-7-5, "Precautions in face machining."

Note 1: This item can be specified, when **[CUT G01]** is selected for **AFD**.

Note 2: This item is available only for the following five units: STEP, POCKET, PCKT MT PCKT VLY and SLOT.

11. PK-DEP

Specify the first cutting depth per pass for pecking operation.

Note: This item can be specified, when **[PECKING]** is selected for **TYPE** in 10.

12. DEP-A

In roughing, a maximum axial cutting stroke in one pass is entered. With **[AUTO SET]** menu key pressed, a smaller value is entered, either the data **SRV-A** entered in the machining unit or the maximum cutting depth registered on the **TOOL FILE** display.

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 3-7-5, "Precautions in face machining.")

13. WID-R

A maximum cutting depth per pass is entered in roughing or bottom finishing.

With the **[AUTO SET]** menu item selected, the data **WID-R** is automatically calculated and determined by the parameter **E10** or **E14** and nominal tool diameter.

$$\text{WID-R} = \text{Norminal tool diameter} \times \frac{\mathbf{E10}}{10} \quad \text{: FCE MILL, TOP EMIL and STEP}$$

$$\text{WID-R} = \text{Norminal tool diameter} \times \frac{\mathbf{E14}}{10} \quad \text{: POCKET, PCKT MT and PCKT VLY}$$

14. C-SP (Cutting conditions)

A spindle surface speed is entered in meters per minute.

With **[AUTO SET]** menu key selected, optimum cutting conditions are automatically calculated and entered, based on the materials of both workpiece and tool and on the cutitng depth.

15. FR

A feedrate ot the tool is entered in millimeters per revolution. Same as the surface speed, the entry of data is done by means of menu keys or numeric keys.

16. M (M-code)

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 three M-codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu. (Refer to the separate Parameter List/Alarm List/M-Code List.)

3-7-5 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:

- Cutting start position in the axial direction
- Type of routing through approach point

For each factor refer to A and B below.

[Basic tool pat]

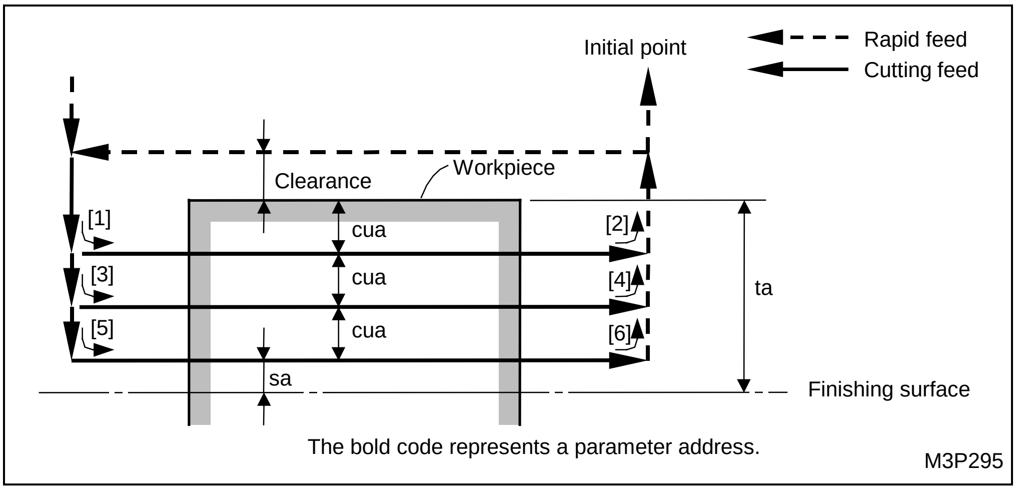


Fig. 3-25 Basic tool path

cua: Axial cutting depth per pass (Axial cutting depth **DEP-A** to be entered in the tool sequence)

Calculation of cua:

$$cua = \frac{ta - sa}{n}$$

$$n = \frac{ta - sa}{cua}$$

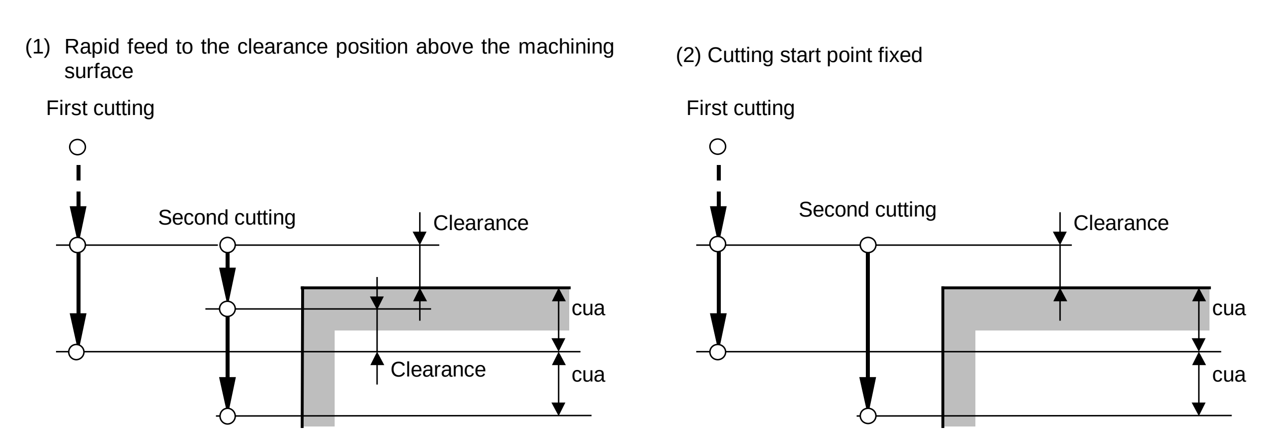
ta: Axial machining allowance **SRV-A** to be entered in the machining unit

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

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

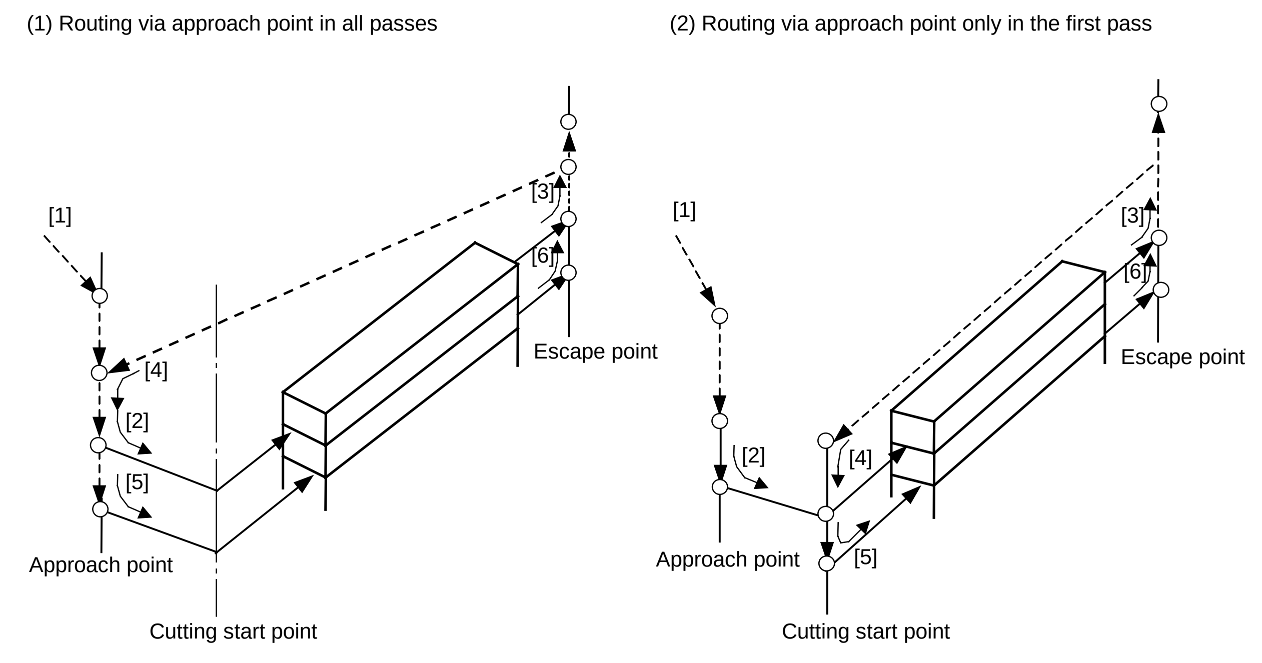
Select one of the following two types:



M3P237

B. Setting the type of routing via approach point

Select one of the following two types:



M3P238

<Tool path setting parameters>

Parameter related to the tool path setting in each face machining unit is shown below.

End milling-top unit (TOP EMIL): **E97**

End milling-step unit (STEP): **E91**

Pocket milling unit (POCKET): **E92**

Pocket milling-mountain unit (PCKT MT): **E93**

Pocket milling-valley unit (PCKT VLT): **E94**

End milling-slot unit (SLOT): **E96**

(The bold codes represent user parameter addresses.)

For A: bit 4 of each parameter = 0: Cutting start point fixed, (2)
1: Rapid feed to the clearance position 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 clearance) from the second cutting when the following conditions are satisfied:

- Bit 1 of **E96** (for SLOT unit) or bit 2 of the other parameters concerned is set to “1”, and
- The unit concerned is TOP EMIL, STEP, POCKET, PCKT MT, PCKT VLY or SLOT.

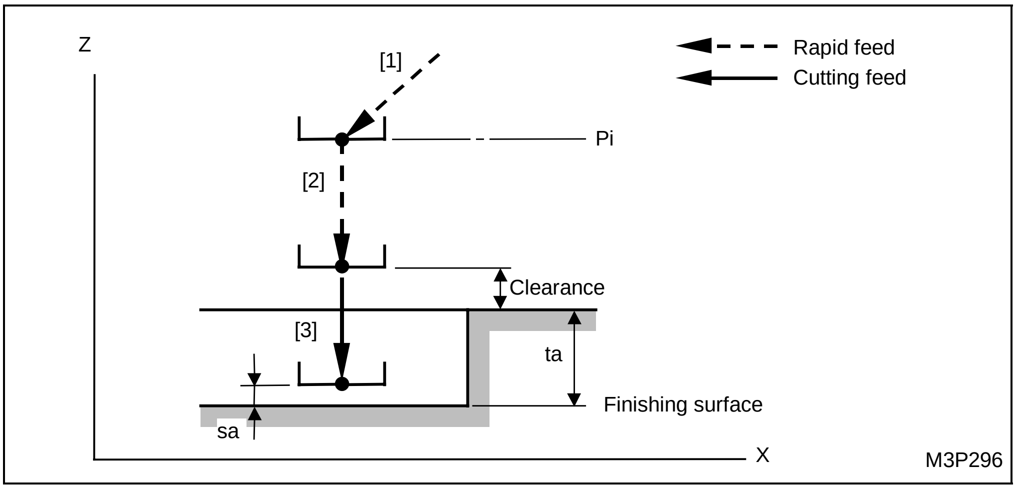
For B: bit 2 of parameter **E95** = 0: Routing via approach point only in the first pass, (2)
1: Routing via approach point in all passes, (1)

Note 1: B can only be used in the end milling-slot (SLOT) unit.

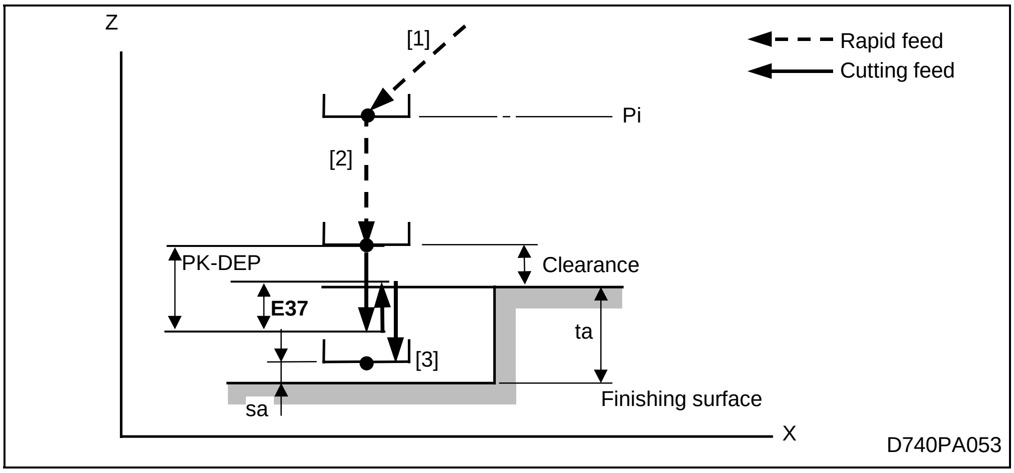
Note 2: The tool path shown at basic tool path above is selected automatically for face machining unit that is not assigned by these parameters.

2. Detail tool path of an axial cutting

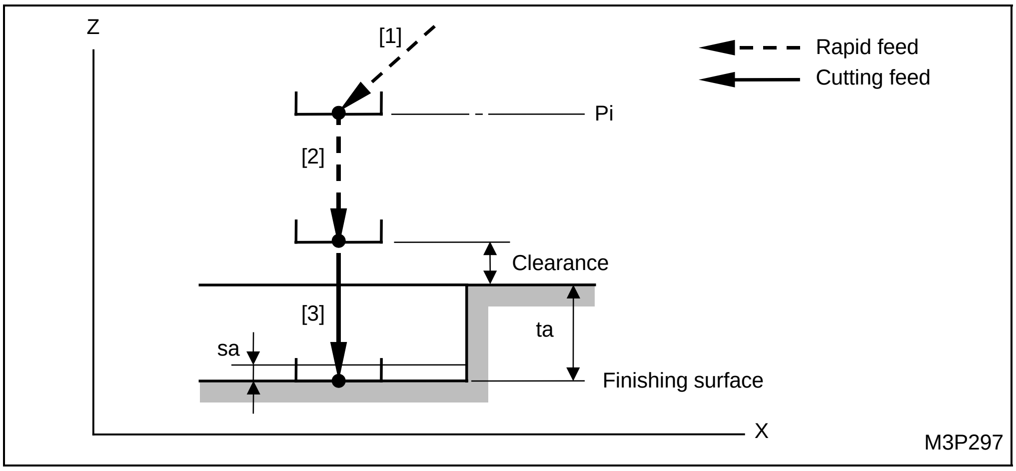
- Roughing (**[STANDARD]** is selected)



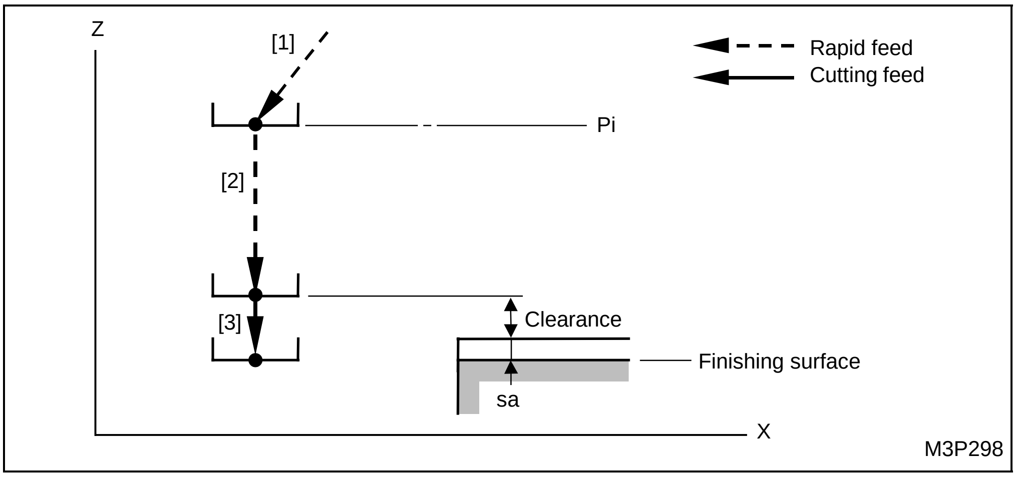
- Roughing (**[PECKING]** is selected)



- Bottom finishing
  (End milling-step, pocket milling, pocket milling-mountain, pocket milling-valley)



- Finishing
  (End milling-top, end milling-slot)



The bold codes represent parameter addresses.

Pi: Initial point

ta: Axial machining 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 starting allowance of axial cutting, specified by the (safety) clearance, 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.

Unit	Parameter	Unit	Parameter
End milling-top	**E97**, bit 2	Pocket milling-mountain	**E93**, bit 2
End milling-step	**E91**, bit 2	Pocket milling-valley	**E94**, 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 **E2**, will become equal to parameter **E5** if the following two states occur at the same time:

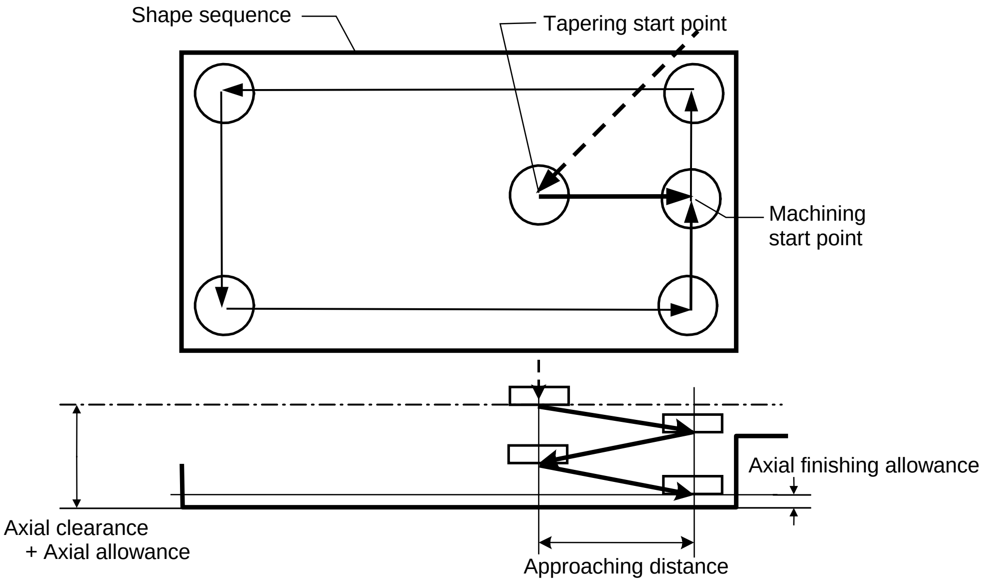
- The designaed parameter for the intended unit is set to 1.

Unit	Parameter	Unit	Parameter
End milling-step	**E91**, bit 3	Pocket milling-mountain	**E93**, bit 3
Pocket milling	**E92**, bit 3	Pocket milling-valley	**E94**, bit 3

- A pre-machining tool is included in that tool sequence.

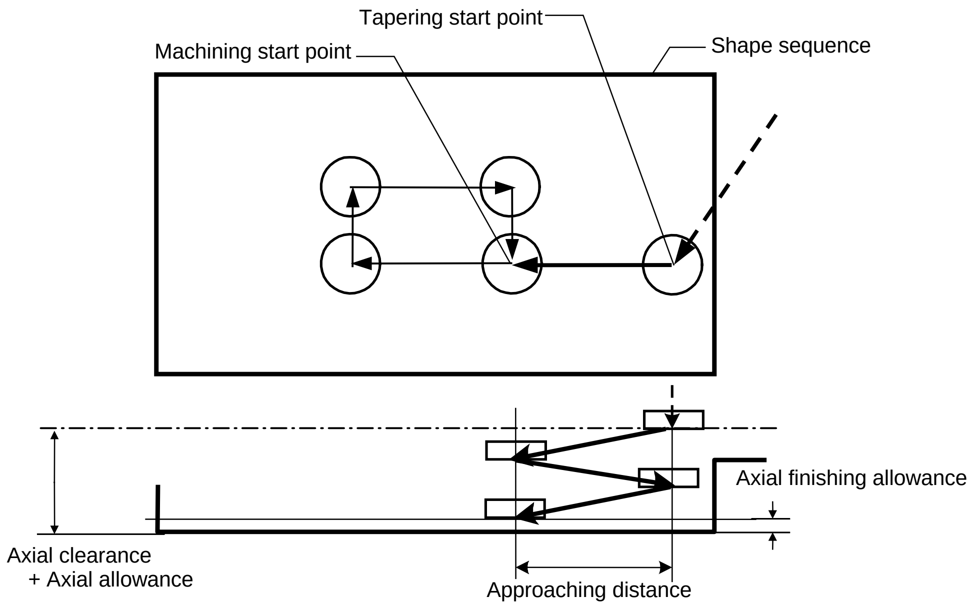
3. Tool paths in tapered approach scheme and helical approach scheme

- Tapered approach scheme
  (machining pattern: outside → inside, approach point: automatically determined)



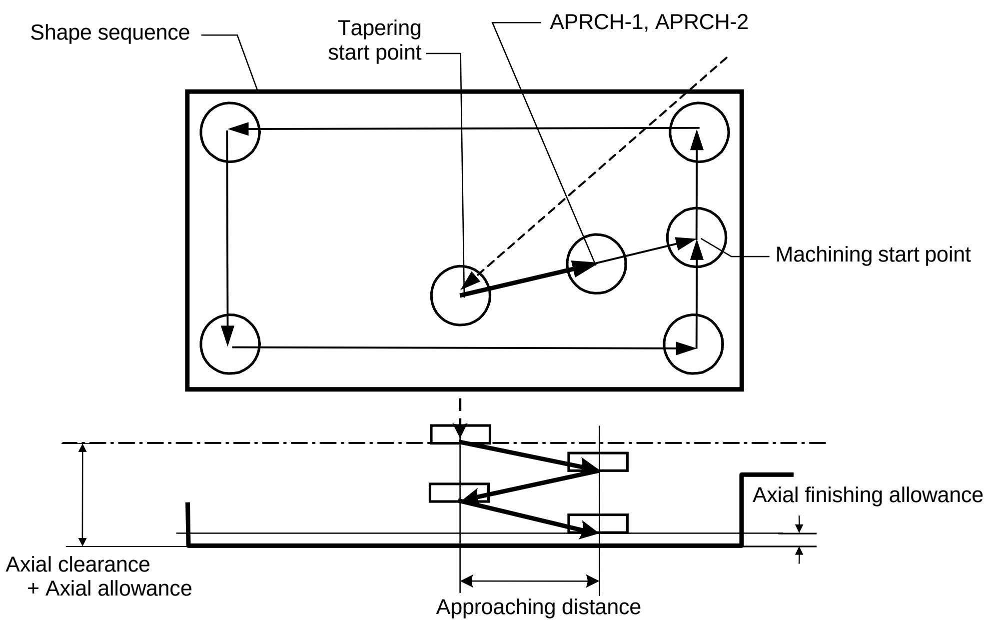
D740PA054

- Tapered approach scheme
  (machining pattern: inside → outside, approach point: automatically determined)



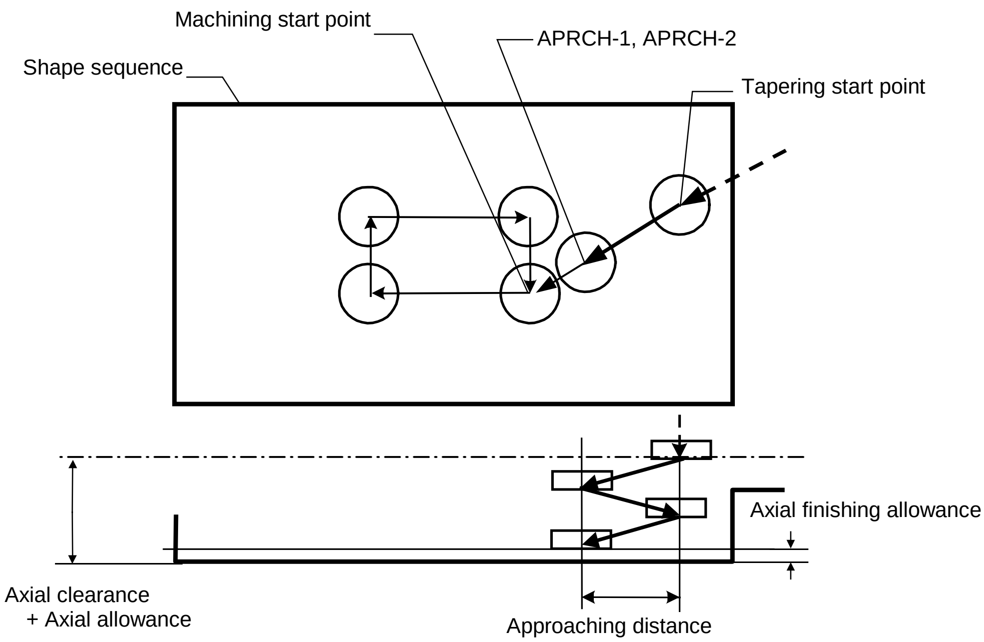
D740PA055

- Tapered approach scheme
  (machining pattern: outside → inside, approach point: manually determined)



D740PA056

- Tapered approach scheme
  (machining pattern: inside → outside, approach point: manually determined)



D740PA057

Note 1: The approaching distance in a tapered approach scheme is automatically determined by a parameter, as described below.

Approaching distance = Nominal diameter of tool × $\dfrac{\mathbf{E34}}{10}$

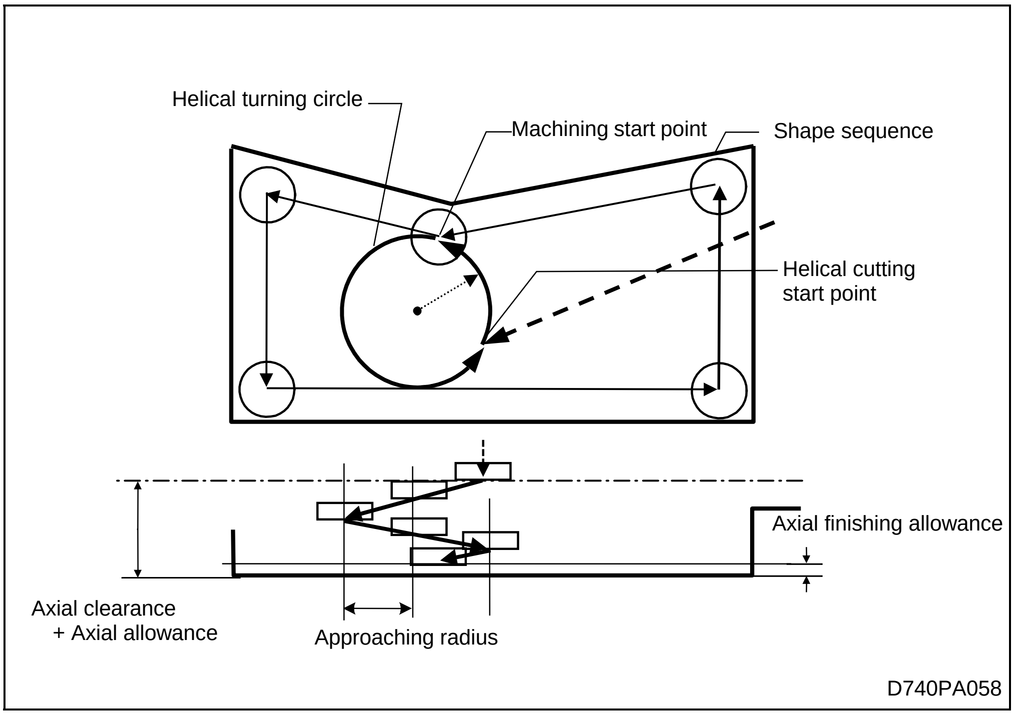
Note 2: The approaching gradient in a tapered approach scheme can be changed with the parameter **E35**.

Note 3: An approaching direction is automatically determined, as described below.

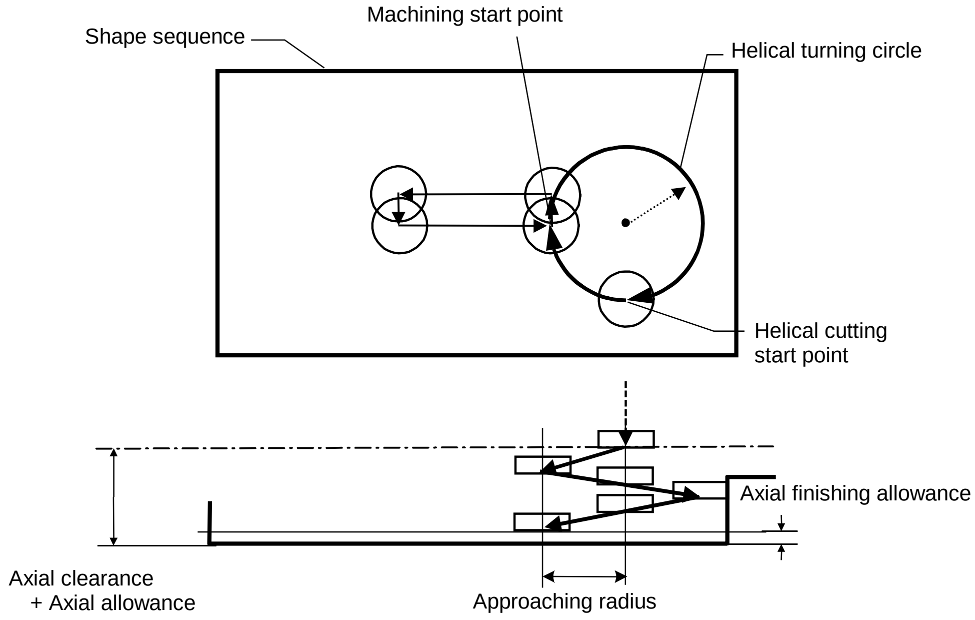
- If an approach point is set automatically
  Outside to inside: Direction of the line that equally divides the angle formed by the two sides of the shape that sandwich the machining start point.
  Inside to outside: Direction of the tangent to the line that connects the machining start point to the next machining point
- If an approach point is set manually
  Direction of the tangent to the line that connects the approach point to the machining start point

Note 4: If the approaching distance is so long that it interferes with the shape sequence or the tool path, the alarm **705 APPROACH POINT ERROR** is issued.

- Helical approach scheme
  (machining pattern: outside → inside, approach point: automatically determined)

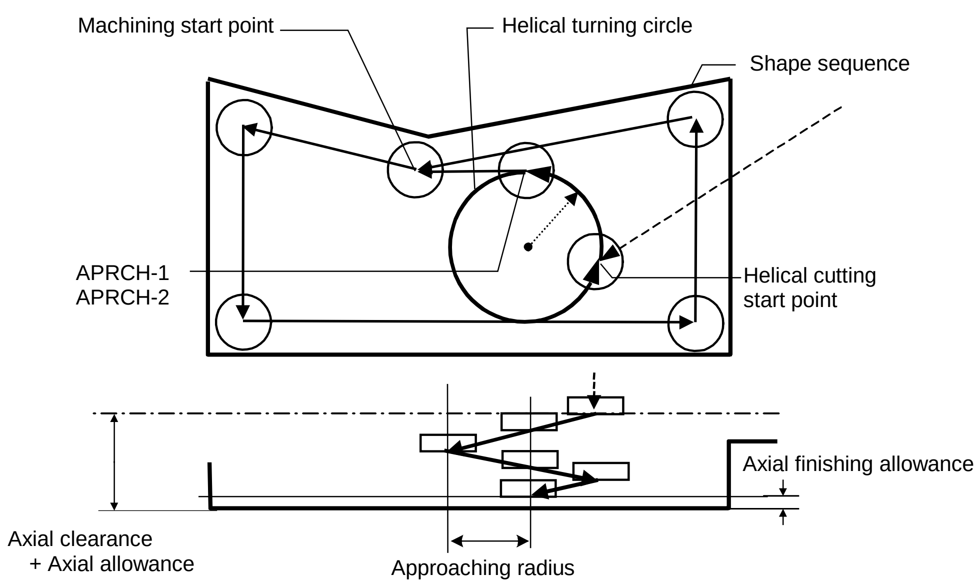


- Helical approach scheme
  (machining pattern: inside → outside, approach point: automatically determined)



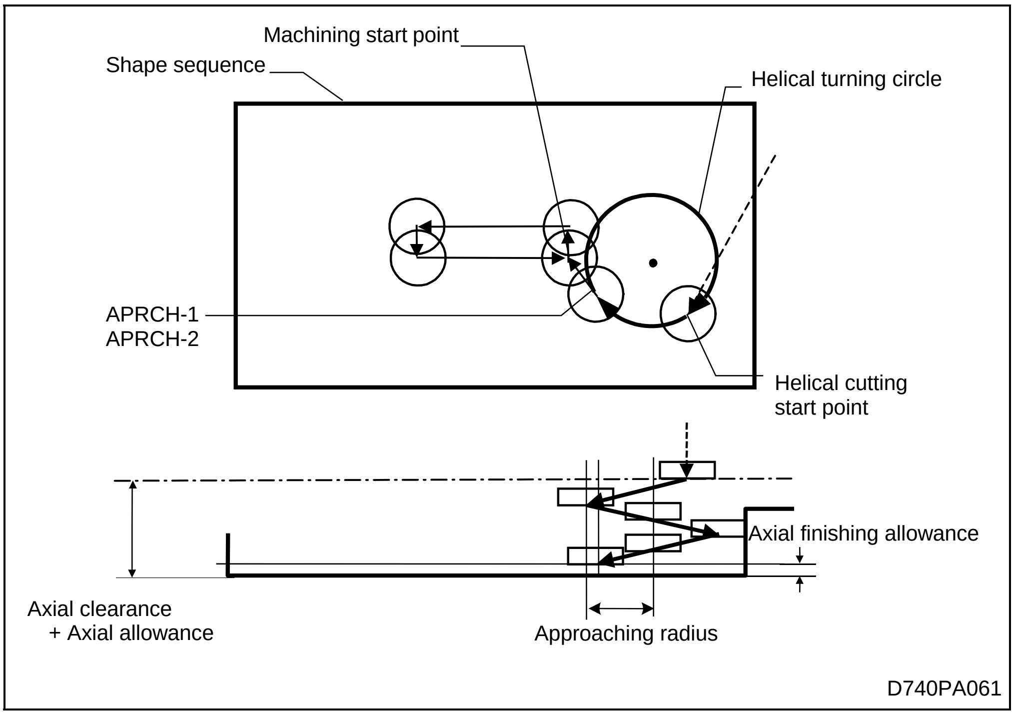
D740PA059

- Helical approach scheme
  (machining pattern: outside → inside, approach point: manually determined)



D740PA060

- Helical approach scheme
  (machining pattern: inside → outside, approach point: manually determined)



Note 1: The approaching radius in a helical approach scheme is automatically determined by a parameter, as described blow.

Approaching radius = Nominal diameter of tool × $\frac{\mathbf{E32}}{10}$

Note 2: The approaching gradient in a helical approach scheme can be changed with the parameter **E33**.

Note 3: The helical turning circle computed is tangent to the path that connects the approach point to the machining start point.

Note 4: The rotational direction of the helical turning circle that the tool approaches is automatically determined, as described below.

- Outside to inside: Rotational direction specified as **TYPE** of tool sequence data
- Inside to outside: Opposite to the rotational direction specified as **TYPE** of tool sequence data

Note 5: If the approach circle is so large that it interferes with the shape sequence or the tool path, the alarm **705 APPROACH POINT ERROR** is issued.

4. Other precautions

1. If shape data, tool data or parameters are modified after the automatic determination of the data **APRCH-1**, **-2** in the tool sequence (displayed in yellow), the cutting start point will be changed accordingly and the tool path will be also modified (tool moves axially in the position of the approach point and then radially to the cutting start point).
2. When machining is performed in the cutting direction reverse mode or when a concave form is pocket milled, a portion remains uncut. Up to a maximum of 32 portions so remaining uncut, machining is automatically carried out. In excess of 32, however, if will result in an alarm.

3-7-6 Override in case of the overall width cutting

In the pocket milling, the cutting width inside the machining form is determined by the data WID-R in the tool sequence. In the first pass, etc., however, the cutting width is equal to the tool diameter. Therefore the cutting load in such a case is bigger than that applied in the next pass. To make the cutting load equal in all passes, the feedrate is automatically reduced in the first pass, etc.

1. Operating conditions

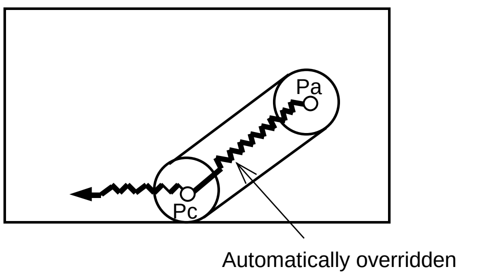
The overall width cutting override is valid in pocket milling on the following four cases on which the cutting width equals the tool diameter:

A. To move from the approach point to the cutting start point:

Illustration below is an example of the pocket milling unit.

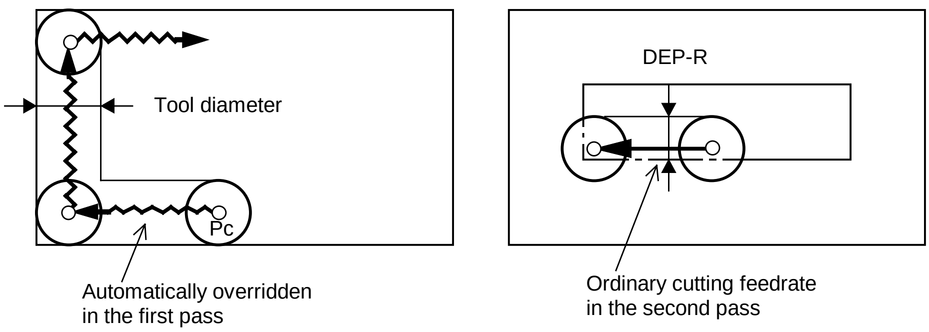
Pa: Approach point

Pc: Cutting start point



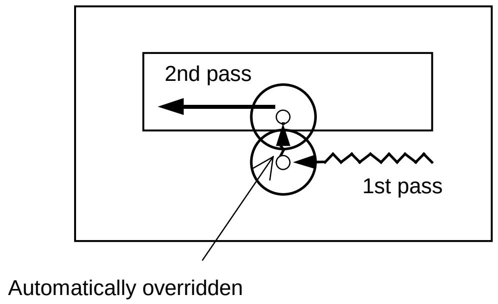
M3P299

B. First pass



M3P300

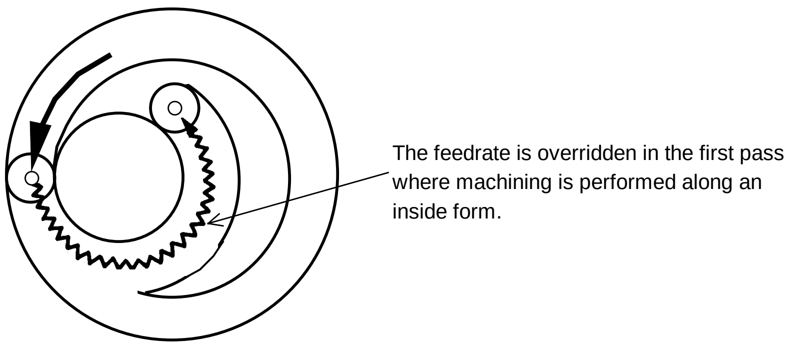
C. To move to the next cutting start point



M3P3001

D. First pass after the inversion of the tool path

Illustration below is an example of the pocket milling-mountain unit.



M3P302

2. Machining available

The overall width override is valid in roughing for pocket milling, pocket milling-mountain, pocket milling-valley and end milling-step units.

3. Override rate

An override on the programmed cutting feedrate is determined by the parameter **E18**.

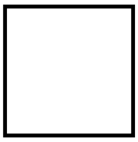
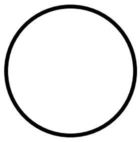
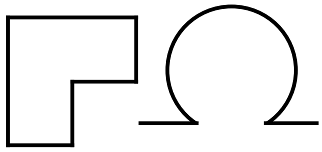
With the parameter set to 0, the overall width cutting override is invalid.

3-7-7 Shape sequence data of the line/face machining unit

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.

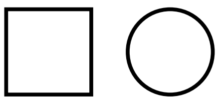
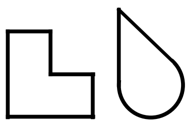
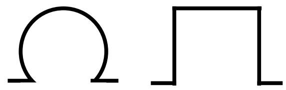
Fixed form		Arbitrary form
SQUARE	CIRCLE	ARBITRY
		 M3P303

The arbitrary forms can be divided into two types shown below. The fixed forms belong to closed type.

A. Closed form and open form

Depending on the machining units, machining form can be divided into the following two types:

Table 3-3 Closed form and open form

	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	Machining to one defined shape	FCE MILL, TOP EMIL, POCKET	SLOT
	Machining with at least two defined shapes	STEP, PCKT MT, PCKT VLY	

A fixed shape is not provided for the SLOT unit.

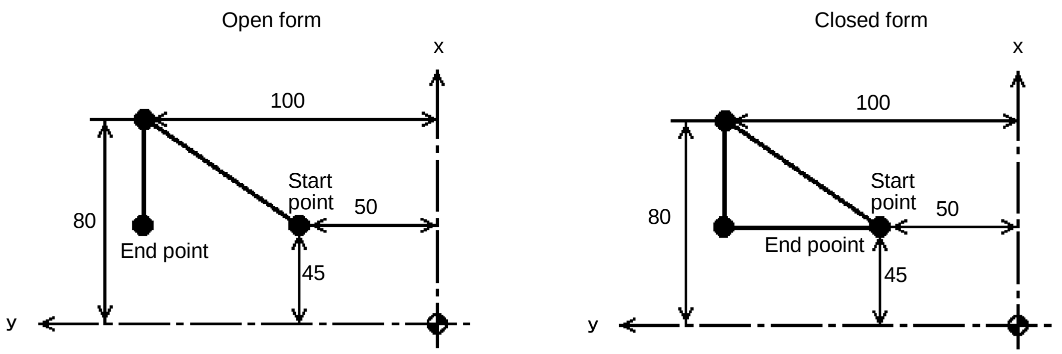
2. Precautions to be taken when defining an arbitrary form

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	PTN	SHIFT-R	X	Y	R/th	I	J	P	CNR	R-FEED	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.

Note 1: Although a maximum of 200 lines of shape sequence data can be set in one milling unit, the maximum usable number of shape sequence data lines may be less than 200 in the following cases:

- When **CW-SH**, **CCW-SH** or **FIG-SH** is included in the shape sequence of the line or face machining unit
  In this case, the maximum usable number of shape sequence data lines is determined by the expression below.
  (No. of shape sequence data lines on the graphics to be shifted or rotated) ×
  (No. of shape repetitions) + (No. of other shape sequence data lines) ≤ 200
- When corner R/C is defined for a complex shape
  Alarm **708 BLOCK DATA LIMIT EXCEEDED** will be displayed and operation brought to a stop, even before the maximum usable number of shape sequence data lines is reached.

Note 2: If the maximum usable number of shape sequence data lines is exceeded, alarm **723 EXCEEDS NUMBER OF SHAPES** will be displayed during tool path checking, shape checking, shape drawing, or automatic operation.

A. When the mode selected in the unit is ZC

1. Fixed form

- Square (**SQR**)

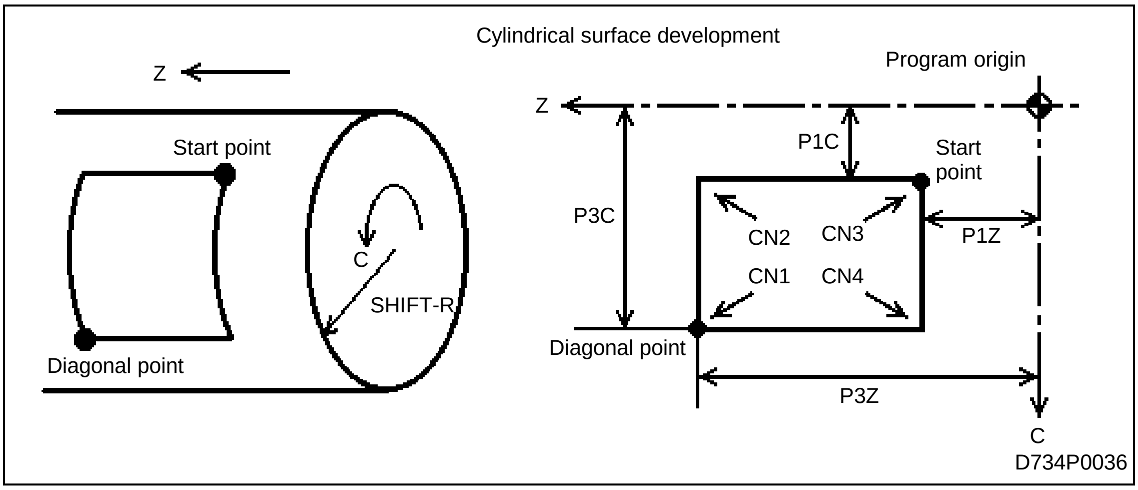


FIG	PTN	SHIFT-R	P1Z/CZ	P1C/CC	P3Z/R	P3C	CN1	CN2	CN3	CN4
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[6]	[6]	[6]

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] CN1 - CN4	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

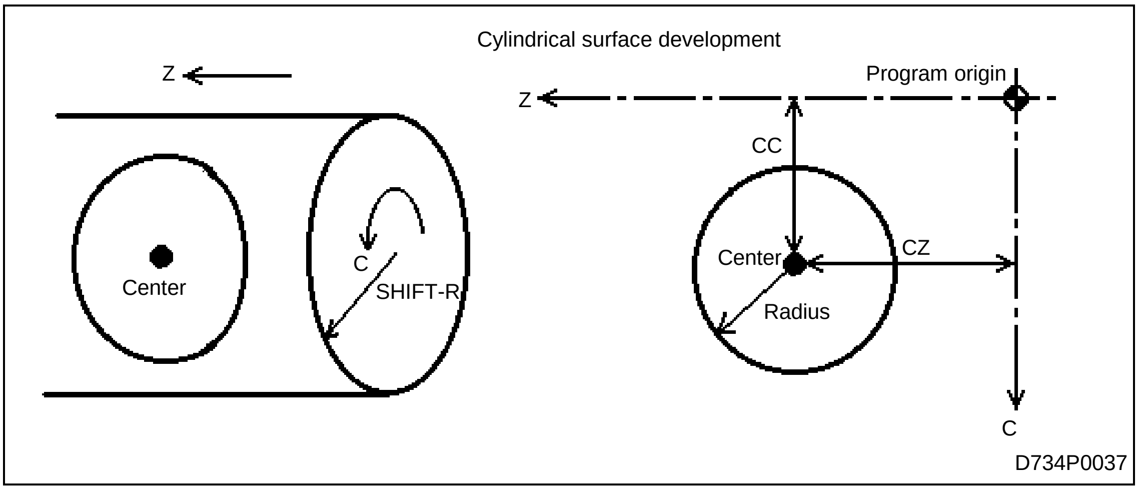


FIG	PTN	SHIFT-R	P1Z/CZ	P1C/CC	P3Z/R	P3C	CN1	CN2	CN3	CN4
1	CIR	[1]	[2]	[3]	[4]	◆	◆	◆	◆	◆

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**)

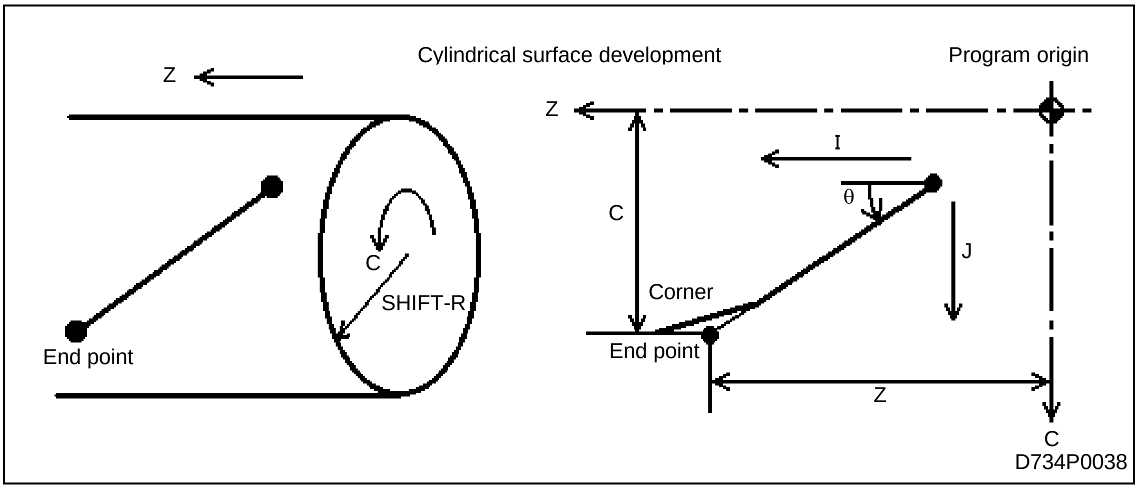


FIG	PTN	SHIFT-R	Z	C	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW**, **CCW**)

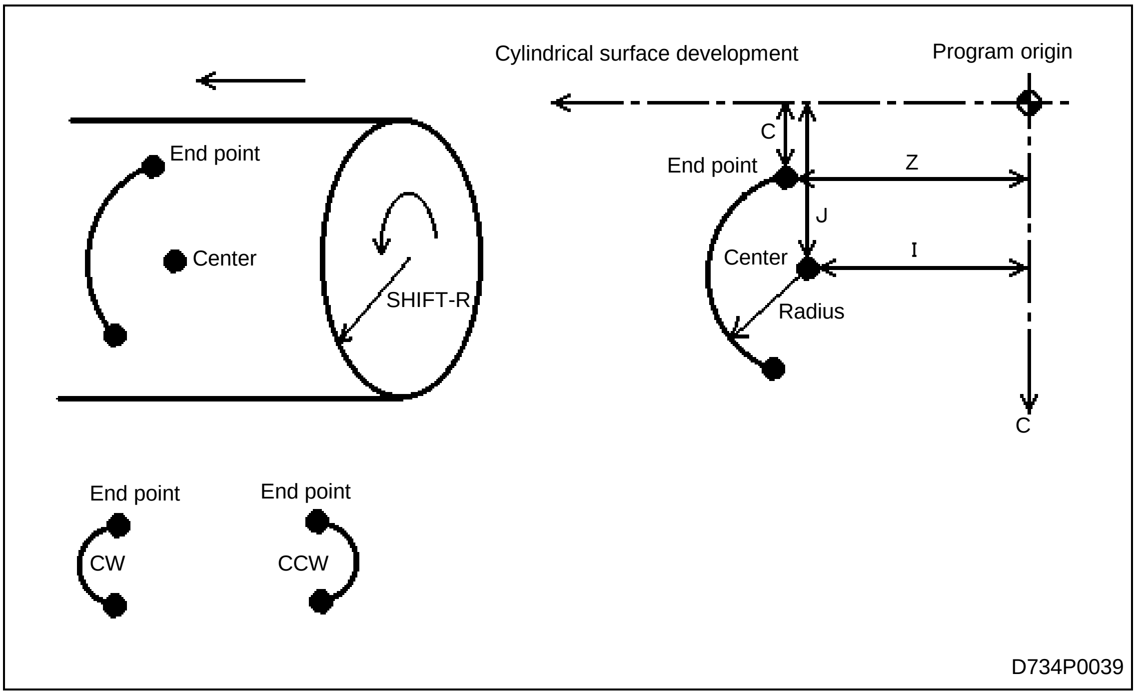
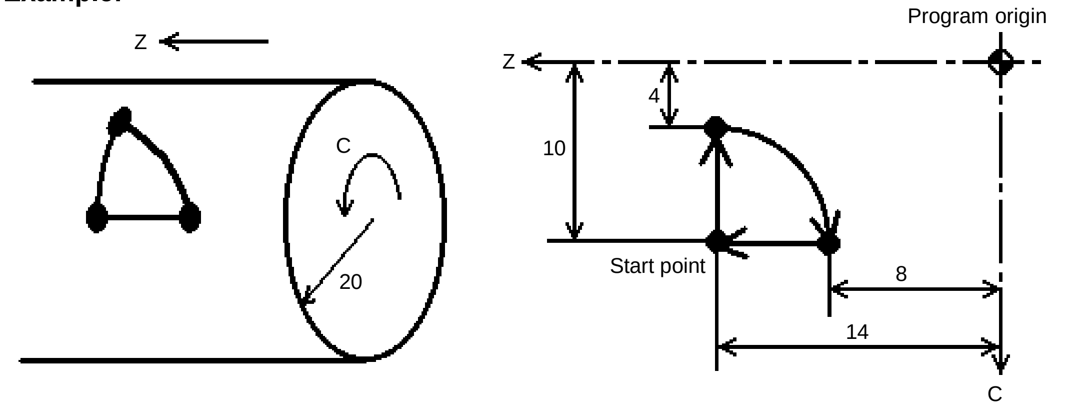


FIG	PTN	SHIFT-R	Z	C	R/th	I	J	P	CNR	R-FEED	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] 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:



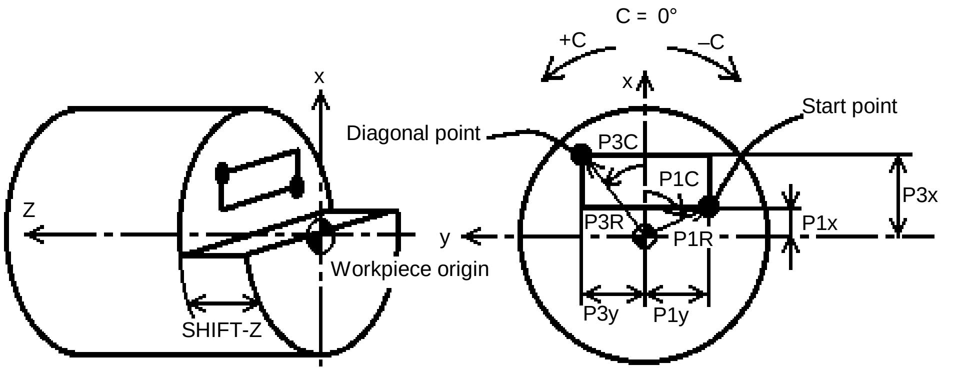
D734P0040

FIG	PTN	SHIFT-R	Z	C	R/th	I	J	P	CNR	R-FEED	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, $\boxed{XC}$, XY or $\boxed{XY}$

1. Fixed form

   - Square (**SQR**)



D734P0041

FIG	PTN	SHIFT-Z	P1Rx/CRx	P1Cy/CCy	P3Rx/R	P3Cy	CN1	CN2	CN3	CN4
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[6]	[6]	[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] CN1 - CN4	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

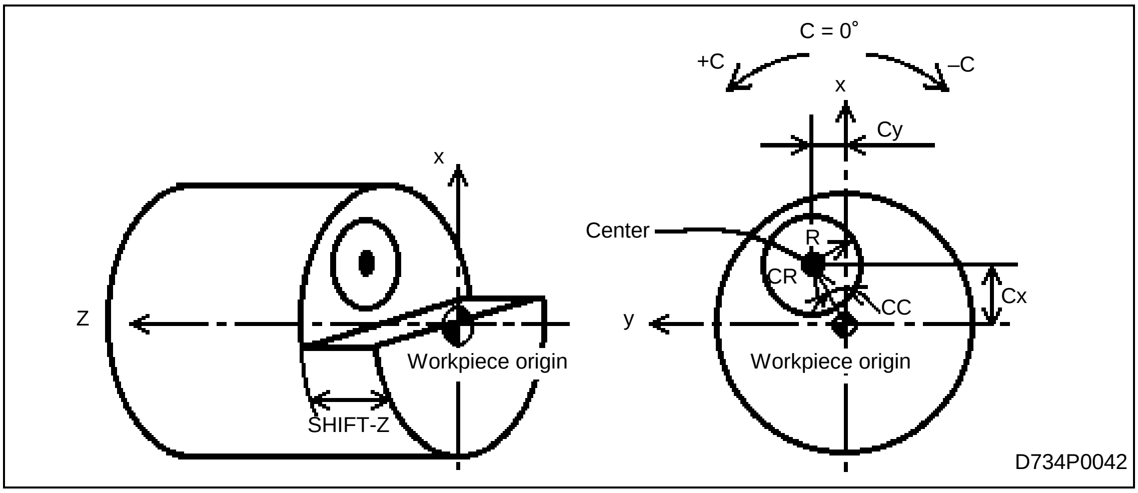


FIG	PTN	SHIFT-Z	P1Rx/CRx	P1Cy/CCy	P3Rx/R	P3Cy	CN1	CN2	CN3	CN4
1	CIR	[1]	[2]	[3]	[4]	◆	◆	◆	◆	◆

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**)

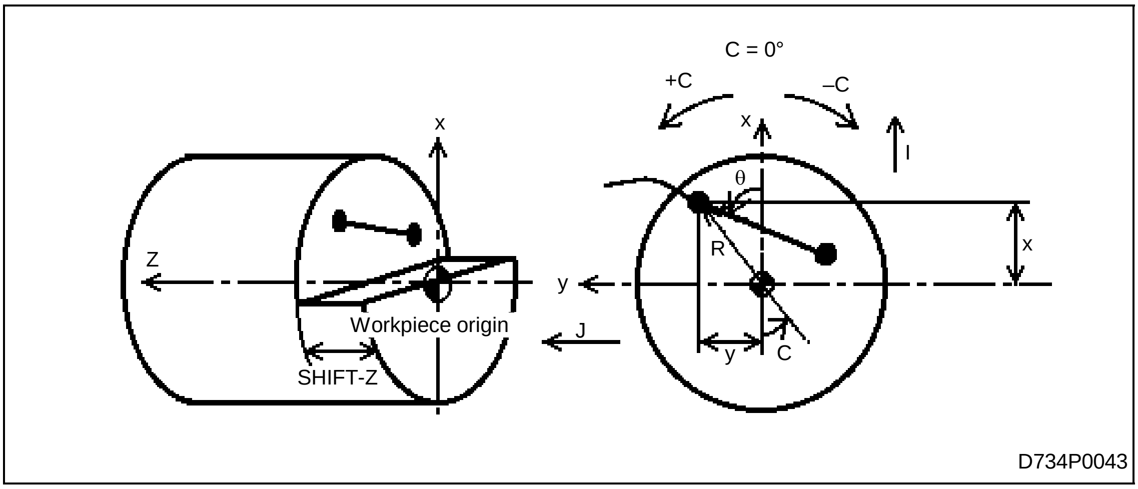


FIG	PTN	SHIFT-Z	R/x	C/y	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW**, **CCW**)

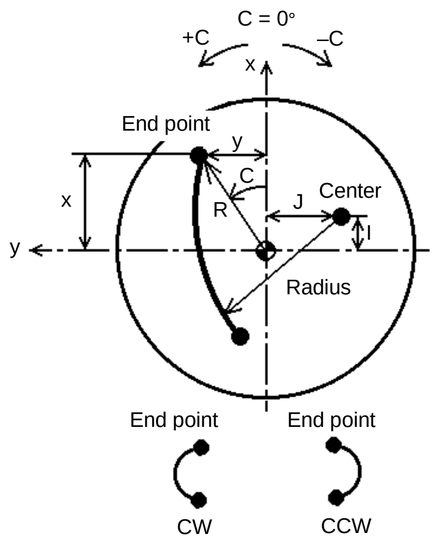
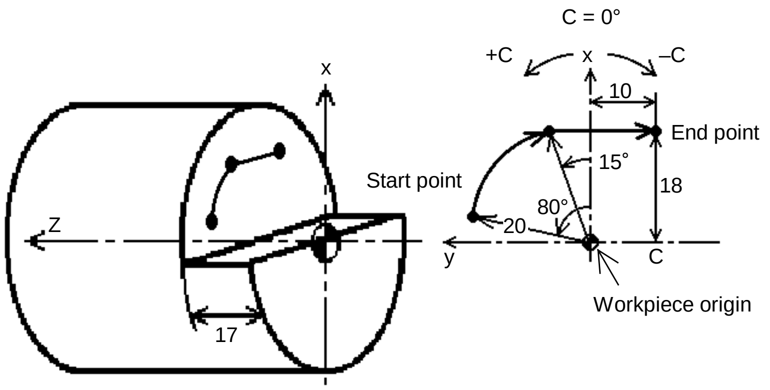


FIG	PTN	SHIFT-Z	R/x	C/y	R/th	I	J	P	CNR	R-FEED	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] 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:



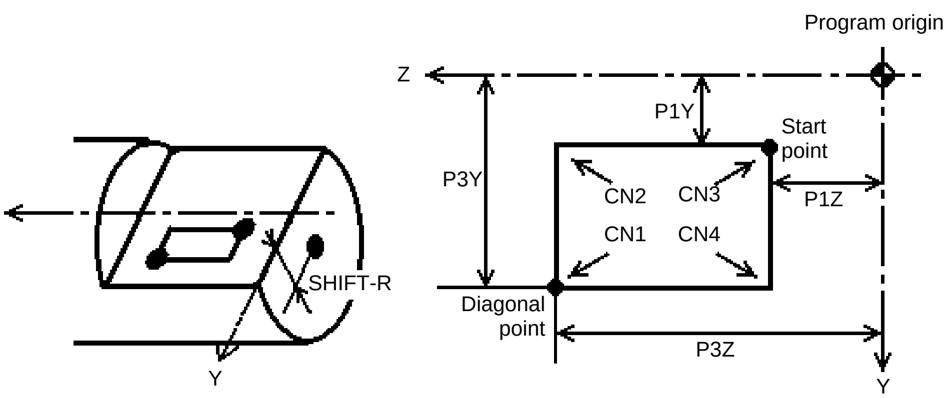
D734P0045

FIG	PTN	SHIFT-Z	R/x	C/y	R/th	I	J	P	CNR	R-FEED	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	PTN	SHIFT-R	P1Z/CZ	P1Y/CY	P3Z/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[6]	[6]	[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] CN1 - CN4	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)

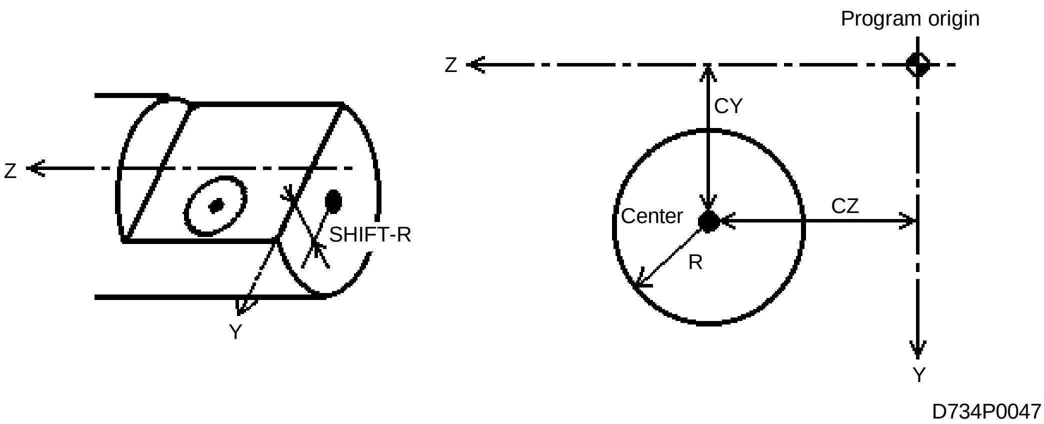
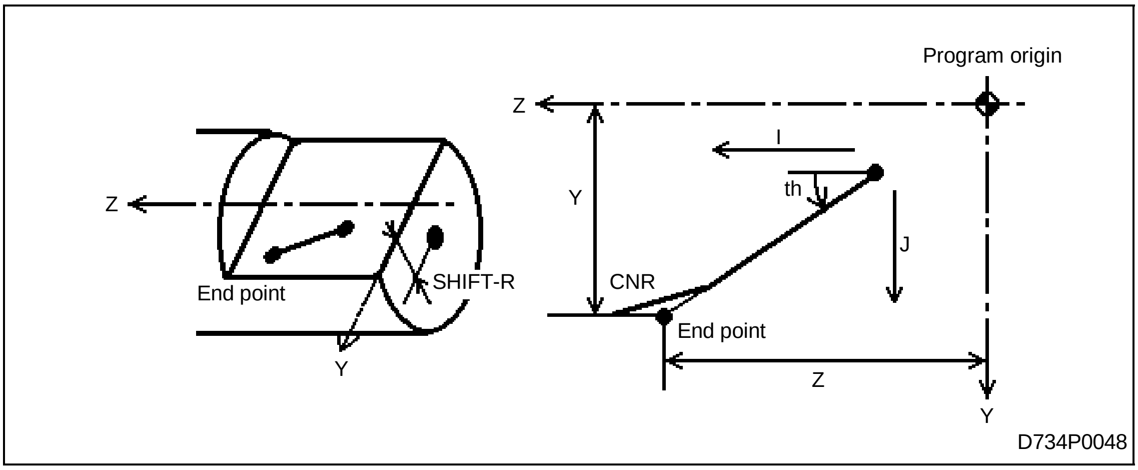


FIG	PTN	SHIFT-R	P1Z/CZ	P1Y/CY	P3Y/R	P3Y	CN1	CN2	CN3	CN4
2	CIR	[1]	[2]	[3]	[4]	◆	◆	◆	◆	◆

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   PTN    SHIFT-R    Z      Y      R/th     I    J    P    CNR   R-FEED   RGH
 1    LINE     [1]     [2]    [3]     [4]     [5]  [6]  [7]   [8]     [9]    [10]
```

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW**, **CCW**)

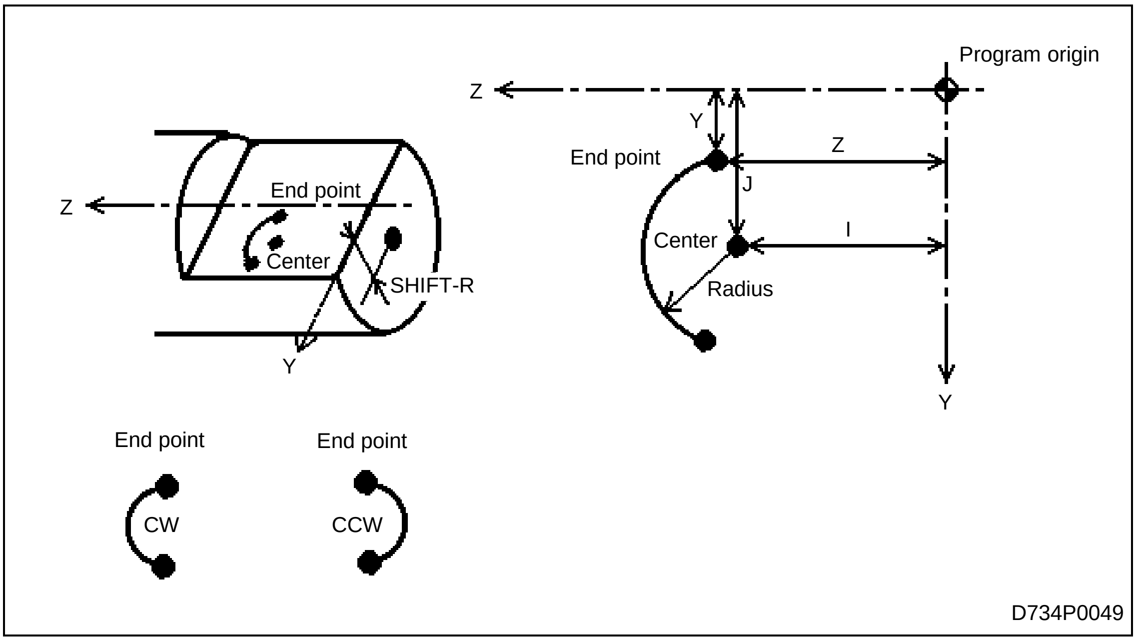
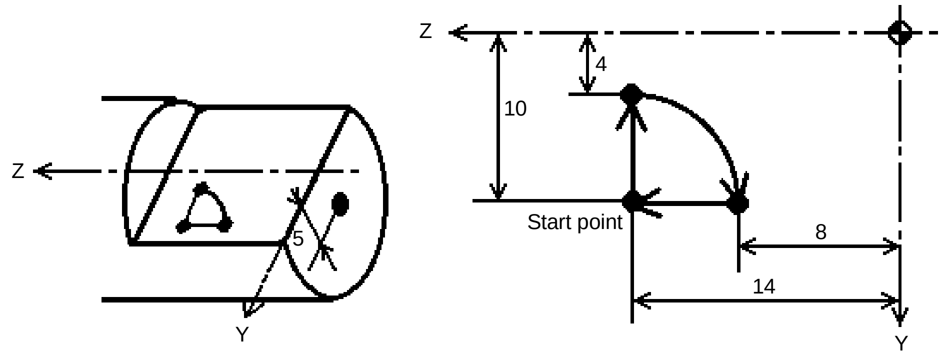


FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

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] R/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 “Automatic Crossing-Point Calculation Function” for details.
[8] CNR	Specify a machining form at the corner of the end point. See Remark 2 for further details.
[9] R-FEED	Specify the roughing feedrate.
[10] 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:



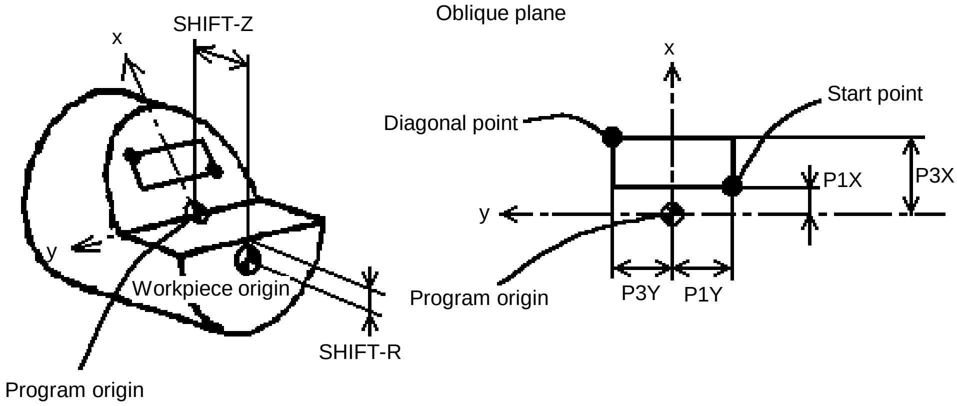
D734P0050

FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	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 /Y

1. Fixed form

- Square (**SQR**)

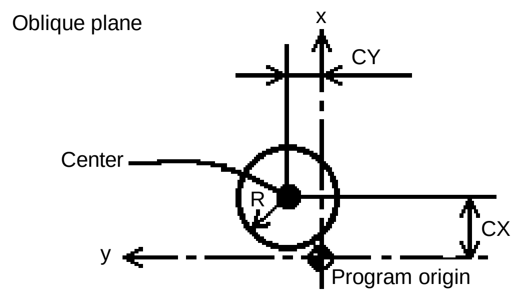


D734P0051

FIG	PTN	SHIFT-Z	SHIFT-R	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[7]	[7]	[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] CN1 - CN4	Specify a machining form at four corners. See Remark 2 for further details.

- Circle (**CIR**)



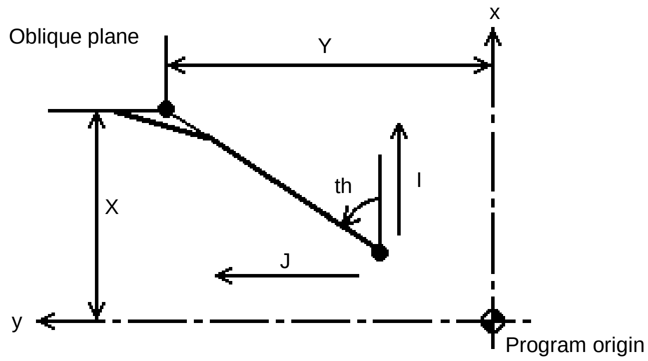
D734P0052

FIG	PTN	SHIFT-R	SHIFT-R	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
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	PTN	SHIFT-Z	SHIFT-R	X	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]	[11]

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] R/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] R-FEED	Specify the roughing feedrate.
[11] RGH	Specify the finishing feedrate according to the particular roughness of the surface. See Remark 1 for further details.

- Arc (**CW**, **CCW**)

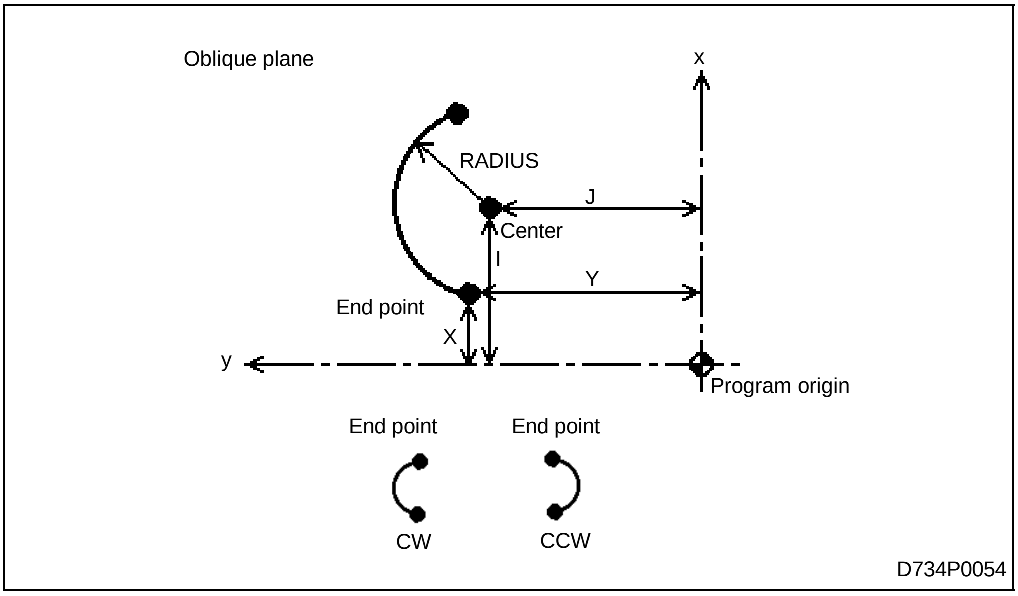
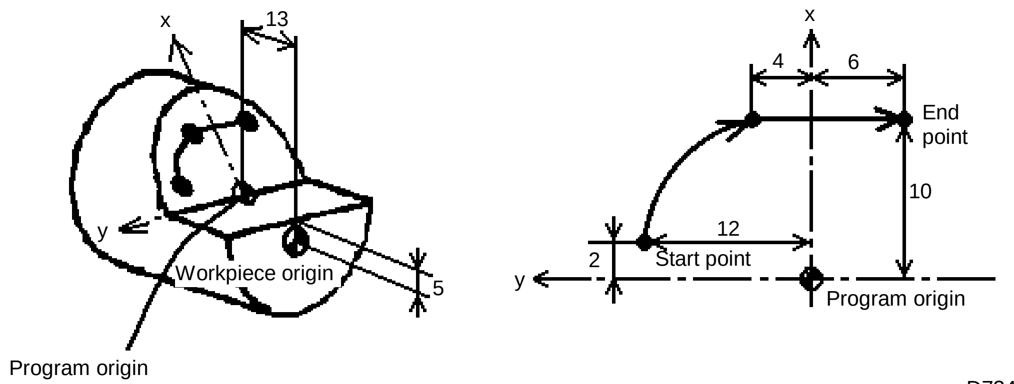


FIG	PTN	SHIFT-Z	SHIFT-R	X	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	CW	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]	[11]

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] R/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] R-FEED	Specify the roughing feedrate.
[11] 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:



D734P0055

FIG	PTN	SHIFT-Z	SHIFT-R	X	Y	R/th	I	J	P	R-FEED	CNR	RGH
1	LINE	13.	5.	2.	12.							
2	CW	◆	◆	10.	4.	8.	2	4.				
3	LINE	◆	◆	10.	-6.	90.	.					

Remark 1: 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 < **E27**............ $Ff_1 = \mathbf{E28} \times \frac{D}{\mathbf{E27}} \times Kf \times Z$

  In case of D ≥ **E27**............ $Ff_1 = \mathbf{E28} \times Kf \times Z$

E27	:	Parameter used to set a reference diameter for the feedrate of finishing during milling
Ff_1	:	Radial-direction finishing feedrate
E28	:	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.

$$Ff_2 = Ff_1 \times \frac{\mathbf{E26}}{100}$$

Ff_2 : Axial-direction finishing feedrate
E26 : 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.

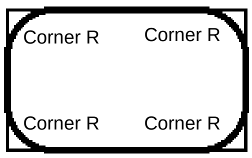
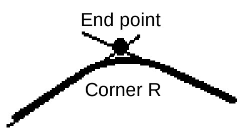
CAUTION

- 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.

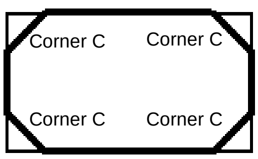
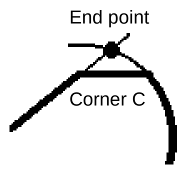
Remark 2: Corner

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.

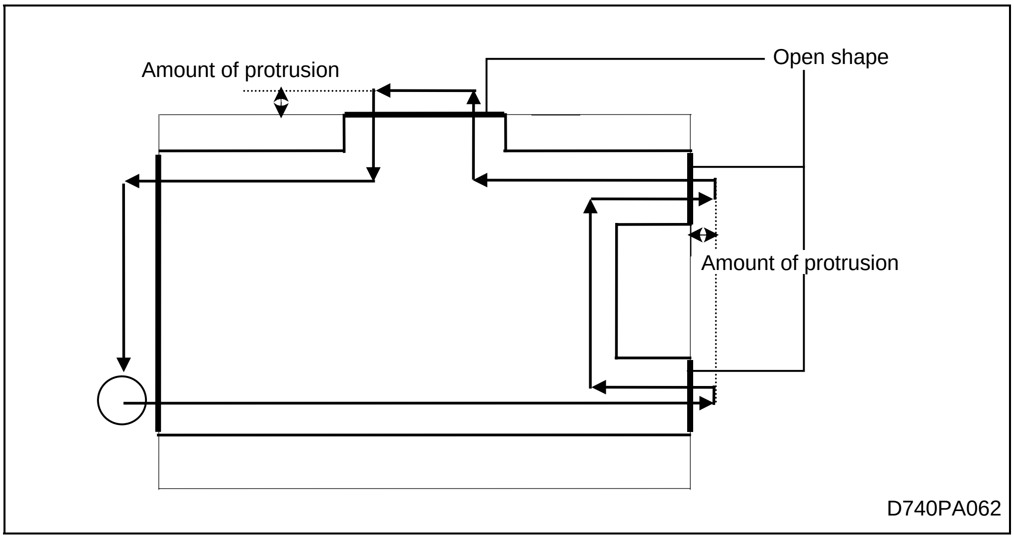
[Fixed form] corner C of the square	[Arbitrary form] corner C of the end point
	

- 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.

Remark 3: Shape of open pocket

For pocket milling units, pocket milling-mountain units and pocket milling-valley units, an open attribute can be specified for each side of any shape.

FIG	PTN	SHIFT-R	X	Y	R/th	I	J	P	CNR	ATTRIB	FR
1	LINE									OPEN	



The amount of protrusion is automatically determined by a parameter, as described below.

$$\text{Amount of protrusion} = \text{Tool diameter} \times \frac{\mathbf{E31}}{10}$$

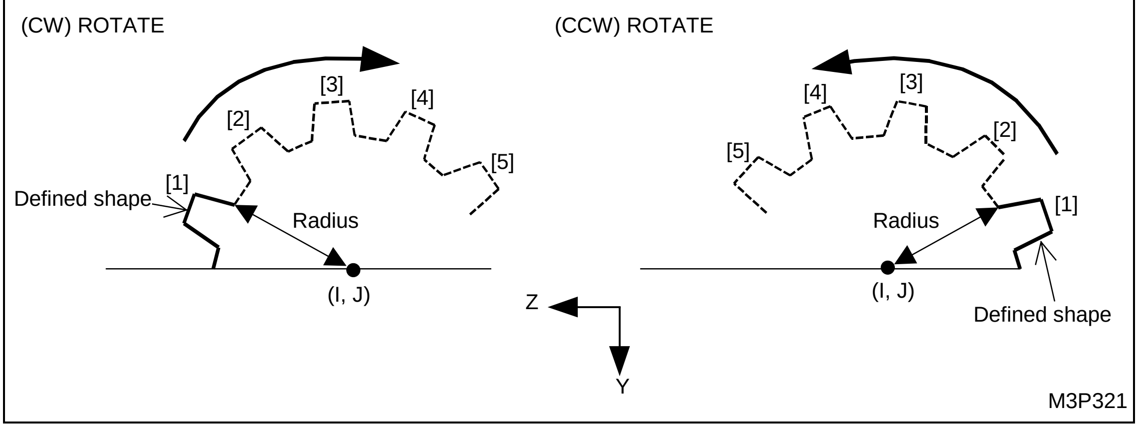
4. Shape rotation and shifting

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 **[CW SHIFT]** or **[CCW SHIFT]** menu key.

2. Data setting in shape sequence **CW**/**CCW-SH** (see figure above)

FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	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] R/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.

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

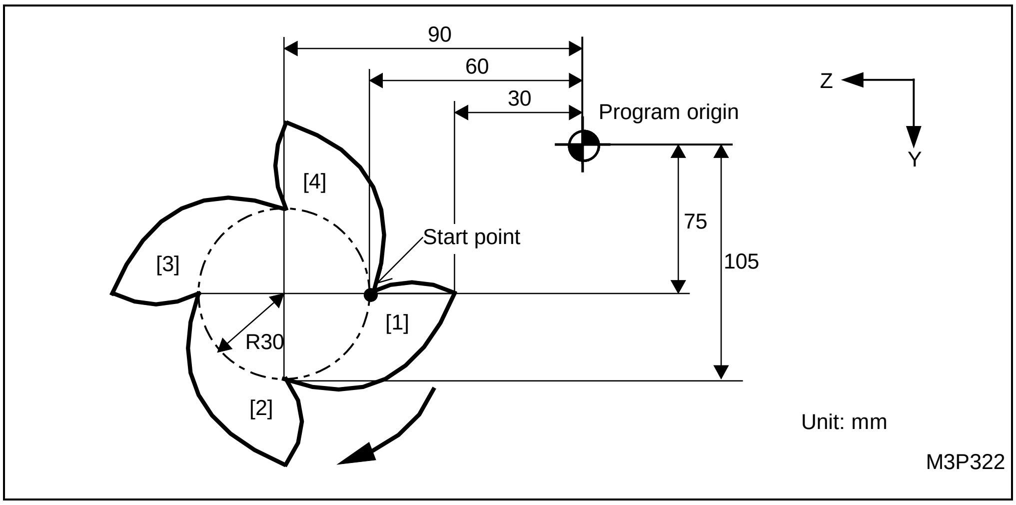
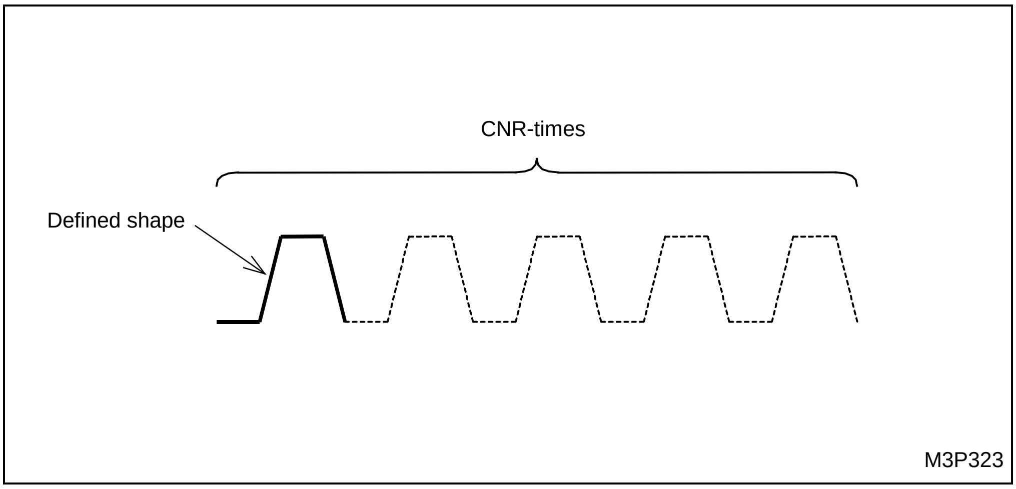


FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	CW-SH	10.	◆	◆	30.	90.	75.	◆	4	◆	◆
2	LINE	◆	60.	75.							
3	CW	◆	30.	75.	100.						
4	CW	◆	90.	105.	50.						
5	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆	◆

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	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	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 **[REPEAT END]** menu key and a shape sequence of **FIG-SH** will be brought to the end.

Example:

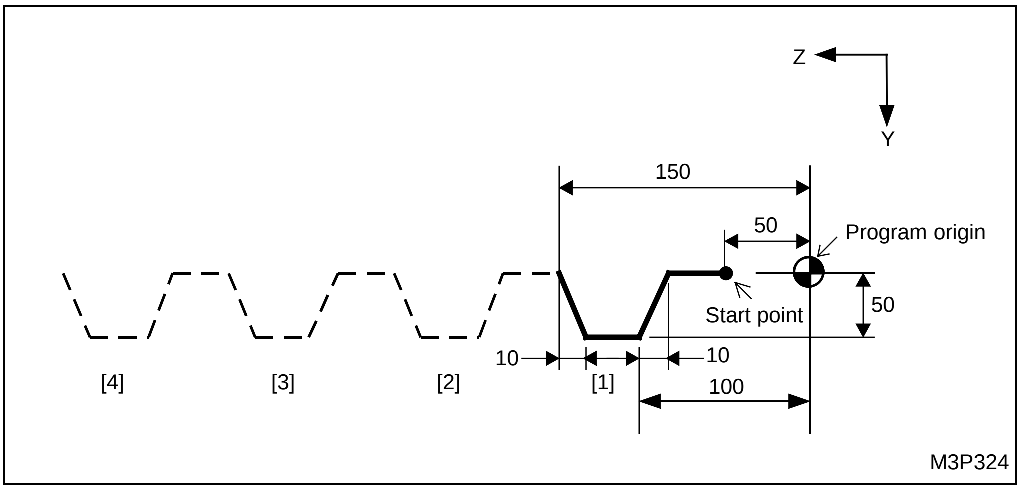


FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	FIG-SH	10.	◆	◆	◆	◆	◆	◆	4	◆	◆
2	LINE	◆	50.	0.							
3	LINE	◆	90.	0.							
4	LINE	◆	100.	50.							
5	LINE	◆	140.	50.							
6	LINE	◆	150.	0.							
7	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆	◆	◆

3-8 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.

3-8-1 Types of turning units

Nine 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)
- Mill-turning unit (MILLTURN)

3-8-2 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[TURNING MACH-ING]** menu key.

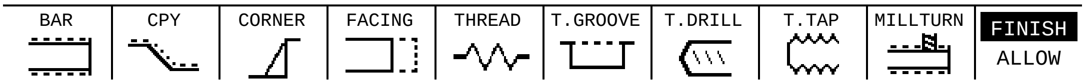
➔ The following menu is displayed.

BAR	CPY	CORNER	FACING	THREAD	T.GROOVE	T.DRILL	T.TAP	MILLTURN	FINISH ALLOW

(3) Press the menu key corresponding to the desired machining unit.

Notes on the menu option **[FINISH ALLOW]**:

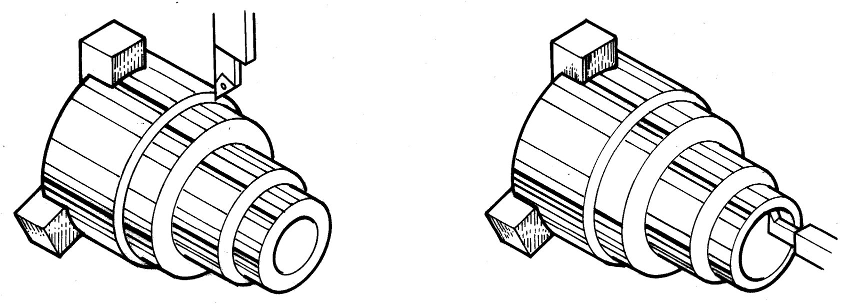
When the **[FINISH ALLOW]** menu item is selected from the turning unit selection menu and then a turning unit is created with the **[FINISH ALLOW]** menu item highlighted as follows, if another turning unit already exists in front of that created unit, the operator can automatically set the same values as the finishing allowances specified in the existing turning unit.



- 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.

3-9 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.

3-9-1 Setting unit data

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
*	BAR	[1]	[2]	[3]	[4]	[5]	[6]

[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

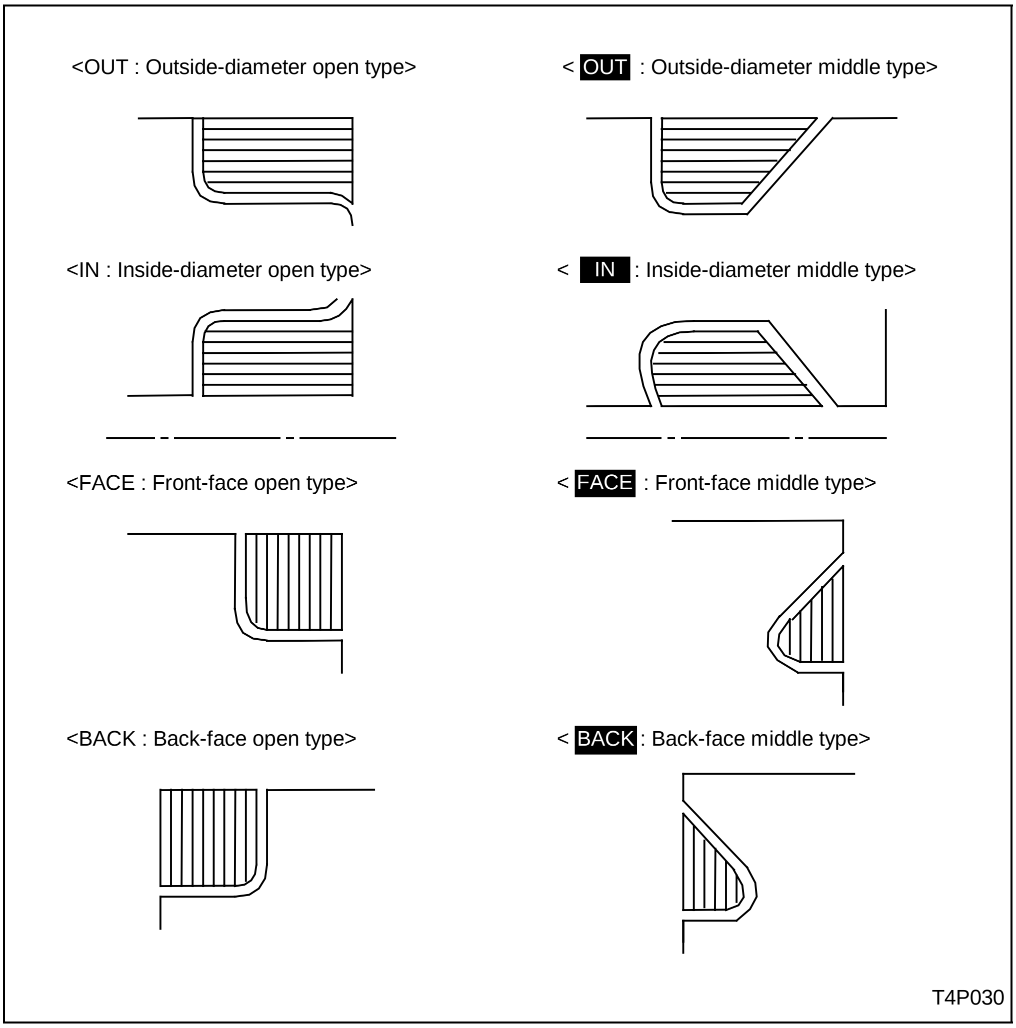
IN : Inner periphery (Cutting from the middle of the inner periphery)
.........................Inside-diameter middle type

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] POS-B

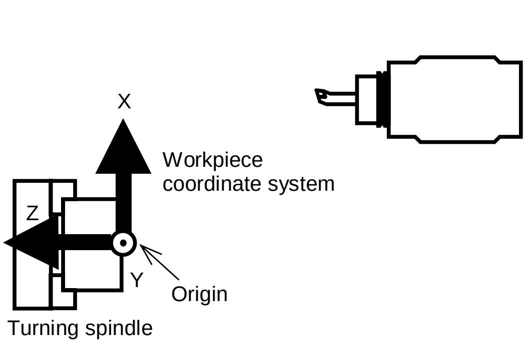
When the cursor is on this item, the following menu is displayed.

←	↓	→							

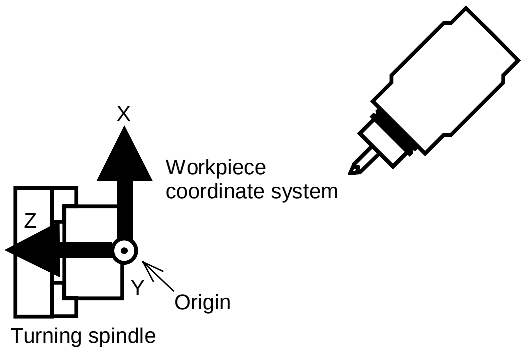
From this menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys.

Note: If the program for the M640 series has been loaded, this section is left blank, so you need to add angles on your own.

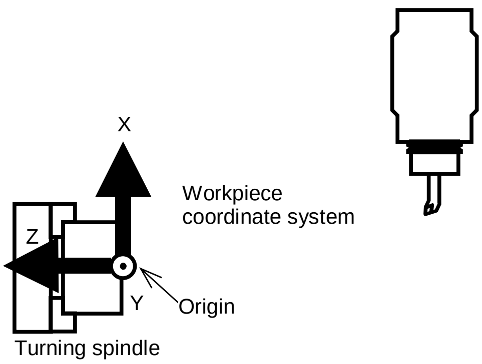
POS-B = 0



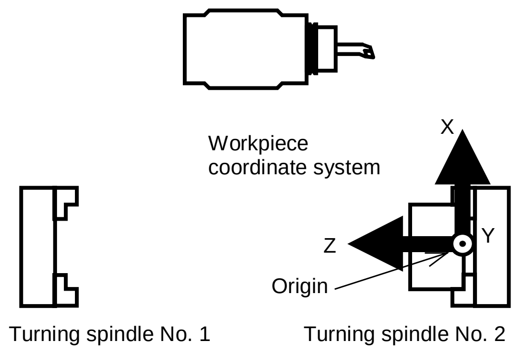
POS-B is oblique



POS-B = 90



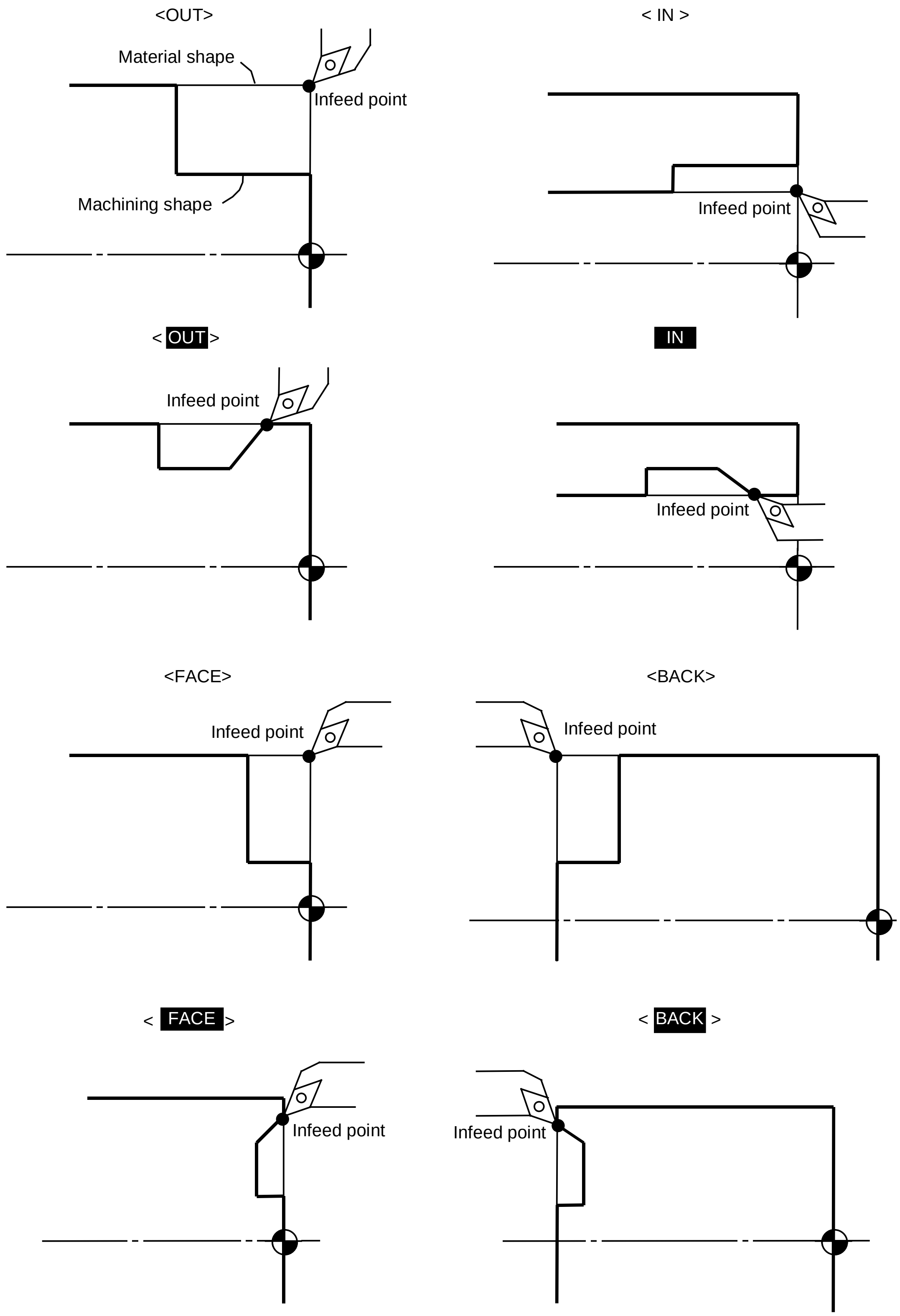
POS-B = 180 (Machine with the turning spindle No. 2)



Note: In the facing unit, the Z-axis faces in opposite direction of the above figure.

[3] CPT-X, [4] CPT-Z

Set the X- and Z-coordinates of the desired infeed point.

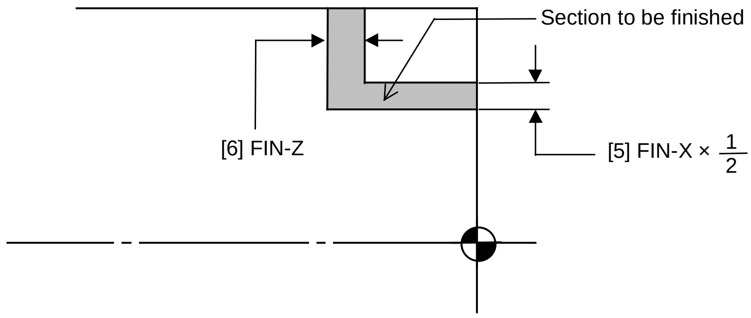


T4P036

- 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.

[5] FIN-X, [6] 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 3-8-2.

3-9-2 Setting tool sequence data

```
SNo.   TOOL    NOM.  No.  #   PAT.  DEP-1 DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP  FR   M    M    M
 R1                                                    ◆      ◆      ◆
 F2                            ◆     ◆       ◆         ◆
       ↑     ↑   ↑  ↑  ↑   ↑    ↑    ↑     ↑       ↑              ↑      ↑      ↑     ↑    ↑    ↑    ↑
      [1]   [2] [3] [4] [5] [6] [7]  [8]   [9]     [10]           [11]   [12]   [13]  [14] [15] [15] [15]
```

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		SIMUL DRILL ROTATION	

[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 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 “Suffix” 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. (Suffix)

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	HEAVY TOOL	>>>

[5] NOM. (Turret selection)

For the machine with the lower turret, select the turret in which the tool to be used is mounted. The following menu is displayed (if **[SET UPPER TURRET]** is selected, the column will remain blank, and if **[SET LOWER TURRET]** is selected, "▼" will be displayed). See Section 5, LOWER-TURRET CONTROL FUNCTIONS, for further details:

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. The following menu is displayed. A press of a menu key displays the menu item in reverse mode, allowing a priority number to be assigned.

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN			PRI.No. ALL ERAS	SUB PROG PROC END	
	(a)		(b)	(c)			(d)	(e)	

The function of menu item (a) to (e) is described below:

Menu item	Function
(a)	Select to conduct subsequent-machining.
(b)	Select to change the priority number for the tool within the particular process. If the cursor is present at a blank space, assign a new number in a usual manner. Entry of an existing priority number displays alarm **420 SAME DATA EXISTS**.
(c)	Select to assign a priority number to the tool to be used repeatedly in the particular process. Alarm **420 SAME DATA EXISTS** will be displayed if the assigned priority number has already been set on any other unit line.
(d)	Selection of this item displays message **ALL ERASE (PROC:0, PROG:1)?**. Setting 0 will erase the priority numbers preassigned to the tool to be used repeatedly in the process. Setting 1 will erase the priority numbers preassigned to the tool to be used repeatedly in the program.
(e)	Select to terminate the process with the subprogram unit.

[7] # (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.
For details see Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS".

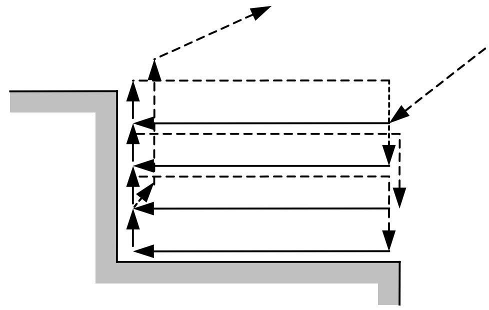
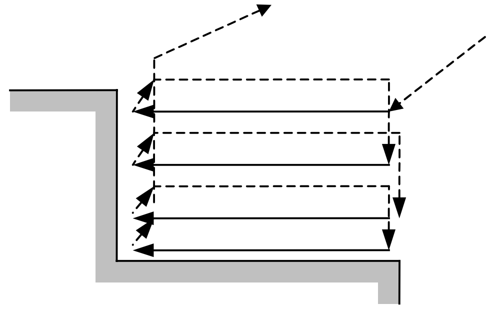
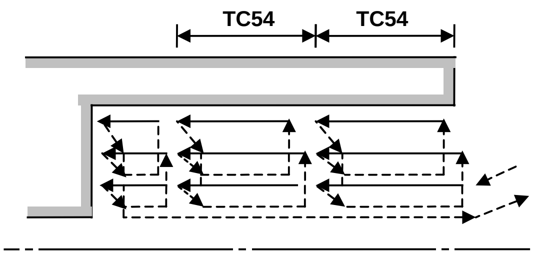
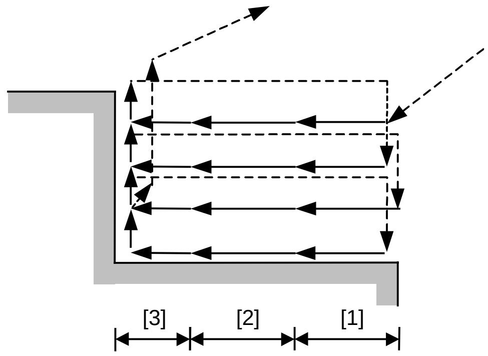
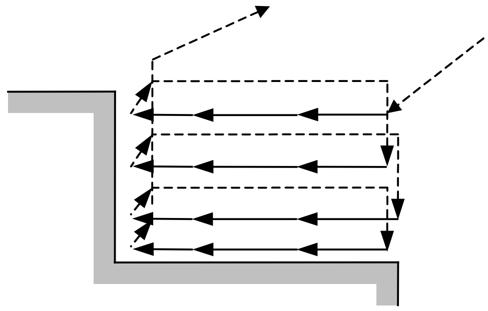
	BALANCE FEED 2								LOWER TURRET ESCAPE

[8] PAT. (Machining pattern)

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

#0 	#1 	#2 	#3 	#4 					
(a)	(b)	(c)	(d)	(e)					

Select the machining pattern from (a) through (e) above. The data of the displayed menu denote the following machining patterns:

(a) #0: Perpendicularly upward cutting by each cutting pass	(b) #1: Diagonally upward cutting by each cutting pass (High-speed rough-machining cycle)
	
(c) #2: Gradual inside-diameter machining from the open end of the workpiece (Inside-diameter deep-hole dividing cycle)	(d) #3: Perpendicularly upward roughing cycle + Chip cutting cycle
	 [1], [2] .....Length of cutting per pass [3] ....Last travel to the end point
(e) #4: Diagonally upward rapid roughing cycle + Chip cutting cycle	
	Rapid feed Cutting feed

#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. Alarm **719 REVERSE SHAPE CONTOUR** will occur in such cases.
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 **TC54** to specify the depth of cut per pass.

#3 and #4 machining can be used only when **OUT** or **IN** is selected for item [1] of the unit.
During outside- or inside-diameter machining, chips may wind around the workpiece, acting as resistance to cutting and preventing the workpiece from being cut properly, depending on the material and the machining conditions. If machining pattern #3 or #4 is selected, the feeding of the tool is stopped halfway through the cutting path and restarted after the spindle has made the number of revolutions specified with the parameter **TC71** in order to cut off chips and prevent them from winding around the workpiece.

Note: Expected results may not be produced under specific cutting conditions.

[9] 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 [9] **DEP-1**, [13] **C-SP**, and [14] **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 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.

[10] DEP-2 (NUM.) (Length of cutting per pass)

Specify a cutting length in the Z-axis direction at intervals of which the feeding of the tool is stopped temporarily during rough machining.

To specify a cutting length, you need to select #3 or #4 in advance for [8] **PAT.** When #0, #1 or #2 is selected, ◆ is displayed in this section and no data can be specified.

Note: The length of cutting per pass needs to be specified by entering the travel in the Z-axis direction. The feeding of the tool is not stopped temporarily for machining only in the Z-axis direction. For sections to be tapered and corners to be chamfered or rounded, the length of cutting per pass also need to be specified as travel in the Z-axis direction.

[11] FIN-X, [12] 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 7-3 "Line 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.

[13] C-SP

Specify the surface speed for the turning spindle.
This surface speed, as with item [9] **DEP-1** (Maximum cutting depth), can be selected from the menu or entered using the numeric keys.

[14] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution. Use the numeric keys to enter the value.
For the roughing tool sequence, this feedrate, as with items [9] **DEP-1** (Maximum cutting depth) and [13] **C-SP**, can be selected from the menu or entered using the numeric keys.

[15] M

Specify the M-code to be issued for the tool immediately after it is selected.

Select the desired code from the menu or enter the desired code using the numeric keys.

3-9-3 Setting shape sequence data

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th	RGH
	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] PTN

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)

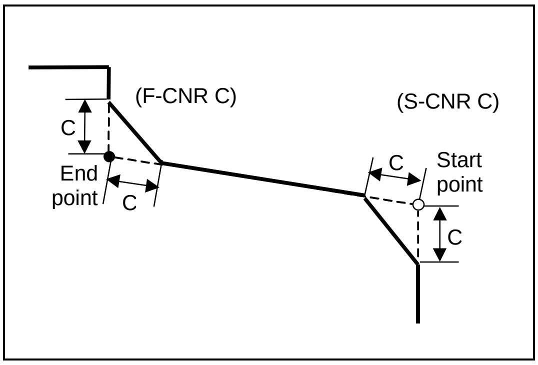
Select the type of machining shape pattern from the above four types (a) to (d).

Menu item	Function
(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.

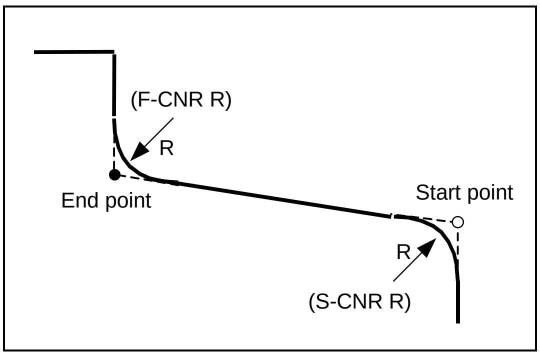
[2] S-CNR

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).

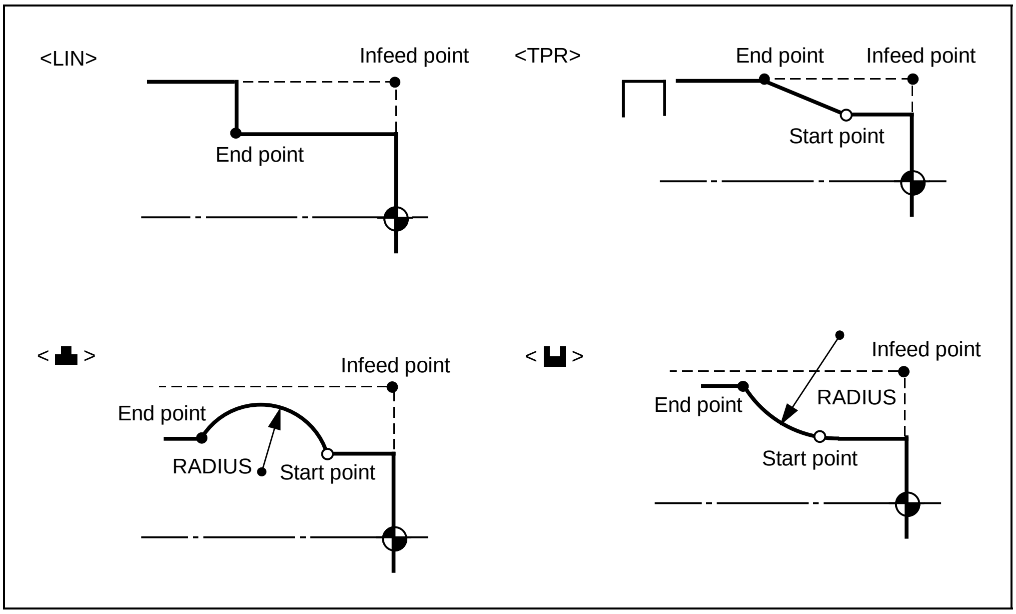


- If R-chamfering is to be done:
  After pressing the **[CORNER R]** menu key, set the radius of rounding (R in the diagram).

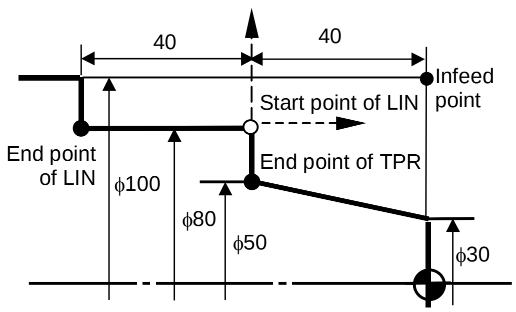


[3] SPT-X, [4] SPT-Z, [5] FPT-X, [6] FPT-Z

- 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	PTN	SPT-X	SPT-Z	FPT-X	FPT-Z
	TPR	30.	0.	50.	40.
	LIN	◆	◆	80.	80.

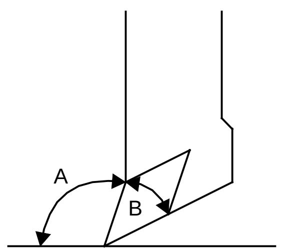
[7] F-CNR/$

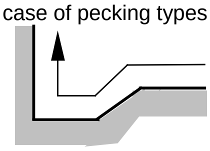
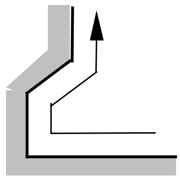
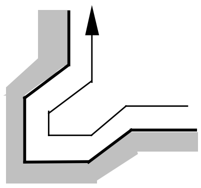
Set data for this item when C-chamfering, R-chamfering, or pecking is to be done at the end point of the shape.

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 **TC27** to **TC34** to select the dimensions of pecking. (See the separate Parameter List/Alarm List/M-Code 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] R/th

- If you selected  or  for item **PTN** [1] above, set the radius of the desired circle (see the diagram shown previously for items [3] to [6]).
- If you selected **TPR** for item [1] above and typed the question mark "?" for one of the four items from [3] to [6], set a taper angle.
  See "Automatic Crossing-Point Calculation Function" 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 numeric 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 [14] **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 [14] **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 [14] **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**.

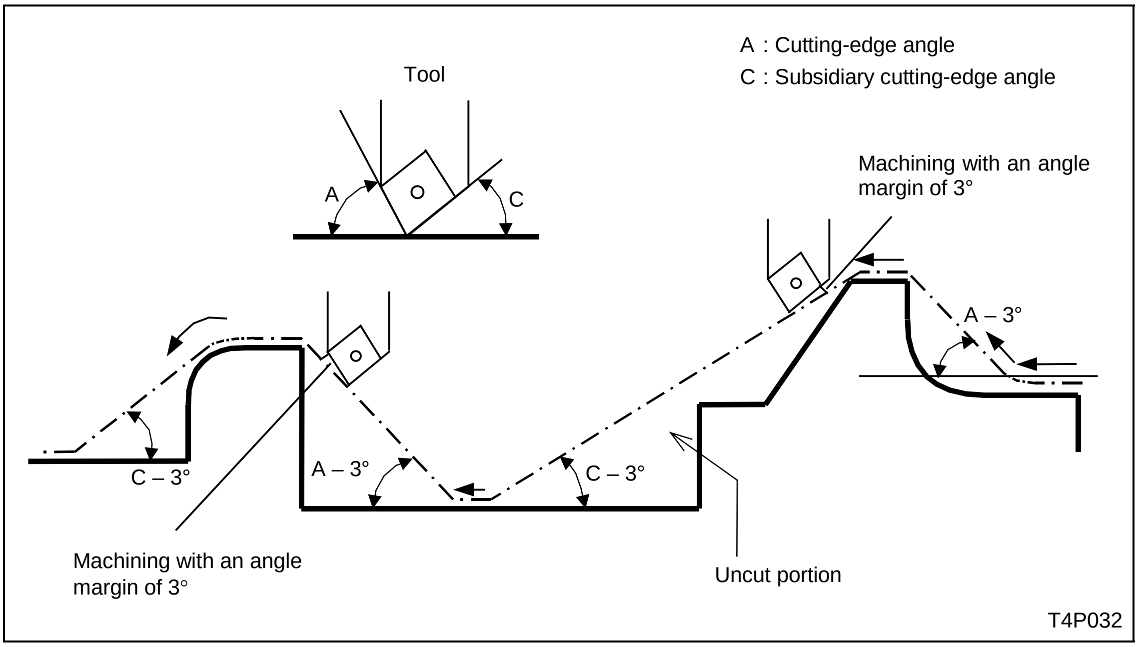
Note 5: Although a maximum of 200 lines of shape sequence data can be set in one turning unit, the maximum usable number of shape sequence data lines may be less than 200 when corner R/C is defined for a complex shape. In that case, alarm **723 EXCEEDS NUMBER OF SHAPES** will be displayed, even before the maximum usable number of shape sequence data lines is reached. This maximum value applies only to BAR, CPY, and THREAD units of all turning units. One shape sequence data line only can be entered for other turning units, namely, FACING, CORNER, T. GROOVE, T. DRILL, and T. TAP.

Note 6: If the maximum usable number of shape sequence data lines is exceeded, alarm **723 EXCEEDS NUMBER OF SHAPES** will be displayed during tool path checking, shape checking, shape drawing, or automatic operation.

<Precautions for BAR unit>

Some parts may remain uncut because of the tool shape.

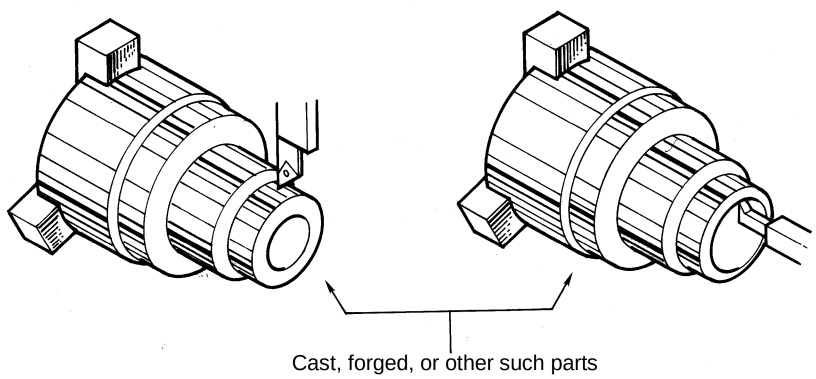
For a tool having a cutting-edge angle of A and a subsidiary cutting-edge angle of C, uncut portions occur at an angle of A – 3° in the machining direction and an angle of C – 3° in an opposite direction because machining will occur generally with an angle margin of 3°.



* The description given above also applies to CPY unit.

3-10 Copy-Machining Unit (CPY)

Select the copy-machining unit when cast, forged, or other such parts are to be cut along their profiles.



Press the **[CPY]** menu key to select this unit.

3-10-1 Setting unit data

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	SRV-X	SRV-Z	FIN-X	FIN-Z
*	CPY	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]

[1] PART

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

OUT	OUT	IN	IN	FACE	FACE	BACK	BACK		

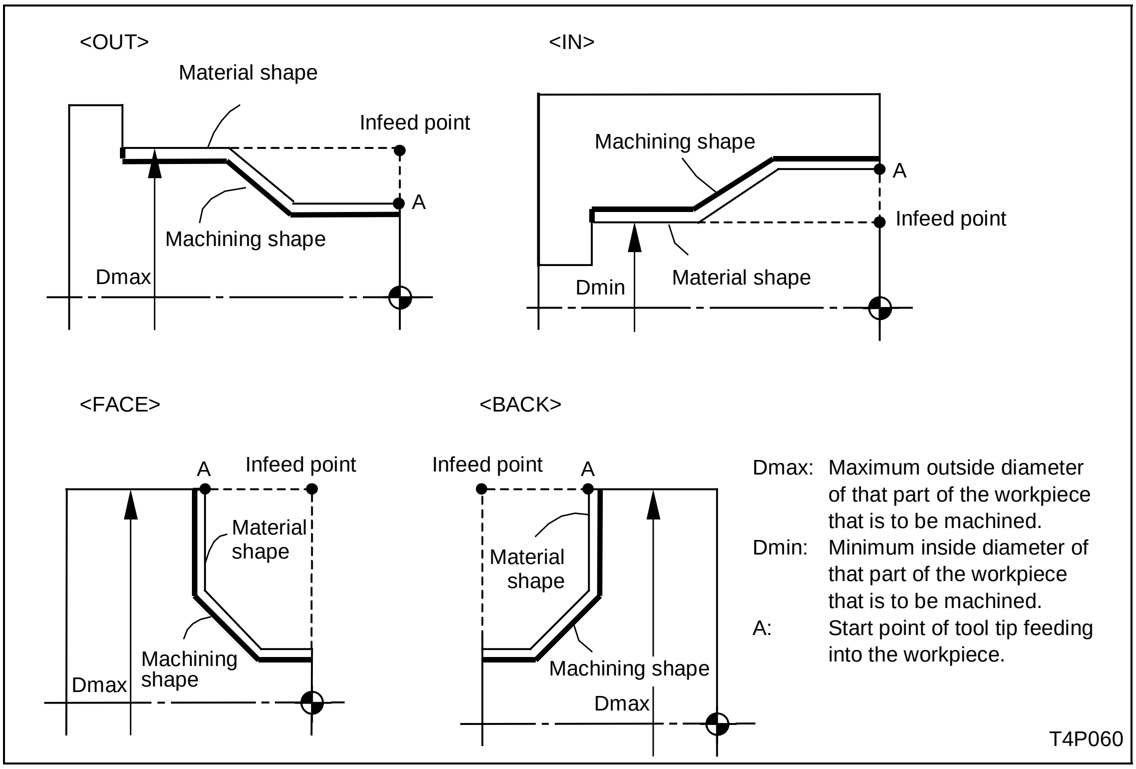
From the menu, select the section to be machined. The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[2] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

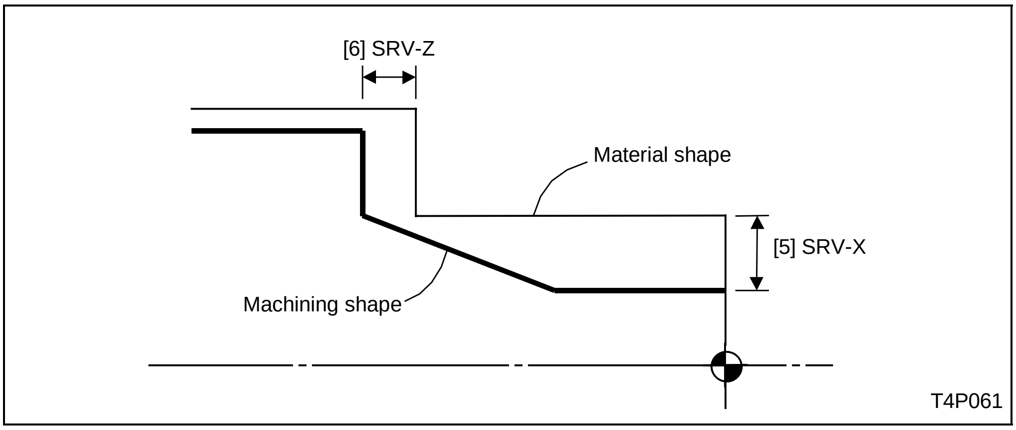
[3] CPT-X, [4] CPT-Z

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.



[5] SRV-X, [6] 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).



[7] FIN-X, [8] FIN-Z

Set the finishing allowances for the X-axis and Z-axis directions (removal allowances during finishing). See the description of the relevant items for BAR unit. See the description of the relevant items for BAR unit.

3-10-2 Setting tool sequence data

SNo.	TOOL		NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R1						◆		◆	◆	◆	◆					
F2						◆	◆	◆	◆							
	↑ [1]	↑ [2]	↑ [3] ↑ [4] ↑ [5]	↑ [6]	↑ [7]		↑ [8]			↑ [9]	↑ [10]	↑ [11]	↑ [12]	↑ [13]	↑ [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)	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		SIMUL DRILL ROTATION	

[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 keys.
See the description of the relevant items for BAR unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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								LOWER TURRET ESCAPE

Note: See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for details of items [5] and [7].

[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 surface 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 it is selected.
See the description of the relevant item for BAR unit.

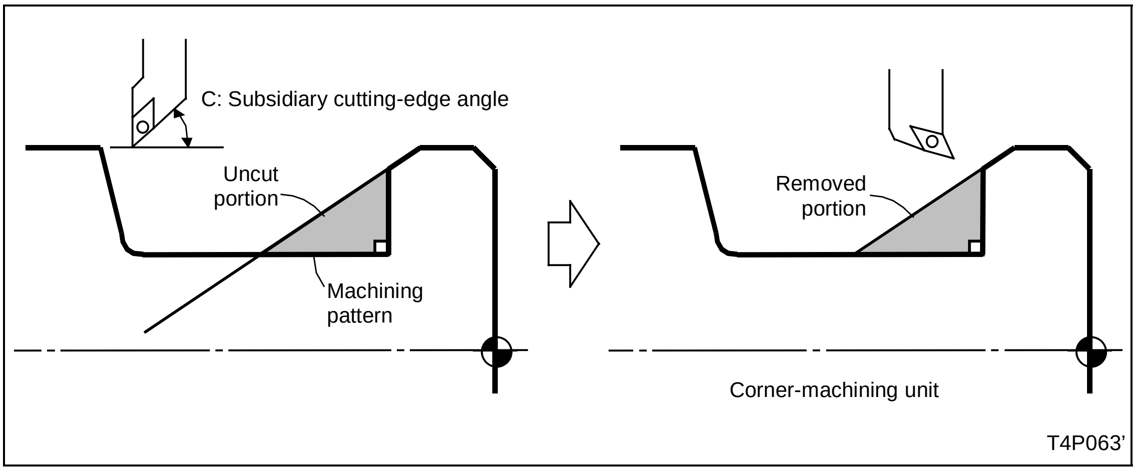
3-10-3 Setting shape sequence data

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/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 3-9 “Bar-Materials Machining Unit (BAR)”.

3-11 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.

3-11-1 Setting unit data

UNo.	UNIT	PART	POS-B	FIN-X	FIN-Z
*	CORNER	[1]	[2]	[3]	[4]

[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] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[3] FIN-X, [4] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

3-11-2 Setting tool sequence data

```
SNo.   TOOL    NOM.   No.  #  PAT. DEP-1 DEP-2/NUM.  DEP-3  FIN-X FIN-Z C-SP  FR   M   M   M
 R1                                         ◆          ◆      ◆     ◆
 F2                               ◆     ◆   ◆          ◆
       ↑    ↑   ↑  ↑  ↑   ↑    ↑    ↑    ↑                    ↑     ↑     ↑    ↑   ↑   ↑   ↑
      [1]  [2] [3][4][5] [6]  [7]  [8]  [9]                  [10]  [11]  [12] [13] [14][14][14]
```

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		SIMUL DRILL ROTATION	

[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 keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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								LOWER TURRET ESCAPE

Note: See Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [7].

[8] PAT. (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.

[9] 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.

[10] FIN-X, [11] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

[12] C-SP

Specify the surface speed for the turning spindle.
See the description of the relevant item for BAR unit.

[13] 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.

[14] M

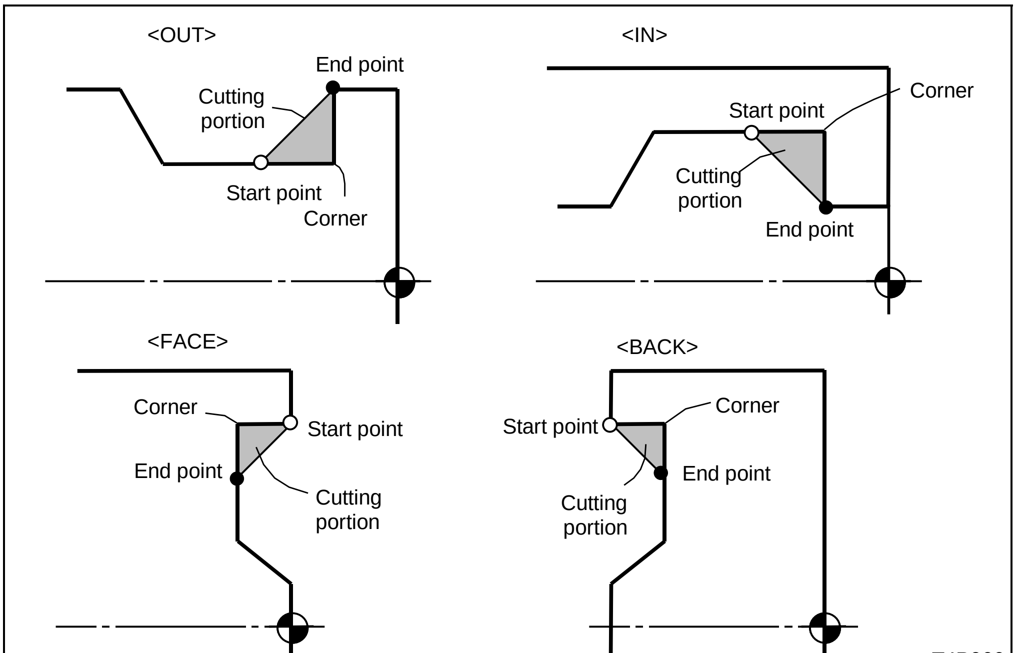
Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

3-11-3 Setting shape sequence data

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	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.



T4P066

[5] F-CNR/$

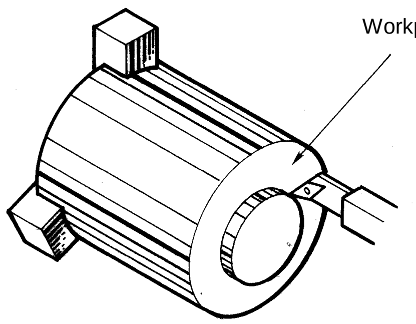
This item applies to the corners shown in the figure below, not the ending-point section. Enter data in this item to perform C-chamfering, R-chamfering, or polish-necking operations.
For data setting method, see the relevant items for BAR unit.

[6] RGH

Set the appropriate, finish-machining feedrate for particular finishing surface roughness.
For data setting method, see the relevant items for BAR unit.

3-12 Facing Unit (FACING)

Select the facing unit when chipping off any protrusions of the workpiece edges (front face or back face).



Press the **[FACING]** menu key to select this unit.

3-12-1 Setting unit data

UNo.	UNIT	PART	POS-B	FIN-Z
*	FACING	[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

[2] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[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.

3-12-2 Setting tool sequence data

```
SNo.   TOOL    NOM.   No.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR  M  M  M
 R1                            ◆           ◆          ◆     ◆     ◆
 F2                            ◆     ◆     ◆          ◆     ◆
       ↑    ↑    ↑  ↑  ↑   ↑    ↑        ↑                               ↑     ↑     ↑   ↑   ↑   ↑
      [1]  [2]  [3] [4] [5] [6]  [7]      [8]                             [9]   [10]  [11] [12] [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)	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		SIMUL DRILL ROTATION	

[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 keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for details of items [5] and [7].

[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-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 surface 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

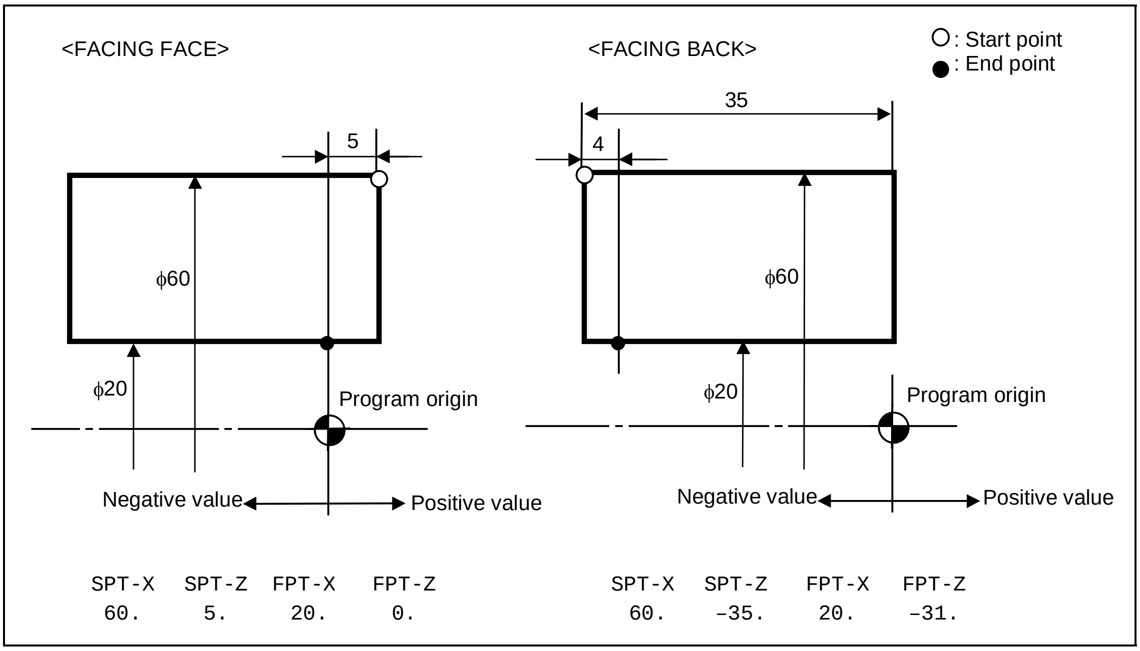
Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

3-12-3 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.

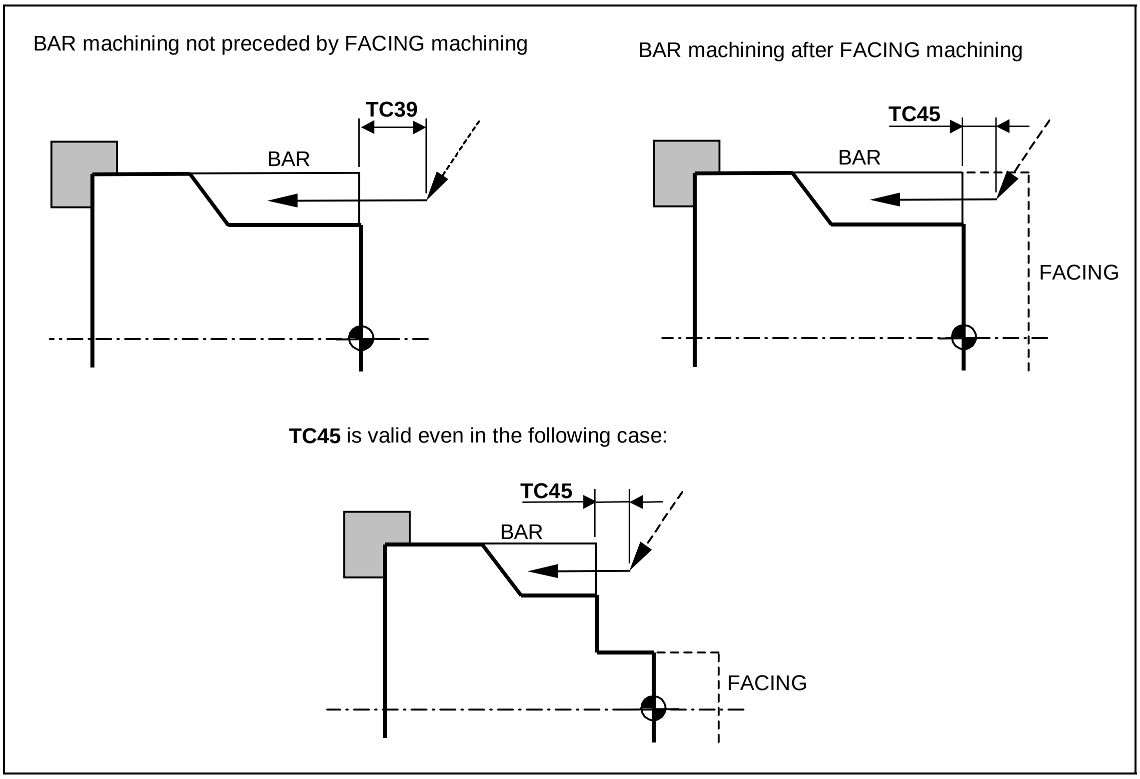


[5] RGH

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.

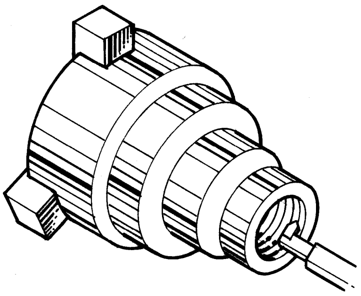
See related items of "Bar-materials machining unit (BAR)".

Note: The tool path for BAR and CPY units is calculated in general with the end-face clearance set in parameter **TC39**.
For a BAR or CPY unit preceded by a FACING unit, in particular, the value of parameter **TC45** is used as the end-face clearance.



3-13 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.

3-13-1 Setting unit data

UNo.	UNIT	PART	POS-B	CHAMF	LEAD	ANG	MULTI	HGT
*	THREAD	[1]	[2]	[3]	[4]	[5]	[6]	[7]

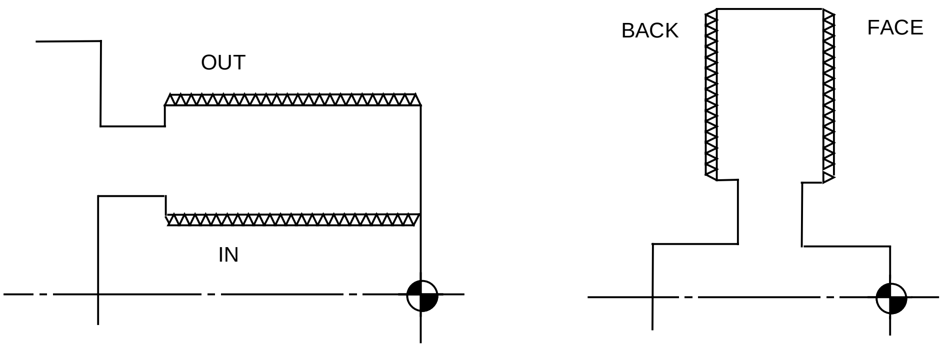
[1] PART

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

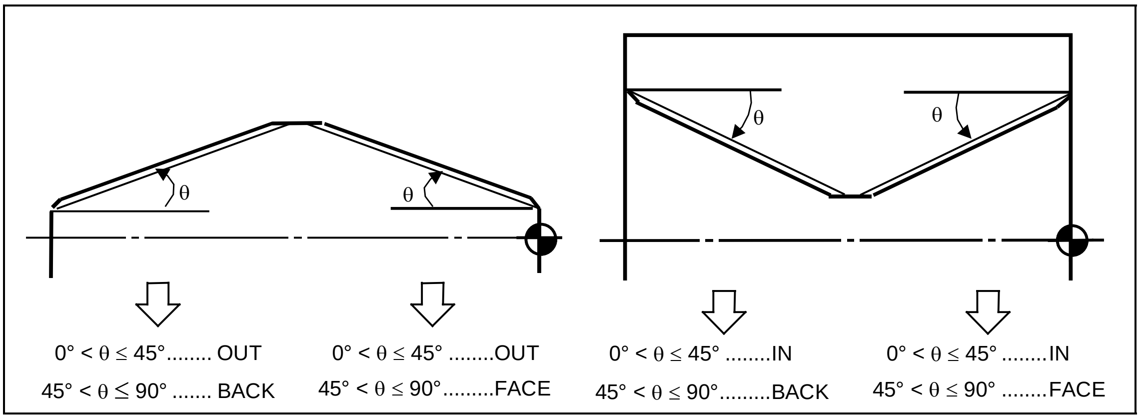
OUT	IN	FACE	BACK						

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] POS-B

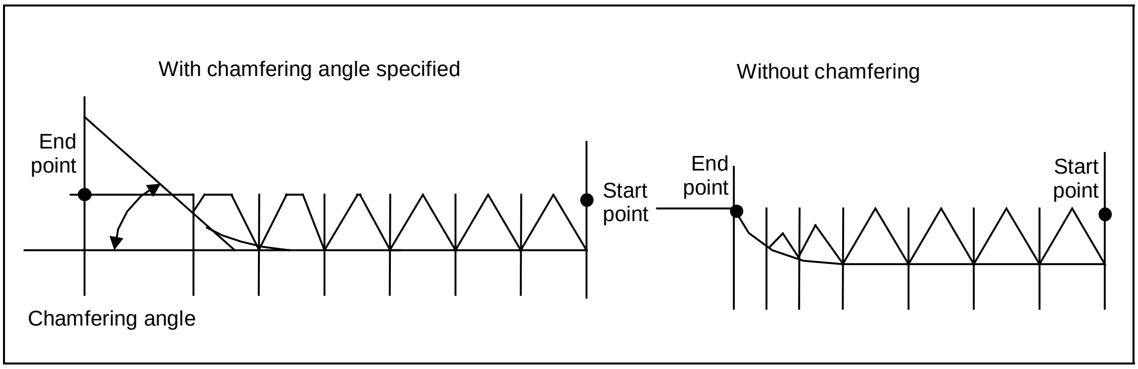
From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[3] CHAMF

Set a chamfering angle for the section you want to thread.

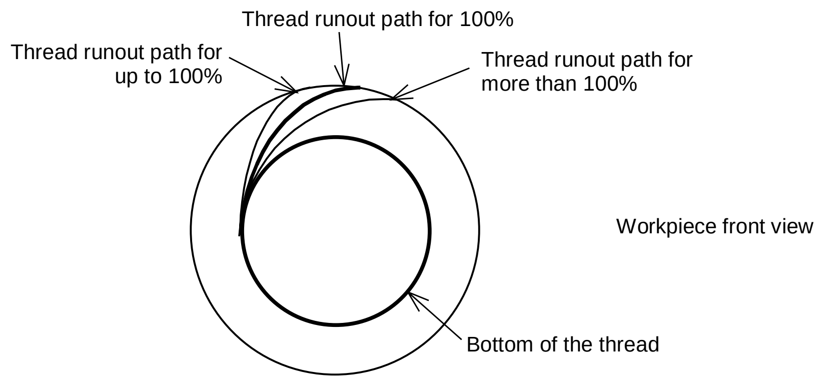
- Set 0 if chamfering is not required.
- Set 1 for a chamfering angle of 45 degrees.
- Set 2 for a chamfering angle of 60 degrees.

Designate chamfering to maintain the lead up to the ending point of threading.



Use the parameter **TC82** 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

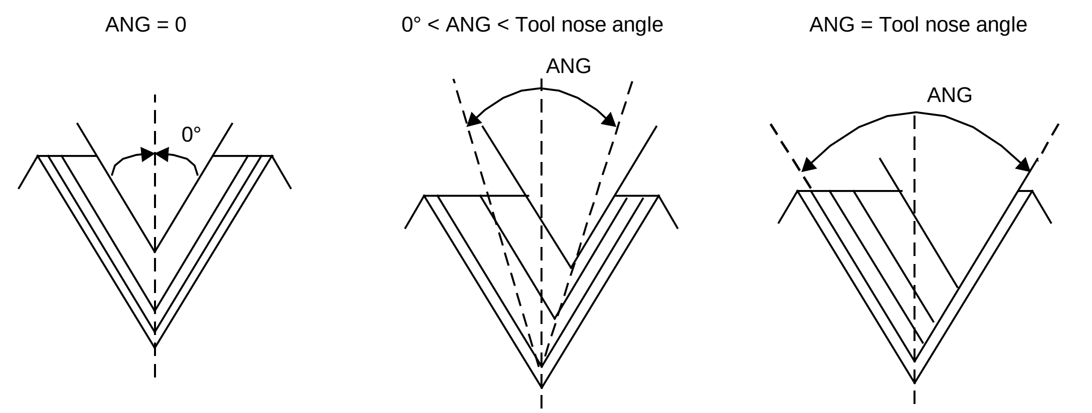
[4] LEAD

Set the threading lead given by the following expression:
(Lead) = (Pitch) × (Number of threads)

[5] ANG

Set a threading angle.

- Usually, set a several degrees smaller value than the nose angle of the tool.



[6] MULTI

Set the desired number of threads.

[7] HGT

Set the threading height.

Pressing the **[AUTO SET]** menu key with the cursor at item [7] will automatically set data into items [7].

3-13-2 Setting tool sequence data

```
SNo.  TOOL   NOM.  No. # PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR  M  M  M
 1                                              ◆     ◆     ◆         ◆
      ↑    ↑   ↑ ↑ ↑  ↑   ↑    ↑     ↑        ↑                 ↑        ↑  ↑  ↑
     [1]  [2] [3][4][5][6] [7]  [8]   [9]      [10]              [11]    [12][12][12]
```

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		SIMUL DRILL ROTATION	

[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 keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for details of items [5] and [7].

[7] PAT. (Machining pattern)

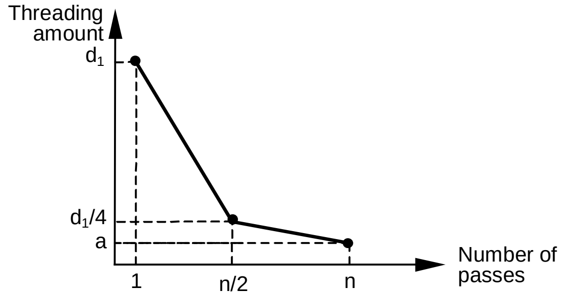
The following menu will be displayed when the cursor is placed at this item:

#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA	#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA				
(a)	(b)								

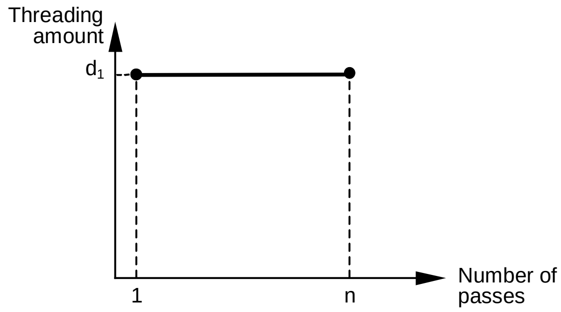
From the menu, select the threading pattern to be machined.

Patterns to be machined that correspond to the data of the displayed menu are as follows.

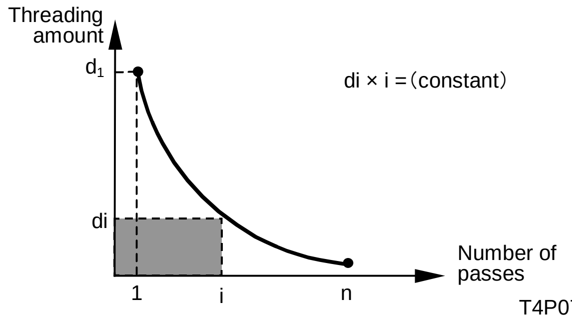
#0, #0 : Standard patterns
The threading amount gradually decreases as the number of passes increases.



#1, #1 : Constant-threading pattern
The threading amount always stays constant, irrespective of the number of passes.



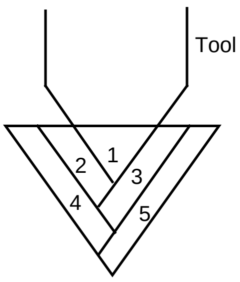
#2, #2 : The threading amount is inversely proportional to the number of passes.



n: Number of passes
d_1: The first threading amount
a: The n-th threading amount
di: The i-th threading amount

T4P077

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 [5], ANG in the unit data.



1 to 5: Threading order

[9] 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 **[AUTO SET]** menu key.

[10] 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.

[11] C-SP

Specify the surface speed for the tool in terms of turning spindle.
See the description of the relevant item for BAR unit.

[12] M

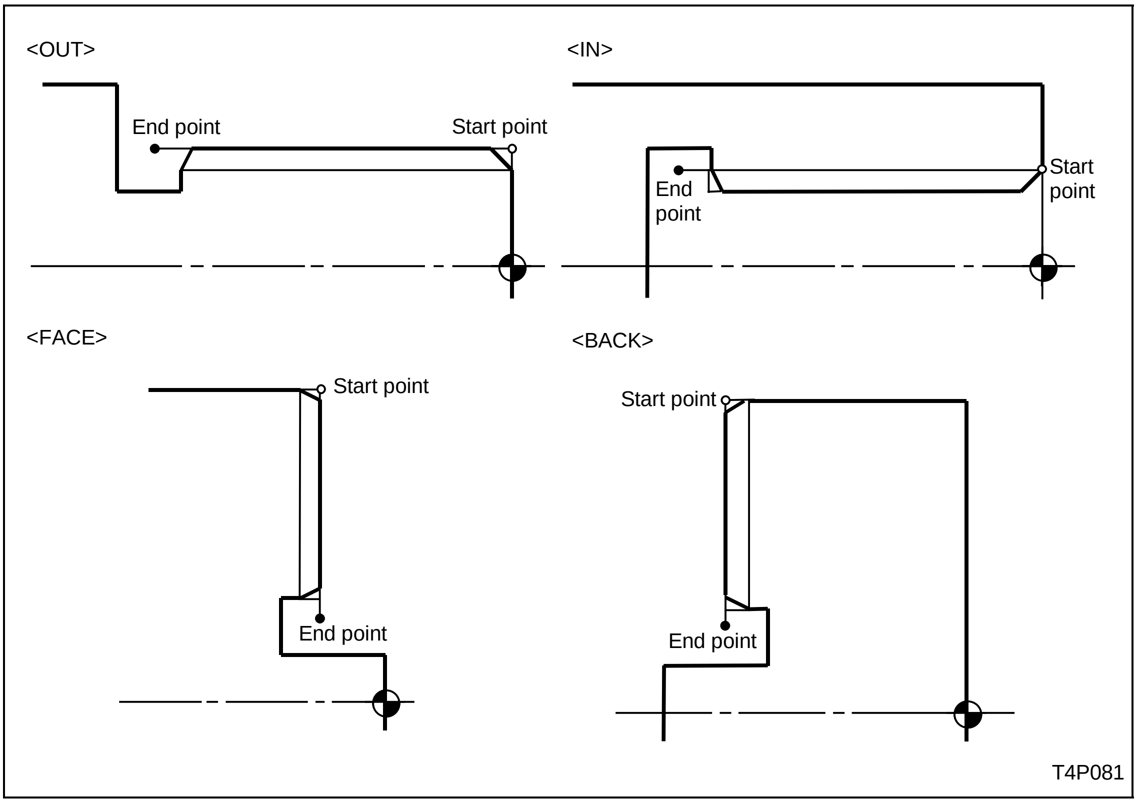
Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

3-13-3 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 ◆).

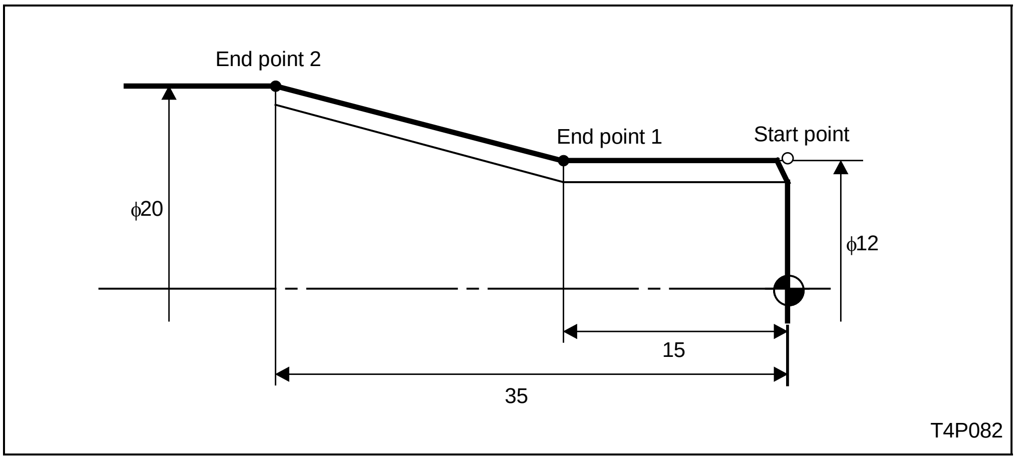
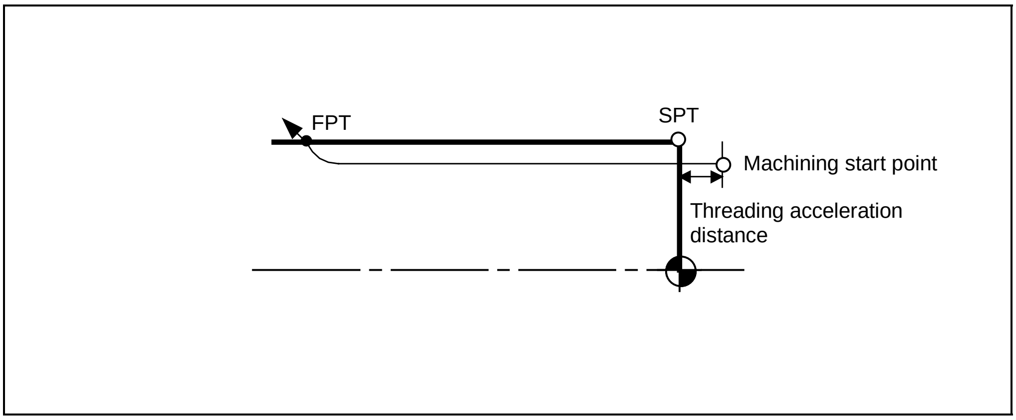


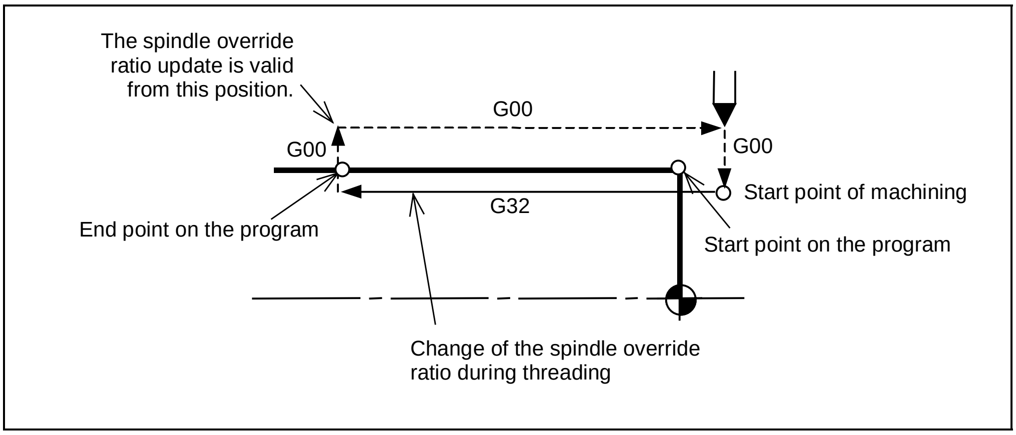
FIG	SPT-X	SPT-Z	FPT-X	FPT-Z
1	12.	0.	12.	15.
2	◆	◆	20.	35.

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.

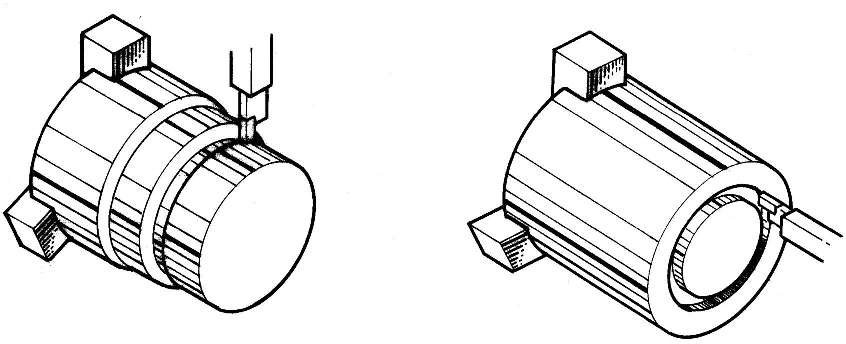


Note 5: Threading start position automatic correction and re-threading functions (both optional) are only valid for a longitudinal threading by cutting feed on the Z-axis: THREAD OUT or IN. (Invalid for THREAD FACE and BACK)

Note 6: Re-threading function (optional) is only valid for constant lead threading.

3-14 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.

3-14-1 Setting unit data

UNo.	UNIT	PART	POS-B	PAT.	No.	PITCH	WIDTH	FINISH
*	T.GROOVE	[1]	[2]	[3]	[4]	[5]	[6]	[7]

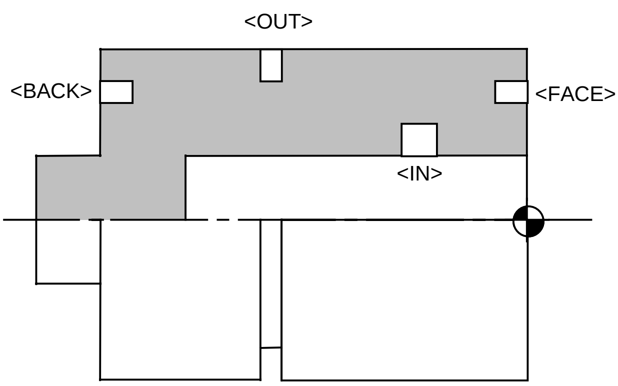
[1] PART

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)



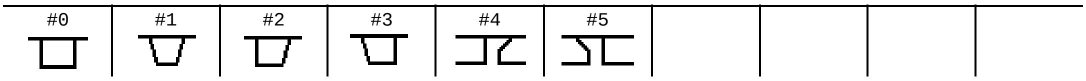
T4P089

[2] POS-B

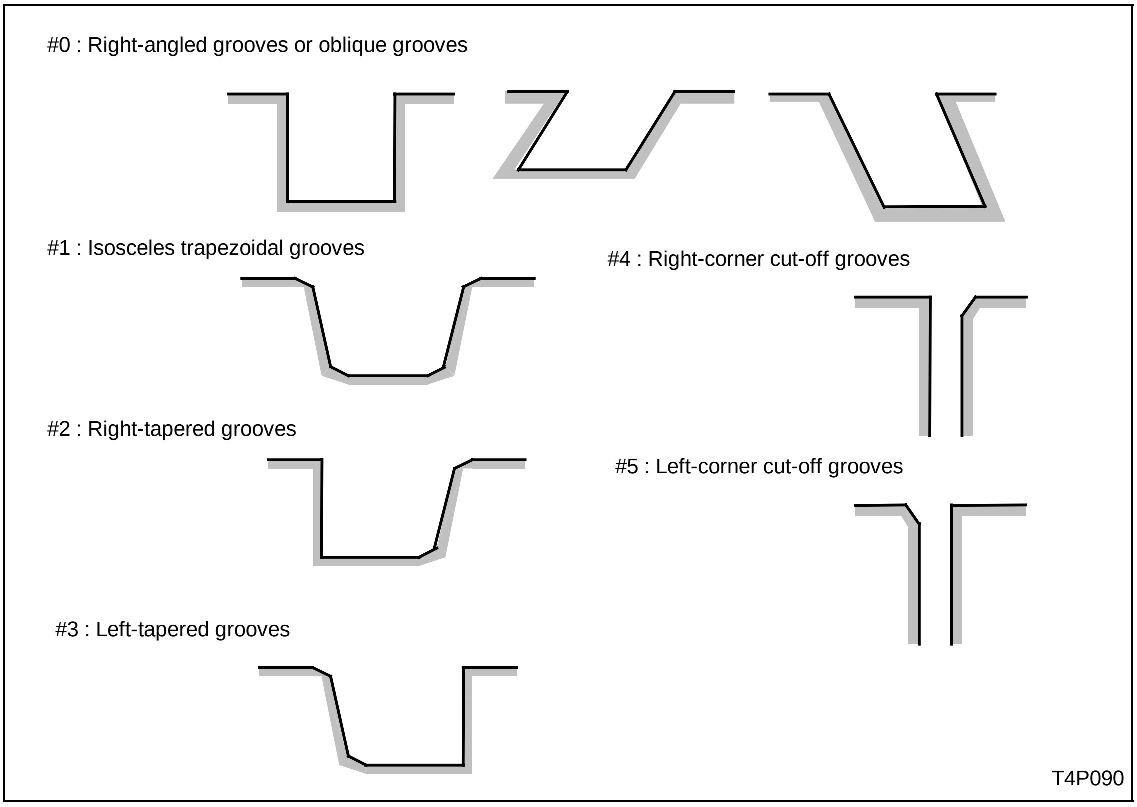
From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[3] 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 **TC50**.

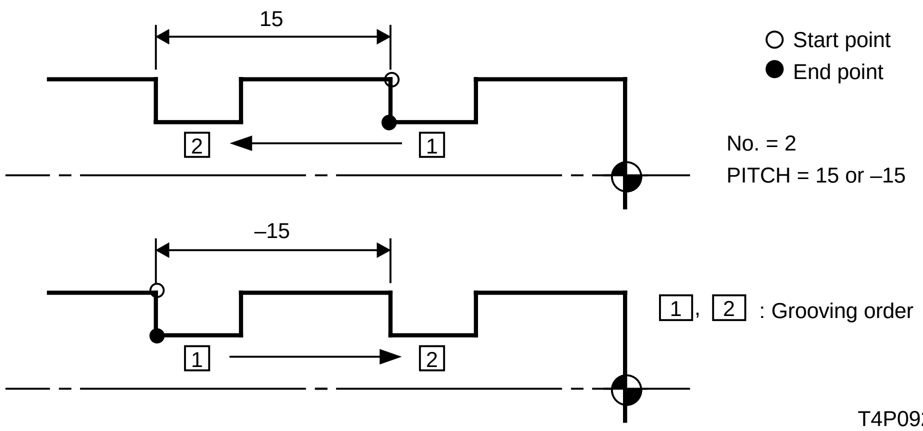
[4] No.

Set the number of grooves when multiple grooves of the same shape are to be machined at fixed spacings.

[5] PITCH

Set a pitch when multiple grooves of the same shape are to be made at fixed spacings.

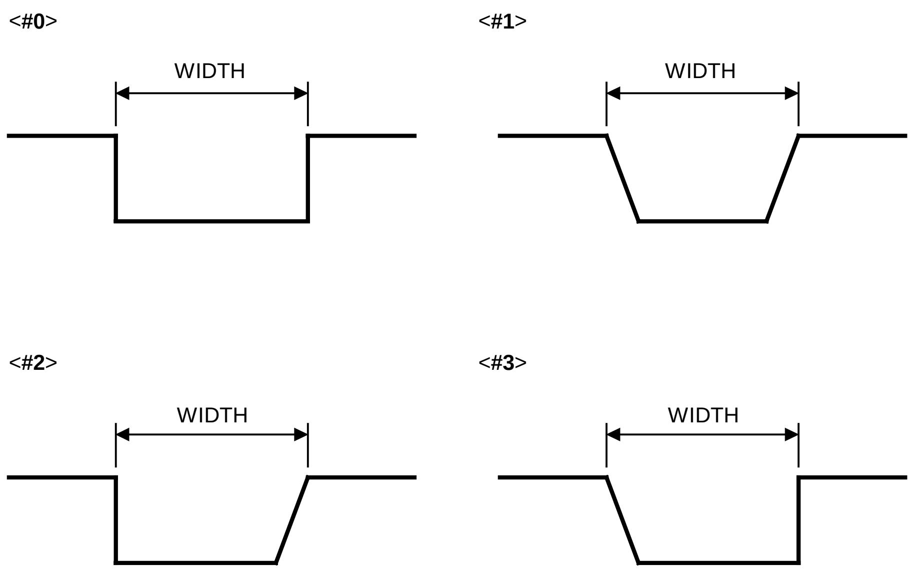
The pitch can be set as either a plus value or a minus value. Setting the pitch as a plus value causes sequential grooving in a forward direction. Setting the pitch with a minus sign causes sequential grooving in a reverse direction.



T4P092

[6] WIDTH

Set a grooving width.

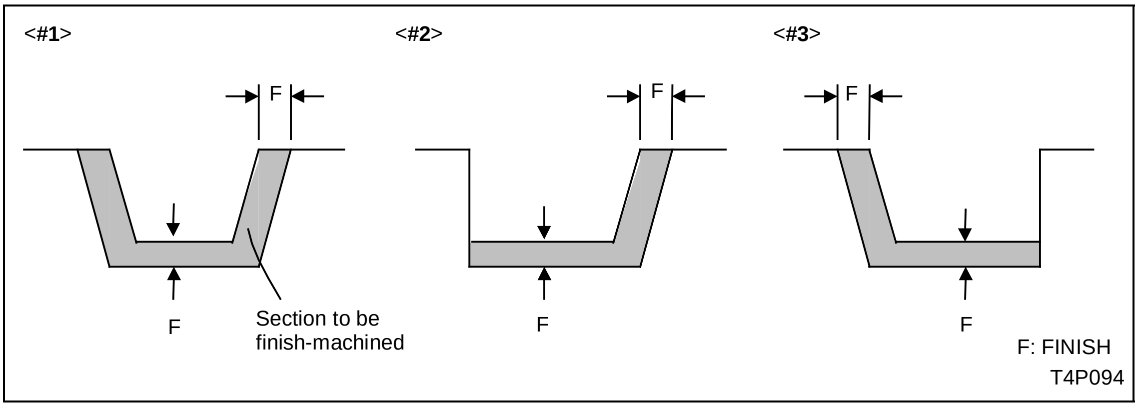


T4P093

- If you have selected grooving pattern #4 or #5, a cutting-off tool tip width is considered as a grooving width.

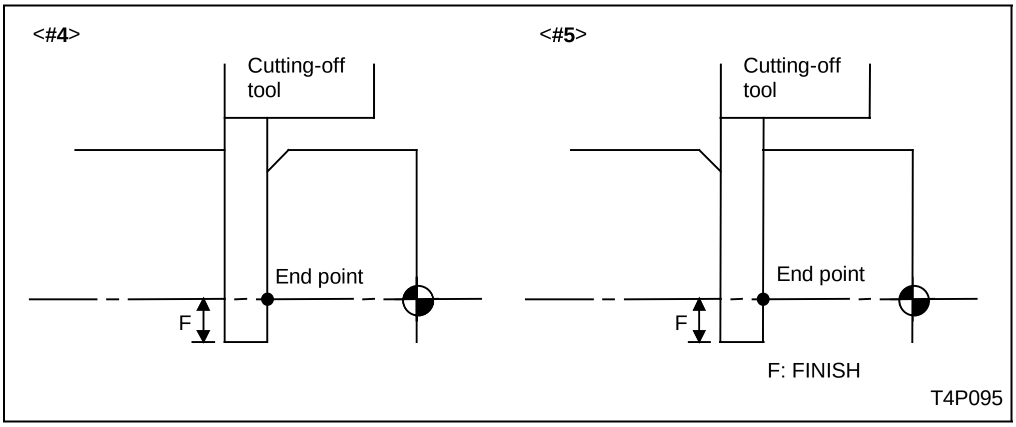
[7] FINISH

- 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.



3-14-2 Setting tool sequence data

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R1					◆		◆	◆	◆	◆					
F2					◆		◆	◆		◆					
	↑ [1] ↑ [2]	↑ [3] ↑ [4] ↑ [5]	↑ [6]	↑ [7]		↑ [8]			↑ [9]		↑ [10]	↑ [11]	↑ [12]	↑ [12]	↑ [12]

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		SIMUL DRILL ROTATION	

[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 keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [7].

[8] DEP-1 (Max. depth of cut), [9] C-SP, [10] FR

For each grooving pattern, set data into these items as follows:

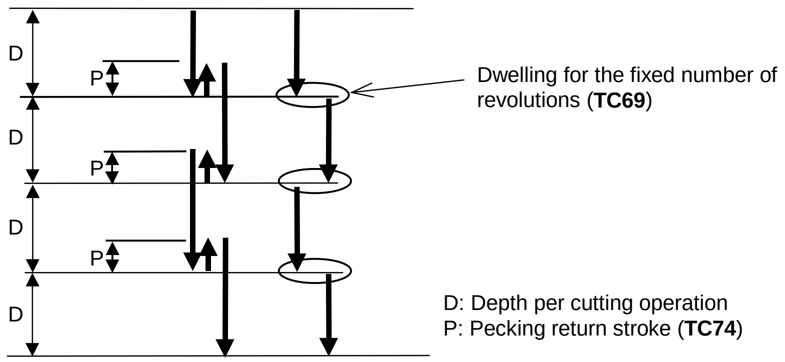
Pattern	Sequence	**DEP-1** (Max. depth of cut)	**C-SP**	**FR**
#0	F (Finishing)	Max. depth of cut per pass (Designate in radius for **OUT** or **IN**)	Surface speed during grooving	Feedrate during grooving (Shape seq. data **RGH** ineffective)
#1, #2 or #3	R (Roughing)		Surface speed during roughing	Feedrate during roughing
	F (Finishing)	—	Surface speed during finishing	Feedrate during finishing
#4 or #5 (Parameter **TC50** = 0, 1)	F (Finishing)	Max. depth of cut per pass (Designate in radius; without pecking if 0 is set)	Surface speed during grooving (limited by the rotational speed specified by the parameter **TC49**)	Feedrate during grooving (Shape seq. data **RGH** effective for cutting-off area specified by the parameter **TC9**)
#4 or #5 (Parameter **TC50** ≥ 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 **TC9**) 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 **TC49**.

** The feedrate is reduced in several steps (set by the parameter **TC50**) to the value of shape sequence data RGH.

<Pecking during grooving>

During grooving, pecking is repeated with each cutting operation in the direction of the groove depth. Use parameter **TC74** to specify the returning stroke of pecking. If **TC74** = 0, pecking will not occur and the machine will come to a dwelling stop while the spindle rotates the number of revolutions specified in parameter **TC69**.



D740PA158

Note: Setting pecking return stroke parameter **TC74** to zero allows the machining time required to be reduced since pecking is not conducted. However, vibration and/or unusual operating sounds may occur under specific cutting conditions. If this is the case, machine the workpiece in normal pecking mode.

[11] FIN-X

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items 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.

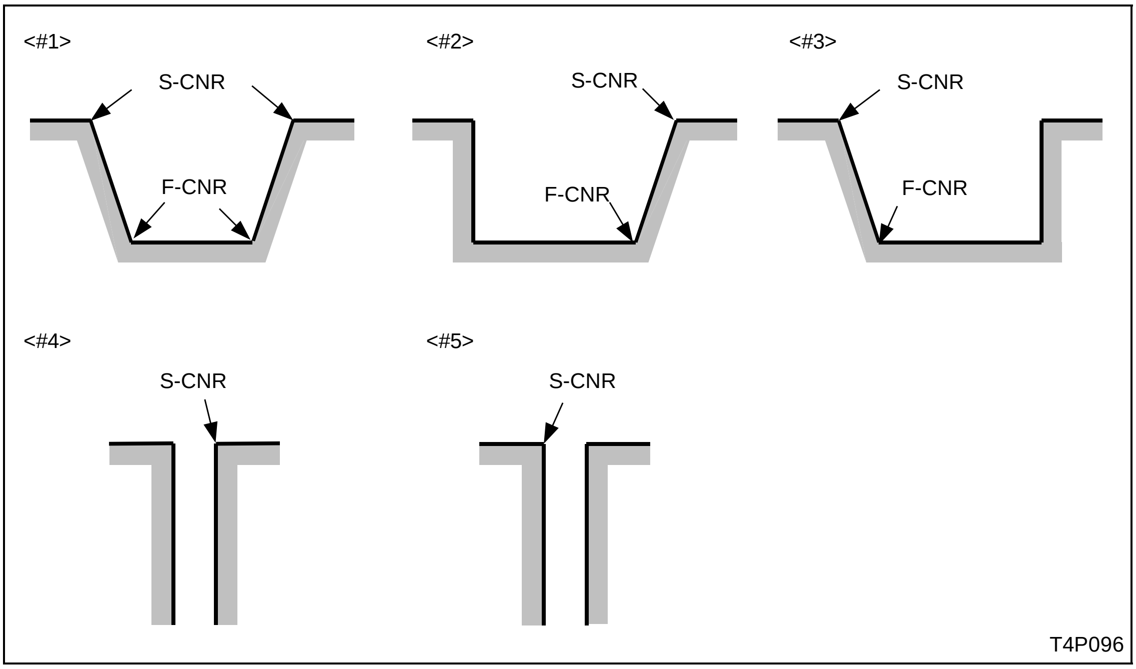
3-14-3 Setting shape sequence data

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-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.

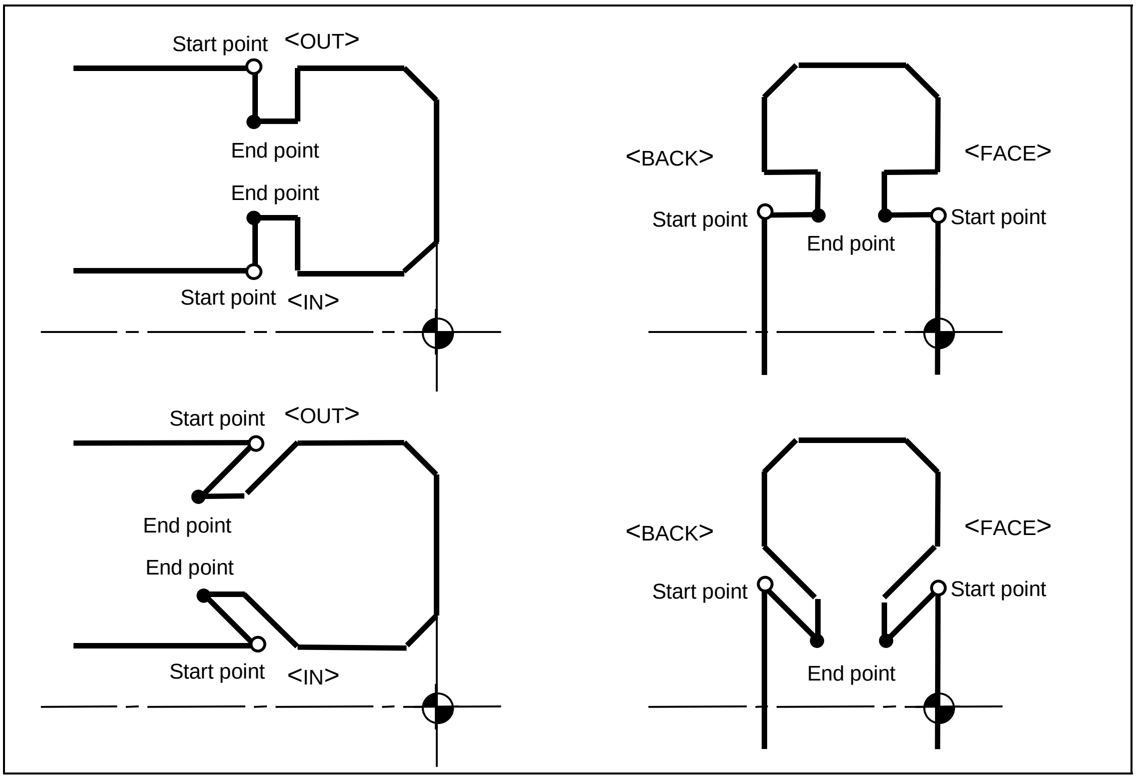


[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z

Set the coordinates of the start point and end point of grooving.

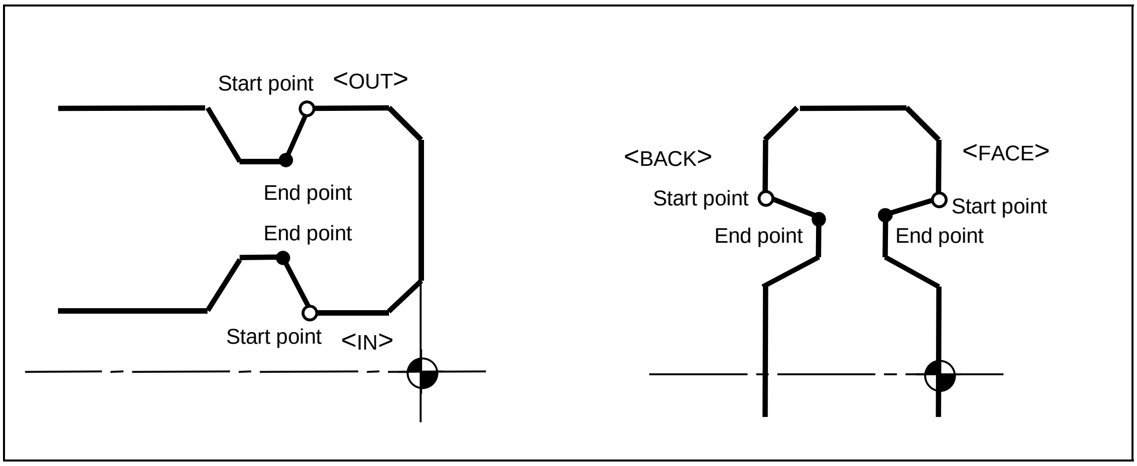
Set the position of the start point and the end point as follows according to the selected grooving pattern.

- For grooving pattern #0:

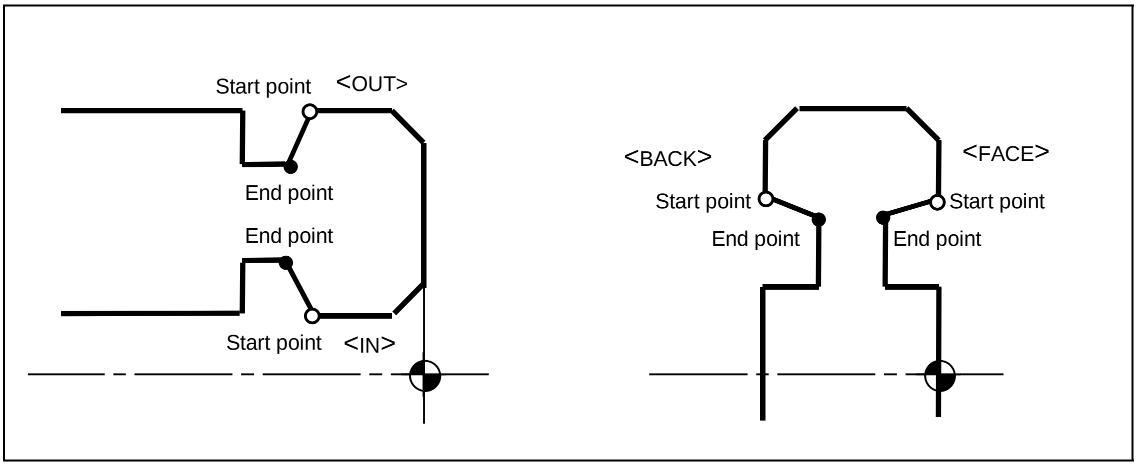


For pattern #0, oblique grooves can be machined by setting the start and end points as shown above.

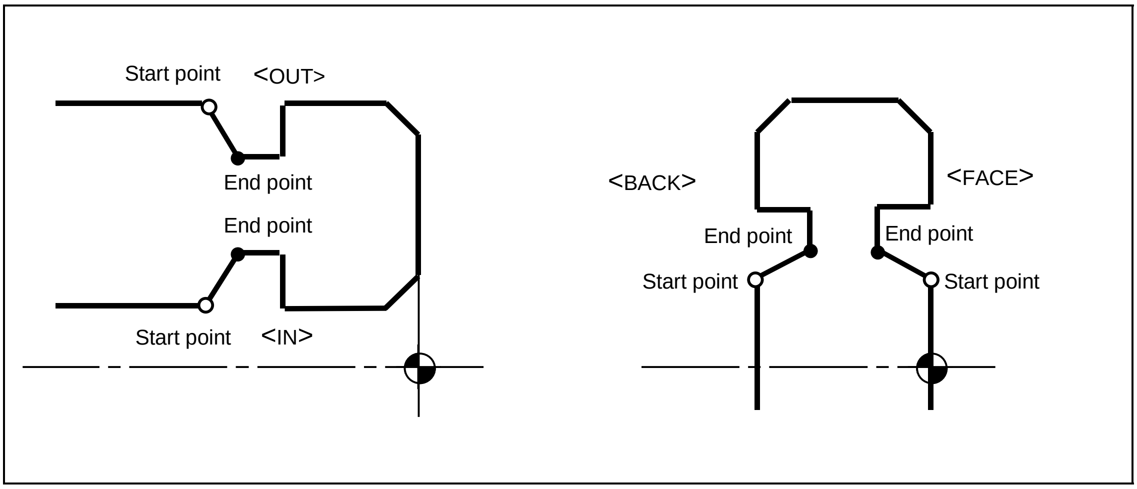
- For grooving pattern #1



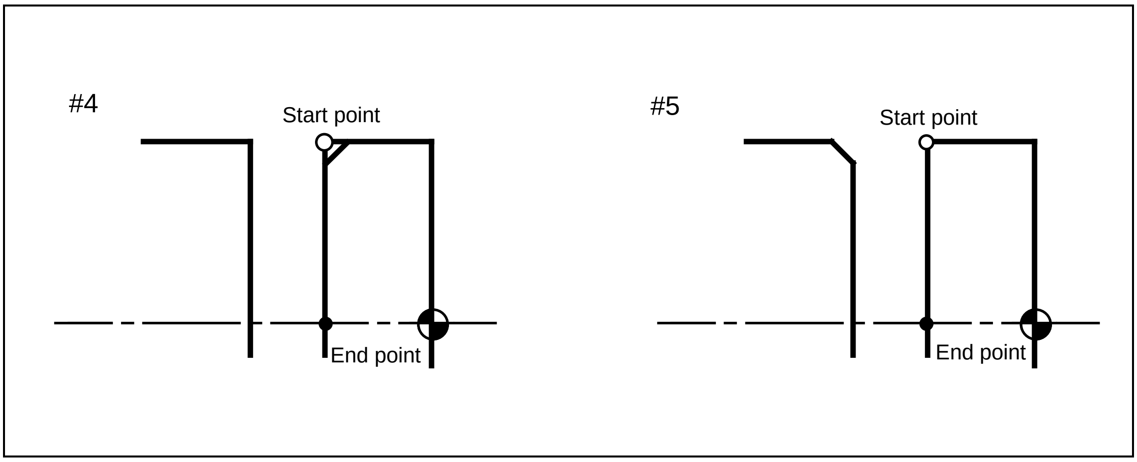
- For grooving pattern #2:



- 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 set 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] F-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 set 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.

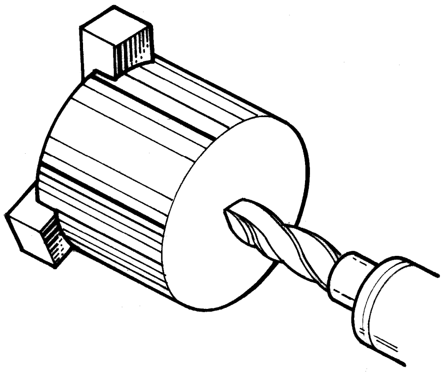
[8] RGH

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.)

3-15 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.

3-15-1 Setting unit data

UNo.	UNIT	PART	POS-B	DIA
*	T.DRILL	[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: **PART** may not be specified for special machine specifications.

[2] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[3] DIA

Set the diameter of the hole to be drilled (nominal diameter of the turning drill).

3-15-2 Setting tool sequence data

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
1									◆	◆					
	↑ ↑	↑ ↑ ↑	↑	↑	↑	↑	↑	↑			↑	↑	↑	↑	↑
	[1] [2]	[3] [4] [5]	[6]	[7]	[8]	[9]	[10]	[11]			[12]	[13]	[14]	[14]	[14]

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.

Remark 3: When **[SIMUL DRILL ROTATION]** is selected for item [1] **TOOL** (Name), the following tool sequence is displayed:

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	RPM	SPDL ROT.	C-SP	FR	M	M	M
1															
	↑ ↑	↑ ↑ ↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑
	[1] [2]	[3] [4] [5]	[6]	[7]	[8]	[9]	[10]	[11]	[15]	[16]	[17]	[18]	[14]	[14]	[14]

[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		SIMUL DRILL ROTATION	

[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 or milling drill).

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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.
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [7].

[8] PAT. (Machining pattern)

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

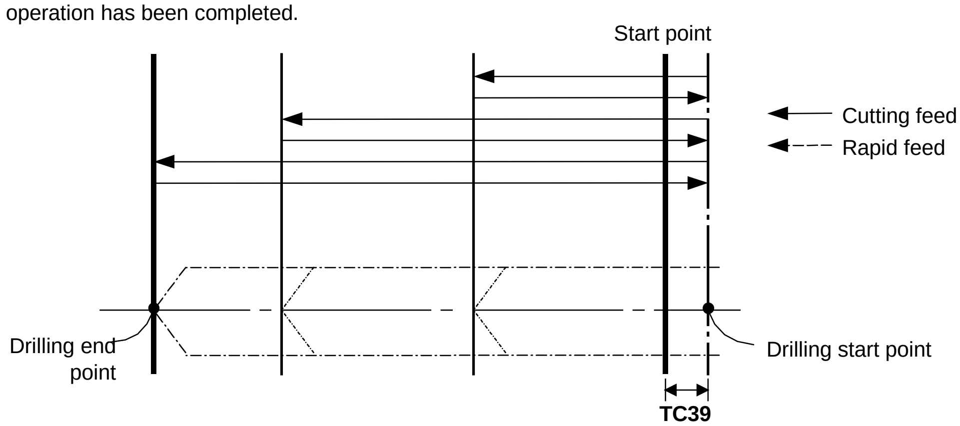
#0 DRILLING BOTTOMED	#1 PECKING BOTTOMED	#2 PECKING BOTTOMED	#3 REAMER BOTTOMED	#4 PECKING BOTTOMED					>>>

#0 DRILLING THROUGH	#1 PECKING THROUGH	#2 PECKING THROUGH	#3 REAMER THROUGH	#4 PECKING THROUGH					>>>

From the menu, select the turning drilling pattern.

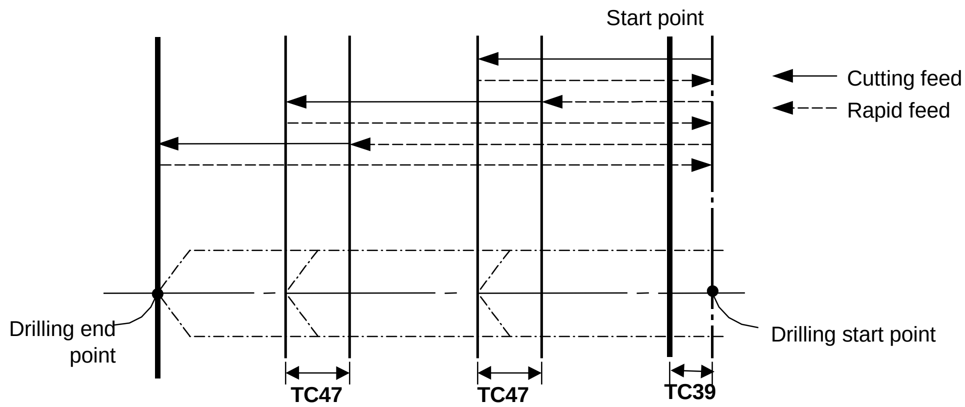
The data of the displayed menu represent following drill patterns.

#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.



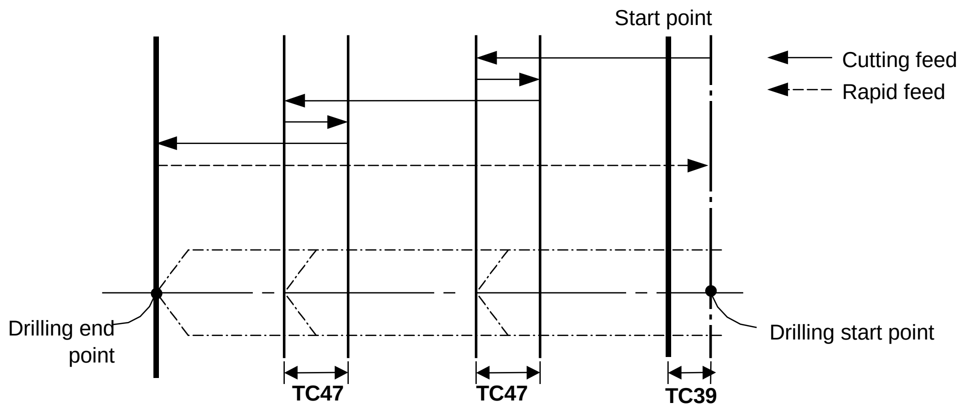
T4P114

#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.



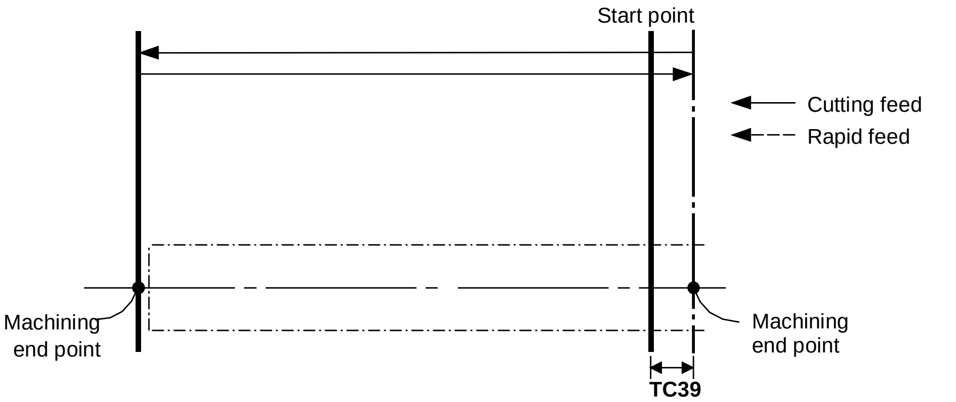
T4P115

#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.



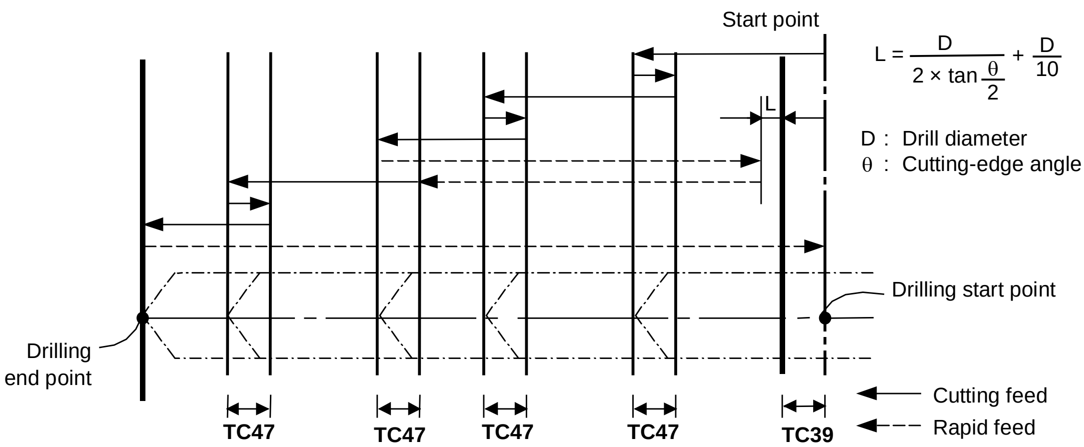
T4P116

#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.



$$L = \frac{D}{2 \times \tan\frac{\theta}{2}} + \frac{D}{10}$$

D : Drill diameter
θ : Cutting-edge angle

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 **D52**.

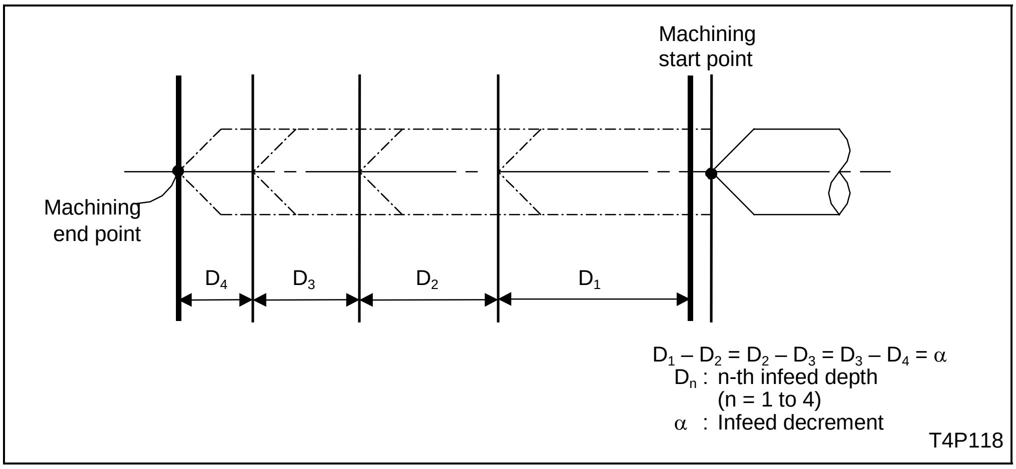
[9] DEP-1, [10] DEP-2/NUM., [11] DEP-3

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 [8] and setting a tool will automatically set the data that has been calculated by the NC unit into items [9] through [11].

When machining pattern #3 or #3 is selected.
Items [9] through [11] 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

$$D_n\ (= D_{n+1} = D_{n+2} = \cdots\cdots) = D_{min}$$

[12] C-SP

Specify the surface speed for the turning spindle.
See the description of the relevant item for BAR unit.

[13] 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.

[14] M

Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

[15] RPM

If **[SIMUL DRILL ROTATION]** is selected for item [1] **TOOL** (name), specify the number of revolutions of the turning spindle.

Note: The number of revolutions of the drilling tool (milling spindle) is calculated from the relative number of revolutions calculated from the surface speed specified in item [17] and the number of revolutions of the turning spindle.

[16] SPDL ROT.

If **[SIMUL DRILL ROTATION]** is selected for item [1] **TOOL** (name), specify the rotational direction of the turning spindle. The following menu is displayed.

FWD	REV								

To rotate the spindle in the forward direction, select **[FWD]**.

To rotate the spindle in the reverse direction, select **[REV]**.

Note: The drilling tool (milling spindle) rotates in the forward (clockwise) direction, as with the drill used for point machining.

[17] C-SP

If **[SIMUL DRILL ROTATION]** is selected for item [1] **TOOL** (name), specify the relative surface speeds of the turning spindle and milling spindle. The relative number of revolutions of the turning spindle and that of the milling spindle are calculated from their relative surface speeds.

[18] FR

If **[SIMUL DRILL ROTATION]** is selected for item [1] **TOOL** (name), specify the feedrate of the milling/drilling tool by entering the rate at which the tool is fed each time the turning spindle and the milling spindle make one revolution relative to each other.

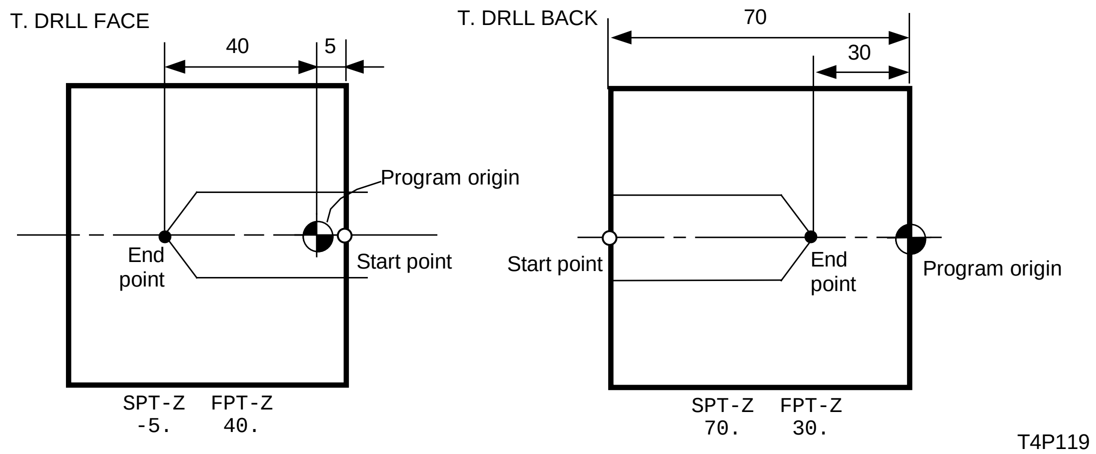
3-15-3 Setting shape sequence data

FIG	SPT-Z	FPT-Z
1	[1]	[2]

<u>[1] SPT-Z, [2] FPT-Z</u>

Set the coordinates of the start point and end point of the intended drilling pattern.

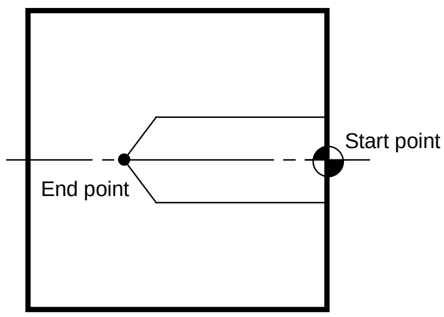
- The start point and end point for drilling stop-holes, for example, are positioned as shown below.



T4P119

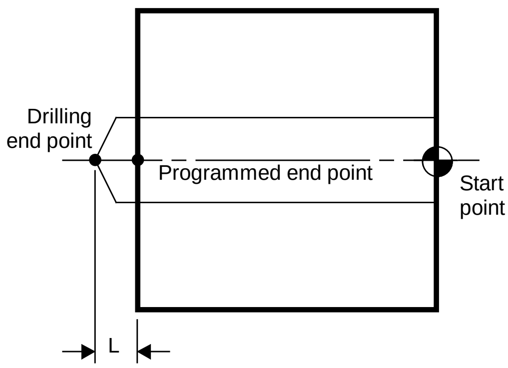
- The position of the end point differs from stop-hole to through-hole as follows:

For stop-hole



The drilling feed will be stopped when the tool tip has reached the programmed end point.

For through-hole



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:

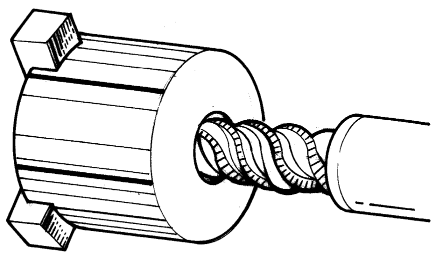
$$L = \frac{D}{2 \times \tan\frac{\theta}{2}} + \frac{D}{10} \quad (0 < \theta < 180°)$$

$$L = \frac{D}{10} \quad (\theta = 180°)$$

D : Tool diameter
θ : Cutting-edge

3-16 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.

3-16-1 Setting unit data

UNo.	UNIT	PART	POS-B	NOM-DIA	PITCH
*	T.TAP	[1]	[2]	[3]	[4]

[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] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numeric keys. See the description of the relevant item for BAR unit.

[3] 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 [INPUT] 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 [INPUT] 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 [INPUT] 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 [INPUT] 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 [INPUT] 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 [INPUT] 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 [INPUT] 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	

[4] 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 [3] **NOM-DIA**, data will be auto-set, except for special threads. Any data, however, can also be entered instead.

3-16-2 Setting tool sequence data

```
SNo.   TOOL    NOM.   No.  #  PAT.  DEP-1 DEP-2/NUM.  DEP-3 FIN-X FIN-Z  C-SP   FR   M   M   M
 1                                ◆      ◆       ◆          ◆     ◆     ◆            ◆
       ↑     ↑    ↑  ↑  ↑   ↑    ↑                                       ↑            ↑   ↑   ↑
      [1]   [2]  [3][4][5] [6]  [7]                                     [8]          [9] [9] [9]
```

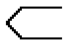
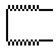
Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is automatically selected.

<u>[1] TOOL</u> (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		SIMUL DRILL ROTATION	

<u>[2] TOOL</u> (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.

<u>[3] NOM.</u> (Nominal diameter)

Set the diameter of the hole to be tapped (nominal diameter of the turning tap). See the description of [3] **NOM-DIA** in the unit.

[4] NOM. (Suffix)

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	HEAVY TOOL	>>>

[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
See the description of the relevant item for BAR unit.

SET UPPER TURRET	SET LOWER TURRET								

[6] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[7] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS", for details of items [5] and [7].

[8] C-SP

Specify the surface speed for the turning spindle.
See the description of the relevant item for BAR unit.

[9] M

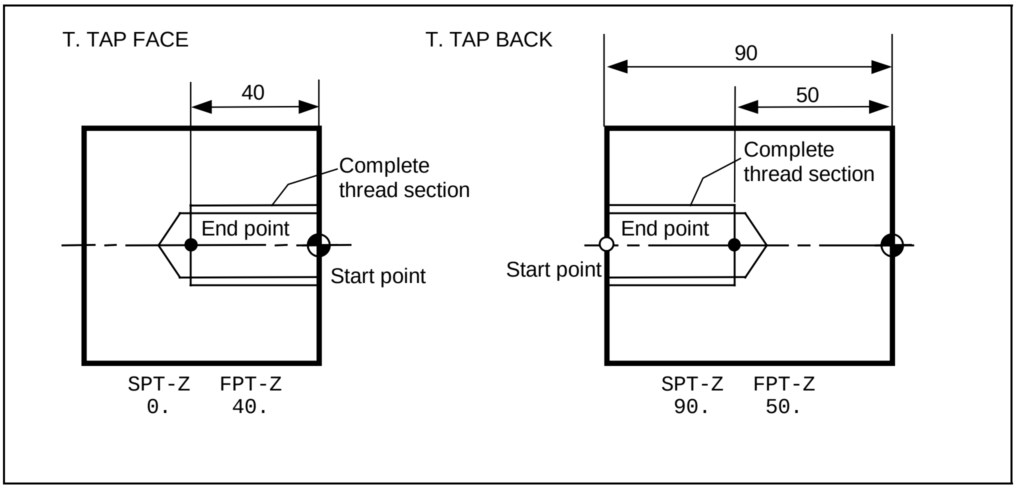
Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

3-16-3 Setting shape sequence data

FIG	SPT-Z	FPT-Z
1	[1]	[2]

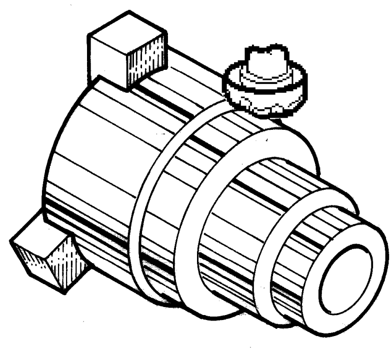
[1] SPT-Z, [2] FPT-Z

Set the coordinates of the start point and end point of the thread shape.



3-17 Mill-Turning Unit (MILLTURN)

Select the mill-turning unit to lathe the outer peripheries of round-bar-materials using milling tools.



D740PA162

Press the **[MILLTURN]** menu key to select this unit.

3-17-1 Setting unit data

UNo.	UNIT	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z	SHIFT-Y
*	MILLTURN	[1]	[2]	[3]	[4]	[5]	[6]

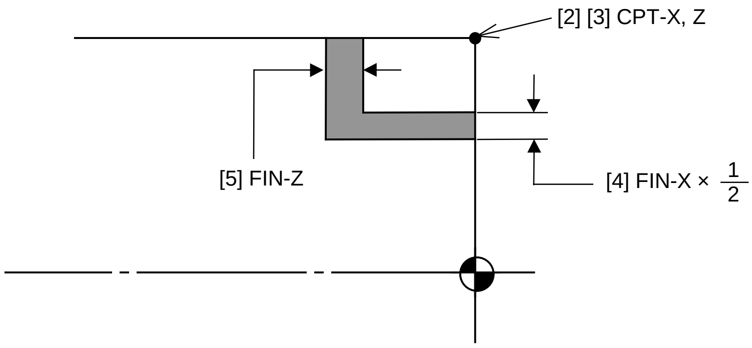
[1] POS-B

Select an angle at which the B-axis is to be indexed. The desired angle can likewise be set up using the numeric keys. See the description of the relevant item for BAR unit.

Note: If an angle other than 90 degrees is set up as **POS-B**, interference with the workpiece may occur at specific inclination angles of the tool. The workpiece may also be machined too much or partly remain unmachined. Modify the program in such cases.

[2] CPT-X, [3] CPT-Z, [4] FIN-X, [5] FIN-Z

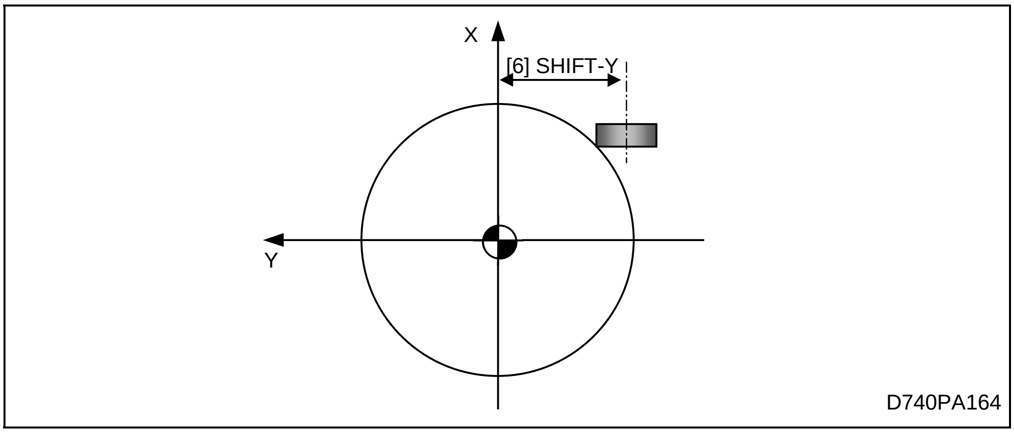
Set the X- and Z-coordinates of the desired infeed point. After that, set the allowances to be left for the X-axis and Z-axis directions.



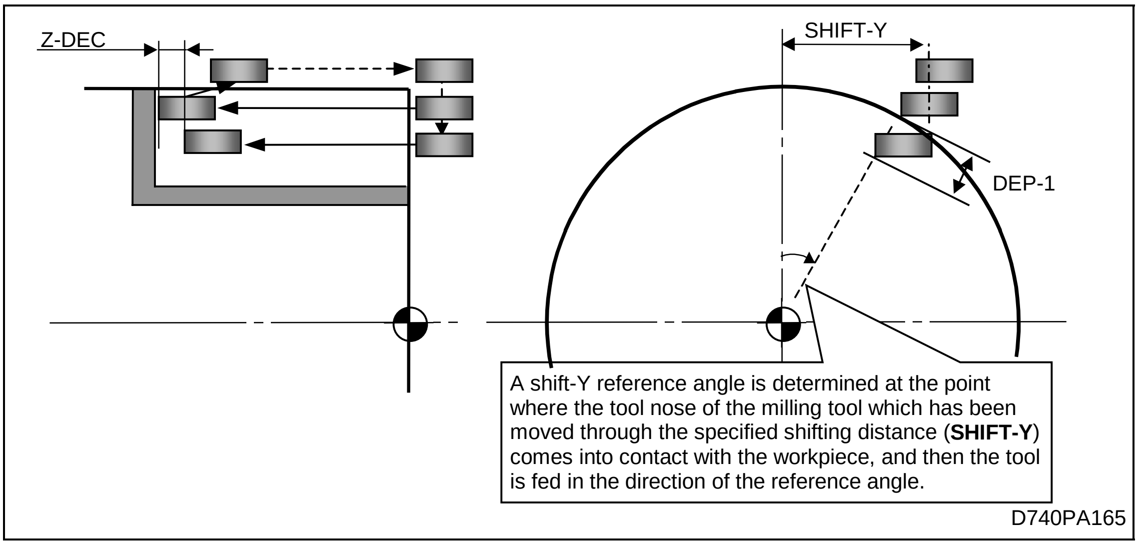
D740PA163

[6] SHIFT-Y

Set the Y-axial shifting distance.



When **SHIFT-Y** is set, the tool path will be as follows:



Note 1: During machining with the Y-axis shifted, part of the workpiece is not machined.

Note 2: Enter 0 under **SHIFT-Y** if an angle other than 90 degrees is set up as **POS-B**.

3-17-2 Setting tool sequence data

SNo.	TOOL	NOM-ϕ	No.	#	PAT.	DEP-1	Z-DEC	RPM	FIN-X	FIN-Z	C-SP	FR	M	M	M
R1									◆	◆					
	↑	↑ ↑	↑	↑	↑	↑	↑	↑			↑	↑	↑	↑	↑
	[1]	[2] [3]	[4]	[5]	[6]	[7]	[8]	[9]			[10]	[11]	[12]	[12]	[12]

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
R1 (Roughing)	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:

ENDMILL	FACEMILL								

[2] NOM-ϕ (Nominal diameter)

Enter the nominal diameter of tools using the numeric keys.
See the description of the relevant item for BAR unit.

[3] NOM-ϕ (Suffix)

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	HEAVY TOOL	>>>

Note: This unit is applicable only for the upper turret. Therefore tool in the lower turret cannot be selected.

[4] No. (Priority No.)

Assign priority levels in the order of machining. See the description of the relevant item for BAR unit.

[5] # (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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

[6] PAT. (Rotational direction of the turning spindle)

Set rotational direction of the turning spindle.
The following menu will be displayed:

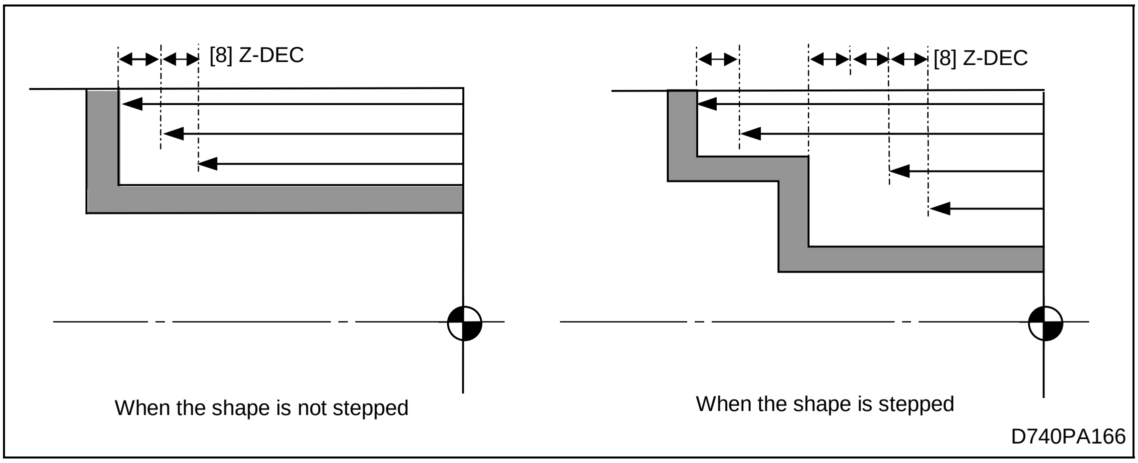
CW	CCW								

[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] Z-DEC

The amount of Z-axial relief per cutting operation can be set to prevent interference between the workpiece and the tool. **Z-DEC** is accumulated with each cutting operation.



[9] RPM

Set the turning spindle speed using the numeric keys. Constant surface speed control cannot be conducted.

Note: When the spindle override value is changed during the rotations of both the turning spindle and the milling spindle, both spindle speeds will change according to the entered percentage value (%). Both spindles will stop if 0% is entered.

[10] C-SP

Enter the the surface speed for milling tool.

[11] FR

Enter the desired feedrate of the milling tool in terms of turning spindle speed per revolution. Set the feedrate using the menu or numeric keys.

Note: When VFC is conducted during the execution of this function, the surface speed and feedrate of the milling tool will be updated.

[12] M

Specify the M-code to be issued for the tool immediately after it is selected.
See the description of the relevant item for BAR unit.

3-17-3 Setting shape sequence data

FIG	PTN	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th
	[1]	[3]	[4]	[5]	[6]	[7]	[8]

The shape sequence data for the mill-turning unit is the same as that for the bar-materials machining unit. See the description of the relevant item in Section 3-9 “Bar-Materials Machining Unit (BAR)”.

Note 1: An arc shape, corner R/C at the starting or ending point, and roughness cannot be specified.

Note 2: Since nose R of the tool is not taken into account, part of the workpiece may remain unmachined.

3-18 Other Units

Twelve units are provided moreover:

- Manual program machining unit
- M-code unit
- Head selection unit
- Workpiece transfer unit
- Subprogram unit
- Add-in MAZATROL unit
- End unit
- Simultaneous machining unit
- 2-workpiece machining unit
- Coordinate measuring unit
- Workpiece measuring unit
- Tool measuring unit

3-19 Manual Program Machining Unit (MANL PRG)

The manual program machining unit complements the turning and milling units described so far (BAR, CPY, CORNER, FACING, THREAD, T. GROOVE, T. DRILL and T. TAP units and point/line/face machining units).

These machining 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.

3-19-1 Setting unit data

UNo.	UNIT	TOOL	NOM-ϕ	No.	#	POS-B
*	MANL PRG	[1]	[2]	[3]	[4]	[5]

[1] TOOL

Specify the tool to be used. If a tool is not specified, the currently valid tool will be used as it is.

Select menu key corresponding to the tool used. Pressing the **[>>>]** menu key changes the menu **a** → **b** → **c** → a in this order.

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR				>>>	**a**
CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR	CHIP VACUUM		>>>	**b**
GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			>>>	**c**

[2] NOM-ϕ

Enter the nominal diameter of the tool by means of numeric keys. Input range is 0.1 to 999.9.

In the case of the identical designation and length but of different materials, it is necessary to differentiate them by identification code.

The identification code is selected from the menu:

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>

In order to designate the heavy tool, select the desired menu item after having displayed the menu for heavy tool identification code by pressing the **[HEAVY TOOL]** menu key.

For the machine with the lower turret, select the turret in which the tool to be used is mounted. The following menu is displayed (if **[SET UPPER TURRET]** is selected, the column will remain blank, and if **[SET LOWER TURRET]** is selected, "▼" will be displayed). See Section 5, LOWER-TURRET CONTROL FUNCTIONS, for further details:

SET UPPER TURRET	SET LOWER TURRET								

Note: When one of the menu items **[END MILL]**, **[FACE MILL]**, **[CHAMFER CUTTER]** and **[BALL ENDMILL]** is selected in the article **TOOL**, the alarm **434 NO ASSIGNED TOOL IN TOOL FILE** is displayed if a tool with the specified nominal diameter has not been previously recorded in the **TOOL FILE** display.

[3] No.

Determine the priority machining number (prior machining, subsequent machining). The input of data is done in accordance with the following 3 methods;

- Priority No. for prior machining:
  Enter the number by means of numeric keys. Input range is 0 to 99.
- Priority No. for subsequent machining:
  Press the **[DELAY PRIORITY]** menu key, then enter the number by means of numeric keys. Input range is 0 to 99.

MACHINING PRIORITY No?

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END		

- Without input: Normal machining:
  The machining order is not specified.

Note: For details, refer to Chapter 4, "PRIORITY FUNCTION FOR THE SAME TOOL."

[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:

LOWER TURRET POS.1	LOWER TURRET POS.2								

Note: See Chapter 5, "LOWER-TURRET CONTROL FUNCTIONS", for details of item [4].

[5] POS-B

From the menu, select an angle for indexing the B-axis. You can specify an angle using numerical keys. See the description of the relevant item for BAR unit.

3-19-2 Setting sequence data

In the manual program machining unit, one sequence data line corresponds to one EIA/ISO program block. For the details of each of the commands, refer to the Programming Manual (Programming EIA/ISO).

SEQ	G1	G2	DATA-1	DATA-2	DATA-3	DATA-4	DATA-5	DATA-6	S	M/B
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

[1] G1, [2] G2

Enter the G-codes (preparatory function) by means of menu keys or numeric keys.

G00	G01	G02	G03	G40 CANCEL	G41 LEFT	G42 RIGHT	G98 /MIN	G99 /REV	MANUAL END

Note: Up to two G-code commands can be designated in one sequence data line.

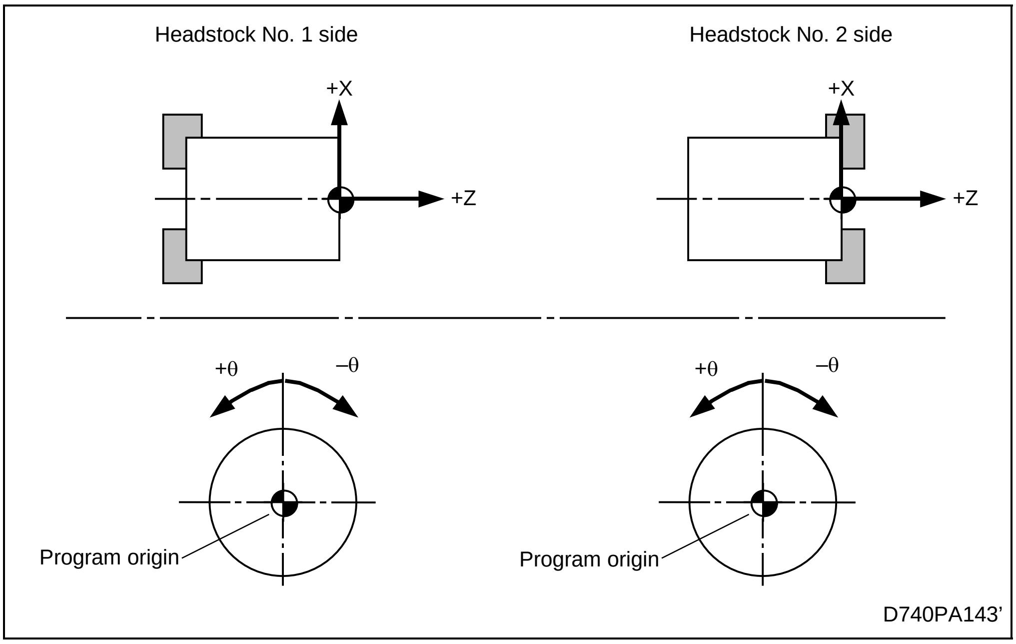
[3] DATA-1, [4] DATA-2, [5] DATA-3, [6] DATA-4, [7] DATA-5, [8] DATA-6

Select the address of the data to be entered from the following menus. Pressing the **[>>>]** menu key changes the menu **a** → **b** → **a** in this order.

X	Z	C	Y	F	U	W	H	V	>>>	**a**
I	J	K	P	D					>>>	**b**

After the address has been set, enter and set data using the numeric keys.

For the 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.



Note: In the conversion of programs created with another type of NC (M640T, M640MT or M640MT Pro) each dimensional information of incremental data input in a manual program turning or milling unit is converted as appropriate into a value of absolute data input.

[9] S

Enter the rotational speed or surface speed of the spindle by means of numeric keys.
Input range is 0 to 99999.

Note 1: The rotational speed or surface speed that you have set remains valid until new such value is set.

Note 2: In the conversion of programs created with another type of NC (M640T, M640MT or M640MT Pro) this item of spindle speed function (S) remains blank (unset) when the corresponding sequence data item in a manual program milling unit of the source program is of surface speed.

[10] M/B

Enter the M-code (auxiliary function) or the B-code (second auxiliary function, 3 digits) by means of numeric keys. For B-code setting, press the **[B CODE INPUT]** menu key and enter a numeric value. For M-code, refer to the separate Parameter List/Alarm List/M-Code List.

If the sequence data contains G65 in **G1** or **G2** item, the data set here will be handled as an argument.

Note 1: Difference between the Manual Program and the EIA/ISO Program
The manual program can be prepared the same as the EIA/ISO program but with certain differences as shown below.

1. In the manual program machining unit, 1 line of tool sequence data corresponds to one block of the EIA/ISO program, which limits the number of data which can be entered in one block. (G: 2 data, S: 1 data, M/B: 1 data, Others: 6 data)
2. In the manual program machining unit, the tool is deviated on a distance entered in the article **ACT-ϕ** on the **TOOL DATA** display (on the nominal diameter in case of the data absence) when the tool diameter correction command (G41 or G42) is given.
3. In G98 mode, value of F cannot be entered with a decimal point. If the method of feed is modified from G98 to G99 or from G99 to G98, the designated feed value of F in the unit data will become blank (****).
4. A MAZATROL program cannot be called up by the use of a subprogram call such as G65.
5. Do not enter G109L for **G1** or **G2**.
6. Do not use address P (waiting) for **DATA-1** to **DATA-6**.

Note 2: When a turning tool is selected for the manual program machining unit, the rotational direction of the turning spindle is determined by the rotational direction set in the tool data. For a milling tool set the rotational direction by the M-code.

Note 3: When the control axis for the cross machining is specified by G110 in the manual program machining unit, cancel the specification in the same unit by G111.

3-20 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 the **[M CODE]** menu key for this unit.

3-20-1 Setting unit data (M-code)

UNo.	UNIT	No.	#		#1	#2	#3	#4	#5	#6	#7	#8	#9	#10	#11	#12
*	M-CODE	[1]	[2]	[3]	[4]	[4]	[4]	[4]	[4]	[4]	[4]	[4]	[4]	[4]	[4]	[4]

[1] No.

Determine the priority machining number (prior machining, subsequent machining). The input of data is done in accordance with the following 3 methods;

- Priority No. for prior machining:
  Enter the number by means of numeric keys. Input range is 0 to 99.
- Priority No. for subsequent machining:
  Press the **[DELAY PRIORITY]** menu key, then enter the number by means of numeric keys. Input range is 0 to 99.

MACHINING PRIORITY No?

	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END		

- Without input: Normal machining:
  The machining order is not specified.

For details, refer to Chapter 4, "PRIORITY FUNCTION FOR THE SAME TOOL."

[2], [3] # (Turret selection and simultaneous machining No.)

- For the machine with the lower turret, select the turret for which M-codes are to be output. The following is displayed:

SET UPPER TURRET	SET LOWER TURRET								

- For the machine with the lower turret, set the simultaneous machining number to be used to operate the upper and lower turrets at the same time.

[4] #1 to #12

Set the desired M-code number in each of the 12 items by means of the following menu or numeric keys. Pressing the **[>>>]** menu key changes the menu **a** → **b** → **c** → a in this order.

01 OPTIONAL STOP	08 FLOOD COOL ON	09 FLOOD COOL OFF	153 MILL-THR COOL ON	154 MILL-THR COOL OFF	206 HD1 CHK OPEN	207 HD1 CHK CLOSE	306 HD2 CHK OPEN	307 HD2 CHK CLOSE	>>>	**a**
901 HEAD1 SELECT	902 HEAD2 SELECT	202 HEAD1 TRN MODE	302 HEAD2 TRN MODE		03 T-SP1 CW	04 T-SP1 CCW	303 T-SP2 CW	304 T-SP2 CCW	>>>	**b**
200 C1-AXIS CONNECT	300 C2-AXIS CONNECT	203 MILL-SP CW	204 MILL-SP CCW	205 MILL-SP STOP					>>>	**c**

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.

Note 3: The M-codes listed in the menu vary from machine to machine.

Note 4: For details of M-codes, refer to the Operating Manual of the machine.

3-21 Head Selection Unit (HEAD)

With the head selection 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 **[SELECT HEAD]** menu key to select this unit.

3-21-1 Setting unit data

UNo.	UNIT	TYPE	HEAD	SPDL
*	HEAD	[1]	[2]	◆

[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).

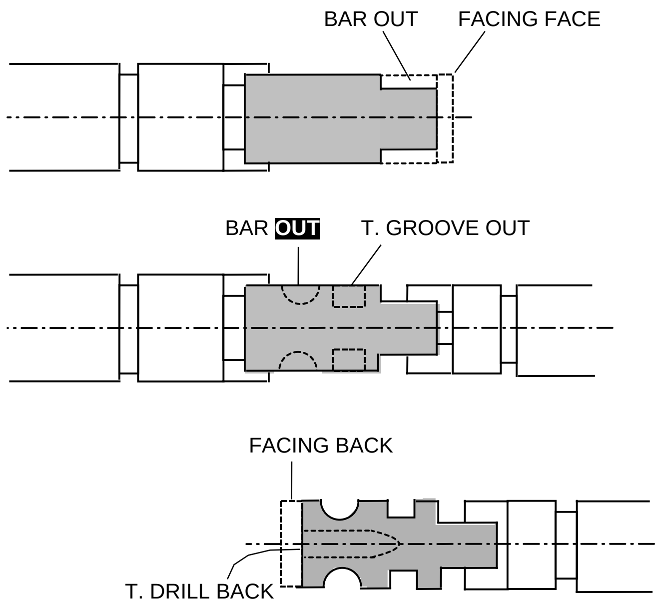
[2] HEAD

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].

```
    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     PAT.   ~  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     PAT.   ~  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

3-22 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.
Press the **[TRANSFER WORKPICE]** menu key to select this unit.

3-22-1 Setting unit data

UNo.	UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	Z-OFFSET	C1	C2	C-OFFSET	LTUR	ESC	TNo.
*	TRANSFER	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]	[11]		[12]	[13]

[1] PAT.

From the following menu, select a transfer pattern.

CHUCK WORK	BAR LOOP	T-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] 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.

[3] SPDL

From the following menu, specify a spindle action (status) during the unit.

0 KEEP	1 FORWARD	2 REVERSE	3 ORIENT	4 C-AXIS POSITION	5 KEEP POSITION				

Specify a spindle action pattern during workpiece rechucking/delivering by selecting one of the items 0 through 5.

0 KEEP:	The spindle does not rotate.
1 FORWARD:	The spindle rotates forward at the speed set previously in parameter **TC58**.
2 REVERSE:	The spindle rotates backward at the speed set previously in parameter **TC58**.
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 POSITION:	The spindle status specfied in the previous unit is held.

[4] 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.

[5] 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.

[6] W1

- When **CHUCK** is selected for data item [1] above, set W-axial machine coordinate of the spindle No. 2 for workpiece delivering.
- When **BAR** is selected for data item [1] above, set W-axial machine coordinate of the spindle No. 2 for workpiece rechucking.

[7] W2

- When **CHUCK** is selected for data item [1] above, set W-axial machine coordinate of the spindle No. 2 after workpiece delivering.
- When **BAR** is selected for data item [1] above, set W-axial machine coordinate of the spindle No. 2 after workpiece rechucking.
- When **MOVE** is selected for data item [1] above, set W-axial machine coordinate of the spindle No. 2 after movement.

[8] Z-OFFSET

Set the Z offset value which becomes valid after delivering the workpiece.

[9] C1

Set C-axial machine coordinate of the spindle No. 1 for workpiece delivering.

[10] C2

Set C-axial machine coordinate of the spindle No. 2 for workpiece delivering.

[11] C-OFFSET

Set the C offset value which becomes valid after delivering the workpiece.

[12] LTUR ESC

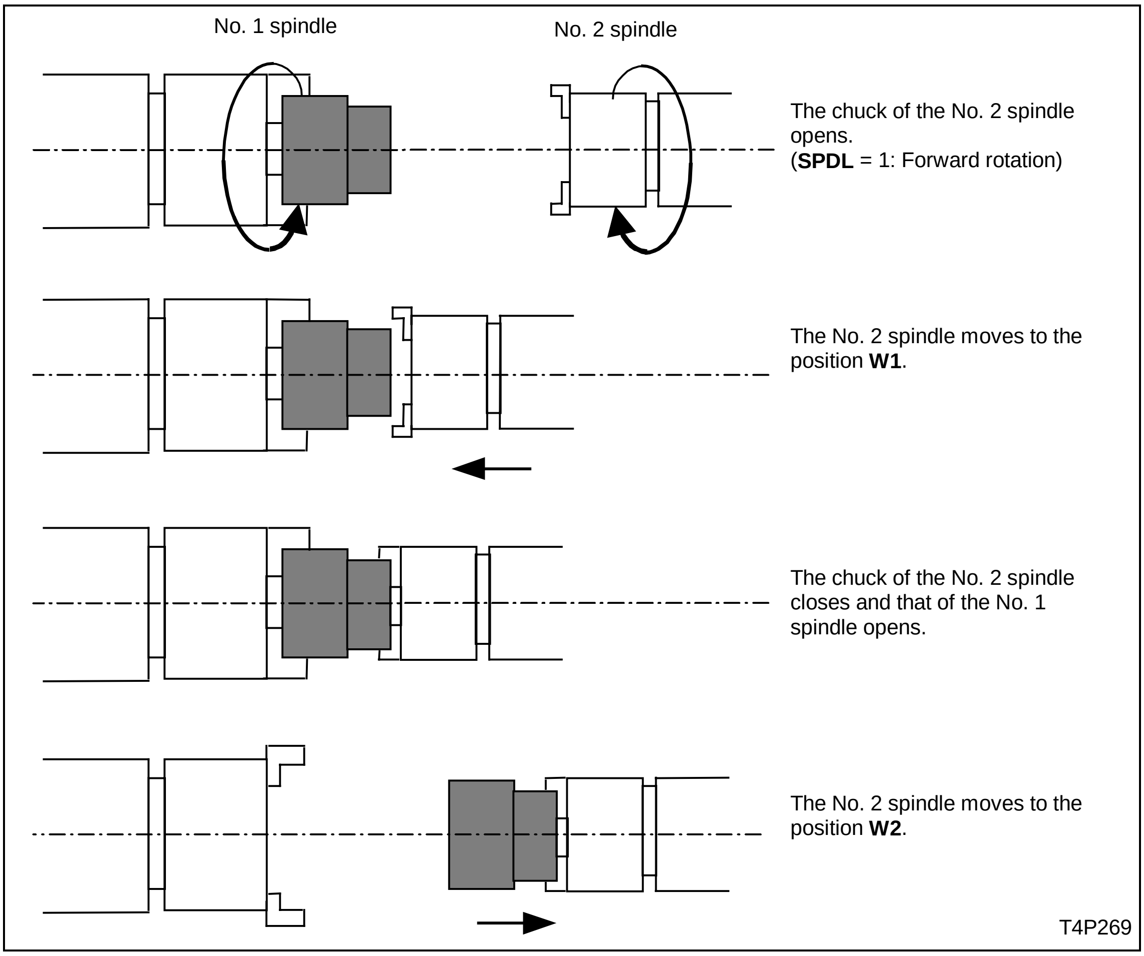
Set the Z-axial machine coordinate of the anti-interference standby position to which the lower turret is to be moved for workpiece delivering.

[13] TNo.

Specify the TNo. of the lower-turret tool to be indexed during workpiece delivering.

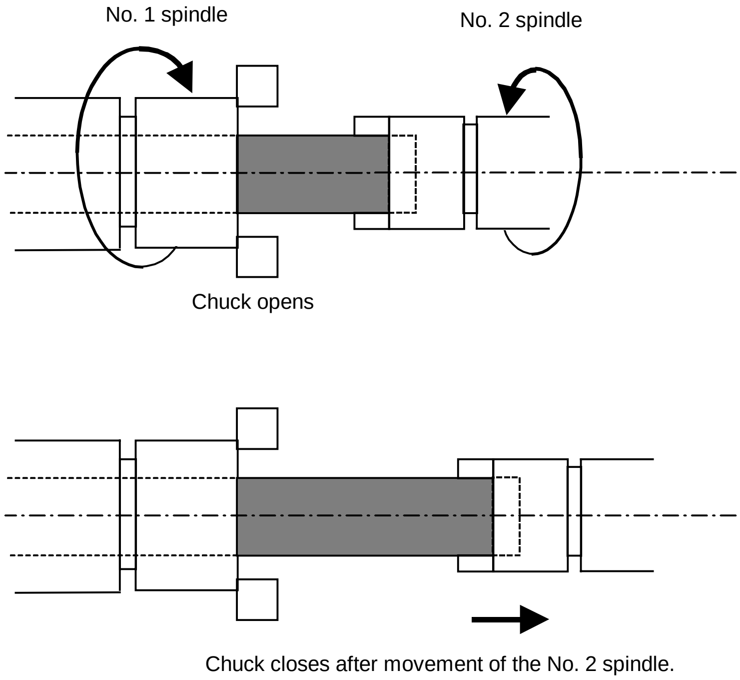
Example 1: TRANSFER CHUCK

UNo.	UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	Z-OFFSET	C1	C2	C-OFFSET	LTUR ESC	TNo.
*	TRANSFER	CHUCK	1→2	1	1	◆	***	***	***	◆	◆	◆	***	*



Example 2: TRANSFER BAR

UNo.	UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	Z-OFFSET	C1	C2	C-OFFSET	LTUR	ESC	TNo.
*	TRANSFER	BAR	1	2	◆	1	◆	***	***	◆	◆	◆	***		*



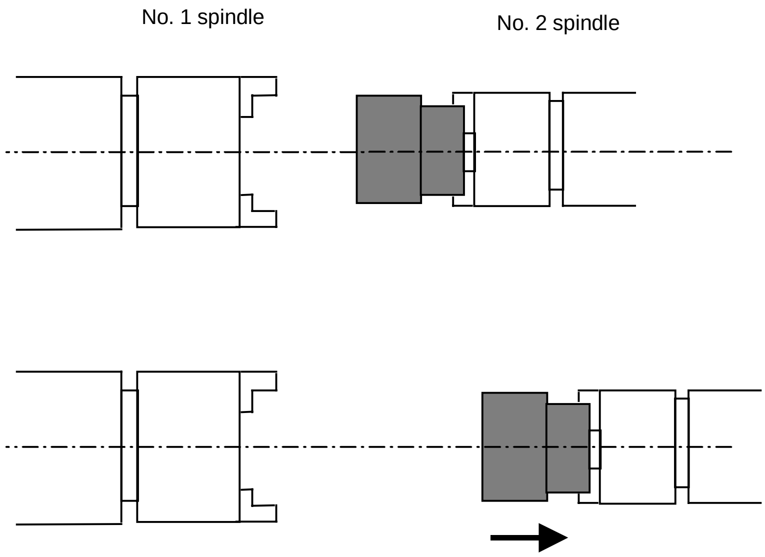
The chuck of the No. 1 spindle opens.
(**SPDL**=2: Reverse rotation)

The No. 2 spindle moves to the position **W2**.
Then the chuck of the spindle indicated in item **HEAD** will close according to **CHUCK**=1.

T4P270

Example 3: TRANSFER MOVE

UNo.	UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	Z-OFFSET	C1	C2	C-OFFSET	LTUR	ESC	TNo.
*	TRANSFER	MOVE	◆	◆	◆	◆	◆	***	***	◆	◆	◆	***		*



The No. 2 spindle moves to the position **W2**.

T4P271

3-23 Subprogram Unit (SUB PRO)

When the same movement is repeated in a machining process or when a common movement is used in several programs, it is desirable to prepare and call up a special program for this movement with this subprogram unit.

In this case, the calling side is referred to as the Main program and the called side is referred to as the Subprogram. Moreover, the call of the sub-program is called Nesting. The subprogram can be used both in the MAZATROL program and in the EIA/ISO program.

However, the maximum number of nesting is nine for the MAZATROL program and eight for the EIA/ISO program.

Press the **[SUB PROGRAM]** menu key to select this unit.

3-23-1 Setting unit data

UNo.	UNIT	WORK No.	NUM.	#	
*	SUB PRO	[1]	[2]	[3]	[4]

[1] WORK No.

Enter the work number of the desired subprogram. The following menu is displayed:

MEASURE MACRO						PROGRAM FILE			

Enter the work number using the numerical keys, or press the **[PROGRAM FILE]** menu key and select the work number from the work-Nos. listing window.

Note: To call up as a subprogram the measuring macro whose execution will result in origin coordinates or tool correction data being changed, press the **[MEASURE MACRO]** menu key and then select the work number while the menu is displayed in reverse mode. Selecting the program while the menu is displayed in reverse mode displays the selected work number in yellow. Selecting the program while the menu is not displayed in reverse mode displays the selected work number in white.

[2] NUM.

Enter the number of repetitions of the subprogram.

If no data is entered here, the subprogram will be executed one time.

[3], [4] # (Turret selection and simultaneous machining No.)

For the machine with the lower turret, select the turret for which subprogram call are to be output. The following menu is displayed:

SET UPPER TURRET	SET LOWER TURRET								

For the machine with the lower turret, set the simultaneous machining number to be used to operate the upper and lower turrets at the same time.

3-23-2 Setting sequence data

UNo.	UNIT	WORK 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]

<u>[1] [2] ARGM</u>

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 Z/C offsets in the subprogram inherit the Z/C offsets of the main program, registered in the **SET UP MANAG.** display.

Note 3: When control returns from the subprogram to the MAZATROL program, the settings of the coordinate systems in the subprogram are canceled and control is returned to the coordinate systems existing before execution of the subprogram unit.

Note 4: The subprogram does not inherit the active modal information within the main program. The subprogram is executed in accordance with reset modal information.

Note 5: During the return of control from the subprogram to the MAZATROL program, modal information is reset to the status existing before subprogram unit execution.

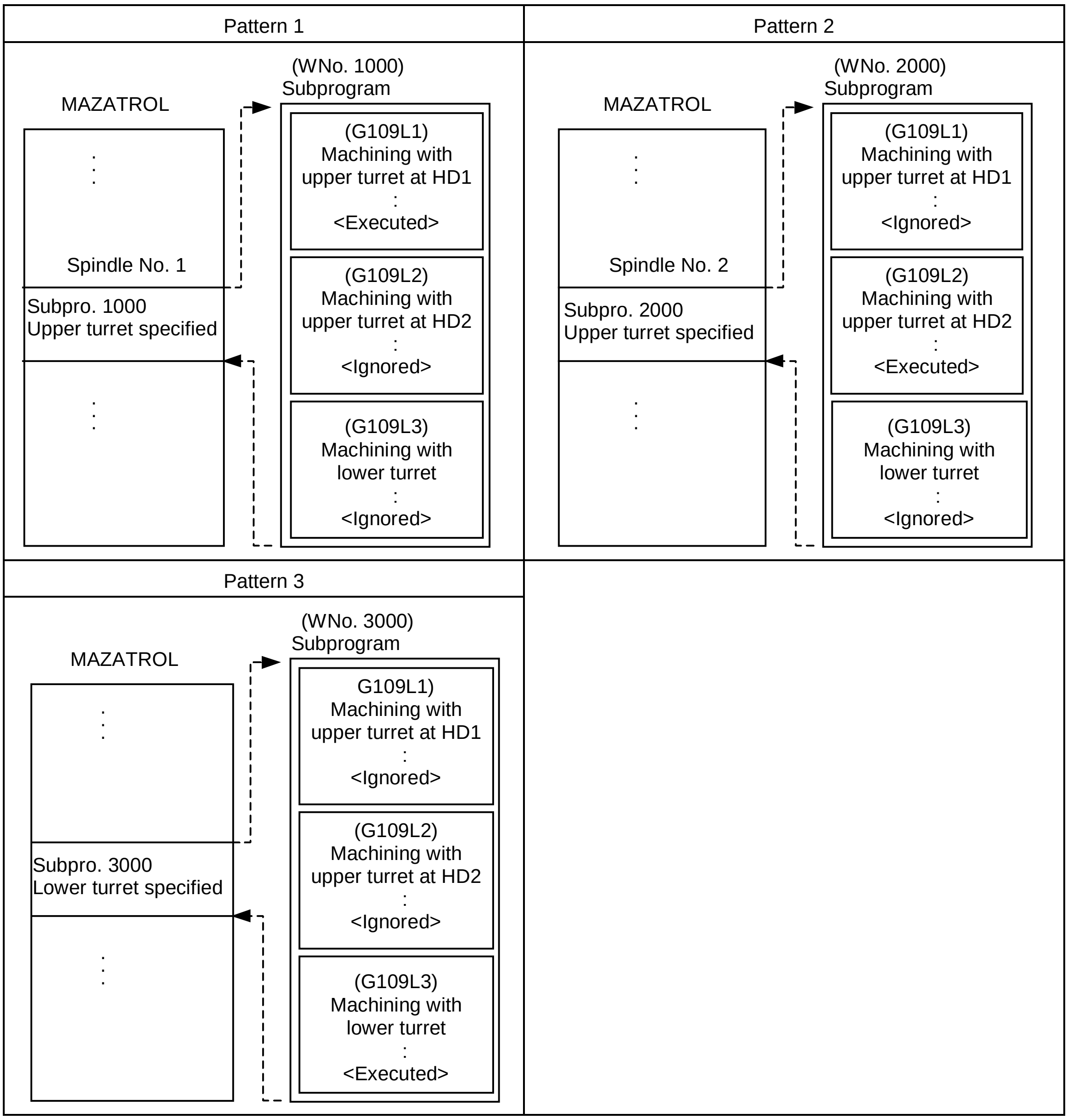
Note 6: The subprogram will be executed only once, even if “0” is set as the number of program repetitions.

Note 7: Do not use a Cross command (G110) in the subprogram to be called.

Note 8: Do not use a queuing P-code in the subprogram to be called.

Note 9: The MAZATROL program cannot be called up from the EIA/ISO subprogram that has been called up by the subprogram unit.

Note 10: Execution of the subprogram including the upper/lower turret command (G109L_) uses only the program sections for the head/turret specified in the subprogram unit. Other program sections are ignored. See the diagram below.



3-24 Add-In MAZATROL Unit

Select the Add-in MAZATROL unit to call up an Add-in MAZATROL as a subprogram from the MAZATROL program.

3-24-1 Setting unit data

UNo.	UNIT	WORK No.	REPEAT	#	
*	[1]	[2]	[3]	[4]	[5]

[1] UNIT

Name of the unit is displayed.

[2] WORK No.

The work number of the Add-in MAZATROL is displayed.

[3] REPEAT

Enter the number of repetitions of the Add-in MAZATROL.
If no data is entered here, the Add-in MAZATROL will be executed one time.

[4], [5] # (Turret selection and simultaneous machining No.)

- For the machine with the lower turret, select the turret for which command for calling Add-in MAZATROL is to be output. The following is displayed:

SET UPPER TURRET	SET LOWER TURRET								

- For the machine with the lower turret, set the simultaneous machining number to be used to operate the upper and lower turrets at the same time.

3-24-2 Setting sequence data

UNo.	UNIT	WORK No.		REPEAT
*	***	***		***
SEQ	ARGM 1	ARGM 2	ARGM 3	ARGM 4
1	[1] [2]	[1] [2]	[1] [2]	[1] [2]

[1] [2] ARGM 1 to 4

Address and title are displayed in [1].
Input the data of an argument in [2].

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: 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 2: In the control mode where the coordinate system of the main program is conveyed to the subprogram (bit 7 of **F161** is set to 1), the Z-offset data registered in the **SET UP MANAG.** display for the main program will remain valid for a subprogam in which the MAZATROL coordinate system is selected.

3-24-3 Help function on Add-in MAZATROL

When the name of the Add-in MAZATROL to be called up, the titles of macro arguments, and other information are defined in any internal text file of the hard disk, the titles of the macro arguments can be displayed in sequence lines, and massage can be displayed for argument setting, just by specifying that Add-in MAZATROL as a subprogram.
A special help display for Add-in MAZATROL can also be created.
Thus, macro arguments can be easily modified by providing the help display.

Note 1: Assign a file name of "EIAMACRO.txt" to the text file to be created.

Note 2: The language of the text file is selected according to the setting of parameter **K11**. Store the text file into the folder matching the display language.
Example: For English "C:\nm64mdata\eng\EIAMACRO.txt".

Note 3: The help display does not always need to be created.

Note 4: This function becomes valid, only when, after the text file and the bmp or png file of the help display have been saved on the hard disk, bit 3 in parameter **F81** is set to "1" and then power is turned off and back on.

A. Format of the text file

```
[EIAMACRO-[1]]
UnitName=[2]
MenuName=[3]
WNo.=[4]
HelpFileName=[5]
Data[6]-[7]=[8], [9], [10], [11], [12], [13]
Data[6]-[7]=[8], [9], [10], [11], [12], [13]
Data[6]-[7]=[8], [9], [10], [11], [12], [13]
                  :
                  :
                  :
```

Argument-specifying area
(Up to 24 arguments)

EIA macroprogram-specifying area
(Up to 9 programs)

[1] EIA macro number

Specify the position of the desired macroprogram menu item. Macroprogram menu items are displayed at the time of unit selection in the MAZATORL programs.
A maximum of nine menu items can be displayed.

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

[2] Title to be displayed in the **PROGRAM** display

Enter the title for the Add-in MAZATROL which is to be displayed on the **PROGRAM** display when the menu key corresponding to the position specified in data item [1] above is pressed.
Up to eight ASCII characters can be displayed as the title.

Example: `UnitName=PTHR OUT`

```
UNo.    UNIT     WORK No.    REPEAT
 *    PTHR OUT
```

[3] Menu item for the **PROGRAM** display

Enter the menu item to be displayed in the menu item display position that was specified in [1] above.
In both upper and lower lines, up to eight ASCII characters can be displayed as the menu item.

Example: `MenuName=P-THR   OUT`

```
P-THR
 OUT
```

[4] Program number of the Add-in MAZATROL

Specify the program number of the Add-in MAZATROL.
Program numbers 1 to 999999999 can be displayed.

Example: `WNo.=777`

UNo.	UNIT	WORK No.	REPEAT
*	PTHR OUT	777	

Remark: Program numbers are automatically assigned when the unit is selected.

[5] File name of the help display

Assign a file name to the graphics within the help display that is to be displayed when macro arguments are entered.
This file name must be entered in up to 30 characters, except for an extension.

Example: `HelpFileName=PTHROUT`

Remark: This line can be omitted when a help display is not to be created.

Note 1: Use the following folder to store the graphics within the help display:
c:\nm64mdata________.bmp (.png)

Note 2: When drawing graphics on the help display, use a format of 900 dots (horizontal) by up to 440 dots (vertical).

[6] Sequence number of the argument, [7] Display position, [8] Address, [9] Title, [10] Message

[6]: Enter the sequence number of the macro argument to be displayed. Up to six SEQ lines can be entered.
[7]: Enter the display position (ARGM 1 to 4) of the macro argument.
[8]: Enter the address of the macro argument in the alphabet (A to Z).
[9]: Enter the title of the macro argument. Up to seven ASCII characters can be displayed as the title.
[10]: Enter the message to be displayed when the macro argument is set. Up to 29 ASCII characters can be displayed as the message.

Example: Enter data as follows to specify 2 as the sequence number, 4 as the ARGM column number, I as the address, "STP-Z" as the title, and "SPT Z" as the message:
`Data2-4=I,STP-Z,SPT Z`

SEQ	ARGM 1	ARGM 2	ARGM 3	ARGM 4
1	[8]:[9]	[8]:[9]	[8]:[9]	[8]:[9]
2	[8]:[9]	[8]:[9]	[8]:[9]	I:STP-Z
3	[8]:[9]	[8]:[9]	[8]:[9]	[8]:[9]
4	[8]:[9]	[8]:[9]	[8]:[9]	[8]:[9]
5	[8]:[9]	[8]:[9]	[8]:[9]	[8]:[9]
6	[8]:[9]	[8]:[9]	[8]:[9]	[8]:[9]

[11] Display of the argument title on the help display

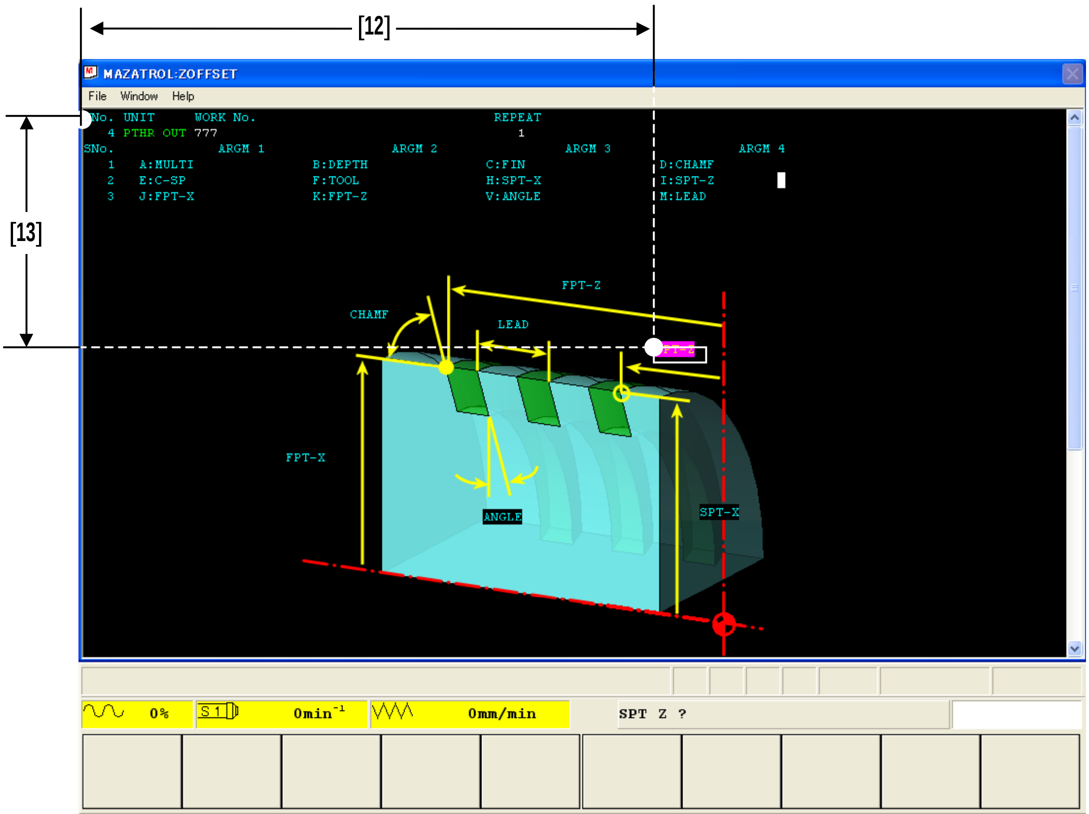
Select whether the title of the macro argument that was specified in data item [9] above is to be displayed on the help display.
Enter 1 to display the title. Enter 0 if the title is not to be displayed.

[12] Horizontal display position in the help display, [13] Vertical display position in the help display

[12]: Specify in dots where in the horizontal direction of the help display the title of the macro argument that was specified in data item [9] is to be displayed.
[13]: Specify in dots where in the vertical direction of the help display the title of the macro argument that was specified in data item [9] is to be displayed.

Note: If 0 was entered in [11], enter 0 in both [12] and [13].

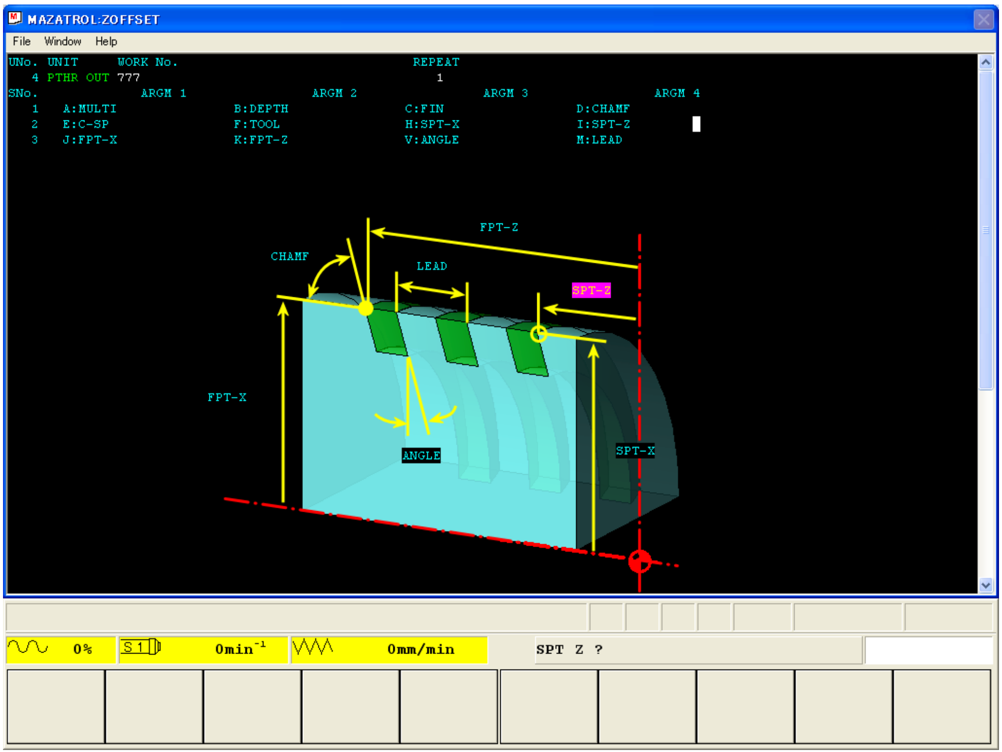
Example: Specify the upper left of the title as the display position and enter data as follows:
For "STP-Z", enter 580 in [12] and 235 in [13].



D740PA167E

B. Example of text file entry

Create the text file [EIAMACRO-1] for the Add-in MAZATROL numbered “777” that contains the following arguments:



D740PA168E

```
[EIAMACRO-1]
UnitName=PTHR OUT
MenuName=  P-THR   OUT
WNo.=777
HelpFileName=PTHROUT
Data1-1=A,MULTI,NUMBER OF ENTRANCE,0,0,0
Data1-2=B,DEPTH,DEPTH PER PASS,0,0,0
Data1-3=C,FIN,FINISH ALLOWANCE,0,0,0
Data1-4=D,CHAMF,CHAMFERING ANGLE,1,270,200
Data2-1=E,C-SP,CUTTING SPEED,0,0,0
Data2-2=F,TOOL,TOOL NUMBER,0,0,0
Data2-3=H,SPT-X,SPT X,1,625,400
Data2-4=I,SPT-Z,SPT Z,1,580,235
Data3-1=J,FPT-X,FPT X,1,205,345
Data3-2=K,FPT-Z,FPT Z,1,485,170
Data3-3=V,ANGLE,THREAD ANGLE,1,405,405
Data3-4=M,LEAD,THREAD LEAD,1,420,210
```

Remark 1: Enter a comment after the semicolon (;).
[Ex] `UnitName=PTHR OUT` ; `Specification of unit name`

Remark 2: The file of the graphics in the above help display is “c:\nm64mdata\PTHROUT.bmp”.

3-25 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 to select this unit.

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING	MANUAL PROGRAM			END	SHAPE CHECK	>>>

3-25-1 Setting unit data

UNo.	UNIT	CONTI.	REPEAT	SHIFT	NUMBER	ATC	RETURN	LOW RET.	WORK No.	EXECUTE
*	END	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] CONTI.

Specify whether you want to carry out the machining operation repeatedly in succession.

- Set 0 to execute the current program once.
- Set 1 to execute the current program perpetually.
- Always set 1 for the subprogram. At this time, the settings in **REPEAT** and **SHIFT** become invalid.

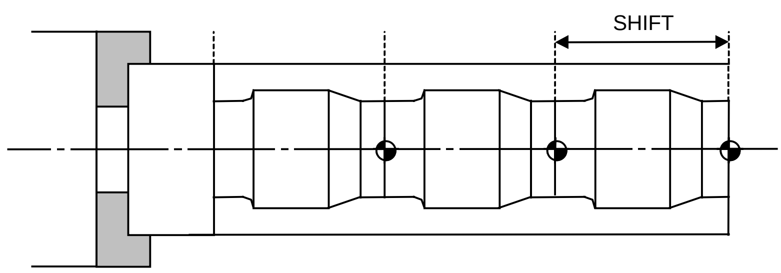
Note: If no data has been set in this item, the NC will interpret that 0 has been set.

[2] REPEAT

If the current program is to be executed repeatedly, set the desired number of times of execution.

[3] 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.



T4P280

For such machining, set the desired shift amount of the program origin in this item.

Note 1: If no data has been set in this item, the shift amount will be regarded as 0.

Note 2: Data must not be set in this item if a measurement unit is to be executed. Setting data other than 0 will cause alarm **657 ILLEGAL NUMBER INPUT** at the second measurement.

Note 3: As for repetitive machining on a single workpiece, the following condition must be satisfied:

LENGTH	>	REPEAT × SHIFT	+	WORK FACE
(Common data)		(END unit)		(Common data)

[4] NUMBER

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.

[5] ATC

Specify the ATC movement at the end of machining.

- Set 0 if the tool is not returned.
- Set 1 if the axes are moved after returning the tool.
- Set 2 if the tool is returned after axis movement

Note: If no data is set, it is regarded that 1 has been set.

[6] RETURN

Specify from the menu the position to which the turret is to be returned after machining.

TOOL CHANGE POSITION	ZERO POSITON	FIXED POSITION			ARBITRAR POSITION				

Note 1: The tool change position of the turret is specified by parameter **SU10**, the fixed position by parameter **M5**. To return the turret to a specific position, do not change the setting of parameter **M5**. Instead, select **[ARBITRAR POSITION]** and specify the coordinates of that position. (Refer to the separate Parameter List/Alarm List/M-Code List for further details.)

Note 2: If no data is set in this item, the tool change position will be regarded as having been specified.

Note 3: During returning, linear axes move at first and then rotational axes.

Note 4: If the setting for **CONTI.** is “1”, whether **RETURN** is to be executed can be selected by assigning the appropriate value to bit 5 in parameter **TC144**.
0: Non-execute, 1: Execute
Even when “0” (Non-execute) is set, however, **RETURN** will be executed if **REPEAT** is specified or if the target count in the **POSITION** display is reached with **NUMBER** set equal to “1” and the machine comes to a stop.

Note 5: The axes for which the returning function is valid differs according to the value specified in **RETURN**. The returning function is performed on either of the following axes:

RETURN	Axes for which the returning function is valid
TOOL CHG (**SU10**)	X-axis, Z-axis
HOME	X-axis, Y-axis, Z-axis
FIXED PT (**M5**)	X-axis, Y-axis, Z-axis
ARB PT	Axes for which the returning position has been set in arbitrary position sequence

Even if **RETURN** = **HOME**, axes other than the X-, Y-, or Z-axes will not be returned to the respective home positions. To return axes other than the X-, Y-, or Z-axes to the home positions, select **[ARBITRAR POSITION]** and then specify the desired axes in the arbitrary position sequence.

Example: If "X0, Y0, Z0, B0, C0" is set in the arbitrary position sequence, the X-, Y-, Z-, B-, and C-axes will be returned to the respective home positions.

[7] LOW RET.

Specify from the menu the position to which the lower turret is to be returned after machining.

TOOL CHANGE POSITION	ZERO POSITON	FIXED POSITION	LOWER TURRET POS.1	LOWER TURRET POS.2	ARBITRAR POSITION				

Note 1: The tool change position of the lower turret is specified by parameter **SU10**, the fixed position by parameter **M5**, lower-turret retraction position 1 by parameters **SU97**, **SU98**, and lower-turret retraction position 2 by parameters **SU99**, **SU100**.
When lower-turret retraction position 1 is set, the lower turret moves to the retraction position, after the tool specified by parameter **SU52** is indexed. When lower-turret retraction position 2 is set, the lower turret moves to the retraction position, after the tool specified by parameter **SU53** is indexed. No tool index occurs when 0 is set to **SU52** or **SU53** respectively.
(Refer to the separate Parameter List/Alarm List/M-Code List.for further details.)

Note 2: If no data is set in this item, the tool change position will be regarded as having been specified.

Note 3: When the lower turret is present at a returning position, selection of that retraction position in **LOW RET.** does not move the turret. Selection of a different returning position or of the home position, fixed position, or any other position moves the lower turret directly to the particular selected position.

[8] WORK 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.

[9] EXECUTE

Set whether operation is to be executed after call of the program specified in item [8].

- Set 1 to execute operation based on the called program.
- Set 0 if the program is only to be called and operation is not to be executed.

Examples of program execution mode

The program execution mode is determined by the data that has been set in items [1], [2], [3], [8] and [9] above.

If the following data has been set for the end unit of the program of work number A:

	CONTI.	REPEAT	SHIFT	WORK No.	EXECUTE
Example **1**	0 or blank	◆	◆	Blank	◆
Example **2**	0 or blank	◆	◆	B	1 (execute)
Example **3**	0 or blank	◆	◆	B	0 (not execute)
Example **4**	1	Blank	◆	◆	◆
Example **5**	1	N	0 or blank	Blank	◆
Example **6**	1	N	0 or blank	B	1 (execute)
Example **7**	1	N	0 or blank	B	0 (not execute)
Example **8**	1	N	s	Blank	◆
Example **9**	1	N	s	B	1 (execute)
Example **10**	1	N	s	B	0 (not execute)

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 following this, the program of work number **B** is executed.

3: 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.

4: The program of work number **A** is executed repeatedly.

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 following this, the program of work number **B** is executed.

7: 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.

8: 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.

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 following this, the program of work number **B** is executed.

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.

3-25-2 Setting sequence data

Any position can be specified in the arbitrary-position specification sequence by specifying **ARB PT** in **RETURN** or **LOW RET.**

```
SNo.   DATA-1   DATA-2   DATA-3   DATA-4   DATA-5   DATA-6   DATA-7   DATA-8   DATA-9
  *
```

The arbitrary-position specification sequence allows coordinates to be specified for each axis and moves the axis to the specified coordinate position in rapid feed mode.

Note: “Arbitrary Position” (**ARB PT**) must be selected in **RETURN** or **LOW RET.** before it becomes possible to move the axis for which any position has been specified.

```
UNo.    UNIT     CONTI.    REPEAT  SHIFT   NUMBER    RETURN    LOW RET.  WORK No. EXECUTE
  *      END                                           END       ARB PT

SNo.    DATA-1   DATA-2   DATA-3   DATA-4   DATA-5   DATA-6   DATA-7   DATA-8   DATA-9
        X1 100
```

Since the returning position is the home position (**END**), the X1-axis does not move to position 100. The axis will be moved to the home position.

3-26 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.

See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for further details of this unit.

3-26-1 Procedure for calling up the SIMULTAN 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

			M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPICE SHAPE	>>>

(3) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

SELECT HEAD	TRANSFER WORKPICE	PROCESS END					SIMUL.		>>>

(4) Press the **[SIMUL.]** menu key.

3-26-2 Setting unit data

UNo.	UNIT	No.	SIMUL.No.	RPM
*	SIMULTAN	[1]	[2]	[3]

[1] No.

Specify the priority machining number for the the simultaneous machining

[2] SIMUL. No.

Specify the group number for the simultaneous machining using both turrets.

[3] RPM

Enter the turning spindle speed of the simultaneous machining group specified in [2] above.

3-27 Two-Workpiece Machining Unit (2 WORKPC)

If you are using a machine equipped with a lower turret, select this unit when machining the 2 workpieces on the No. 1 and No. 2 spindles simultaneously, using the upper and lower turrets.

See Chapter 5, “LOWER-TURRET CONTROL FUNCTIONS”, for further details of this unit.

3-27-1 Procedure for calling up the 2 WORKPC 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

			M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPICE SHAPE	>>>

(3) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

SELECT HEAD	TRANSFER WORKPICE	PROCESS END					SIMUL.	2 WORKPC MODE	>>>

(4) Press the **[2 WORKPC MODE]** menu key.

3-27-2 Setting unit data

UNo.	UNIT	PAT.	SP1/SP2
*	2 WORKPC	[1]	[2]

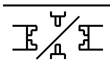
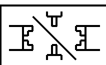
[1] PAT.

Specify the start and end points for machining 2 workpieces simultaneously. The following menu is displayed.

START	END								

[2] SP1/SP2

When machining the workpieces on the No. 1 and No. 2 spindles, specify which turret to use: upper or lower turret. The following menu is displayed.

3-28 Coordinate Measuring Unit (MMS)

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.

3-28-1 Procedure for calling up the MMS 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

			M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPICE SHAPE	>>>

(3) Press the **[WORKPICE MEASURE]** menu key.

3-28-2 Setting unit data

UNo.	UNIT	TOOL	NOM-ϕ	No.	U.SKIP	$
	MMS	TOL SENS	[1]	[2]	[3]	◆

[1] NOM-ϕ

Specify the nominal diameter of the feeler. Enter the approximate diameter of the point of feeler by means of numeric keys.

[2] No.

Specify the priority machining number for the coordinate measuring.

[3] U. SKIP

Specify whether the MMS unit is executed or not.

3-28-3 Setting sequence data

SNo.	PTN	X	Y	Z	C	X	R	D/L	K	DIR
1	[1]	[2]	[2]	[2]	[2]	◆	[3]	[4]	[5]	[6]

[1] PTN

Select the type of measurement from the following menu.

Z FACE	C FACE		C GROOVE		C STEP		CALIBR.	PTN END	>>>

For details, refer to "Type of measurement."

[2] X, Y, Z, C

Specify the measurment starting position by means of numeric keys.

[3] R

Specify the surface coordinates to be measured by means of numeric keys. The contents of the entry vary according to the type of measurement.

[4] D/L

Specify the width of the groove, the width of the projection, etc., or specify for measurement retry the angle of shift of the measurement start point, by means of numeric keys.
The contents of the entry differ according to the type of measurement. For details, see Subsection “Type of measurement.”

[5] K

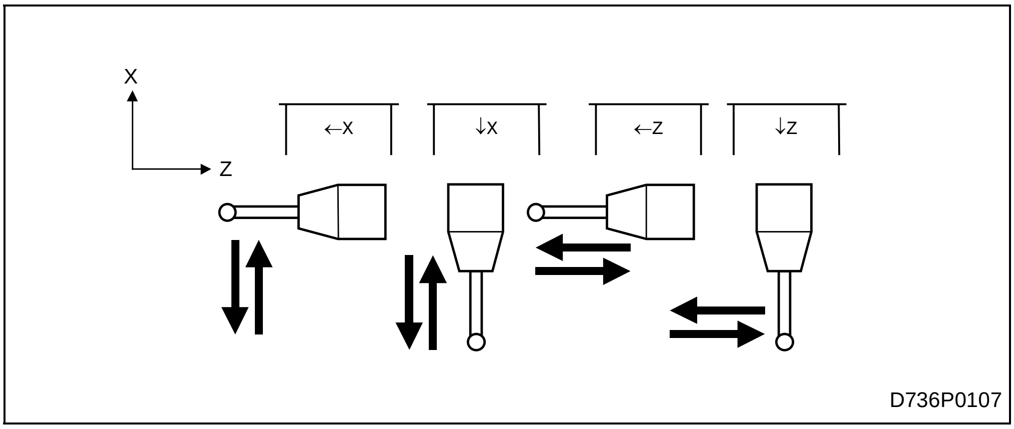
Specify the feed distance at skip speed by means of numeric keys. The term skip speed means the feed speed at the time when the probe comes into contact with the surface to be measured.

[6] DIR.

Select from the menu the direction (arrow) of the milling spindle head during measurement, and the approach/escape direction to be applied to the measurement.

← Z	← X	↓ Z	↓ X	→ Z	→ X				

Selection of the **[←X]** menu key specifies the B-axis (milling spindle head) facing in the direction of the arrow (left) and the approach/escape in X-axial direction during the measurement.



Note 1: During the process of the measurement movement, the block by block stop and rapid feed deceleration are effective, but not the override of skip feed.

Note 2: 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.

Note 3: Before execution of the coordinate measuring unit turn off the symmetrical image function. The measurement movement after contact with the workpiece may not be correctly performed if the symmetrical image function is valid.

3-28-4 Type of measurement

Select the type of measurement for the offset of the coordinates system. The four types of measurement are available.

Measurement of the reference surface .... **Z-FACE**, **C-FACE**
Measurement of groove center .... **C-GRV**
Measurement of center of projection width ... **C-STP**

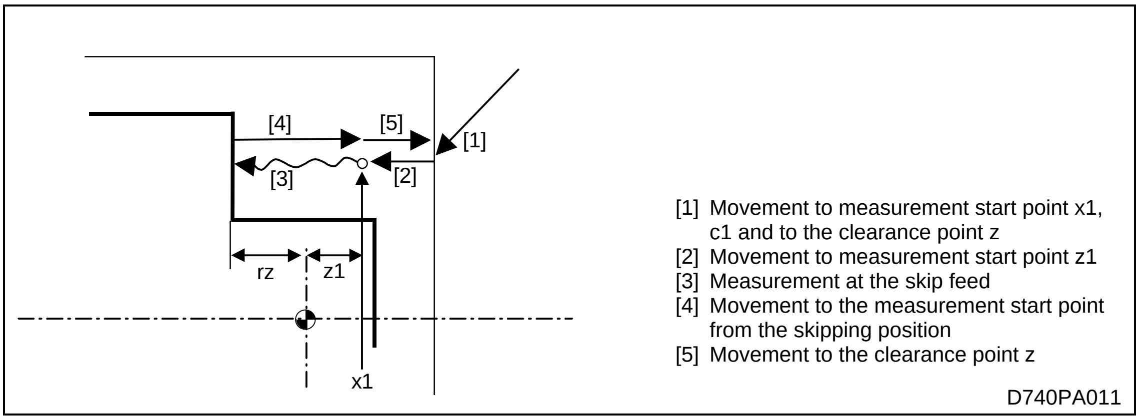
Each measurement type is described in the following descriptions 1 to 4:

1. Z-FACE

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	X	R	D/L	K	DIR.
1	Z-FACE	x1	y1	z1	c1	◆	rz	◆	k	←X

◆: Not necessary to be set here.

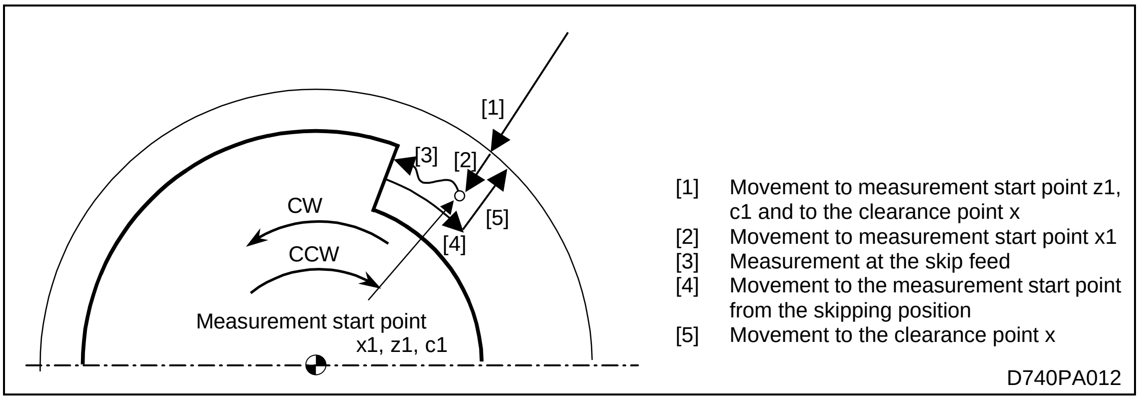


2. C-FACE

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	X	R	D/L	K	DIR.
1	C-FACE	x1	◆	z1	c1	◆	rc	ℓ	◆	←X

◆: Not necessary to be set here.



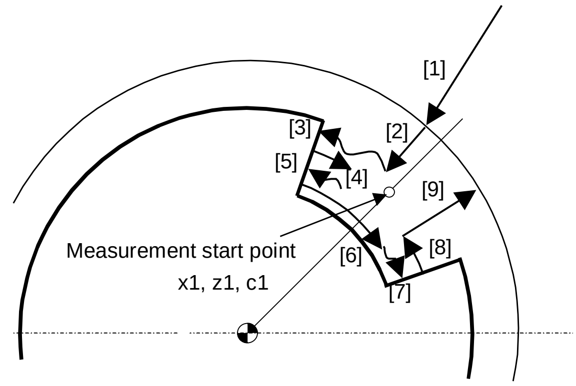
Remark: The measurement direction is determined as follows: CW, when c1 (measurement start point C) ≥ rc (coordinate C of the surface to be measured) and CCW, when c1 (measurement start point C) < rc (coordinate C of the surface to be measured).

3. C-GRV

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	←X

◆: Not necessary to be set here.



[1] Movement to measurement start point z1, c1 and to the clearance point x
[2] Movement to measurement start point x1 at the measuring approach speed
[3] Pre-measurement at the measuring approach speed
[4] Movement through measuring return stroke "k" from the skipping position
[5] The first measurement at the skip feed
[6] Movement to the measuring start point at the opposite side
[7] The second measurement at the skip feed
[8] Movement through measuring return stroke "k" from the skipping position
[9] Movement to the clearance point x

D740PA014

Remark 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 later in this manual.

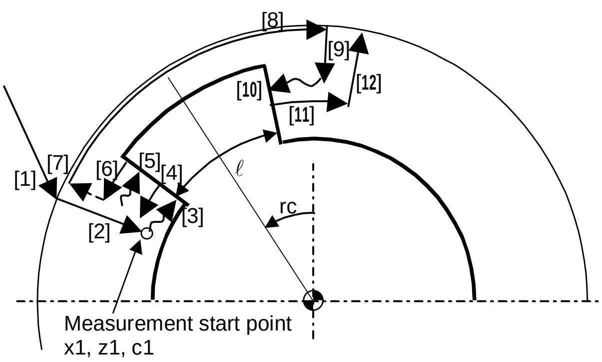
Remark 2: If 0 is entered for the amount of measurement return, only pre-measurement at the measuring approach speed for one side will occur and both-side measurement at the measuring speed will not occur.

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	X	R	D/L	K	DIR.
1	C-STP	x1	◆	z1	c1	◆	rc	ℓ	k	←X

◆: Not necessary to be set here.



[1] Movement to measurement start point z1, c1 and to the clearance point x
[2] Movement to measurement start point x1
[3] Pre-measurement at the measuring approach speed
[4] Movement through measuring return stroke "k" from the skipping position
[5] The first measurement at the skip feed
[6] Movement through measuring return stroke "k" from the skipping position
[7] Movement to the clearance point x
[8] C-axis movement to the measurement start point at the opposite side.
[9] X-axis movement to the measurement start point at the opposite side.
[10] The second measurement at the skip feed
[11] Movement through measuring return stroke "k" from the skipping position
[12] Movement to the clearance point x

D740PA016

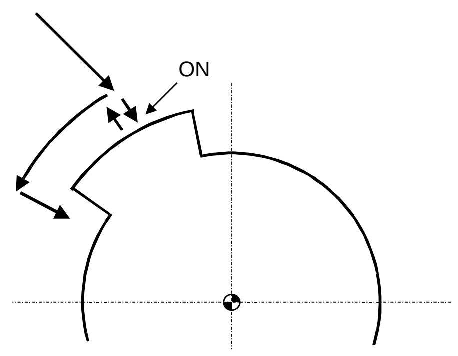
Remark 1: If the sensor operates during movement at the measuring approach speed in step [2], this step will be repeated again (this is referred to as the retry function). The retry function is described below.

Remark 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 **K21** (Measurement retry width) / 100] and then returns to measurement.
The retry function is executed the number of times specified in parameter **K22** (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 **K22** (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 **K22** × **K21** < 100, measurement is repeated as often as the setting of **K22**.
If **K22** × **K21** ≥ 100, measurement is repeated as often as the number of times obtained by rounding any fractions of 100/**K21**.



D736P0139

3-29 Workpiece Measuring Unit (WORK MES)

The workpiece measuring unit uses a touch sensor to measure the dimensions of the workpiece.

Also, measurement results are used for automatic setting of tool correction data.

The workpiece measuring unit measures coordinates after indexing the spindle head in the direction specified by **DIR.** in sequence data, not to the B-axis angle specified in the index unit. Enter the coordinates of actual axes of the machine coordinate system as the coordinates in sequence data.

3-29-1 Procedure for selecting workpiece measuring 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[>>>]** menu key.

➔ The following menu will be displayed.

			M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPICE SHAPE	>>>

(3) Press the **[WORKPICE MEASURE]** menu key.

3-29-2 Setting the unit data

UNo.	UNIT	COMPENSATE	OFS-TOOL	COMP.DATA	SNS-TOOL	No.	#	INTERVAL	OUTPUT
	WORK MES	[1]	[2]	[3]	[4] TOL SENS	[5]	[6]	[7]	[8]

[1] COMPENSATE

Select from the menu whether the measurement results are to be used to correct tool data.

YES	NO								

[2] OFS-TOOL

Specify by its tool name, nominal diameter (nominal size) and identification code, the tool for which the measurement results are to be incorporated.

If **NO** has been specified in **COMPENSATE**, a ◆ mark is displayed to indicate that no data can be entered.

[3] 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 **COMPENSATE**, a ◆ mark is displayed to indicate that no data can be entered.

[4] SNS-TOOL

Enter the nominal diameter and identification code of the touch sensor.

[5] No.

Enter the machining priority number.

[6] #

Specify the retraction position of the lower turret during measurement.

[7] INTERVAL

Specify the intervals at which the workpiece measuring unit is to be executed.

[8] 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. The text files that can be sent to HDD are up to 100 MB in file size. (This maximum size can be changed using parameter **DPR8**.)

2: Output to a serial printer via an RS-232C interface

Note: Specify output items in parameter **F112**.

3-29-3 Setting the sequence data

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM−	BASE	DIR
1	[1]	[2]	[2]	[2]	[3]	[3]	[3]	[4]	[5]	[6]	[7]

[1] PTN

Select a workpiece measuring pattern from the menu.

A press of the **[>>>]** menu key displays menus in the order of [1]→[2] →[3] →[1].

OUTER X DIA	OUTER Y DIA	INNER X DIA	INTER Y DIA		X GRV	Y GRV	Z GRV		>>>	[1]
X WIDTH	Y WIDTH	Z WIDTH	+X STEP	-X STEP	+Y STEP	-Y STEP	+Z STEP	-Z STEP	>>>	[2]
INNER GRV	INNER WIDTH	EXT M.TOOL	EXT T.TOOL						>>>	[3]

When **[OUTER X DIA]** or **[INNER X DIA]** is selected, specify the measuring method next.

0: Both-side measurement

1: Single-side measurement

[2] SPT-X, SPT-Y, SPT-Z

Specify the starting position of measurement. Setup data items differ according to the selected measuring pattern.

[3] FPT-X, FPT-Y, FPT-Z

Specify the ending position of measurement. Setup data items differ according to the selected measuring pattern.

[4] T LIM+

Set the upper-limit value of the tolerance.

[5] T LIM–

Set the lower-limit value of the tolerance.

[6] BASE

Set the reference position for measurement.

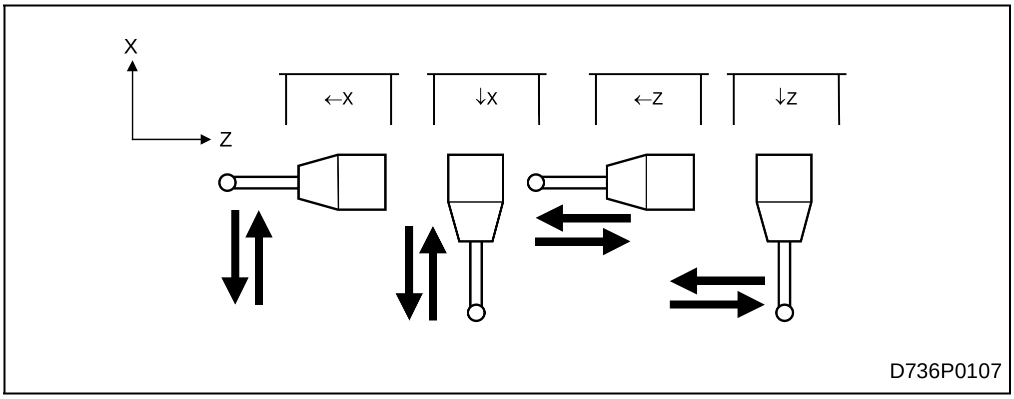
0: The starting position is defined as reference.
1: The ending position is defined as reference.

[7] DIR.

Select from the menu the direction (arrow) of the milling spindle head during measurement, and the approach/escape direction to be applied to the measurement.

← Z	← X	↓ Z	↓ X	→ Z	→ X				

Selection of the **[←X]** menu key specifies the B-axis (milling spindle head) facing in the direction of the arrow (left) and the approach/escape in X-axial direction during the measurement.



3-29-4 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 X-axial points on the outside-diameter section of the workpiece. Select **OUTER Y** to measure Y-axial points on the outside-diameter section of the workpiece.

Also, a measuring method can be selected for **OUTER X**.

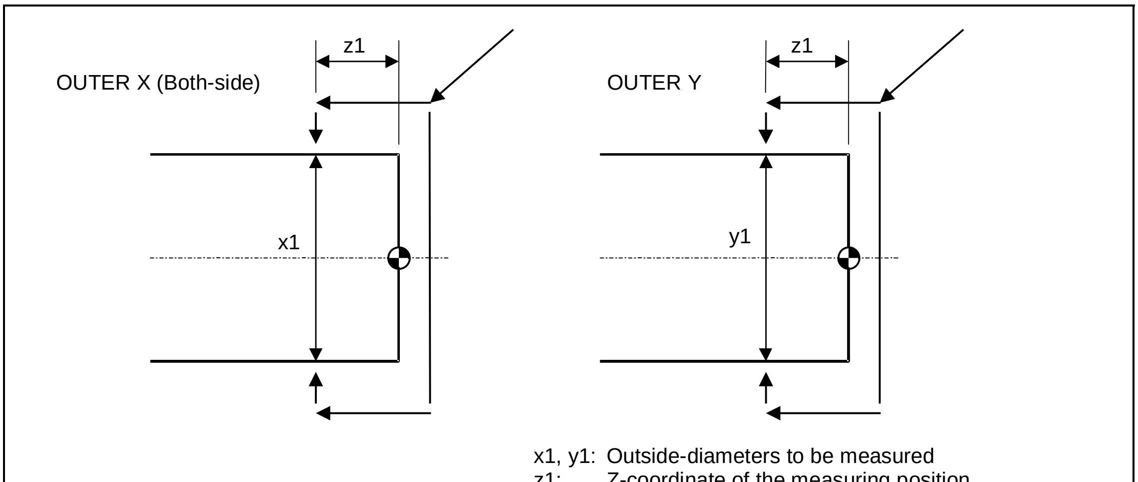
0: Both-side measurement
(Measuring the distance between two points with the position of X = 0 as its center)

1: Single-side measurement
(Measuring the distance between the reference point and measuring point within the plus area of the X-axis)

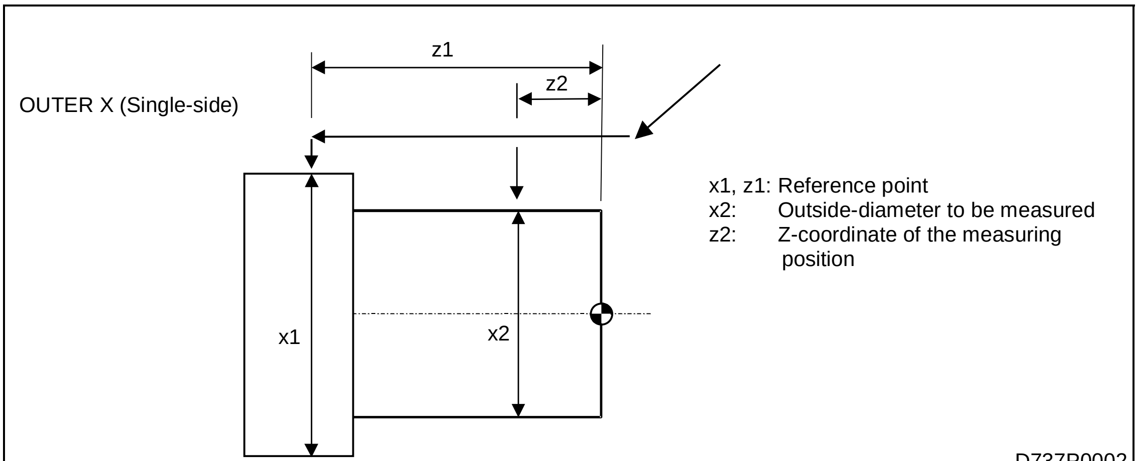
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	OUTER X 0	x1	◆	z1	◆	◆	◆	t1	t2	◆	←X

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	OUTER X 1	x1	◆	z1	x2	◆	z2	t1	t2	S	←X

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	OUTER Y	◆	y1	z1	◆	◆	◆	t1	t2	◆	←X



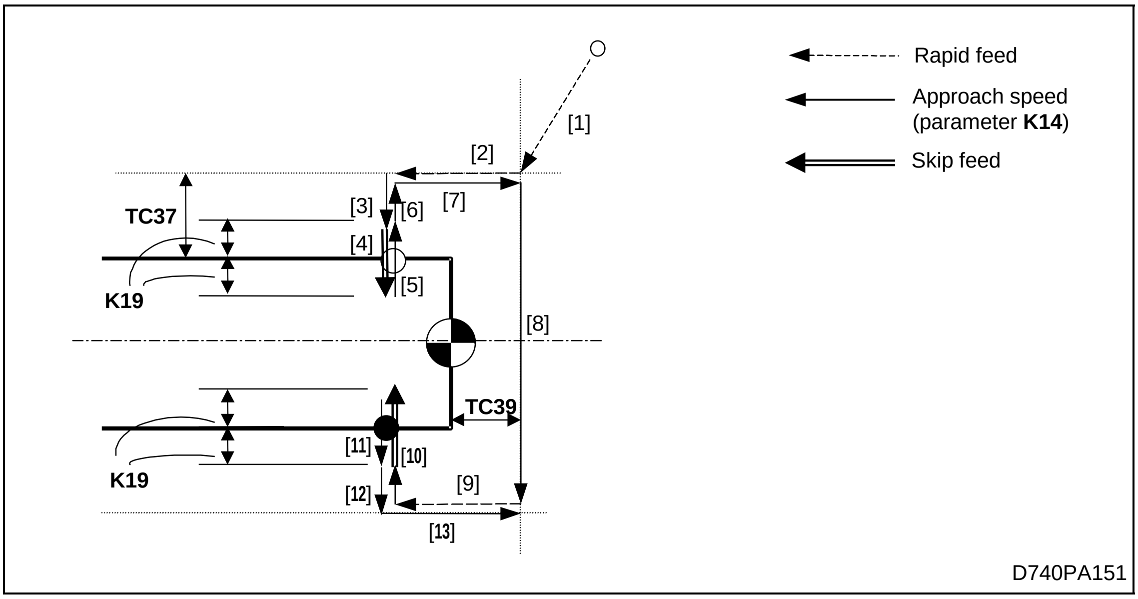
D736P0113



D737P0002

Set “0” as the reference position in the case of single-side measurement.

[Measurement movement (**OUTER**)]



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.

Also, a measuring method can be selected for **INNER X**.

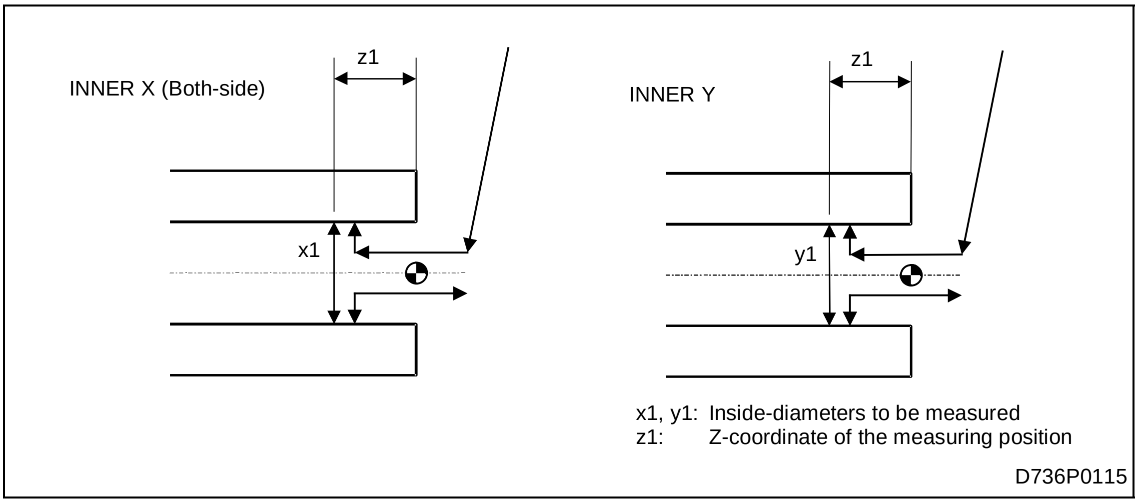
0: Both-side measurement
(Measuring the distance between two points with the position of X = 0 as its center)

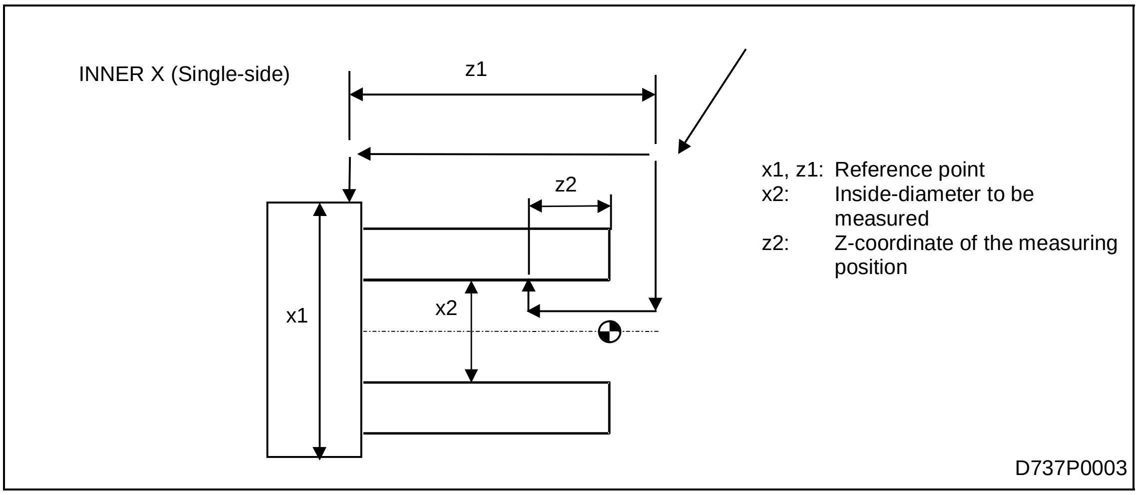
1: Single-side measurement
(Measuring the distance between the reference point and measuring point within the plus area of the X-axis)

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	INNER X 0	x1	◆	z1	◆	◆	◆	t1	t2	◆	←Z

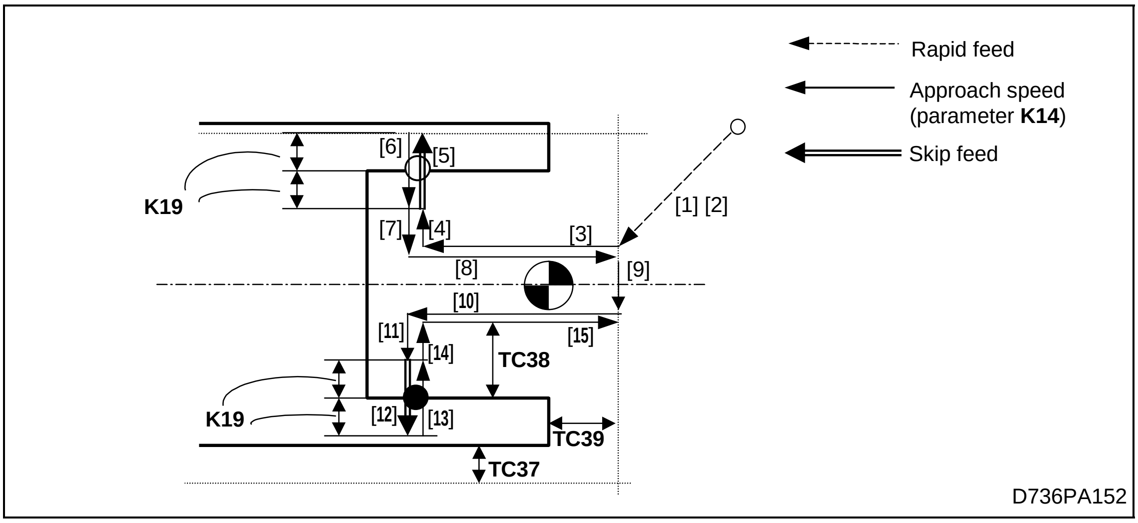
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	INNER X 1	x1	◆	z1	x2	◆	z2	t1	t2	S	←Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	INNER Y	◆	y1	z1	◆	◆	◆	t1	t2	◆	←Z





[Measurement movement (**INNER**)]

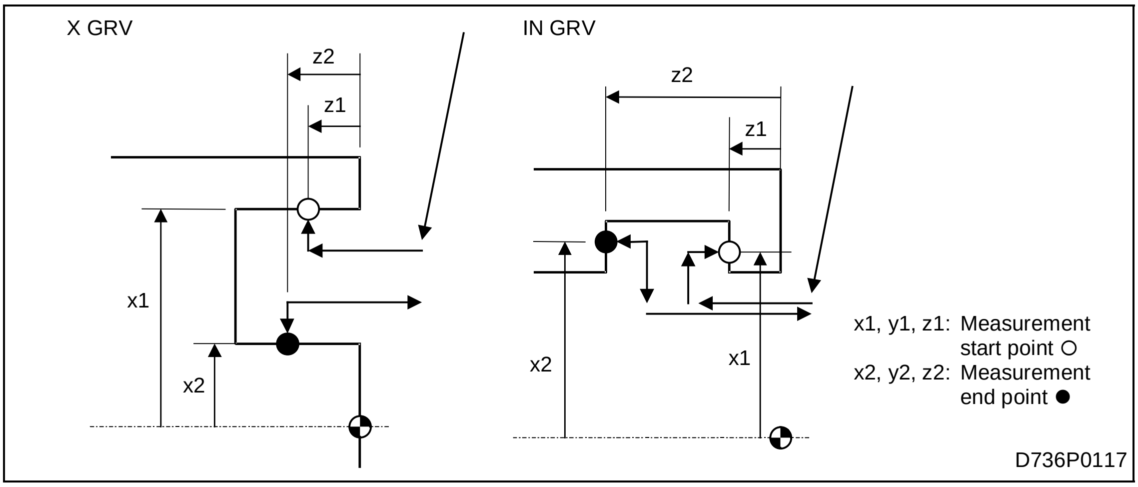


3. Groove width measurement

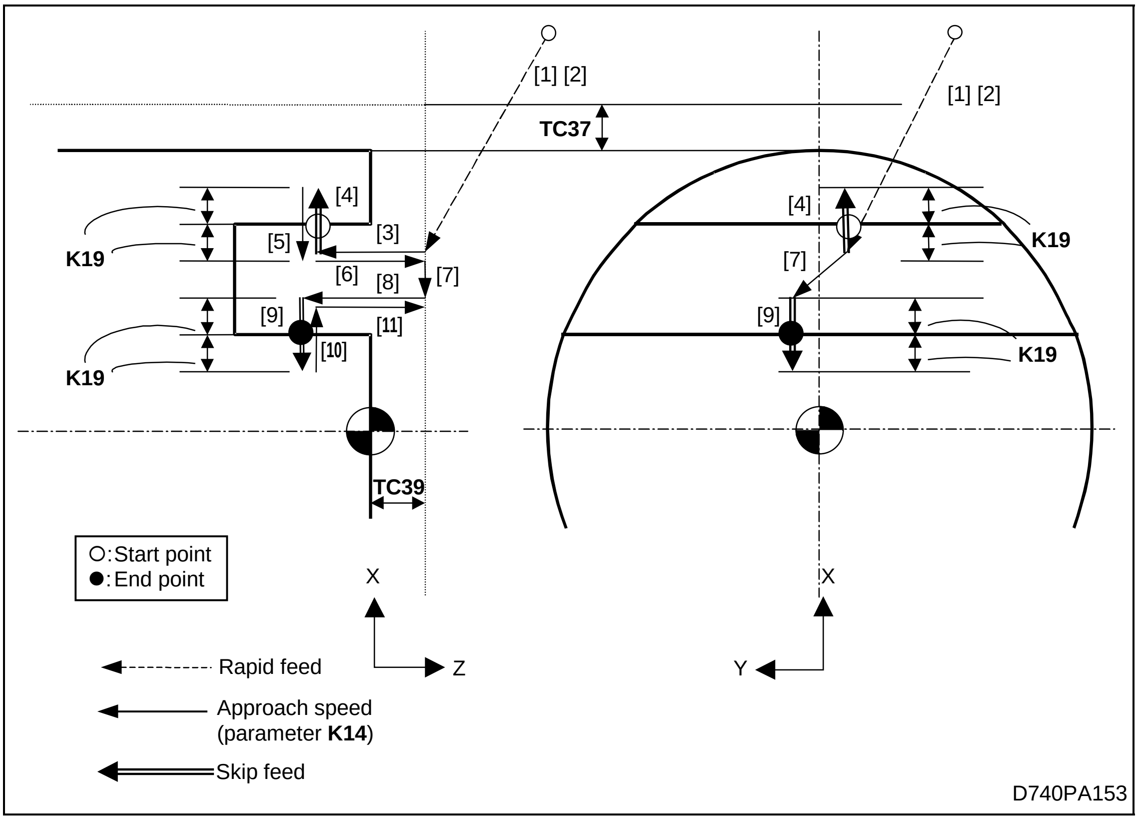
For **X GRV**, **Y GRV**, and **Z GRV** measurements are performed on X-axial, Y-axial, and Z-axial groove widths, respectively.
For **IN GRV**, the groove width at the inside diameter side is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	X GRV	x1	y1	z1	x2	y2	z2	t1	t2	S	←Z

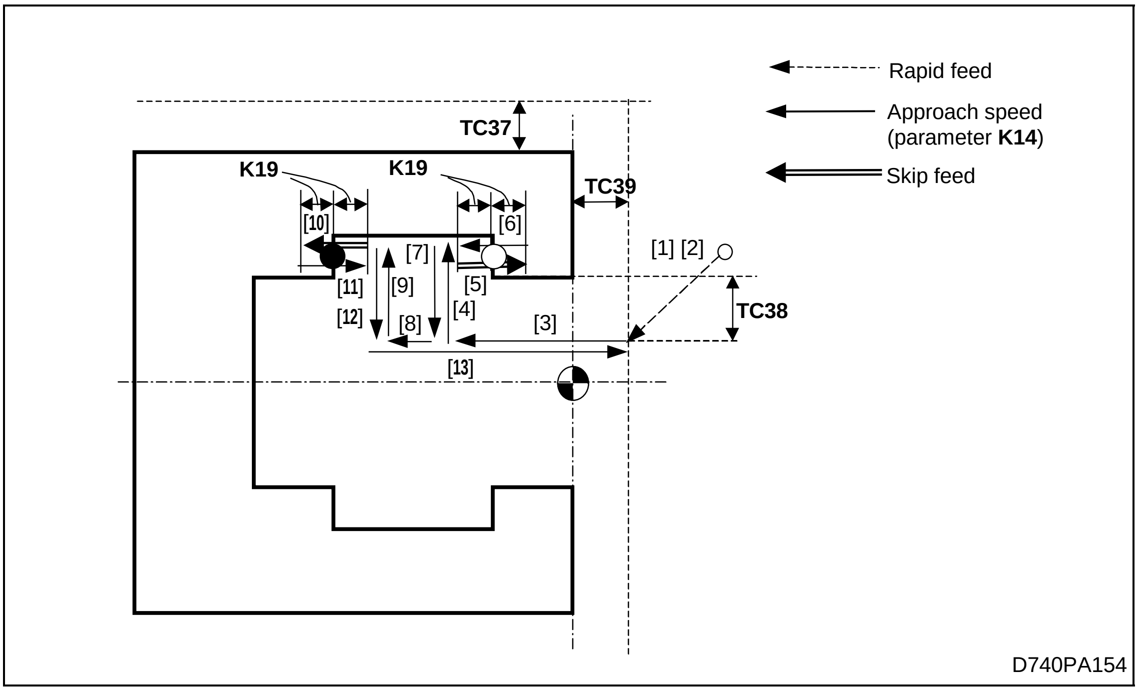
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	IN GRV	x1	y1	z1	x2	y2	z2	t1	t2	S	←Z



[Measurement movement (**X GRV**)]



[Measurement movement (**IN GRV**)]

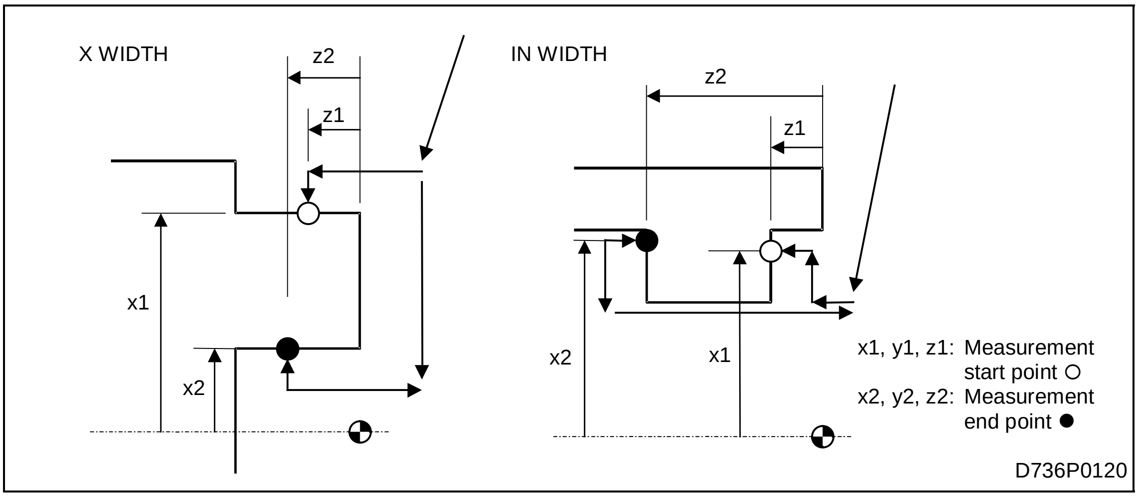


4. Protrusion width measurement

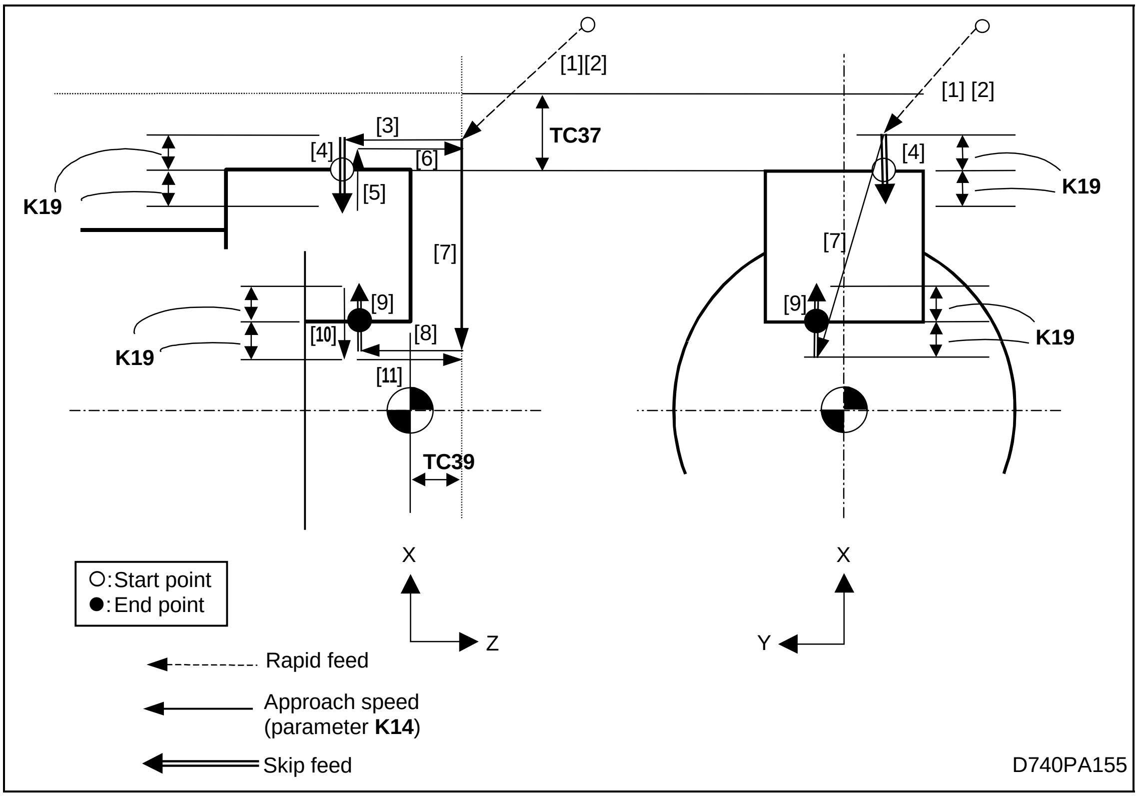
For **X WIDTH**, **Y WIDTH**, and **Z WIDTH** measurements are performed on X-axial, Y-axial, and Z-axial protrusion widths, respectively.
For **IN WIDTH**, the protrusion width at the inside diameter side is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	T LIM+	T LIM-	BASE	DIR.
1	X WIDTH	x1	y1	z1	x2	y2	z2	t1	t2	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	x1	y1	z1	x2	y2	z2	t1	t2	S	←Z

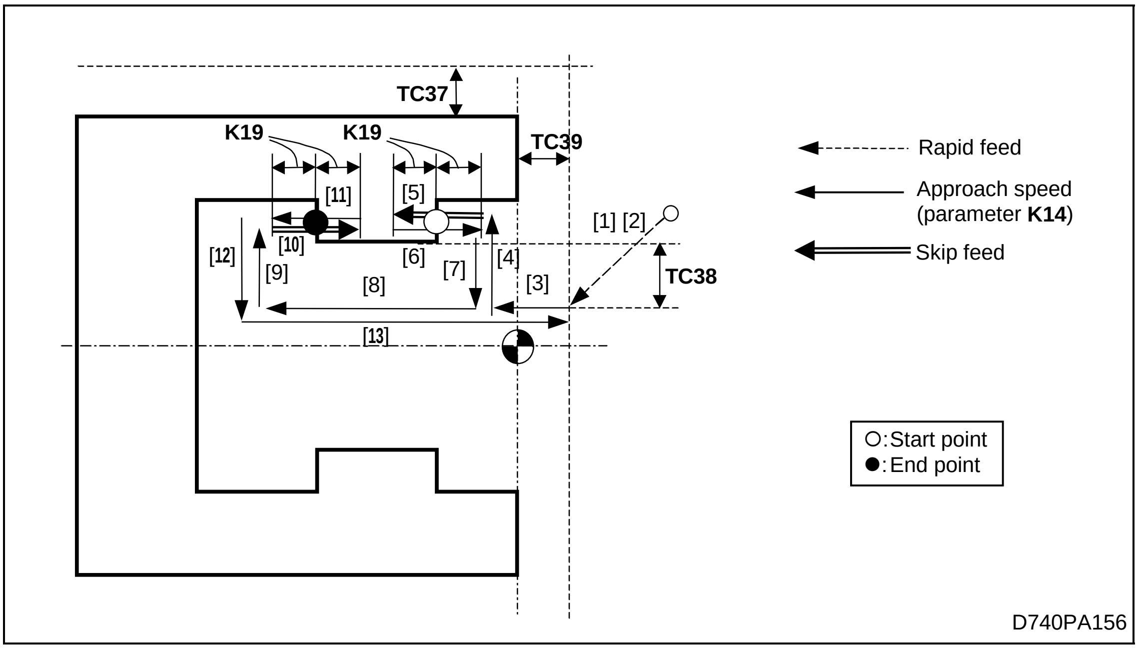


[Measurement movement (**X WIDTH**)]



Note: The safety profile clearance back (**TC40**) is used for measurement at the No. 2 spindle side.

[Measurement movement (**IN WIDTH**)]

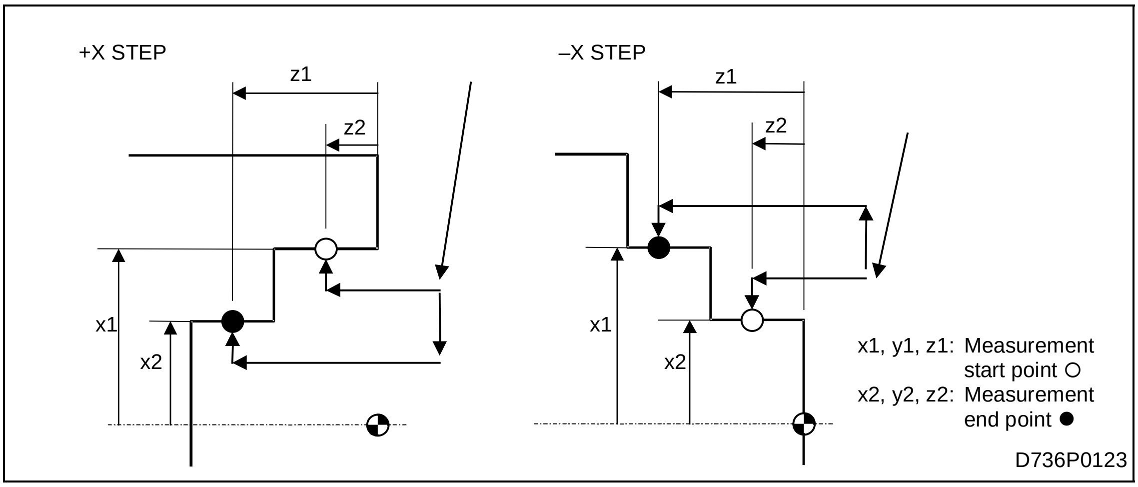


5. Step distance measurement

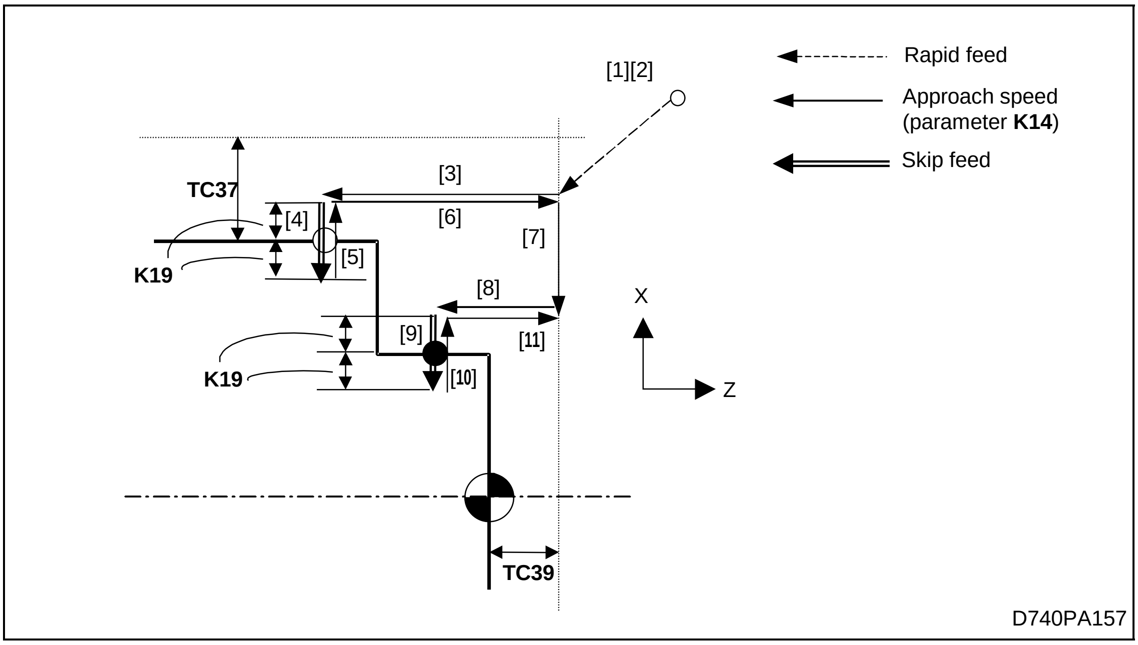
For **+X STEP**, **+Y STEP**, and **+Z STEP**, surface level differences in the plus directions of the X-axis, Y-axis, and Z-axis, respectively, are measured.

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	x1	y1	z1	x2	y2	z2	t1	t2	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	z1	◆	◆	◆	t1	t2	◆	◆

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	z1	◆	◆	◆	t1	t2	◆	◆

P: Select an offset item from the menu below.

WEAR X	WEAR Z	TOOL DIAMETER							

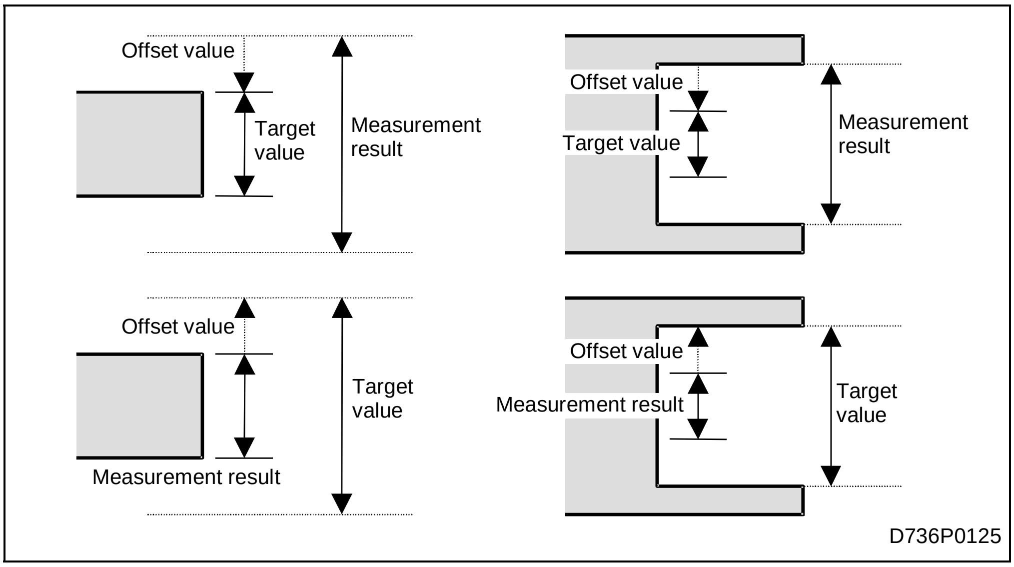
- For **EXT MILL**, **COMP. DATA** item on the unit line is invalid.
- **TOOL DIAMETER** is displayed only for **EXT MILL**.

N: Enter the number of the section to be measured using an external measuring unit.

z1: Enter the target value for the section which is to be measured using an external measuring unit.

3-29-5 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

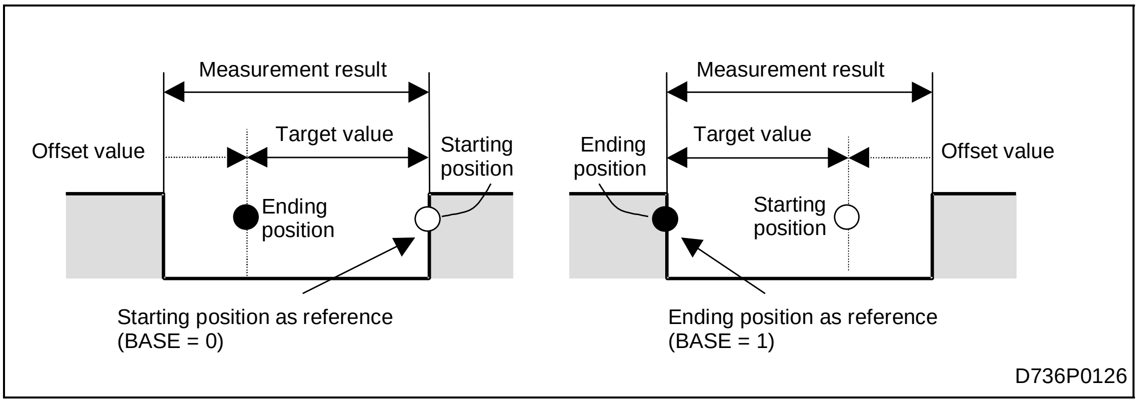
= | (Measurement point #1 – Measurement point #2) | – 2 × Touch sensor stylus ball radius

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

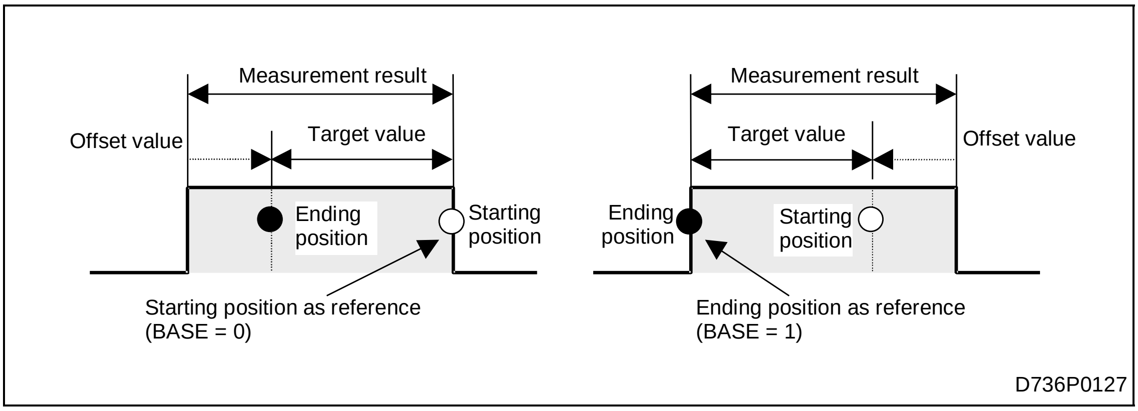
Measurement result Y (Z)
= | Measurement point #1 – Measurement point #2 | + 2 × Touch sensor stylus ball radius

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

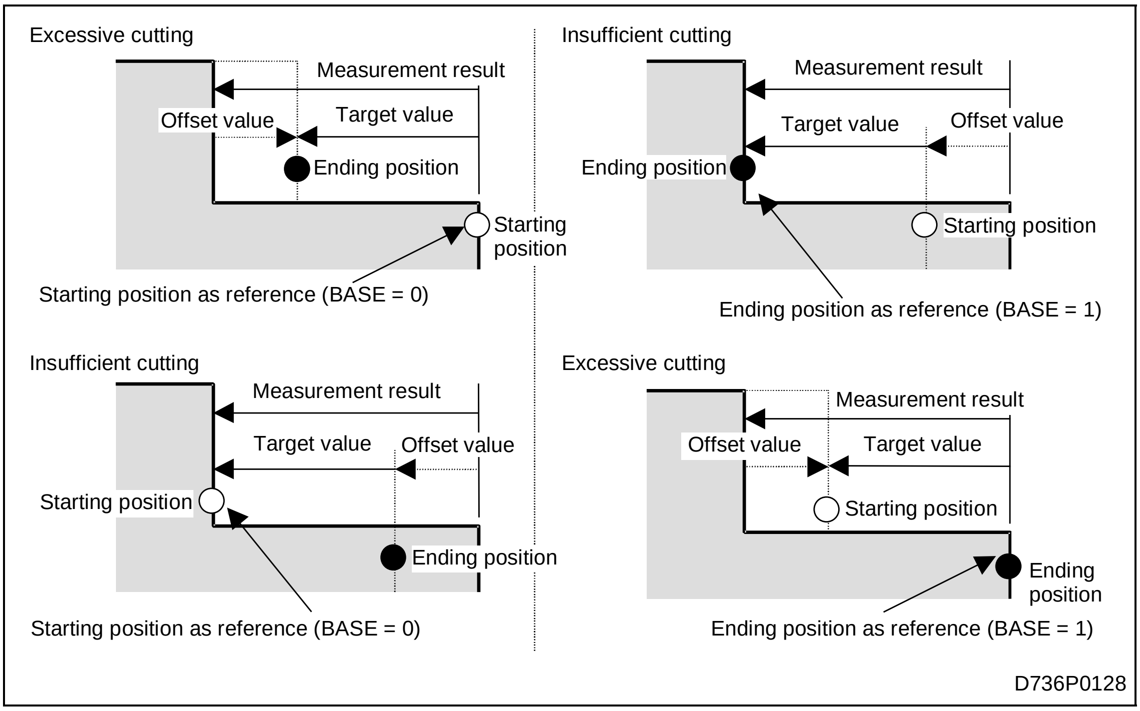
Measurement result Y (Z)
= | Measurement point #1 – Measurement point #2 | – 2 × Touch sensor stylus ball radius

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

3-29-6 Offset judgment

Offset data that has been obtained from measurements is handled as follows:

[Offset judgment conditions]

Condition	Result
$\text{Offset value} > \dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18}$	An alarm is displayed.
$\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18} \geq \text{Offset value} \geq \dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17}$	The offset is performed.
$\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17} > \text{Offset value} > -\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17}$	The offset is not performed.
$-\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17} \geq \text{Offset value} \geq -\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18}$	The offset is performed.
$-\dfrac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18} > \text{Offset value}$	An alarm is displayed.

Parameter **K17**: Lower-limit value of the measurement tolerance

Parameter **K18**: 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.

3-30 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.

3-30-1 Procedure for selecting tool measuring unit

(1) Procedure for selecting tool measuring unit 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	MANUAL PROGRAM			END	SHAPE CHECK	>>>

(2) Press the **[>>>]** menu key. The following menu will be displayed.

			M CODE	SUB PROGRAM	WPC MSR	WORKPICE MEASURE	TOOL MEASURE	WORKPICE SHAPE	>>>

(3) Press the **[TOOL MEASURE]** menu key.

3-30-2 Setting the unit data

UNo.	UNIT	COMPENSATE	OFS-TOOL	No.	#	INTERVAL	OUTPUT
	TOOL MES	[1]	[2]	[3]	[4]	[5]	[6]

[1] COMPENSATE

Select from the menu whether the measurement results are to be used to correct tool data.

YES	NO								

[2] OFS-TOOL

Enter the name, nominal diameter (nominal size), identification code and turret number of the tool to be measured.

[3] No.

Enter the machining priority number.

[4] #

Specify the retraction position of the lower turret during measurement.

[5] INTERVAL

Specify the intervals at which the tool measuring unit is to be executed.

[6] 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 **F112**.

3-30-3 Setting the sequence data

1. Setting TOOL MES sequence data

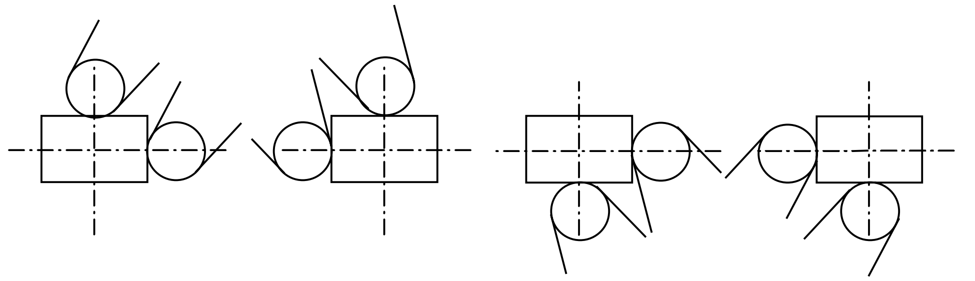
SNo.	PTN	T-LIM-X	T-LIM-Z	TOOLEYE	DIR.
1	[1]	[2]	[3]	[4]	[5]

[1] 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:



[2] 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.

[3] 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.

[4] 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.

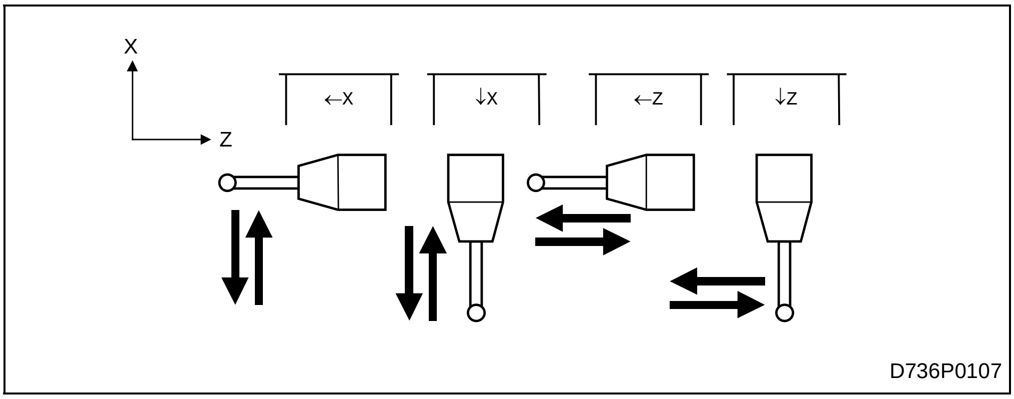
However, enter 0 for the last tool whose nose is to be measured using this unit. If 1 remains entered, the next machining unit will be executed with TOOL EYE extended.

[5] DIR.

Select from the menu the direction (arrow) of the milling spindle head during measurement, and the approach/escape direction to be applied to the measurement.

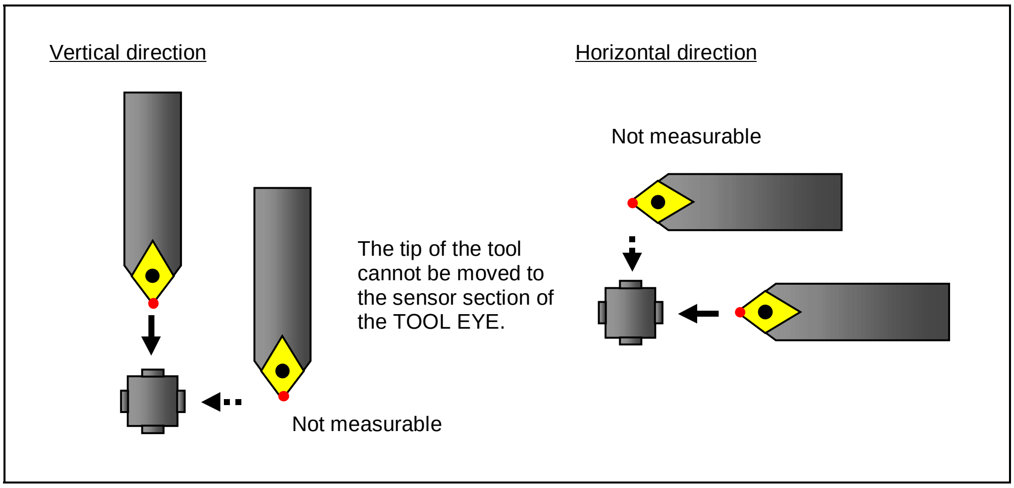
← Z	← X	↓ Z	↓ X	→ Z	→ X				

Selection of the **[←X]** menu key specifies the B-axis (milling spindle head) facing in the direction of the arrow (left) and the approach/escape in X-axial direction during the measurement.



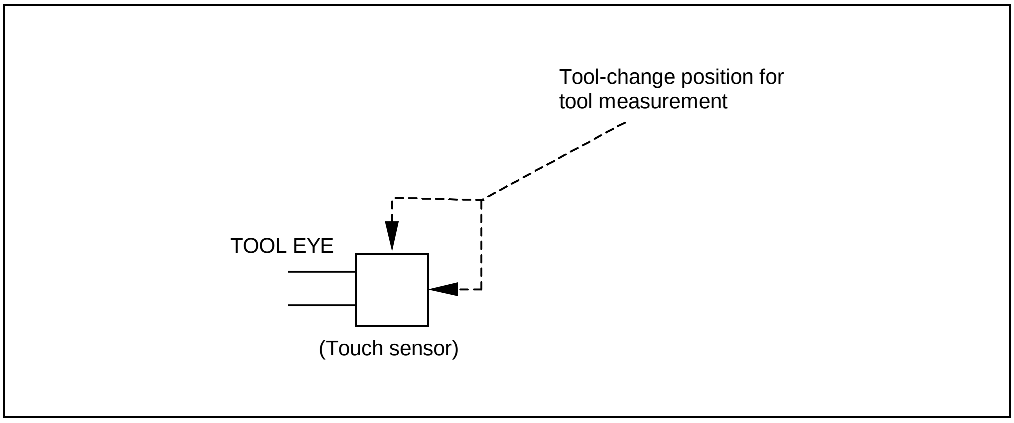
Note: The measurement may not be possible according to the particular type of tool.

Example: A sharply pointed cutting tool with a cutting angle less than 90 degrees



3-30-4 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.

4 PRIORITY FUNCTION FOR THE SAME TOOL

The program is executed by numeric order from its head. Consequently, tool change cycle is repeated for each of the tools specified in the tool sequence. This priority function for the same tool is intended to reduce the frequency of tool change and therefore the machining time by assigning the priority number to tools developed and by performing the machining according to the numbers thus assigned.

It is in the following units and tool sequences that the priority number can be specified.

- Tool sequence of machining unit
- Manual program machining unit
  (In the case of absence of a tool, the priority number cannot be specified.)
- Coordinate measuring unit
- Workpiece measuring unit
- M-code unit

4-1 Priority Machining Order

In a program containing specified priority numbers, the machining is done in the following order.

Prior machining ....The priority number is specified if the necessity of machining with complete priority occurs; for example, in the case of turning rough-machining, machining with a centering drill, etc. The machining is done by numeric order of the tools with priority number (displayed in yellow).

↓

Ordinary machining ....The machining is done in the programmed order of the tools developed by the tool sequence (tools without priority number).

↓

Subsequent machining....The priority number is specified when the need for machining lastly is arises; for example, in the case of chamfering cutter. The machining is done by the numeric order of the tools with priority numbers (displayed in reversed status).

Program without priority number

```
UNo.   MAT.     OD-MAX     ID-MIN      LENGTH      WORK FACE
 0    CBN STL    150.        0.          73.           3.
UNo.   UNIT      PART                FIN-Z
 1    FACING     FACE                 0.1
SNo.     TOOL        NOM.      No.   #    PAT.
 R 1  GENERAL OUT    25. A                 ◆
 F 2  GENERAL OUT    20. B                 ◆
 FIG     SPT-X       SPT-Z      FPT-X      FPT-Z
  1       150.         3.         0.         0.
UNo.   UNIT      PART       CPT-X      CPT-Z
 2      BAR       OUT        150.        0.
SNo.     TOOL        NOM.      No.   #    PAT.
 R 1  GENERAL OUT    25. A                 ◆
 F 2  GENERAL OUT    20. B                 ◆
 FIG     PTN      S-CNR      SPT-X      SPT-Z      FPT-X      FPT-Z
  1      LIN      C 0.5        ◆          ◆         120.       15.
  2      TPR                  120.       15.        130.       35.
  3      LIN                   ◆          ◆         135.       40.
UNo.   UNIT    MODE    POS-B     POS-C     DIA     DEPTH   CHMF
 3   DRILLING   XC       ◆         ◆       10.      50.     5.
SNo.   TOOL    NOM-ϕ  No.   #    HOLE-ϕ   HOLE-DEP  PRE-DIA
 1    CTR-DR    20.              10.        ◆         ◆
 2     DRILL    10.              10.        0.       100.
 3    CHAMFER   99.             999.        0.        16.
 FIG     PTN   SPT-R/x.  SPT-C/y.    SPT-Z     NUM.     ANG
  1      ARC      50.       0          0.        3       90.
UNo.   UNIT    MODE    POS-B     POS-C    NOM.  MAJOR-ϕ PITCH TAP-DEP
 4   TAPPING    XC       ◆         ◆       M16.    16     2.    30.
SNo.   TOOL    NOM-ϕ  No.   #    HOLE-ϕ   HOLE-DEP  PRE-DIA
 1    CTR-DR    20.              10.        ◆         ◆
 2     DRILL    14.2             14.2      27.        0.
 3    CHAMFER   99.             999.        0.        16.
 4      TAP    M16               16.       20.       TAP
 FIG     PTN   SPT-R/x.  SPT-C/y.    SPT-Z     NUM.     ANG
  1      ARC      30.      -90.        0.        3      120.
UNo.   UNIT    CONTI.   REPEAT   SHIFT    NUMBER    RETURN   LOW RET.
 5      END       0
```

Program with priority number

```
UNo.   MAT.      OD-MAX     ID-MIN      LENGTH      WORK FACE
 0    CBN STL     150.        0.          73.           3.
UNo.   UNIT       PART                FIN-Z
 1    FACING      FACE                 0.1
No.      TOOL         NOM.     No.   #    PAT.
R 1   GENERAL OUT     25. A     1          ◆
F 2   GENERAL OUT     20. B     2          ◆
FIG      SPT-X         SPT-Z     FPT-X       FPT-Z
 1        150.           3.        0.          0.
UNo.   UNIT       PART       CPT-X      CPT-Z
 2     BAR        OUT        150.         0.
SNo.     TOOL         NOM.     No.   #    PAT.
R 1   GENERAL OUT     25. A     1          ◆
F 2   GENERAL OUT     20. B     2          ◆
FIG    PTN      S-CNR     SPT-X      SPT-Z      FPT-X      FPT-Z
 1     LIN      C 0.5       ◆          ◆         120.        15.
 2     TPR                 120.        15.        130.        35.
 3     LIN                  ◆          ◆         135.        40.
UNo.   UNIT     MODE    POS-B     POS-C     DIA     DEPTH    CHMF
 3   DRILLING    XC       ◆         ◆       10.      50.      5.
SNo.   TOOL    NOM-ϕ   No.   #    HOLE-ϕ   HOLE-DEP  PRE-DIA
 1    CTR-DR    20.    3            10.        ◆         ◆
 2    DRILL     10.                 10.        0.       100.
 3   CHAMFER    99.    1           999.        0.        16.
FIG    PTN    SPT-R/x.  SPT-C/y.    SPT-Z     NUM.     ANG
 1     ARC       50.        0         0.        3       90.
UNo.   UNIT     MODE    POS-B     POS-C    NOM.  MAJOR-ϕ PITCH TAP-DEP
 4   TAPPING     XC       ◆         ◆       M16.    16.    2.     30.
SNo.   TOOL    NOM-ϕ   No.   #    HOLE-ϕ   HOLE-DEP  PRE-DIA
 1    CTR-DR    20.    3            10.        ◆         ◆
 2    DRILL    14.2                14.2       27.        0.
 3   CHAMFER    99.    1           999.        0.        16.
 4     TAP      M16                 16.       20.       TAP
FIG    PTN    SPT-R/x.  SPT-C/y.   SPT-Z     NUM.     ANG
 1     ARC      30.      -90.        0.        3      120.
UNo.   UNIT    CONTI.   REPEAT    SHIFT   NUMBER   RETURN   LOW RET.
 5     END       0
```

Prior machining priority number

Subsequent machining priority number

If one reclassifies these two programs by machining order, the following tables are obtained.

Program without priority number

UNo.	SNo.	Tool	Tool change
1	1	GENERAL OUT 25 A	○
1	2	GENERAL OUT 20 B	○
2	1	GENERAL OUT 25 A	○
2	2	GENERAL OUT 20 B	○
3	1	CTR-DR 20	○
3	2	DRILL 10	○ 11 times
3	3	CHAMFER 99	○
4	1	CTR-DR 20	○
4	2	DRILL 14.2	○
4	3	CHAMFER 99	○
4	4	TAP M16	○

Program with priority number

UNo.	SNo.	Tool	Tool change
1	1	GENERAL OUT 25 A	○
2	1	GENERAL OUT 25 A	
1	2	GENERAL OUT 20 B	○
2	2	GENERAL OUT 20 B	
3	1	CTR-DR 20	○
4	1	CTR-DR 20	7 times
3	2	DRILL 10	○
4	2	DRILL 14.2	○
4	4	TAP M16	○
3	3	CHAMFER 99	○
4	4	CHAMFER 99	

Without a specified priority number, the machining is done by the programmed order and the tool change cycle is executed for each tool. Consequently, in this example, the tool change cycle is executed 11 times. By specifying the priority number, two machinings of the same type are done at the same time by the same tool, which permits reducing the number of tool change cycles to 7.

Note 1: If a different priority number is assigned to the same tool, the machining is done in the order of the priority number.

Note 2: When the priority number is assigned to all the tools of the same process, the M-code unit without a priority number is executed once for extra between the prior machining and the subsequent machining.

4-2 Priority Machining Zone

The priority machining zone for the same tool is delimited by the following units. The zone delimited by these units is called process. In the case of different processes, it is possible to specify the same priority number for a different tool.

- Process end unit
- Workpiece transfer unit
- Head selection unit
- End unit (if **WORK No.** is specified)

Example: Case where the process end unit has been programmed.

If the program is executed which contains the priority numbers specified for GENERAL, CTR-DR and CHAMFER, the machining is done in the following order.

Process1

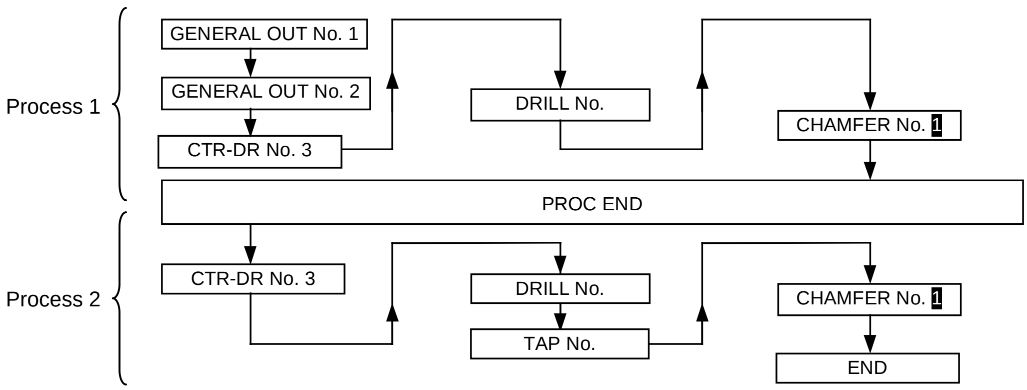
```
UNo.   MAT.      OD-MAX    ID-MIN    LENGTH    WORK FACE
 0     CBN STL   150.      0.        73.       3.
UNo.   UNIT      PART                FIN-Z
 1     FACING    FACE                0.1
SNo.   TOOL            NOM.     No.   #    PAT.
R 1    GENERAL OUT     25. A    1          ◆
F 2    GENERAL OUT     20. B    2          ◆
FIG    SPT-X     SPT-Z     FPT-X     FPT-Z
 1     150.      3.        0.        0.
UNo.   UNIT      PART      CPT-X     CPT-Z
 2     BAR       OUT       150.      0.
SNo.   TOOL            NOM.     No.   #    PAT.
R 1    GENERAL OUT     25. A    1          ◆
F 2    GENERAL OUT     20. B    2          ◆
FIG    PTN     S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z
 1     LIN     C 0.5    ◆        ◆        120.     15.
 2     TPR              120.     15.      130.     35.
 3     LIN              ◆        ◆        135.     40.
UNo.   UNIT       MODE   POS-B   POS-C   DIA    DEPTH   CHMF
 3     DRILLING   XC     ◆       ◆       10.    50.     5.
SNo.   TOOL      NOM-ϕ   No.  #   HOLE-ϕ   HOLE-DEP   PRE-DIA
 1     CTR-DR    20.     3        10.      ◆          ◆
 2     DRILL     10.              10.      0.         100.
 3     CHAMFER   99.     1        999.     0.         16.
FIG    PTN     SPT-R/x.   SPT-C/y.   SPT-Z   NUM.   ANG
 1     ARC     50.        0          0.      3      90.
UNo.   UNIT
 4     PROC END
```

Process 2

```
UNo.   UNIT      MODE   POS-B   POS-C   NOM.   MAJOR-ϕ  PITCH  TAP-DEP
 5     TAPPING   XC     ◆       ◆       M16.   16.      2.     30.
SNo.   TOOL      NOM-ϕ   No.  #   HOLE-ϕ   HOLE-DEP   PRE-DIA
 1     CTR-DR    20.     3        10.      ◆          ◆
 2     DRILL     14.2             14.2     27.        0.
 3     CHAMFER   99.     1        999.     0.         16.
 4     TAP       M16              16.      20.        TAP
FIG    PTN     SPT-R/x.   SPT-C/y.   SPT-Z   NUM.   ANG
 1     ARC     30.        -90.       0.      3      120.
UNo.   UNIT    CONTI.   REPEAT   SHIFT   NUMBER   RETURN   LOW RET.
 6     END     0
```

Priority number for prior machining: 1, 2, 3

Priority number for subsequent machining 1



4-3 Editing Function and Input Method of Priority Numbers

4-3-1 Input of priority numbers

The priority number is of two types: for prior machining and for subsequent machining, it is entered by means of menu keys and numeric keys. The priority number is entered in ascending order of the tool sequence.

(1) In creating mode, move the cursor to the item **No.**

```
UNo.      UNIT        PART
 2         BAR         OUT
SNo.      TOOL        NOM.    No.    #    PAT.
 1     GENERAL OUT    25.A    █
```

➔ The message **ACHINING PRIORITY No.?** is displayed and the menu changes as follows.

```
                                          MACHINING PRIORITY No.?
        | DELAY  |        |PRI. No.|PRI. No.|        |PRI. No.|SUB PROG|        |
        |PRIORITY|        | CHANGE | ASSIGN |        |ALL ERAS|PROC END|        |
```

(2) Enter the priority number. There are the three following entry methods (refer to Note 1 to 3 below):

Prior machining priority number

To be set by means of numeric keys.

➔ The number is displayed in yellow.

Subsequent machining priority number

1) Press the **[DELAY PRIORITY]** menu key.

➔ The display of **[DELAY PRIORITY]** is reversed.

2) Set the subsequent machining priority number by means of numeric keys.

➔ The priority number is displayed in reversed status.

Without entry (ordinary machining)

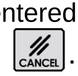
The priority number is not entered. Move the cursor to the following article.

➔ When the priority number is entered, the cursor moves to the following article.

```
SNo.      TOOL        NOM.    No.    #    PAT.
 1     GENERAL OUT    25.A     1     █
```

Note 1: The prior machining and subsequent machining can receive a priority number from 1 to 99, respectively. Moreover, it is not always necessary to mark the priority of the sequence numbers.

Note 2: It is possible to assign the same priority number or a different priority number to the same tool, but it is impossible to assign the same priority number to different tools; this will cause the alarm **420 SAME DATA EXISTS** to display.

Note 3: In order to cancel a priority number after it has been entered, move the cursor to the position concerned and press the data cancellation key .

4-3-2 Assignment of priority numbers

This function is used to make the assignment of priority numbers entered for all the identical tools in the same process.

Note: This function is only useful for a program in the process of editing.

Menu selection: **[PRI. No. ASSIGN]** (→ **[DELAY PRIORITY]**)

(1) In creating mode, move the cursor to the item **No.**

```
UNo.      UNIT        PART
 2         BAR         OUT
SNo.      TOOL        NOM.     No.    #    PAT.
 1     GENERAL OUT    25.A     █← Cursor
```

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

➔ The display of **[PRI. No. ASSIGN]** is reversed and the message **MACHINING PRIORITY No.?** is displayed in the message zone of the display.

(3) Enter the priority number by means of numeric keys.

Example: Entry of 2

Press the following keys: [2] [INPUT]

- For subsequent machining, enter the intended number after having pressed the **[DELAY PRIORITY]** menu key.

Note: If the data cancellation key [CANCEL] is pressed, all the priority numbers for the identical tools in the process will be erased.

➔ The same priority number is assigned to all the identical tools in the process and the cursor moves to the following article.

```
SNo.      TOOL        NOM.     No.    #    PAT.
 1     GENERAL OUT    25.A      2     █
UNo.      UNIT        PART
 3         BAR         OUT
SNo.      TOOL        NOM.     No.    #    PAT.
 1     GENERAL OUT    25.A      2
```

Note: Regardless of whether the priority number is entered or not, all the identical tools in the process are marked with the same priority number.

4-3-3 Change of priority numbers

This function is used for changing the priority number entered for all the identical tools in a process.

Note: This function is only useful for a program in the process of editing.

Menu selection: **[PRI. No. CHANGE]** (→ **[DELAY PRIORITY]**)

(1) In creating mode, move the cursor to the item **No.**

```
UNo.    UNIT          PART
 2       BAR           OUT
SNo.    TOOL          NOM.     No.   #    PAT.
 1    GENERAL OUT     25.A     █← Cursor
```

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

➔ The display of **[PRI. No. CHANGE]** is reversed and the message **MACHINING PRIORITY No.?** is displayed in the message zone of the display.

(3) Enter the priority number by using numeric keys.

Example: Entry of 5

Press the following keys: [5] [INPUT]

- For subsequent machining, press the **[DELAY PRIORITY]** menu key, and then enter the intended number.

Note: If the data cancellation key [CANCEL] is pressed, all the priority numbers for the identical tools in the process will be erased.

➔ The same priority number is assigned to all the identical tools in the process and the cursor moves to the next item.

```
SNo.    TOOL          NOM.    No.   #    PAT.
 1    GENERAL OUT     25.A     5    █
UNo.    UNIT          PART
 3       BAR           OUT
SNo.    TOOL          NOM.    No.   #    PAT.
 1    GENERAL OUT     25.A     5
```

Note: Regardless of whether the priority number is entered or not, the same priority number is assigned to all the identical tools in the process.

4-3-4 Deletion of all the priority numbers

This function is used for deleting all the priority numbers contained in the process or in the program.

Note: This function is only useful for a program in the process of editing.

Menu selection: **[PRI. No. ALL ERAS]**

(1) In creating mode, move the cursor to the item **No.**

```
UNo.      UNIT        PART
 2         BAR         OUT
SNo.      TOOL        NOM.     No.   #    PAT.
 1     GENERAL OUT    25.A     █← Cursor
```

(2) Press the **[PRI No. ALL ERAS]** menu key.

➔ The display of **[PRI. No. ALL ERAS]** is reversed and the message **ALL ERASE <PROC:0, PROG:1>?** is displayed in the message zone of the screen.

(3) By means of numeric keys, specify the zones to be deleted.

- Enter 1 to delete all the priority numbers contained in the program.
- Enter 0 to delete all the priority numbers contained in a process where the cursor is located.

Example: Deletion of all the priority numbers contained in a program

Press the following keys: [1] [INPUT].

➔ All the priority numbers in the specified zone are deleted.

```
SNo.      TOOL        NOM.     No.   #    PAT.
 1     GENERAL OUT    25.A           █
UNo.      UNIT        PART
 3         BAR         OUT
SNo.      TOOL        NOM.     No.   #    PAT.
 1     GENERAL OUT    25.A
```

4-3-5 How to use the SUB PROG PROC END function

When the priority number has been edited in the main program, it is necessary to perform the same editing for the subprogram.

If in the process constituting the subject of the editing, there is a subprogram containing a process delimitation unit (process end unit), press the **[SUB PROG PROC END]** menu key in order to invert the display, which has the result that the subprogram is treated the same as the process end unit. (See Fig. 4-1.)

SUB PROG PROC END — Menu display normal

SUB PROG PROC END — Menu display reversed

(Main program)

```
UNo.   MAT.     OD-MAX    ID-MIN    LENGTH   WORK FACE
 0    CBN STL    150.       0.        73.       3.
UNo.   UNIT      PART               FIN-Z
 1    FACING     FACE                0.1
SNo.    TOOL        NOM.       No.     #     PAT.
 R 1  GENERAL OUT    25. A                    ◆
 F 2  GENERAL OUT    20. B                    ◆
 FIG    SPT-X    SPT-Z    FPT-X    FPT-Z
 1      150.      3.       0.       0.
UNo.    UNIT        PART       CPT-X      CPT-Z
 2       BAR         OUT        150.        0.
SNo.    TOOL        NOM.       No.     #     PAT.
 R 1  GENERAL OUT    25. A                    ◆
 F 2  GENERAL OUT    20. B                    ◆
 FIG    PTN    S-CNR   SPT-X   SPT-Z    FPT-X    FPT-Z
 1      LIN    C 0.5     ◆       ◆      120.      15.
 2      TPR             120.    15.     130.      35.
 3      LIN              ◆       ◆      135.      40.
UNo.
 3      Subprogram
UNo.     UNIT      MODE   POS-B   POS-C   DIA   DEPTH    CHMF
 4     DRILLING     XC      ◆       ◆     10.    50.      5.
SNo.   TOOL    NOM-ϕ  No.   #    HOLE-ϕ   HOLE-DEP PRE-DIA
 1     CTR-DR   20.                10.        ◆       ◆
 2     DRILL    10.                10.        0.     100.
 3    CHAMFER   99.               999.        0.      16.
 FIG    PTN    SPT-R/x.   SPT-C/y.    SPT-Z    NUM.    ANG
 1      ARC      50.         0.        0.       3      90.
UNo.  UNIT  CONTI.  REPEAT   SHIFT  NUMBER  RETURN  LOW RET.
 5    END     0
```

Process (1): [1] Process (1) — main program units 0 to 3; [4] Process (2) — units 4 and 5. [2] and [3] refer to the subprogram sections below.

(Subprogram)

```
[2]
UNo.    UNIT     MODE   POS-B   POS-C    DIA     DEPTH    CHMF
 1    DRILLING    XC      ◆       ◆      10.      50.      0.
SNo.    TOOL     NOM-ϕ   No.    #    HOLE-ϕ  HOLE-DEP   PRE-DIA
 1     CTR-DR    20.                  10.       ◆          ◆
 2     DRILL     10.                  10.       0.       100.
 FIG     PTN    SPT-R/x.  SPT-C/y.   SPT-Z   NUM.    ANG
 1       ARC       50.       0.        0.      3      90.
UNo.    PROC END
 2
[3]
UNo.    UNIT     MODE   POS-B POS-C   NOM.   MAJOR-ϕ  PITCH TAP-DEP
 3    TAPPING    XC      ◆     ◆      M16.     16       2.     30.
SNo.    TOOL     NOM-ϕ   No.    #    HOLE-ϕ  HOLE-DEP   PRE-DIA
 1     CTR-DR    20.                  10.       ◆          ◆
 2     DRILL     14.2                 14.2     27.         0.
 3    CHAMFER    99.                 999.       0.        16.
 4      TAP      M16                  16.      20.        TAP
 FIG     PTN    SPT-R/x.  SPT-C/y.   SPT-Z   NUM.    ANG
 1       ARC       30.      -90.       0.      3     120.
UNo.    UNIT    CONTI.  REPEAT  SHIFT  NUMBER  RETURN  LOW RET.
 4      END       0
```

Fig. 4-1 Subprogram unit = process end unit

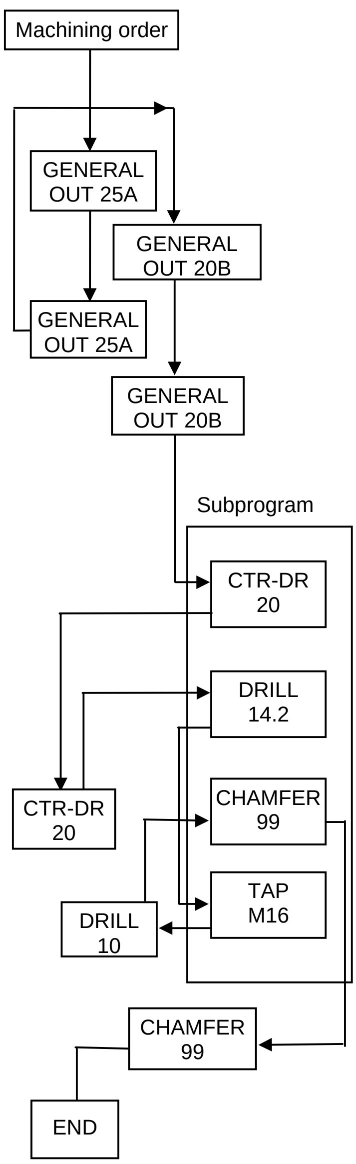
Remark 1: The editing function zone can be divided by the subprogram unit.
Even if the editing function is executed in delimited zone [1], this has no effect in the zone [2], [3] and [4].

Remark 2: Display of **[SUB PROG PROC END]** is reversed: Two processes (1) and (2)
Display of **[SUB PROG PROC END[** is not reversed: One process (1)

4-4 Relation between the Subprogram Unit and the Priority Machining Function

If one program contains a subprogram unit and the priority function for the same tool, the machining order is as mentioned below.

Example: Entry of priority number for GENERAL, CTR-DR and CHAMFER




```
UNo.   MAT.      OD-MAX     ID-MIN     LENGTH   WORK FACE
 0    CBN STL    150.         0.         73.        3.
UNo.    UNIT       PART                FIN-Z
 1     FACING      FACE                 0.1
SNo.     TOOL          NOM.        No.     #     PAT.
 R 1  GENERAL OUT    25. A          1              ◆
 F 2  GENERAL OUT    20. B          2              ◆
 FIG     SPT-X     SPT-Z     FPT-X     FPT-Z
 1      150.        3.         0.        0.
UNo.     UNIT          PART        CPT-X       CPT-Z
 2        BAR           OUT        150.         0.
SNo.     TOOL          NOM.        No.     #     PAT.
 R 1  GENERAL OUT    25. A          1              ◆
 F 2  GENERAL OUT    20. B          2              ◆
 FIG     PTN    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z
 1      LIN    C 0.5      ◆        ◆       120.     15.
 2      TPR              120.     15.      130.     35.
 3      LIN               ◆        ◆       135.     40.
UNo.
 3           Subprogram
UNo.     UNIT      MODE    POS-B   POS-C   DIA   DEPTH   CHMF
 4     DRILLING     XC      ◆       ◆     10.    50.     5.
SNo.    TOOL     NOM-ϕ   No.   #     HOLE-ϕ   HOLE-DEP   PRE-DIA
 1     CTR-DR    20.     3            10.        ◆          ◆
 2     DRILL     10.                  10.        0.        100.
 3    CHAMFER    99.     1           999.        0.         16.
 FIG     PTN    SPT-R/x.    SPT-C/y.     SPT-Z     NUM.     ANG
 1      ARC       50.          0.          0.        3       90.
UNo.   UNIT   CONTI.  REPEAT   SHIFT   NUMBER   RETURN   LOW RET.
 5     END      0
```

Priority number
Prior: 1, 2, 3
Subsequent: 1

(Subprogram)

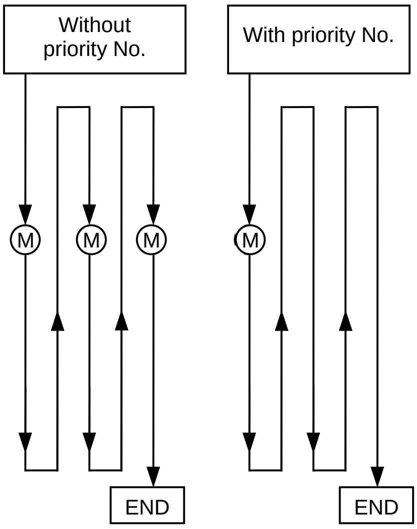
```
UNo.    UNIT     MODE   POS-B  POS-C   NOM.   MAJOR-ϕ  PITCH   TAP-DEP
 3     TAPPING    XC      ◆      ◆     M16.     16       2.      30.
SNo.    TOOL     NOM-ϕ    No.    #     HOLE-ϕ  HOLE-DEP   PRE-DIA
 1     CTR-DR    20.      3             10.       ◆          ◆
 2     DRILL     14.2                   14.2     27.         0.
 3    CHAMFER    99.      1            999.       0.        16.
 4      TAP      M16                    16.      20.        TAP
 FIG    PTN    SPT-R/x.   SPT-C/y.    SPT-Z    NUM.    ANG
 1      ARC      30.        -90.        0.       3     120.
UNo.   UNIT    CONTI.   REPEAT    SHIFT   NUMBER  RETURN  LOW RET.
 4      END      0
```

In the process of priority machining search, the subprogram unit is executed as follows:

- In the case where subprogram is a MAZATROL program, the subprogram unit is always executed. (The machinings specified in the subprogram are executed in the numeric order of the priority numbers.)
- In the case where subprogram is an EIA/ISO program, the subprogram unit is only executed once at the time of ordinary machining.

4-5 Relation between the M-Code Unit and the Priority Machining Function

The machining order differs as follows, according to whether the M-code unit contains the priority code for the same tool or not.



In the process of the search for the priority No., the M-code unit is executed each time it is read.

The M-code unit is executed only once in conformity with the priority No.

```
UNo.   MAT.      OD-MAX     ID-MIN     LENGTH   WORK FACE
 0    CBN STL     150.        0.         73.        3.
UNo.   UNIT        PART                 FIN-Z
 1    FACING       FACE                  0.1
SNo.    TOOL          NOM.        No.     #     PAT.
 R 1  GENERAL OUT    25. A         1             ◆
 F 2  GENERAL OUT    20. B         2             ◆
 FIG    SPT-X     SPT-Z     FPT-X     FPT-Z
  1     150.       3.        0.        0.
UNo.    UNIT
 3     M-CODE                              M-code unit
UNo.    UNIT          PART        CPT-X       CPT-Z
 2       BAR           OUT         150.         0.
SNo.    TOOL          NOM.        No.     #     PAT.
 R 1  GENERAL OUT    25. A         1             ◆
 F 2  GENERAL OUT    20. B         2             ◆
 FIG    PTN    S-CNR   SPT-X   SPT-Z    FPT-X    FPT-Z
  1     LIN    C 0.5     ◆       ◆       120.     15.
  2     TPR             120.    15.      130.     35.
  3     LIN              ◆       ◆       135.     40.
UNo.    UNIT       MODE   POS-B   POS-C   DIA   DEPTH    CHMF
 4    DRILLING      XC      ◆       ◆     10.    50.      5.
SNo.   TOOL     NOM-ϕ  No.  #    HOLE-ϕ  HOLE-DEP  PRE-DIA
 1    CTR-DR     20.                10.       ◆         ◆
 2    DRILL      10.                10.       0.       100.
 3    CHAMFER    99.               999.       0.        16.
 FIG    PTN    SPT-R/x.   SPT-C/y.    SPT-Z     NUM.    ANG
  1     ARC      50.         0          0.        3      90.
UNo.  UNIT   CONTI.  REPEAT  SHIFT   NUMBER   RETURN  LOW RET.
 5    END      0
```

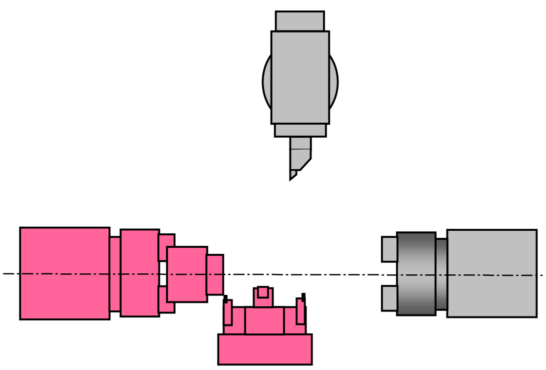
5 LOWER-TURRET CONTROL FUNCTIONS

This chapter describes the programming methods that use the lower turret (turret 2) mounted under twin-turret machine specifications.

5-1 Machining with the Lower Turret

There are four types of machining which uses the lower turret: independent machining with the lower turret, simultaneous machining with the upper and lower turrets, balanced cutting with the upper and lower turrets, and simultaneous machining of processes 1 and 2 with the upper and lower turrets.

5-1-1 Independent machining with the lower turret



D737P0004

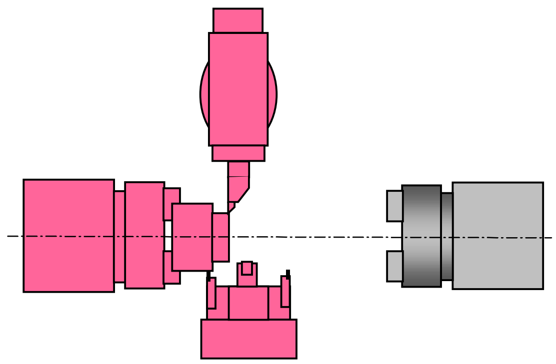
To execute machining that uses only the lower turret, select one of the tools mounted in the lower turret and then specify turret 2 (marked "◤").

```
UNo.  MAT.    OD-MAX ID-MIN LENGTH WORK FACE  ATC MODE       RPM       LTUR DIA
 0   CST IRN  100.    0.     40.      0.                     2000
UNo.   UNIT      PART     POS-B   CPT-X    CPT-Z    FIN-X   FIN-Z
 1   BAR          OUT       90     100.      0.      0.2     0.2
SNo.    TOOL     NOM. No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP   FR   M M M
 R 1 GENERAL OUT 45. A        0     5.       ◆        ◆     ◆     ◆    120  0.45
 F 2 GENERAL OUT 30.  ◤       ◆     ◆        ◆        ◆     0.    0.   196  0.1
 FIG  PTN     S-CNR      SPT-X    SPT-Z    FPT-X    FPT-Z  F-CNR/$  R/th     RGH
 1   LIN                          ◆        80.      55.             ◆        4
UNo.   UNIT    CONTI. REPEAT          NUMBER    RETURN   LOW RET. WORK No.    EXECUTE
 2   END                                                                      ◆
```

◤: Identification mark for the lower turret

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).

5-1-2 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.

1. 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 surface 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	ATC MODE	RPM	LTUR DIA
0	CST IRN	100.	0.	40.	0.		2000	

UNo.	UNIT	No.	SIMUL.No.	RPM
1	SIMULTAN		50	2000

Machining identified by simultaneous machining group number 50 occurs at a speed of 2000 min^{-1}.

When a machining priority number is assigned, all processes assigned the simultaneous machining group number (in this example, 50) will be conducted in the order specified by the machining priority number.

2. 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   ATC MODE       RPM   LTUR DIA
 0   CST IRN     100.      0.      40.       0.                     2000
UNo.    UNIT      No.   SIMUL.No.      RPM
 1   SIMULTAN                50         2000
UNo.  UNIT     PART      POS-B     CPT-X     CPT-Z      FIN-X      FIN-Z
 2   BAR        OUT        90        100.       0.        0.2        0.2
SNo.     TOOL      NOM. No   # PAT.  DEP-1 DEP-2/NUM.  DEP-3  FIN-X  FIN-Z C-SP   FR   M 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   PTN     S-CNR     SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$    R/th     RGH
  1   LIN                  ◆        ◆       80.      55.                ◆        4
UNo.   UNIT       PART      POS-B   PAT.  No.   PITCH  WIDTH  FINISH
 3   T.GROOVE     OUT        90      0     1      0      10.     ◆
SNo.     TOOL     NOM. No.  # PAT. DEP-1 DEP-2/NUM.  DEP-3  FIN-X  FIN-Z C-SP   FR   M M M
 F 1 GROOVE OUT  10.  ◆  50   ◆    2.        ◆         ◆      ◆      ◆     ◆    0.1
 FIG          S-CNR     SPT-X      SPT-Z    FPT-X    FPT-Z   F-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.

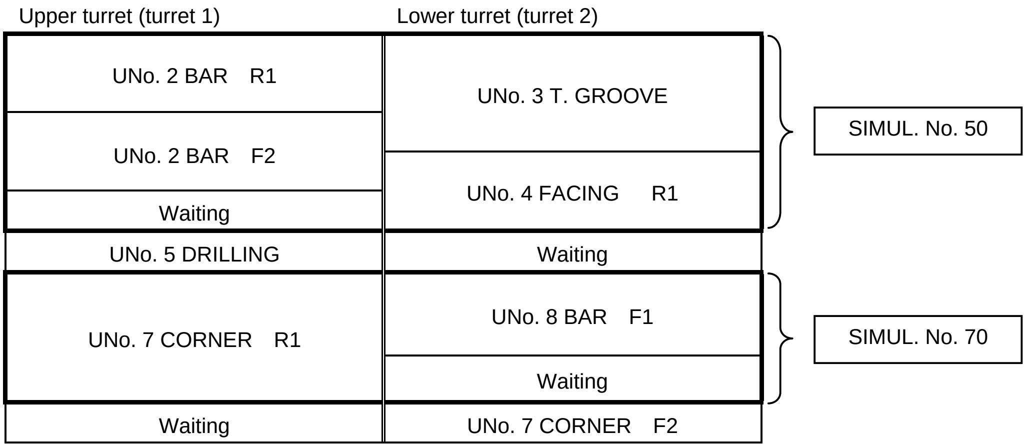
3. Example of multiple sets of simultaneous machining

Multiple sets of simultaneous machining can be performed using multiple simultaneous machining units.

A sample program is shown below.

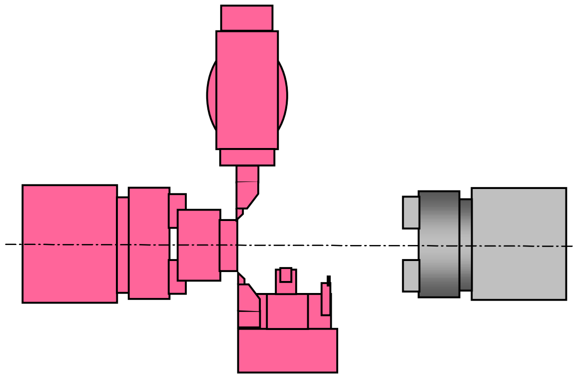
```
UNo.   MAT.    OD-MAX   ID-MIN   LENGTH  WORK FACE  ATC MODE      RPM   LTUR DIA
 0   CST IRN    100.     20.      100.      1.                    2000
UNo.    UNIT        No.      SIMUL.No.     RPM
 1   SIMULTAN                    50        2000
UNo.  UNIT     PART      POS-B     CPT-X    CPT-Z    FIN-X    FIN-Z
 2   BAR        OUT        90       100.      0.      0.2      0.2
SNo.    TOOL    NOM. No.  #  PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z  C-SP   FR    M 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  PTN    S-CNR     SPT-X     SPT-Z     FPT-X     FPT-Z   F-CNR/$     R/th      RGH
  1   LIN               ◆         ◆         80.       55.                 ◆          4
UNo.   UNIT      PART      POS-B  PAT.  No.   PITCH  WIDTH FINISH
 3   T.GROOVE     OUT        90      0     1      0.      10.    ◆
SNo.    TOOL     NOM. No.   #   PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z C-SP  FR    M M M
 F 1  GROOVE OUT  10.     ◤ 50   ◆     2.         ◆         ◆     ◆     ◆     ◆    0.1
 FIG         S-CNR     SPT-X     SPT-Z     FPT-X     FPT-Z     F-CNR     ANG      RGH
  1       C 1.        100.      70.       90.       70.      C 1.
UNo.   UNIT      PART      POS-B       FIN-Z
 4   FACING      FACE       90           0
SNo.    TOOL     NOM. No.  #  PAT. DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z  C-SP   FR    M 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 No. # HOLE-ϕ HOLE-DEP  PRE-DIA  PRE-DEP   RGH   DEPTH C-SP   FR   M M M
 1   DRILL     5.          5.      20.       0.       100    PCK1 T   2.5    25  0.089
 FIG  PTN   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        No.      SIMUL.No.     RPM
 6   SIMULTAN                    70        1500
UNo.  UNIT     PART      POS-B     CPT-X    CPT-Z
 7   CORNER     OUT        90        0.2      0.2
SNo.    TOOL     NOM. No.   #   PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR   M 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      POS-B     CPT-X    CPT-Z    FIN-X    FIN-Z
 8   BAR         IN         0       20.       0.      0.2      0.2
SNo.    TOOL    NOM. No.  #   PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z  C-SP   FR   M M M
 F 1  GENERAL IN 15.A    ◤70   ◆     ◆          ◆         ◆     0.     0.   196   0.1
 FIG  PTN    S-CNR     SPT-X     SPT-Z     FPT-X     FPT-Z   F-CNR/$     R/th      RGH
  1   LIN               ◆         ◆         30.       10.                 ◆          4
UNo.   UNIT     CONTI.   REPEAT  NUMBER      RETURN  LOW RET.  WORK No.     EXECUTE
 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.

5-1-3 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.

1. Programming method

```
UNo.   UNIT     PART      POS-B     CPT-X    CPT-Z    FIN-X    FIN-Z
 2   BAR       OUT        90       100.     45.      0.2      0.2
SNo.     TOOL      NOM.  No. # PAT. DEP-1  DEP-2/NUM.   DEP-3  FIN-X  FIN-Z   C-SP   FR  M 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.

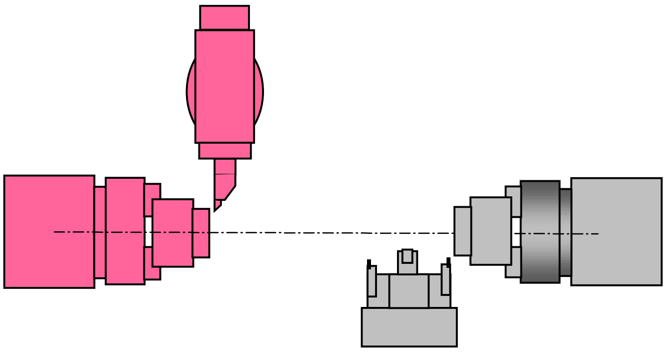
Pressing the **[BALANCE FEED 2]** menu key first and then the enter key adds a lower-turret tool sequence for balanced cutting.

```
UNo.   UNIT     PART      POS-B     CPT-X    CPT-Z    FIN-X    FIN-Z
 2   BAR       OUT        90       100.     45.      0.2      0.2
SNo.     TOOL      NOM.  No. # PAT. DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP    FR   M 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.

5-1-4 Simultaneous machining of processes 1 and 2, using the upper and lower turrets (optional)

The use of the two-workpiece machining unit (2 WORKPC) allows machining in processes 1 and 2 to be performed simultaneously.



D740PA063

1. Flow of processes 1 and 2 simultaneous machining program

For simultaneous machining of processes 1 and 2, you need to set up the workpiece transfer unit before setting up the two-workpiece machining unit. Then, set up the two-workpiece machining unit (START), machining on the No. 1 spindle side, machining on the No. 2 spindle side, and two-workpiece machining unit (END).

```
Common unit
    ⇩       (Removing the workpiece from the No. 2 spindle)
TRANSFER unit (HEAD: 1 → 2)
    ⇩       (Mounting a workpiece in the No. 1 spindle
2 WORKPC unit   (START)
    ⇩
Machining unit for the No. 1 spindle
    ⇩
Machining unit for the No. 2 spindle
    ⇩
2 WORKPC unit   (END)
    ⇩
END unit        (CONTI.: 1)
```

2. Setting data in the 2 WORKPC unit

UNo.	UNIT	PAT.	SP1/SP2
*	2 WORKPC	[1]	[2]

[1] PAT.

Specify **START** when starting simultaneous machining of processes 1 and 2, and specify **END** when stopping it.

Note 1: If **END** is specified before **START**, the alarm **DBL SPDL OPER UNIT ERROR** is issued.

Note 2: If **START** is specified again between **START** and **END**, the alarm **DBL SPDL OPER UNIT ERROR** is issued.

[2] SP1/SP2

When using the upper turret for machining on the No. 1 spindle side and the lower turret for marching on the No. 2 spindle side, select **HI/LOW**.

When using the upper turret for machining on the No. 2 spindle side and the lower turret for marching on the No.1 spindle side, select **LOW/HI**.

3. Example of program

Here is an example of a program for the machining of 10 workpieces that need to be machined in two processes each.

```
UNo.   MAT.    OD-MAX   ID-MIN   LENGTH  WORK FACE  ATC MODE      RPM   LTUR DIA
 0   CST IRN    100.      0.      80.       0.                    2000
UNo.   UNIT    PAT.   HEAD   SPDL  PUSH CHUCK   W1   W2  Z-OFFSET  C1   C2  C-OFFSET LTUR ESC  TNo.
 1   TRANSFER CHUCK  1→2     4      1     ◆    -950   0    750      0    0      0      -450     101
UNo.   UNIT    No.   M1    M2    M3    M4    M5    M6    M7    M8    M9    M10   M11   M12
 2    M-CODE          0
UNo.   UNIT    PAT.      SP1/SP2
 3   2 WORKPC  START     HI/LOW
UNo.   UNIT    No.   M1    M2    M3    M4    M5    M6    M7    M8    M9    M10   M11   M12
 4    M-CODE          0
UNo.   UNIT    TYPE   HEAD   SPDL
 5     HEAD    SIN      1     ◆
UNo.   UNIT    PART     POS-B     CPT-X    CPT-Z    FIN-X    FIN-Z
 6   BAR       OUT       90       100.      0.       0.2      0.2
SNo.    TOOL    NOM. No.  #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z   C-SP    FR    M  M  M
 R 1  GENERAL  OUT  45.A        0     5.       ◆        ◆      ◆      ◆      ◆    0.45
 F 2  GENERAL  OUT  30.E        ◆     ◆        ◆        ◆      0.     0.    196   0.1
 FIG   PTN    S-CNR    SPT-X    SPT-Z    FPT-X    FPT-Z   F-CNR/$    R/th     RGH
  1   LIN                ◆        ◆       80.      30.                ◆        4
UNo.   UNIT     TYPE      HEAD     SPDL
 7   HEAD       SIN        2        ◆
UNo.   UNIT    MODE   POS-B   POS-C   DIA    DEPTH   CHMF
 8   DRILLING   XC      ◆       ◆      5.     20.     0.
SNo.   TOOL    NOM-ϕ No. # HOLE-ϕ HOLE-DEP  PRE-DIA  PRE-DEP   RGH    DEPTH   C-SP    FR    M  M  M
 1   DRILL     5. ◤           5.      20.      0.       100    PCK1 T   2.5     25   0.089
 FIG   PTN   SPT-R/x   SPT-C/y    SPT-Z     SPT-Y     NUM.   ANGLE    Q     R
  1   ARC     50.        0.        30.        0.        4     90.     0     1
UNo.   UNIT     PAT.    SP1/SP2
 9   2 WORKPC   END       ◆
UNo.   UNIT    CONTI.   REPEAT  NUMBER     RETURN  LOW RET.  WORK No.     EXECUTE
 10   END         1        11                                               ◆
```

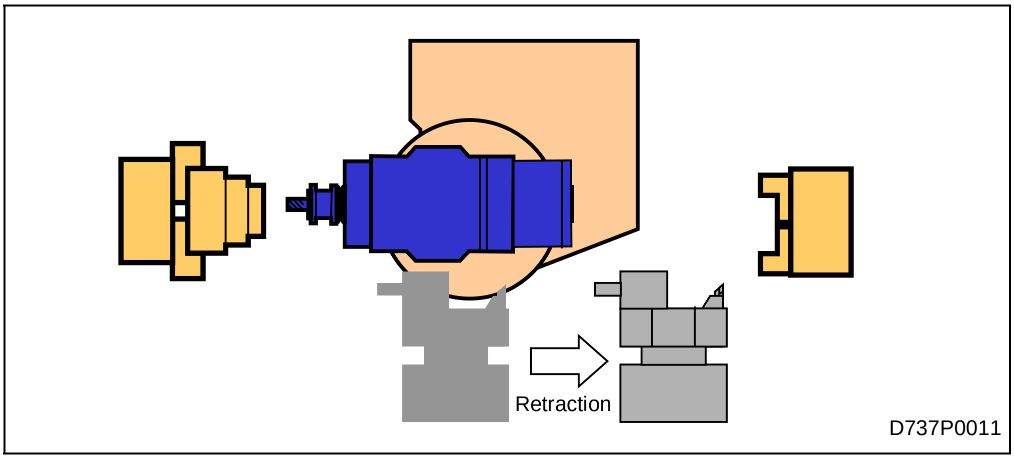
Set **START/END** in the 2 WORKPC unit.

Note: Air cutting occurs in the No. 2 spindle when machining the first workpiece, and in the No. 1 spindle when machining the last workpiece.

4. Other precautions

- If there is a discrepancy between the **SP1/SP2** setting and the turret setting in the tool sequence, the alarm **DBL SPDL OPER ILLEGAL TUR ASIGN** is issued.
- During the machining of 2 workpieces, the item # cannot be specified in the tool sequence.
- During the machining of 2 workpieces, the following operations cannot be performed: transfer of workpieces, simultaneous machining, balanced cutting and retraction.
- During the machining of 2 workpieces, the measuring unit cannot be used.
- During the machining of 2 workpieces, only SINGLE operation in the head selection unit can be selected.
- During the machining of 2 workpieces, cross commands cannot be entered in the subprogram and manual program machining unit.
- In the 2 WORKPC unit with START designation, the upper and the lower turrets are placed in a queue.
- If the turrets interfere with each other during the machining of 2 workpieces, set the 2 WORKPC unit with END designation and use the unit anew to time the operation of the upper and lower turrets.

5-2 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.

1. 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.

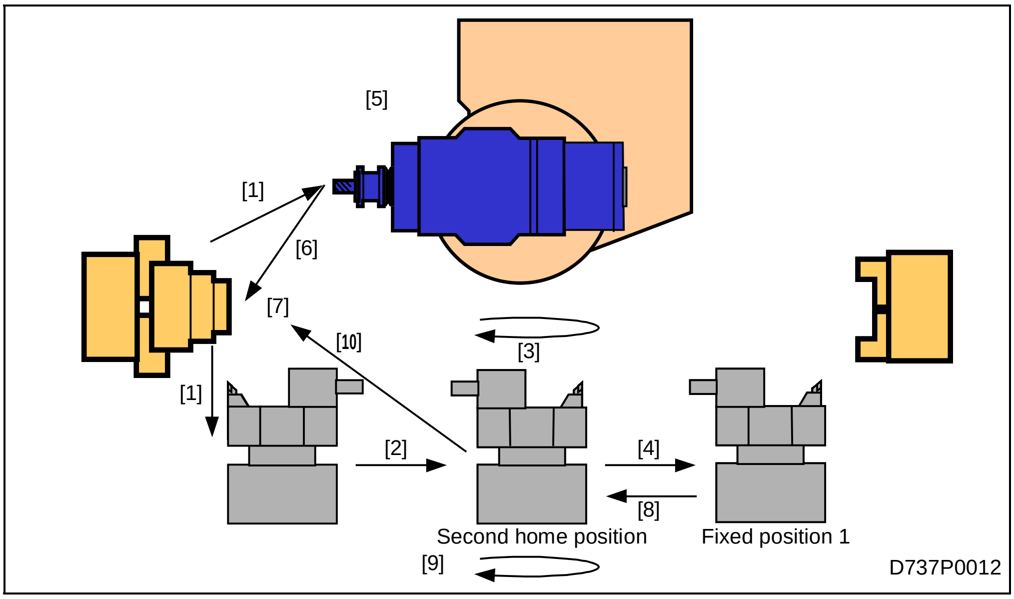
```
UNo.   UNIT     MODE   POS-B   POS-C    DIA    DEPTH    CHMF
 1   DRILLING    XY      ◆      45.     10.     30.      0.
SNo.   TOOL    NOM-φ  #   HOLE-φ HOLE-DEP PRE-DIA PRE-DEP   RGH    DEPTH    C-SP    FR   M M M
 1   DRILL      10.  ◤1   10.      30.       0.      100    DRIL T   5.      36   0.294
 FIG  PTN    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    POS-B   PAT.  No.  PITCH    WIDTH   FINISH
 2   T.GROOVE   OUT      90      0     1     0.       10.      ◆
SNo.     TOOL       NOM.    #   PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z  C-SP   FR  M 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 **SU10**.

2. Description of operation

Upper turret	Lower turret
[1] Movement to tool-rotating position	[1] Movement to lower-turret tool changing position [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.

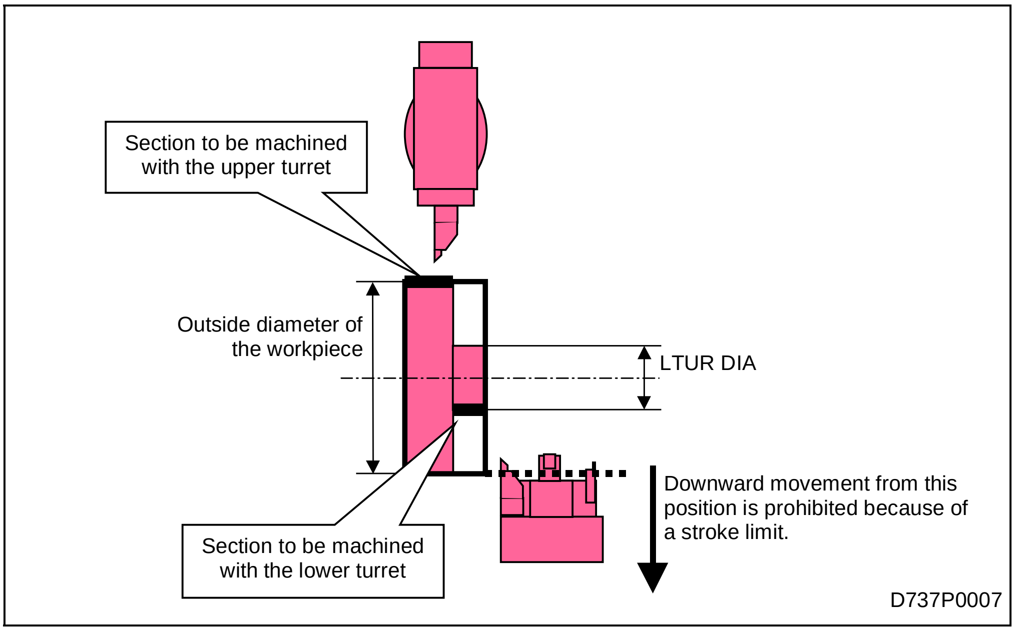
3. Description of parameters

Parameters relating to the lower-turret retraction function are listed below. See the separate Parameter List/Alarm List/M-Code List for further details.

SU52: Tool number of retraction tool 1
SU53: Tool number of retraction tool 2
SU97: X-axis coordinate of fixed position 1
SU98: Z-axis coordinate of fixed position 1
SU199: X-axis coordinate of fixed position 2
SU100: Z-axis coordinate of fixed position 2
M5 (X, Z): Second home position

5-3 Other Setup Items

5-3-1 LTUR 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 **LTUR DIA** in the common unit as the safe outside-diameter value for the lower turret. The lower turret judges the setting of **LTUR 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	ATC MODE	RPM	LTUR DIA
0	CST IRN	280.	0.	55.	0.		2000	130.

In other cases, no data needs to be specified in **LTUR DIA**.

6 TPC DATA SETTING

Tool path control (TPC) data can be set for each unit of the program. The TPC data consists of data items used to adjust tool paths and relay points.

Tool paths are automatically created using the data that has been set on the **PROGRAM** display, and the data that has been set in various parameters. The TPC data is intended to allow unit-by-unit modification of the tool paths that have thus been created, and thus to remove unnecessary paths or prevent interference.

The TPC data, therefore, does not always need to be set to perform machining operations.

The TPC data cannot be set for the following units:

- Common unit
- M-code unit
- End unit
- Subprogram unit
- Process end unit
- Materials shape unit
- Head selection unit
- Simultaneous machining unit

6-1 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  ATC MODE    RPM   LTUR DIA
  0   CBN STL   60.     0.     60.       0.         0       2000
UNo.  UNIT   PART   POS-B  CPT-X  CPT-Z  FIN-X  FIN-Z
  1   BAR      OUT     90.     60.     0.      0.2     0.1
SNo.   TOOL          NOM.    No. #  PAT.   DEP-1  DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP  FR  M M M
 R 1  GENERAL  OUT   20.  A          0     2.5       ◆          ◆      ◆      ◆     25  0.1
 F 2  GENERAL  OUT   20.  B          ◆      ◆        ◆          ◆      0.     0.    63  0.1
FIG   PTN  S-CNR  SPT-X     SPT-Z     FPT-X    FPT-Z    F
  1   LIN   C  2.    ◆         ◆         50.      50.      C  5.       ◆      ▼▼ 4
UNo.    UNIT     PART  POS-B  PAT.  No.  PITCH      WIDTH  FINISH
  2   T.GROOVE   OUT    90.    0     1    20.         5.      ◆
SNo.   TOOL        NOM.    No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M M M
 F 1  GROOVE OUT   5.   A          ◆      2.       ◆        ◆      ◆      ◆     120   0.08
FIG       S-CNR     SPT-X     SPT-Z     FPT-X    FPT-Z     F-CNR     ANG    RGH
  1                  50.       20.       40.      20.                         ▼▼ 4
UNo.  UNIT  CONTI. REPEAT  SHIFT  NUMBER  ATC  RETURN   LOW RET.  WORK No.       EXECUTE
  3   END      0      ◆       ◆       0      0    END      END                      ◆
```

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  POS-B  CPT-X  CPT-Z   FIN-X     FIN-Z
 1     BAR   OUT    90.    60.    0.      0.2       0.1          ← A

PARAMETER      TC37      TC38      TC39      TC40      TC62     TC45
                ■         ■         ■         ■         ■        ■
               TC1       TC67      TC68      TC5       TC6      TC71
                ■         ■         ■         ■         ■        ■
               TC13      TC15
                ■         ■

UPPER ROUGH   ROTATE POSITION SU10 ■  SU50 ■   SU51 ■  X  □   Z   □
              APPROACH RELAY POINT [AUTO]      ESCAPE RELAY POINT [AUTO]
                   X     Y     Z     M     S         X    Y    Z    M    S
               1   □     □     □     □     □     1   □    □    □    □    □
               2   □     □     □     □     □     2   □    □    □    □    □
               3   □     □     □     □     □     3   □    □    □    □    □

      FINISH  ROTATE POSITION SU10 ■  SU50 ■   SU51 ■  X  □   Z   □
              APPROACH RELAY POINT [AUTO]      ESCAPE RELAY POINT [AUTO]
                   X     Y     Z     M     S         X    Y    Z    M    S
               1   □     □     □     □     □     1   □    □    □    □    □
               2   □     □     □     □     □     2   □    □    □    □    □
               3   □     □     □     □     □     3   □    □    □    □    □

LOWER ROUGH   ROTATE POSITION SU10 ■  SU50 ■   SU51 ■  X  □   Z   □
              APPROACH RELAY POINT [AUTO]      ESCAPE RELAY POINT [AUTO]
                     X         Z     M    S            X        Z     M   S
               1     □         □     □    □    1       □        □     □   □
               2     □         □     □    □    2       □        □     □   □
               3     □         □     □    □    3       □        □     □   □

      FINISH  ROTATE POSITION SU10 ■  SU50 ■   SU51 ■  X  □   Z   □
              APPROACH RELAY POINT [AUTO]      ESCAPE RELAY POINT [AUTO]
                     X         Z     M    S            X        Z     M   S
               1     □         □     □    □    1       □        □     □   □
               2     □         □     □    □    2       □        □     □   □
               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 **SU50**: 2500 (in 0.001 mm steps)

↓

SU50 data displayed on the **TPC** display: 2.500 (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, eight blocks of program memory (maximum) will be used.

If TPC data has not been set:

```
UNo.    MAT.     OD-MAX   ID-MIN
 0     CBN STL    60.       0.
UNo.    UNIT      PART     POS-B
 1      BAR       OUT       90.
SNo.     TOOL       NOM.     No.
 F1   GENERAL OUT  20. A
 F2   GENERAL OUT  20. B
FIG      PTN      S-CNR    SPT-X
 1       LIN      C 2.       ◆
UNo.     UNIT      PART    POS-B
 2     T.GROOVE    OUT      90.
SNo.     TOOL       NOM.
 F1    GROOVE OUT   5. A
FIG               S-CNR    SPT-X
 1                          50.
UNo.     UNIT     CONTI.  REPEAT
 3       END        0        ◆
```

+ mark

If TPC data has been set for unit No. 1:

```
UNo.    MAT.     OD-MAX   ID-MIN
 0     CBN STL    60.       0.
UNo.    UNIT      PART     POS-B
 +1     BAR       OUT       90.
SNo.     TOOL       NOM.     No.
 F1   GENERAL OUT  20. A
 F2   GENERAL OUT  20. B
FIG      PTN      S-CNR    SPT-X
 1       LIN      C 2.       ◆
UNo.     UNIT      PART    POS-B
 2     T.GROOVE    OUT      90.
SNo.     TOOL       NOM.
 F1    GROOVE OUT   5. A
FIG               S-CNR    SPT-X
 1       TPR                50.
UNo.     UNIT     CONTI.  REPEAT
 3       END        0        ◆
```

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.

6-2 Description of Each TPC Data Item of Turning Unit and Measurement Unit

```
UNo.  UNIT   PART   POS-B   CPT-X   CPT-Z   FIN-X   FIN-Z     } (a)
 1    BAR    OUT    90.     60.     0.      0.2     0.1
PARAMETER    TC37   TC38    TC39    TC40    TC62    TC45      ⎫
             1.     1.      2.      2.      0       0.2       ⎪
             TC1    TC67    TC68    TC5     TC6     TC71      ⎬ (b)
             100    1.      1.      50      50      1000      ⎪
             TC13   TC15                                      ⎪
             100    100                                       ⎭

UPPER ROUGH  ROTATE POSITION  SU10 6  SU50 2.5  SU51 5.  X  Z   (c)
             APPROACH RELAY POINT [AUTO]   ESCAPE RELAY POINT [AUTO]
               X    Y    Z    M    S         X    Y    Z    M    S
             1                             1
  (f)        2          (d)                2          (e)
             3                             3

      FINISH ROTATE POSITION  SU10 6  SU50 2.5  SU51 5.  X  Z   (c)
             APPROACH RELAY POINT [AUTO]   ESCAPE RELAY POINT [AUTO]
               X    Y    Z    M    S         X    Y    Z    M    S
             1                             1
  (f)        2          (d)                2          (e)
             3                             3

LOWER ROUGH  ROTATE POSITION  SU10 6  SU50 2.5  SU51 5.  X  Z   (c)
             APPROACH RELAY POINT [AUTO]   ESCAPE RELAY POINT [AUTO]
               X     Z     M     S           X     Z     M     S
             1                             1
  (f)        2          (d)                2          (e)
             3                             3

      FINISH ROTATE POSITION  SU10 6  SU50 2.5  SU51 5.  X  Z   (c)
             APPROACH RELAY POINT [AUTO]   ESCAPE RELAY POINT [AUTO]
               X     Z     M     S           X     Z     M     S
             1                             1
  (f)        2          (d)                2          (e)
             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.

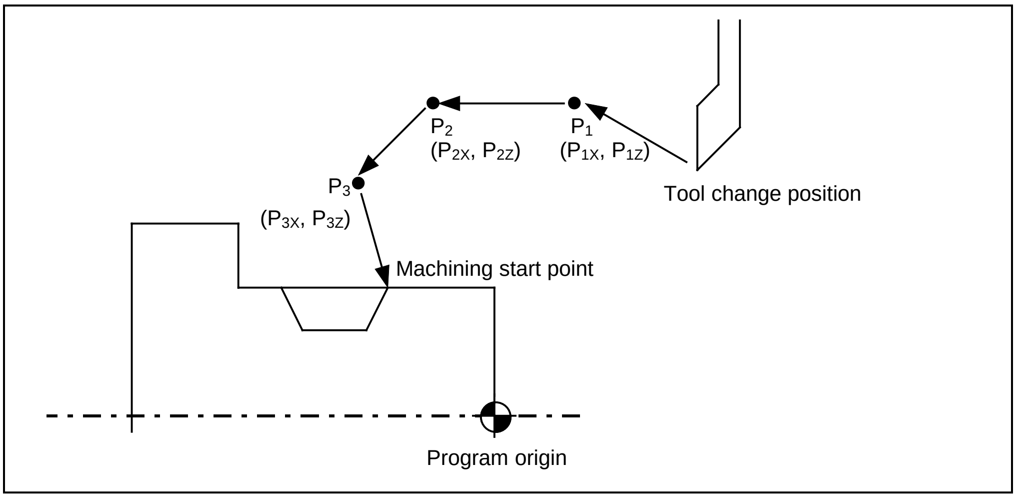
(b) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display. Refer to the separate Parameter List/Alarm List/M-Code 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 of items **(c)**, **(d)**, and **(e)** can be set for each turret and each process. Set TPC data only for the corresponding process. For example, TPC data for the lower turret does not need to be set in a unit intended only for machining with the upper turret, and TPC data for the finishing does not need to be set in a unit intended only for rough machining, either.

(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 separate Parameter List/Alarm List/M-Code List for details of **SU10**, **SU50** and **SU51**.

(d) 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]**". Finally enter the coordinates of the relay points, required M-codes, and S-codes for revolution speed on three lines (1, 2 and 3) in the desired order of relaying. M- and S-codes in the line without axis feed command are invalid.



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:

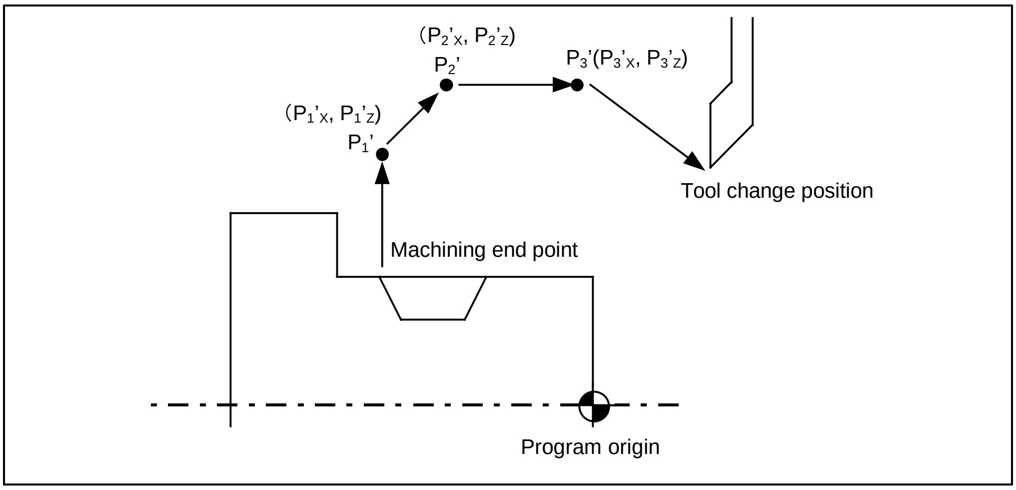
```
APPROACH RELAY POINT [MANU]
```

	X	Y	Z	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 or in radius values, for the turning or milling unit respectively.
- 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.

(e) Use this section to modify the escape path so that interference does not occur.
Refer to the description in (d) for details on data setting.



ESCAPE RELAY POINT [MANU]

	X	Y	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'

(f) For the MMS unit, WORK MES unit, TOOL MES unit, and TRANSFER unit set M-codes to be executed.

- When two M-codes are set here, they are executed simultaneously.

Note: 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.

7 PROGRAM EDITING

This chapter describes operating procedures for editing programs already created. It also describes the various editing functions of the NC unit.

7-1 Operating Procedures for Editing Programs

1. 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.

2. Program editing functions

Pressing the menu changeover key with the **PROGRAM** display remaining on the screen displays the following menu:

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
	1			2	3	4	4	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 or sequence in the program
2	INSERT	To insert one blank line for unit or sequence into any position on the **PROGRAM** display
3	ERASE	To delete a specific unit or sequence existing in the program currently being displayed on the **PROGRAM** display
4	COPY	To copy the data in unit of program, unit or machining shape (sequence)

7-2 Search

There are the following five types of search:

- Search for a unit number
- Search for the end of a program
- Search for a unit name
- Search for a name of a tool
- Search for a workpiece transfer unit

1. Search for a unit number

This function is used for displaying a unit to be checked or modified in the program. The cursor is displayed at the intended unit.

(1) Display the menu including **[SEARCH]**. During program editing, press the menu selector key to display the menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(2) Press the **[SEARCH]** menu key.

➔ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH	TRS SEARCH				

(3) Press the **[UNIT No. SEARCH]** menu key.

➔ This causes the display of **[UNIT No. SEARCH]** to reverse and the screen displays the message **UNIT NUMBER <INPUT>?**.

(4) Enter the number of the unit to be found.

Example: Unit number 10

Press the following keys: [1] [0] [INPUT]

➔ The cursor moves to the number of the unit entered and the unit is displayed on the screen.

```
UNo.  UNIT
  10  █    ← The cursor moves here.
SNo.  TOOL
   1
   2
FIG  PTN
```

Note 1: When the entered number of the unit does not exist in the program, the alarm **407 DESIGNATED DATA NOT FOUND** is displayed.

Note 2: In the UNIT No. SEARCH function mode, the number of the unit entered is searched for from the head of the program, wherever the present position of the cursor is.

2. Search for the end

This LAST SEARCH function moves the cursor to the end of the program. This function is used for resuming the programming whilst in progress.

(1) Display the menu including **[SEARCH]**. During program editing, press the menu selector key to display the menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(2) Press the menu key **[SEARCH]**.

➔ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH	TRS SEARCH				

(3) Press the **[LAST SEARCH]** menu key.

➔ This causes the display of **[LAST SEARCH]** to reverse and the screen displays the message **LAST SEARCH <INPUT>?**.

(4) Press the input key INPUT .

➔ The cursor goes to the end of the program and the last line is displayed on the screen.

```
FIG

UNo.  UNIT
       █  ← The cursor moves here.
```

3. Search for a unit name

This function serves to display the line of a required unit on the basis of the unit name. The cursor is displayed at the intended unit.

(1) Display the menu including **[SEARCH]**. During program editing, press the menu selector key to display the menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(2) Press the **[SEARCH]** menu key.

➔ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH	TRS SEARCH				

(3) Press the **[UNIT SEARCH]** menu key.

➔ The unit menu is displayed and the message **UNIT NAME SEARCH <INPUT>?** is indicated on the screen.

UNIT NAME SEARCH <INPUT>?

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING	MANUAL PROGRAM			END		>>>

(4) Select the name of the unit to be found.

Example: Search under the name of the RGH CBOR machining unit.

1) Presse the **[POINT MACH-ING]** menu key.

➔ The point machining unit menu is displayed.

```
                                          UNIT NAME SEARCH <INPUT>?           (          )
```

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

2) Press the menu key **[RGH CBOR]**.

➔ The display of **[RGH CBOR]** is then reversed.

3) Press the input key INPUT.

➔ The cursor then goes to the line of the entered unit and the unit is displayed on the screen.

```
UNo.   UNIT
 10    RGH CBOR
SNo.   TOOL
   1   CTR-DR
   2   DRILL
```

The cursor moves here.

4) Another pressing of the input key INPUT results in the searching for the following same unit name.

```
UNo.   UNIT
 24    RGH CBOR
SNo.   TOOL
  1    CTR-DR
  2    DRILL
```

The cursor moves to the following same unit name.

Note: The alarm **407 DESIGNATED DATA NOT FOUND** is displayed when the name of the unit specified for the search does not exist after the cursor position.

4. Search for name of a tool

This function, TOOL SEARCH, serves to display the sequence line of the required tool on the basis of the name of the tool. The cursor is displayed at the intended tool sequence line.

(1) Display the menu including **[SEARCH]**. During program editing, press the menu selector key to display the menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(2) Press the **[SEARCH]** menu key.

➔ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH	TRS SEARCH				

(3) Press the **[TOOL SEARCH]** menu key.

➔ The tool names menu is displayed and the screen displays the message **TOOL NAME SEARCH <INPUT>?**.

TOOL NAME SEARCH <INPUT>?

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR				>>>	**a**

- Pressing the **[>>>]** menu key changes the menu **a** → **b** → **c** → a in this order.

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR	CHIP VACUUM		>>>	**b**
GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			>>>	**c**

Example: Search under the tool name: DRILL

1) Press the **[DRILL]** menu key.

➔ This will cause the display of **[DRILL]** to reverse.

2) Press the input key INPUT .

➔ The cursor moves to the sequence line of the tool entered and the sequence line is displayed on the screen.

SNo.	TOOL
1	CTR-DR
2	DRILL
3	CHAMFER

The cursor moves here.

3) Another pressing of the input key INPUT results in finding the following same tool name.

SNo.	TOOL
1	CTR-DR
2	DRILL
3	CHAMFER
4	END MILL

The cursor moves to the following same unit name.

Note: The alarm **407 DESIGNATED DATA NOT FOUND** is displayed when the name of the tool specified for search does not exist after the cursor position.

5. Search for a workpiece transfer unit

This function searches for a workpiece transfer unit and move the cursor to the unit.

(1) Display the menu including **[SEARCH]**. During program editing, press the menu selector key to display the menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

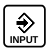
(2) Press the **[SEARCH]** menu key.

➔ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH	TRS SEARCH				

(3) Press the **[TRS SEARCH]** menu key.

➔ The display of menu item is reversed and the message **TRS UNIT SEARCH <INPUT>?** is indicated on the screen.

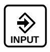
(4) Press the input key  .

➔ The cursor then goes to the line of the workpiece transfer unit and the unit is displayed on the screen.

```
FIG

UNo.    UNIT
 26     TRANSFER
UNo.    UNIT
 27
```

The cursor moves here.

Another pressing of the input key  results in finding the following workpiece transfer unit.

```
FIG

UNo.    UNIT
 38     TRANSFER
UNo.    UNIT
 39
```

The cursor moves to the following workpiece transfer unit.

Note: The alarm **407 DESIGNATED DATA NOT FOUND** is displayed when the workpiece transfer unit does not exist after the cursor position.

7-3 Insertion

This INSERT function is used to insert (add) one unit, one tool sequence or one shape sequence during the creating or editing of a program.
The following three types of insertion are available:

- Insertion of a unit
- Insertion of a tool sequence
- Insertion of a shape sequence

The line to be inserted (unit, tool) is determined depending on the cursor position.

1. Insertion of a unit

Perform the following procedure to insert a unit.

Menu selection: **[INSERT]**

(1) Bring the cursor to the next line of the unit to be inserted.

```
UNo.     MAT
 0     CBN STL
UNo.     UNIT
 1     DRILLING
SNo.     TOOL
 1     CTR-DR
 2     DRILL
```

In the case where a line is inserted here, the cursor is to be located here.

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[INSERT]** menu key.

➔ The display of **[INSERT]** is reversed and the screen displays the message **LINE INSERT <INPUT>?**.

(4) Press the input key INPUT.

```
UNo.    MAT.                 UNo.    MAT.
 0     CBN STL                0     CBN STL
UNo.    UNIT                 UNo.    UNIT
 1     DRILLING               1     █
SNo.    TOOL                 UNo.    UNIT
 1     CTR-DR                 2     DRILLING
 2     DRILL                 SNo.    TOOL
                              1     CTR-DR
                              2     DRILL
```

Unit inserted

Note 1: When the cursor is located on the line of the unit which follows the line of the shape sequence at step (1), the empty shape sequence is inserted. Then, pressing the **[SHAPE END]** menu key results in the insertion of an empty unit.

➔ This causes the insertion of an empty unit.

```
SNo.   TOOL          SNo.   TOOL          SNo.   TOOL
 1     CTR-DR         1     CTR-DR         1     CTR-DR
 2     DRILL          2     DRILL          2     DRILL
FIG    PTN           FIG    PTN           FIG    PTN
 1     PT             1     PT             1     PT
UNo.   UNIT           2     █             UNo.   UNIT
 2     FCE MILL      UNo.   UNIT           2     █
SNo.   TOOL           2     FCE MILL      UNo.   UNIT
 1     FCE MILL      SNo.   TOOL           3     FCE MILL
```

Shape sequence inserted   Unit inserted

Note 2: When the insertion operation is done in UNo. 0 (common unit), the alarm **409 ILLEGAL INSERTION** is displayed.

(5) Enter the data. Refer to the Chapter 3, “PROGRAM CREATION” for the selection of each unit and data setting.

Note: When the machining unit is inserted, the tool sequence and the shape sequence are successively inserted progressively with the development of the operation.

2. Insertion of a tool sequence

Perform the following procedure to insert a tool sequence.

Menu selection: **[INSERT]**

(1) Bring the cursor to the next line of the tool sequence to be inserted.

```
UNo.    MAT.
  0    CBN STL
UNo.    UNIT
  1    DRILLING
SNo.    TOOL      <---- Case where a tool sequence line is to be inserted here
  1    CTR-DR
  2    DRILL
FIG     PTN
  1     PT
```

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[INSERT]** menu key.

➔ The display of **[INSERT]** is reversed and the screen displays the message **LINE INSERT <INPUT>?**.

(4) Press the input key INPUT.

➔ This causes the insertion of an empty tool sequence line.

```
UNo.    MAT.                 UNo.    MAT.
  0    CBN STL                 0    CBN STL
UNo.    UNIT                 UNo.    UNIT
  1    DRILLING                1    DRILLING
SNo.    TOOL          =>     SNo.    TOOL
  1    CTR-DR                  1    █        <---- Tool sequence inserted
  2    DRILL                   2    CTR-DR
FIG      PTN                   3    DRILL
  1    PT                    FIG      PTN
                               1    PT
```

(5) Enter the data. Refer to the Chapter 3, “PROGRAM CREATION” for the selection of each unit and data setting.

Note: When the insertion operation is done in UNo. 0 (common unit), the alarm **409 ILLEGAL INSERTION** is displayed.

3. Insertion of a shape sequence

Perform the following procedure to insert a shape sequence.

Menu selection: **[INSERT]**

(1) Bring the cursor to the next line of the shape sequence to be inserted.

```
UNo.   UNIT
  1    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
FIG    PTN        <-- Case where a shape sequence line is to be inserted here
  1    PT
UNo.   UNIT
  2   FCE MILL
```

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[INSERT]** menu key.

➔ The display of **[INSERT]** is reversed and the screen displays the message **LINE INSERT <INPUT>?**.

(4) Press the input key INPUT .

➔ This causes the insertion of an empty shape sequence line.

```
UNo.   UNIT
  1    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
FIG       PTN
  1    PT
UNo.   UNIT
  2   FCE MILL
```

```
UNo.   UNIT
  1    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
FIG       PTN
  1                <-- Shape sequence inserted
  2    PT
UNo.   UNIT
  2    FCE MILL
```

Note: When the cursor is located on the line of the unit which follows the line of the shape sequence, an empty shape sequence is inserted as follows.

```
SNo.   TOOL                    SNo.   TOOL
  1    CTR-DR                    1    CTR-DR
  2    DRILL                     2    DRILL
FIG    PTN          =>         FIG    PTN
  1    PT                        1    PT
UNo.   UNIT                      2    █            <- Shape sequnce
  2    FCE MILL                                       inserted
SNo.   TOOL                    UNo.   UNIT
  1    FCE MILL                  2    FCE MILL
                               SNo.   TOOL
                                 1    FCE MILL
```

(5) Enter the data. Refer to the Chapter 3, "PROGRAM CREATION" for the selection of each unit and data setting.

Note: When the insertion operation is done in UNo. 0 (common unit), the alarm **409 ILLEGAL INSERTION** is displayed.

7-4 Deletion

This ERASE function is used to erase the unit, the tool sequence or the shape sequence which has become unnecessary during the creating or editing of a program.

The following three types of deletion are available:

- Deletion of the unit
- Deletion of the tool sequence
- Deletion of the shape sequence

1. Deletion of the unit

Menu selection: **[ERASE]**

(1) Place the cursor on the unit to be deleted.

```
UNo.    MAT.
  0    CBN STL
UNo.    UNIT
  1    DRILLING  <------  Case where this unit is to be deleted
SNo.    TOOL
  1    CTR-DR
  2    DRILL
```

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[ERASE]** menu key.

➔ The display of **[ERASE]** is reversed and the screen displays the message: **SELECT PROGRAMS - CURSOR?**.

➔ The unit on which the cursor is located is selected (display of the unit line is reversed).

(4) When multiple units are to be deleted at a time, use the upward and downward cursor keys to designate the area.

(5) Press the input key INPUT .

➔ The selected units are then deleted. The tool sequence and the shape sequence in this unit will equally be deleted.

```
UNo.   MAT.
 0     CBN STL
UNo.   UNIT
 1     DRILLING
SNo.   TOOL
 1     CTR-DR
 2     DRILL
FIG    PTN
 1     PT
UNo.   UNIT
 2     RGH CBOR
SNo.   TOOL
 1     CTR-DR
 2     DRILL
 3     END MILL
 4     CHAMFER
FIG    PTN
 1     LIN
UNo.   UNIT
 3     SLOT
```

← Units to be deleted (reversed display)

```
UNo.   MAT.
 0     CBN STL
UNo.   UNIT
 1     SLOT
```

Note: When the deletion operation is done for UNo. 0 (common unit), the alarm **410 ILLEGAL DELETION** is displayed.

2. Deletion of the tool sequence

Menu selection: **[ERASE]**

(1) Place the cursor on the tool sequnece to be deleted.

```
UNo.    MAT.
  0    CBN STL
UNo.    UNIT
  1    DRILLING
SNo.    TOOL
  1     DRILL
FIG     PTN
  1    PT
UNo.    UNIT
  2    FCE MILL
```

Case where this sequence is to be deleted

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[ERASE]** menu key.

➔ The display of **[ERASE]** is reversed and the screen displays the message: **SELECT PROGRAMS - CURSOR?**.

➔ The tool sequence on which the cursor is located is selected (display of the sequence line is reversed).

(4) When multiple tool sequences are to be deleted at a time, use the upward and downward cursor keys to designate the area.

➔ When unit line is inclueded in the area, deletion occurs in the same manner as in “1. Deletion of the unit.”

➔ When tool sequence line is inclueded in the area, deletion occurs in the same manner as in “3. Deletion of the shape sequence.”

(5) Press the input key INPUT .

➔ The designated tool sequence, unit and shape sequence are deleted.

```
UNo.     MAT.
  0     CBN STL
UNo.     UNIT
  1     DRILLING
SNo.     TOOL
  1     CTR-DR      <-- Tool sequence to be deleted (reversed display)
  2     DRILL
FIG      PTN
  1     PT
```

```
UNo.     MAT.
  0     CBN STL
UNo.     UNIT
  1     DRILLING
SNo.     TOOL
  1     DRILL
FIG      PTN
  1     PT
```

3. Deletion of the shape sequence

Menu selection: **[ERASE]**

(1) Place the cursor on the shape sequnece to be deleted.

```
UNo.   MAT.
  0   CBN STL
UNo.   UNIT
  1   DRILLING
SNo.   TOOL
  1   CTR-DR
  2   DRILL
FIG    PTN
  1    LINE   <---- Case where this sequence is to be deleted
  2    PT
UNo.  UNIT
  2   SLOT
```

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[ERASE]** menu key.

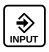
➔ The display of **[ERASE]** is reversed and the screen displays the message: **SELECT PROGRAMS - CURSOR?**.

➔ The shape sequence on which the cursor is located is selected (display of the sequence line is reversed).

(4) When multiple shape sequences are to be deleted at a time, use the upward and downward cursor keys to designate the area.

➔ When unit line is inclueded in the area, deletion occurs in the same manner as in "1. Deletion of the unit."

➔ When tool sequence line is inclueded in the area, deletion occurs in the same manner as in "2. Deletion of the tool sequence."

(5) Press the input key 

.

➔ The designated shape sequence, unit and tool sequence are deleted.

```
UNo.    MAT.
 0     CBN STL
UNo.    UNIT
 1    DRILLING
SNo.    TOOL
 1    CTR-DR
 2    DRILL
FIG     PTN
 1    LINE
 2    PT
UNo.   UNIT
 2    SLOT
```

Shape sequence to be deleted (reversed display)

```
UNo.    MAT.
 0     CBN STL
UNo.    UNIT
 1    DRILLING
SNo.    TOOL
 1    CTR-DR
 2    DRILL
FIG     PTN
 1    PT
UNo.    UNIT
 2    SLOT
```

7-5 Copy

During the process of creating or editing of a program, this COPY function is used to copy another program or one unit/shape sequence of a program in the process of creating or editing.

There are three types of copying depending on the contents to be copied.

- Copying of a program
- Copying of a unit
- Copying of a shape

1. Copying of a program

This PROGRAM COPY function is used to copy another program in the process of creating or editing of a program.

However, the common unit and the end unit cannot be copied.

Menu selection: **[PROGRAM COPY]**

(1) Move the cursor to the line on which another program is inserted.

```
UNo.     MAT.
  0     CBN STL
                  <-------- Case where another program is inserted here
UNo.     UNIT
  1    DRILLING
SNo.     TOOL
  1     CTR-DR
  2      DRILL
```

Note 1: When the cursor is not located on the unit line, the alarm **454 CURSOR POSITION INCORRECT** will be displayed when selecting the **[PROGRAM COPY]** menu key.

Note 2: When the cursor is located on the common unit, the alarm **454 CURSOR POSITION INCORRECT** will be displayed when selecting the menu key **[PROGRAM COPY]**.

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[PROGRAM COPY]** menu key.

➔ The display of **[PROGRAM COPY]** is then reversed and the **WORK No.** window is displayed.

(4) Enter the workpiece number of the program to be copied.

Example: Workpiece number 1000

Press the following keys: [1] [0] [0] [0] [⇒]

➔ The program of workpiece number 1000 is then copied.

Example: Program of WNo. 1000 is copied as follows:

Program in the process of editing

```
UNo.   MAT.
  0    CBN STL
UNo.   UNIT
  1    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
```

Program of WNo. 1000

```
UNo.   MAT.
  0    CST IRN
UNo.   UNIT
  1    WPC-0
UNo.   UNIT
  2    M-CODE
UNo.   UNIT
  3    END
```

Program to be copied

Program after copying

```
UNo.   MAT.
  0    CBN STL
UNo.   UNIT
  1    WPC-0
UNo.   UNIT
  2    M-CODE
UNo.   UNIT
  3    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
```

Program copied

Note 1: When a workpiece number which is not recorded is entered, the alarm **405 PROGRAM No. NOT FOUND** is displayed.

Note 2: When an EIA/ISO program workpiece number is entered, the alarm **440 EIA/ISO PROGRAM DESIGNATED** is displayed.

2. Copying of a unit

In the process of creating or editing a program, this UNIT COPY function is used to perform the copying, unit by unit from the program or from another program. The unit and also the tool sequence and the shape sequence which follow are copied.

Menu selection: **[UNIT COPY]**

(1) Move the cursor to the line on which a unit is copied.

```
UNo.   MAT.
  0   CBN STL
                 <----------  Case where another unit is inserted here
UNo.   UNIT
  1   DRILLING
SNo.   TOOL
  1   CTR-DR
  2   DRILL
```

Note 1: When the cursor is not located on the unit line, the alarm **454 CURSOR POSITION INCORRECT** will be displayed when selecting the menu key **[UNIT COPY]**.

Note 2: When the cursor is located on the common unit (UNo. 0), the alarm **454 CURSOR POSITION INCORRECT** will be displayed when selecting the menu key **[UNIT COPY]**.

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[UNIT COPY]** menu key.

➔ The display of **[UNIT COPY]** is then reversed and the **WORK No.** window is displayed.

(4) Enter the workpiece number of the program containing the unit to be copied.

Example: Workpiece number 1000

Press the following keys: [1] [0] [0] [0] [INPUT]

➔ When the workpiece number is entered, the screen displays the message **UNIT NUMBER <INPUT>?**.

(5) Enter the number of the unit to be copied.

Example: Unit number 1

Press the following keys: 1 INPUT

➔ Unit number 1 in the program of workpiece No. 1000 is then copied.

Example: UNo. 1 of program WNo. 1000 is copied as follows:

Program in the process of editing

```
UNo.   MAT.
  0    CBN STL
UNo.   UNIT
  1    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
```

Program of WNo. 1000

```
UNo.   MAT.
  0    CST IRN
UNo.   UNIT
  1    LINE CTR
SNo.   TOOL
  1    FCE MILL
FIG    PTN
  1    SQR
UNo.   UNIT
  2    RGH CBOR
```

Unit to be copied

Program after copying

```
UNo.   MAT.
  0    CBN STL
UNo.   UNIT
  1    LINE CTR
SNo.   TOOL
  1    FCE MILL
FIG.   PTN
  1    SQR
UNo.   UNIT
  2    DRILLING
SNo.   TOOL
  1    CTR-DR
  2    DRILL
```

Unit copied

Note 1: The common unit UNo. 0 cannot be copied. Any attempt to make such a copy will cause the alarm **402 ILLEGAL NUMBER INPUT** to be displayed.

Note 2: When an unregistered workpiece number is entered, the alarm **405 PROGRAM No. NOT FOUND** is displayed.

Note 3: When a workpiece number of the EIA/ISO program is entered, the alarm **440 EIA/ISO PROGRAM DESIGNATED** is displayed.

3. Copying of shape

This SHAPE COPY function is used to copy the shape sequence in the process of creation or editing of a program. However, it is impossible to perform the copying if the shape sequence line has already been filled with data.

Menu selection: **[SHAPE COPY]**

(1) Move the cursor to the position in which the shape sequence is to be copied.

Example:

```
UNo.     UNIT
  1    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    PT
  2    CIR
  3    SQR
UNo.     UNIT
  2    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    █     <------   Case where the shape sequence of the unit No. 1 is
                       copied on this shape sequence line
```

Note: When the cursor is located in a position other than the shape sequence or when data have already been entered in the shape sequence, the alarm **454 CURSOR POSITION INCORRECT** is displayed.

(2) Display the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(3) Press the **[SHAPE COPY]** menu key.

➔ The display of **[SHAPE COPY]** is then reversed and the screen displays the message **UNIT NUMBER <INPUT>?**.

(4) Enter the number of the unit containing the shape sequence to be copied.

Example: Unit number 1

Press the following keys: [1] [INPUT]

➔ The shape sequence under unit No. 1 is then copied.

Example: The shape sequence under unit No. 1 is copied as follows:

Program before copying

```
UNo.     UNIT
  1    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    PT
  2    CIR
  3    SQR
UNo.     UNIT
  2    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    █
```

Shape sequence to be copied

Program after copying

```
UNo.     UNIT
  1    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    PT
  2    CIR
  3    SQR
UNo.     UNIT
  2    DRILLING
SNo.     TOOL
  1    DRILL
FIG      PTN
  1    PT
  2    CIR
  3    SQR
```

Shape sequence copied

Note 1: When the number of an unestablished unit is entered, the alarm **407 DESIGNATED DATA NOT FOUND** is displayed.

Note 2: When the number of the unit entered does not contain the shape sequence, the alarm **452 NO SHAPE DATA IN UNIT** is displayed.

Note 3: When the type of the shape of the unit entered is different from that of the unit to be copied, the alarm **453 NO SHAPE DATA TO COPY IN UNIT** is displayed.

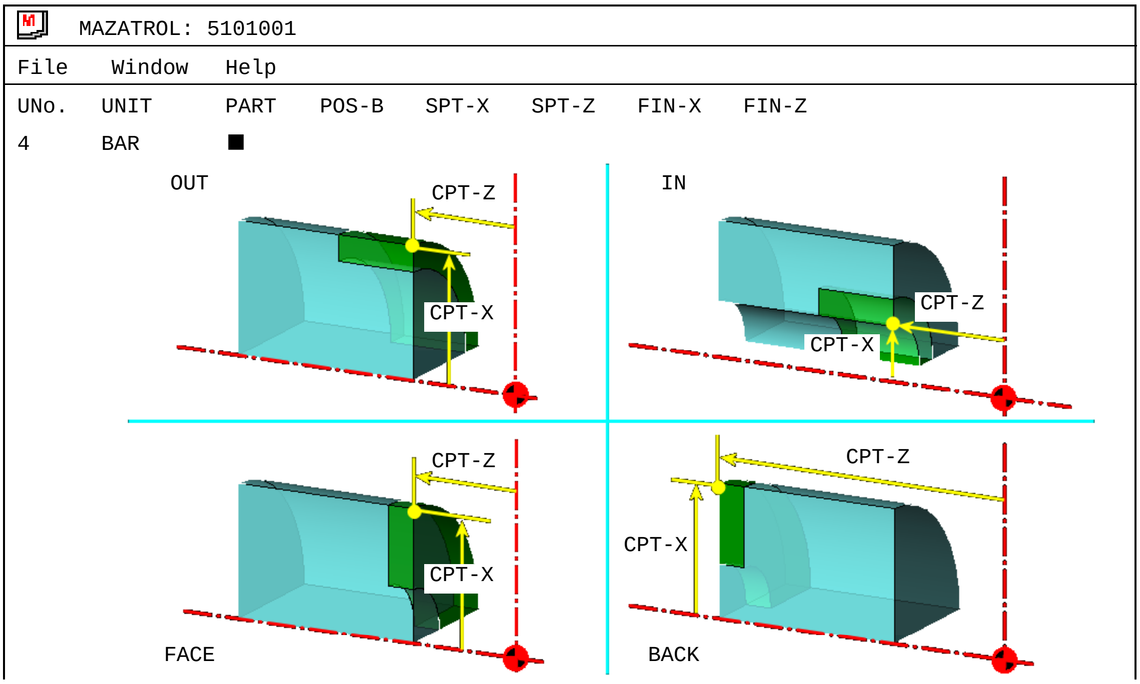
- NOTE -

8 PROGRAM CREATING/EDITING FUNCTIONS

8-1 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.

8-2 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.

8-2-1 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.

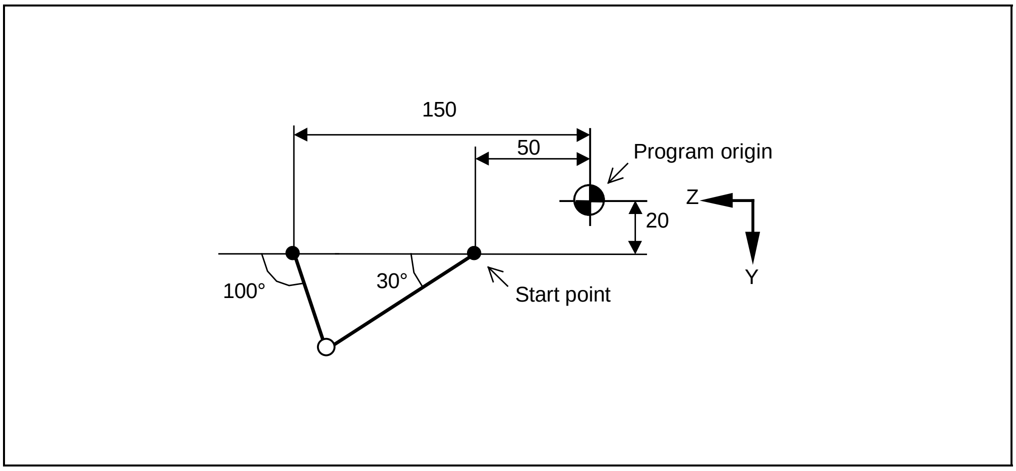
1. Coordinates of the crossing-point

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	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	10.	50.	20.							
2	LINE	◆	?	?	30.						
3	LINE	◆	150.	20.	100.						

FIG	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	10.	50.	20.							
2	LINE	◆	140.7604	72.4005	30.						
3	LINE	◆	150.	20.	100.						

Displayed in yellow

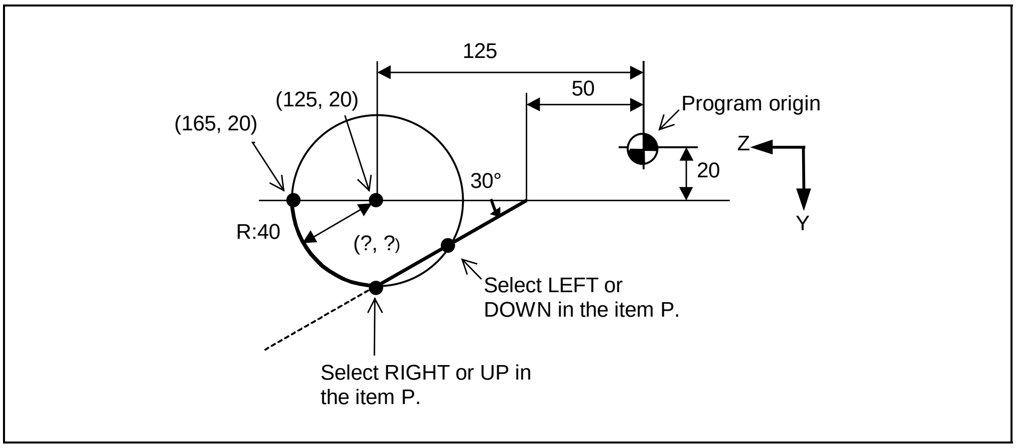


After checking the plane, return to the **PROGRAM** display again and the coordinates so automatically obtained as a crossing-point will be displayed in yellow.

Coordinates are also caluculated when the cursor is set on the **?** item and the **[CALCULAT]** menu key is pressed.

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	PTN	SHIFT-R	Z	Y	R/th	I	J	P	CNR	R-FEED	RGH
1	LINE	10.	50.	20.							
2	LINE	◆	?	?	30.						
3	CW	◆	165.	20.	40.	125.	20.	RGT			

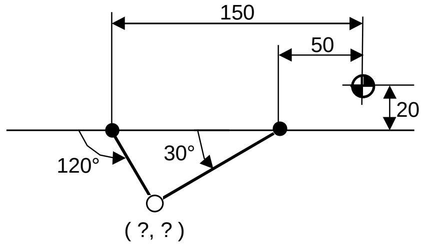
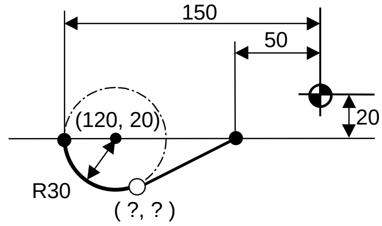
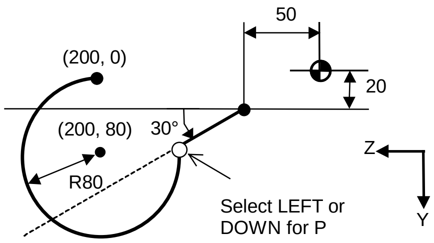
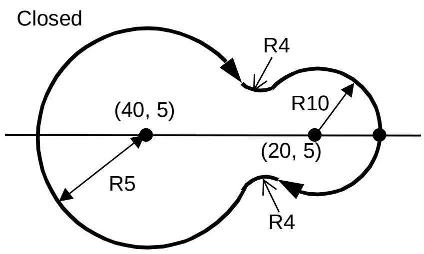
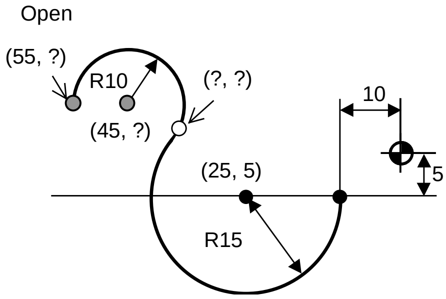


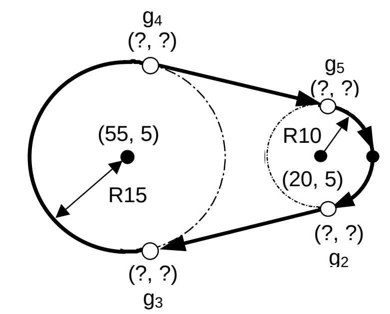
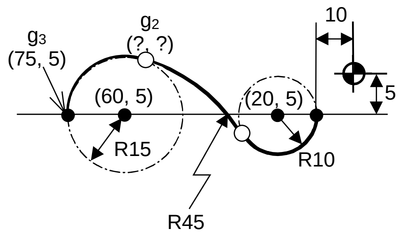
To find a crossing point with the automatic crossing-point calculation function, first view the Y-direction from the origin in either ZY, XY or /Y mode and compare the positions of two crossing points.

If the crossing point at right is the desired one, select **RGT** by pressing the **[RIGHT]** menu key. If the crossing point at left is the desired one, select **LEFT** by pressing the **[LEFT]** menu key. If the crossing point at the plus side is the desired one, this crossing point can likewise be specified by selecting **UP** with the **[UP]** menu key. If the crossing point at the minus side is the desired one, this crossing point can likewise be specified by selecting **DOWN** with the **[DOWN]** 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.

Pattern	Shape	Shape sequence
LINE | LINE		FIG PTN SCHIFT-R X Y R/th I J P CNR 1 LINE 50. 20. 2 LINE ? ? 30. 3 LINE 150. 20. 120.
LINE | ARC (Contact)		FIG PTN SHIFT-R X Y R/th I J P CNR 1 LINE 50. 20. 2 LINE ? ? 3 CW 150. 20. 30. 120. 20.
LINE | ARC (Cross)		FIG PTN SHIFT-R X Y R/th I J P CNR 1 LINE 50. 20. 2 LINE ? ? 30. LEFT 3 CW 200. 0. 80. 200. 80.
ARC | ARC		FIG PTN SHIFT-R X Y R/th I J P CNR 1 CW ? ? 10. 20. 5. UP R4 2 CW ? ? 15. 40. 5. DOWN R4
		FIG PTN SHIFT-R X Y R/th I J P CNR 1 LINE 10. 5. 2 CW ? ? 15. 25. 5. DOWN 3 CCW 55. ? 10. 45. ?

Pattern	Shape	Shape sequence
ARC | LINE | ARC		FIG PTN SHIFT-R X Y R/th I J P CNR 1 LINE ? ? 2 CW ? ? 10. 20. 5. 3 LINE ? ? 4 CCW ? ? 15. 55. 5.
ARC | ARC | ARC		FIG PTN SHIFT-R X Y R/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).

8-2-2 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.

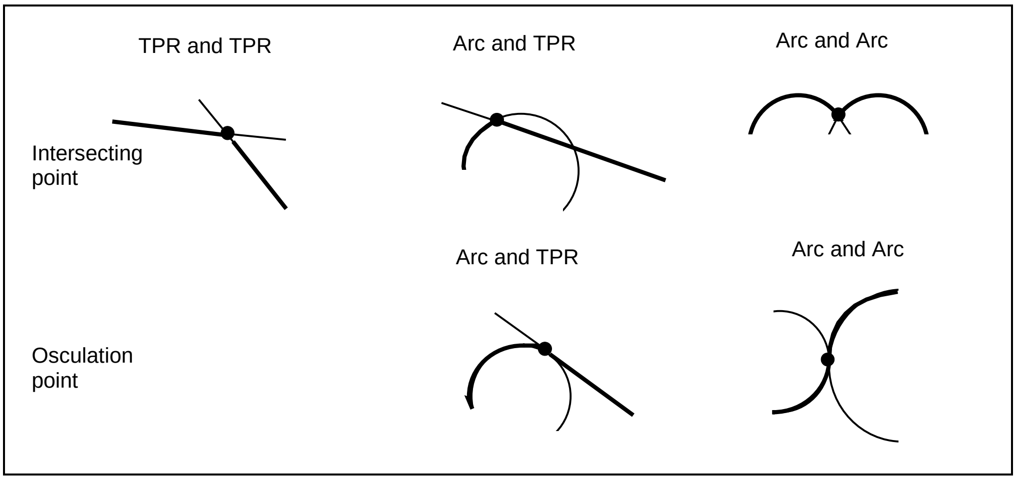
- Automatic calculation within one sequence

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.

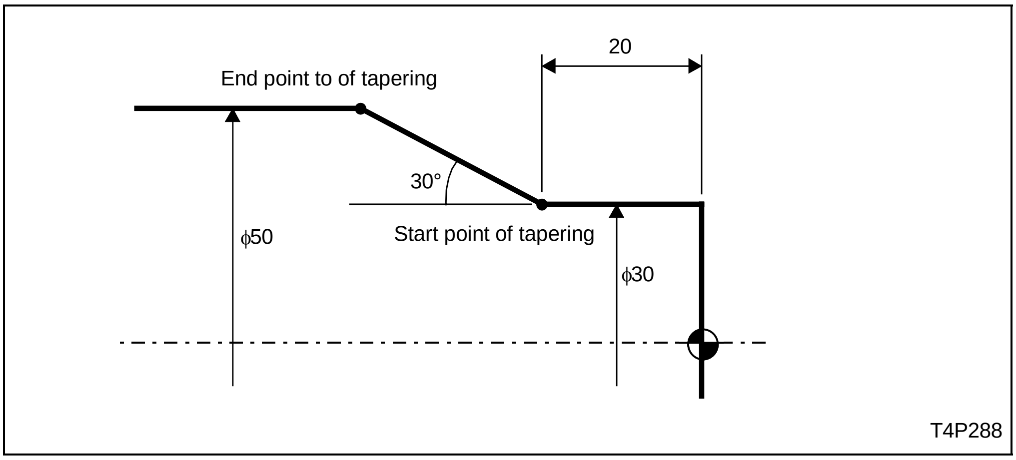


- Automatic calculation can also be performed in grafically checking the programmed data on the **TOOL PATH** or **SHAPE CHECK** display and the result is entered in a program.

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.



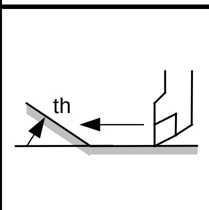
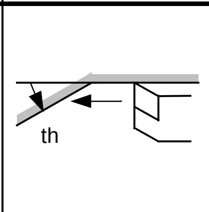
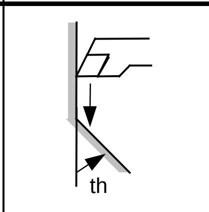
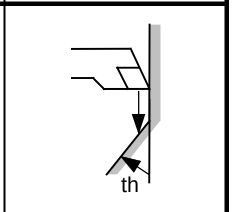
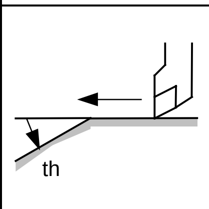
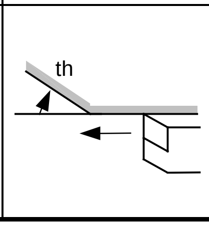
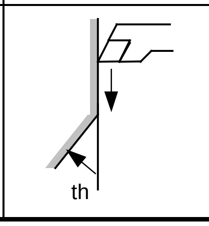
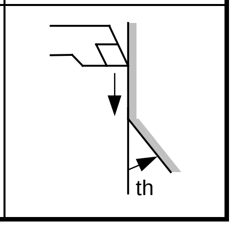
T4P288

Set data as follows:

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z		
*	BAR	OUT	90.	***	***	***	***		
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th	RGH
1	LIN		◆	◆	30.	20.		◆	
2	TPR		30.	20.	50.	?		30.	

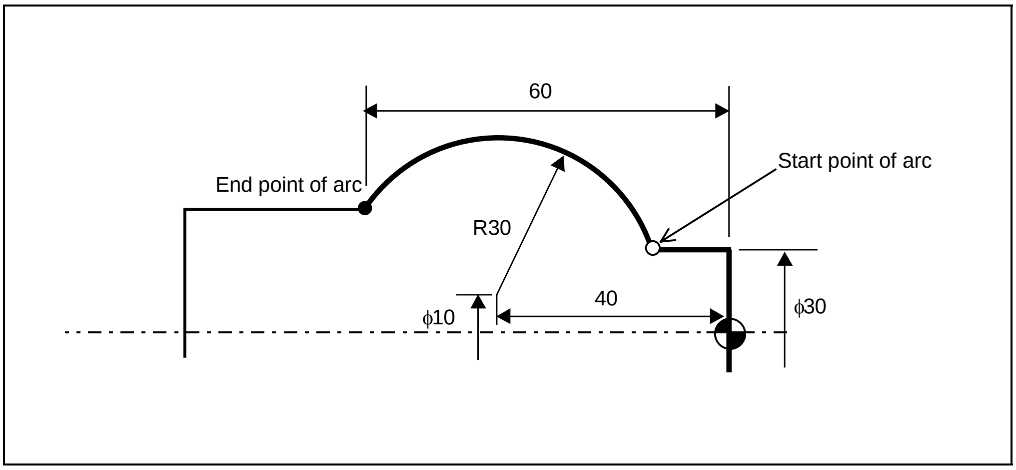
Press the **[INTER PT]** menu key for the unknown **FPT-Z**.
Enter the tapering angle, 30°, for **R/th**.

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	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z		
*	BAR	OUT	90.	***	***	***	***		
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	4	R/th	RGH
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 **R/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:

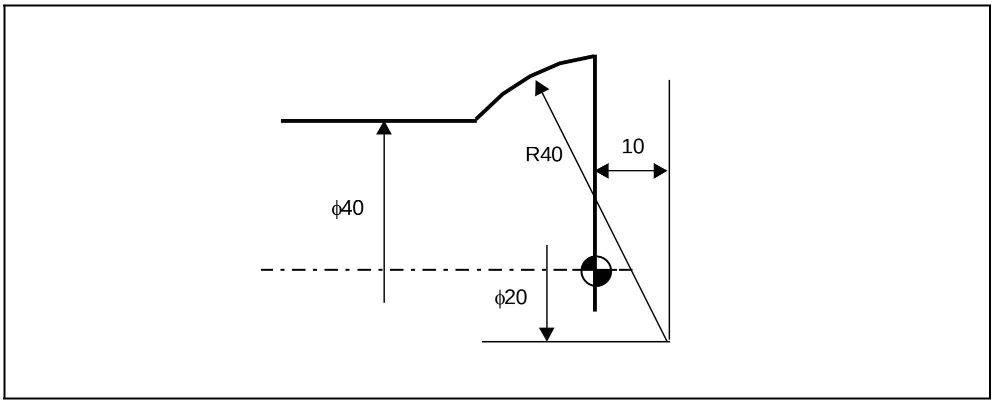


FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th	RGH
1			?	0.	40.	?		40.	
2	CTR	◆	-20.	-10.	◆	◆	◆		LEFT

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 **R/th** item for unknown **SPT** or at **RGH** for **FPT**.
   To find a crossing point with the automatic crossing-point calculation function, first view the X-direction from the origin and compare the positions of two crossing points.
   If the crossing point at right is the desired one, select **RGT** by pressing the **[RIGHT]** menu key. If the crossing point at left is the desired one, select **LEFT** by pressing the **[LEFT]** menu key. If the crossing point at the plus side is the desired one, this crossing point can likewise be specified by selecting **UP** with the **[UP]** menu key. If the crossing point at the minus side is the desired one, this crossing point can likewise be specified by selecting **DOWN** with the **[DOWN]** menu key.

   Example:

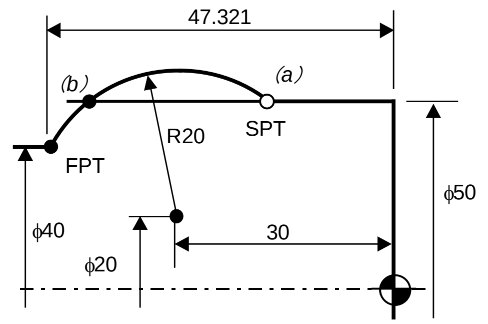


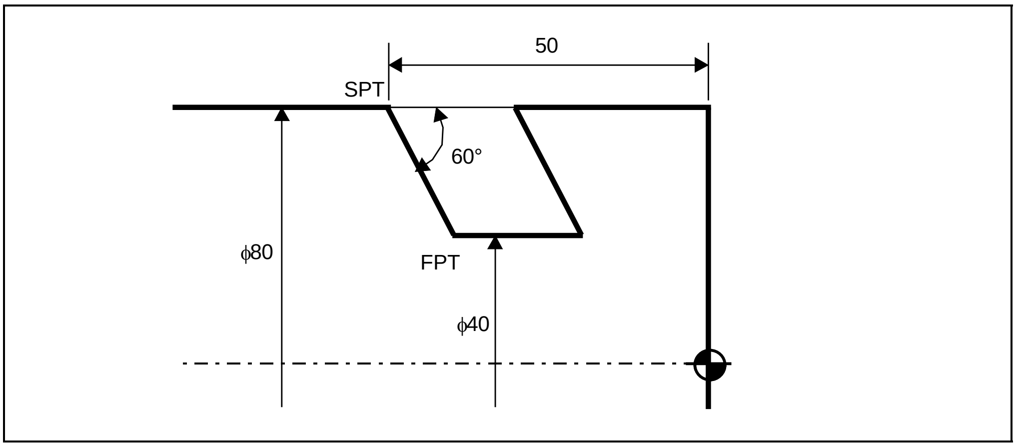
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th	RGH
1	▟▙		50.	?	40.	47.321		20.	
2	CTR	◆	20.	30.	◆	◆	◆	RGT	

To specify (*a*) for calculation of **SPT-Z**, press the **[RIGHT]** menu key at **R/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.



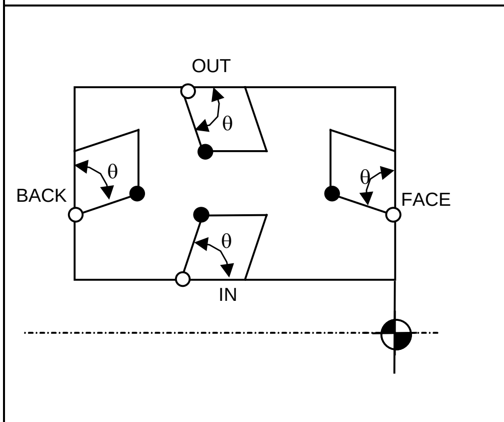
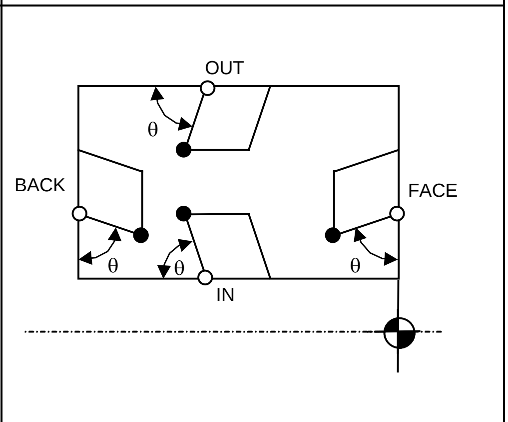
Set data as follows:

UNo.	UNIT	PART	POS-B	PAT.	No.	PITCH	WIDTH	FINISH
*	T.GROOVE	OUT	90.	0	1	0.	30.	◆

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	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.	

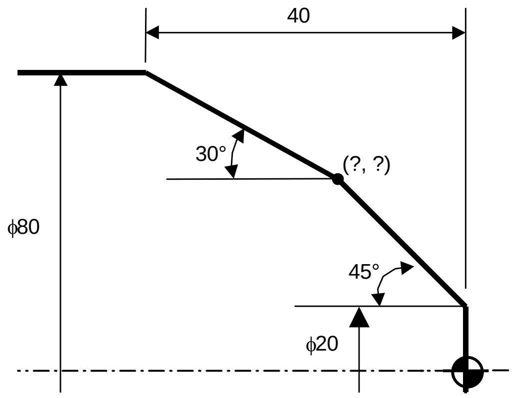
For the grooving pattern #0, the **ANG** data must be entered as a positive or negative value according to the direction of the respective tapering.

Enter positive values for **ANG** (θ).	Enter negative values for **ANG** (θ).
	

For the patterns #1 to #3, the sign of the **ANG** data is insignificant.

4. If intersecting point of two taperings is unknown.

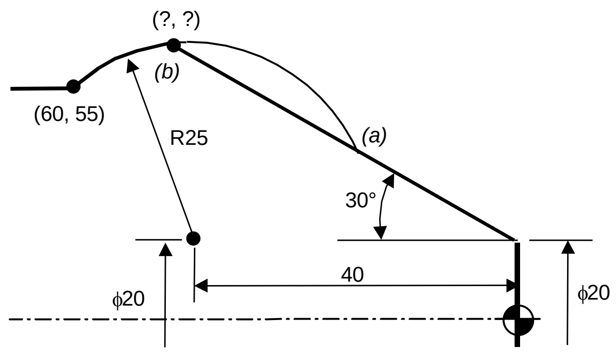
Example:



PTN	SPT-X	SPT-Z	FPT-X	FPT-Z	R/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.

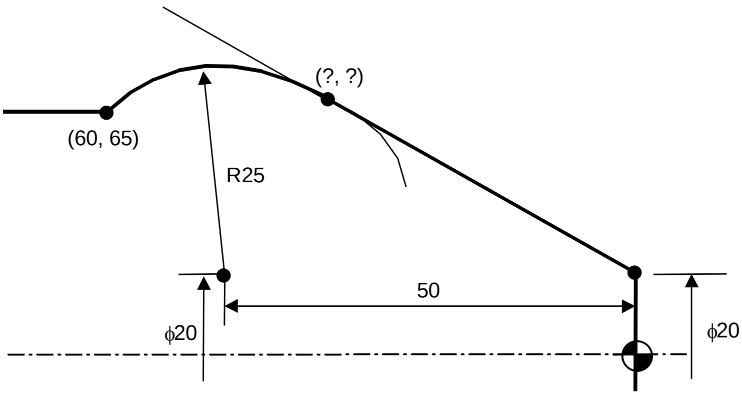
5. If intersecting point of tapering and arc is unknown.



PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	R/th	RGH
TPR		20.	0.	?	? *1		30. *2	
■		?	? *1	60.	55.		25. *3	
CTR	◆	20.	40. *4	◆	◆	◆	LEFT *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.

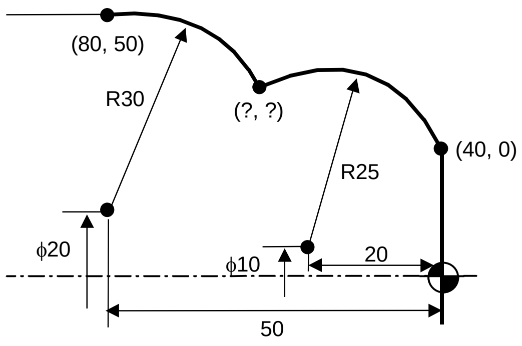
6. If osculation point of tapering and arc is unknown.



PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	R/th	RGH
TPR		20.	0.	? ○	? ○ *1			
■		? ○	? ○ *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.

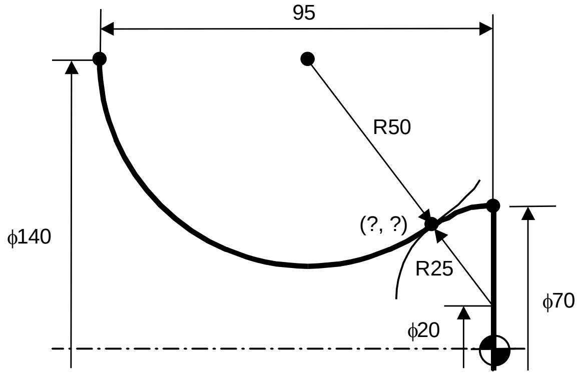
7. If intersecting point of two arcs is unknown.



PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	R/th	RGH
■		40.	0.	?	? *1		25. *2	
CTR	◆	10.	20. *3	◆	◆	◆		UP *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.



PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	R/th	RGH
		70.	0.	? ○	? ○ *1		25. *2	
CTR		20.	0. *3	◆	◆	◆		
	◆	? ○	? ○ *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.

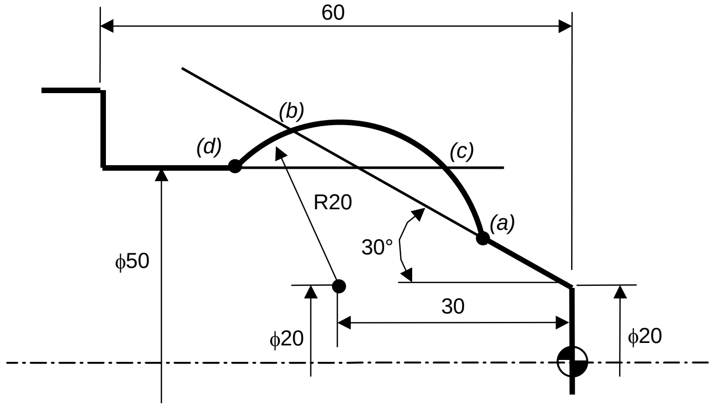
3. Enter the center coordinates of convex arc.

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.



PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	R/th	RGH
TPR		20.	0.	?	? *1		30. *2	
▀		?	? *1	50.	? *3		20. *4	
CTR	◆	20.	30. *5	◆	◆	◆	RGT *6	LEFT *7
LINE		◆	◆	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.
<u>In general, even an unknown coordinate of arc end point can be calculated with the intersecting point of tapering and arc remaining unknown.</u>

***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 **R/th**. and **RGH** to specify one of the two possible intersecting points of arc and tapering.
Press at **R/th** the **[RIGHT]** (or **[DOWN]**) menu key to specify (*a*) from among the two intersecting points of tapering and convex arc.
Press at **RGH** the **[LEFT]** menu key to specify (*d*) from among the two intersecting points of arc and straight line.

8-3 Automatic Cutting-Conditions Setting Function

For machining units except for manual program machining units, 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 speed 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	ATC MODE	RPM	LTUR DIA
0	CBN STL	100.	0.	40.	0.	0	2000	

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	90.	100.	0.	0.	0.

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL OUT	45.	A		0	3.	◆	◆	◆	◆	100	0.3			

After VFC

UNo.	MAT.	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM	LTUR DIA
0	CBN STL	100.	0.	40.	0.	0	2000	

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	90.	100.	0.	0.	0.

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL OUT	45.	A		0	3.	◆	◆	◆	◆	120	0.33			

Registered with "cermet" on the **TOOL DATA** display

After modification of programmed data using the VFC function during machining, the new values of cutting conditions (surface speed, etc.) 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  ATC MODE   RPM    LTUR DIA
 0   CBN STL  100.    0.     40.      0.         0      2000
UNo.   UNIT    PART    POS-B    CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR    OUT         90.     100.      0.       0.       0.
SNo.   TOOL       NOM.  No. #  PAT.  DEP-1  DEP-2/NUM.  DEP-3  FIN-X  FIN-Z C-SP FR M M M
 R 1 GENERAL OUT  45. A          0               ◆          ◆      ◆      ◆
```

Pressing the **[AUTO SET]** menu key.

```
UNo.   MAT.   OD-MAX  ID-MIN  LENGTH  WORK FACE  ATC MODE    RPM   LTUR DIA
 0   CBN STL  100.     0.      40.      0.         0       2000
UNo.   UNIT    PART    POS-B    CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR    OUT         90.     100.      0.       0.       0.
SNo.   TOOL       NOM.  No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3 FIN-X FIN-Z C-SP  FR  M 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  ATC MODE     RPM     LTUR DIA
 0   5052      30.     0.     40.      0.         0        2000
UNo.   UNIT    PART    POS-B    CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR    OUT         90.     100.      0.       0.       0.
SNo.   TOOL       NOM.  No. #  PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z C-SP  FR  M M M
 R 1 GENERAL OUT  45. A          0               ◆          ◆     ◆     ◆
```

Pressing the **[AUTO SET]** menu key.

```
UNo.   MAT.    OD-MAX  ID-MIN LENGTH WORK FACE ATC POS   RPM    LTUR DIA
 0   5052       30.      0.     40.     0.              2000
UNo.   UNIT    PART    POS-B    CPT-X    CPT-Z    FIN-X    FIN-Z
 1   BAR    OUT         90.     100.      0.       0.       0.
SNo.   TOOL       NOM.  No. #  PAT.  DEP-1  DEP-2/NUM.  DEP-3 FIN-X FIN-Z C-SP  FR  M 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 $\dfrac{3.4 \times LEAD}{MULTI} + 3.59$

Inch $\dfrac{3.4 \times LEAD \times 25.4}{MULTI} + 3.59$

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, $\boxed{\#2}$ $D_1 = (D - a)/\sqrt{N}$

8-4 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.

8-5 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. Data in **FW/RV R/L** is displayed blue in reversed status, when the tool index angle is set to "reverse".

- 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	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	90.	100.	0.	0.	0.

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL OUT						◆	◆	◆	◆	▌				

TOOL DATA

TNo.	TOOL		NOM. NOM-Ø	MAT.	STAT.
1	GENERAL	IN	2.		
2	GENERAL	IN	2.		
3	GENERAL	OUT	3.		
4	GENERAL	IN	4.		
5	GENERAL	OUT	5.		
6	GENERAL	IN	6.		
10	GENERAL	OUT	10.		
11	GENERAL	OUT	11.		

1/ 2

Example 2: Milling tool

UNo.	UNIT	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	START	END
3	LINE CTR	ZY	◆	45.	5.	10.	7	0.068	OPEN	OPEN

SNo.	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M	M
R 1	END MILL										▌				

TOOL DATA

TNo.	TOOL		NOM. NOM-Ø	MAT.	STAT.
1	END MILL	EDGE	20.	A	
7	END MILL	OUT	20.		
9	END MILL	EDGE	20.	B	

1/ 1

8-6 Tool File Window

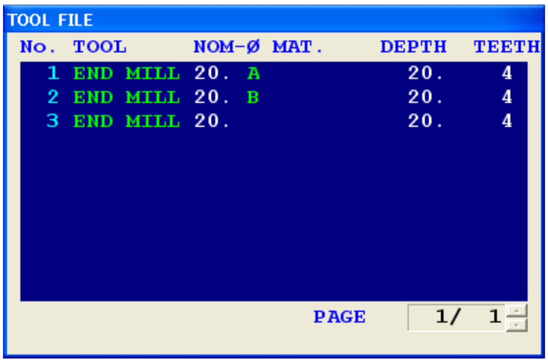
A tool file window can be displayed by pressing the **[TOOL. F WINDOW]** menu key while the cursor remains set at a **NOM-ϕ** item in the tool sequence data for the end mill, face mill, chamfering cutter, or ball end mill of the milling unit. Only the data for tools corresponding to the current tool 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	START	END
3	LINE CTR	ZY	◆	45.	5.	10.	7	0.068	◆	OPEN	OPEN

SNo.	TOOL	NOM-ϕ	No.	#	APRCH-1	APRCH-2	TYPE	AFD	DEP-A	DEP-R	C-SP	FR	M	M	M
R 1	END MILL														


TOOL FILE

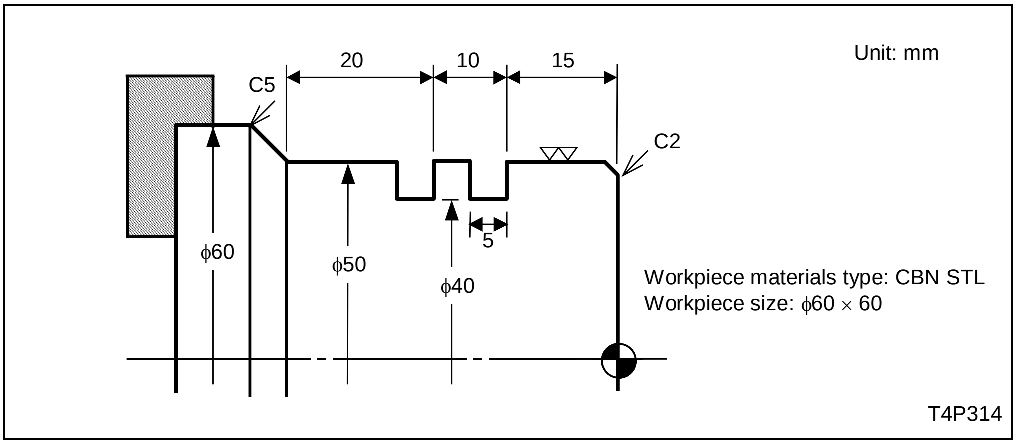
No.	TOOL	NOM-Ø	MAT.	DEPTH	TEETH
1	END MILL	20.	A	20.	4
2	END MILL	20.	B	20.	4
3	END MILL	20.		20.	4

PAGE 1/ 1

9 SAMPLE PROGRAMS

Example 1: 2-axes machining (BAR, T. GROOVE)

Machining drawing



Program

UNo.	MAT.	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM	LTUR DIA
0	CBN STL	60.	0.	60.	0.	0	2000	

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
1	BAR	OUT	90.	60.	0.	0.2	0.1

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL OUT	0.5	A		0	2.5	◆	◆	◆	◆	130	0.3			
F 2	GENERAL OUT	0.1	B		◆	◆	◆	◆	0.	0.	200	0.1			

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/th	RGH
1	LIN	C 2.	◆	◆	50.	50.	C 5.	◆	▼▼4

UNo.	UNIT	PART	POS-B	PAT.	No.	PITCH	WIDTH	FINISH
2	T.GROOVE	OUT	90.	0	2	10.	5.	◆

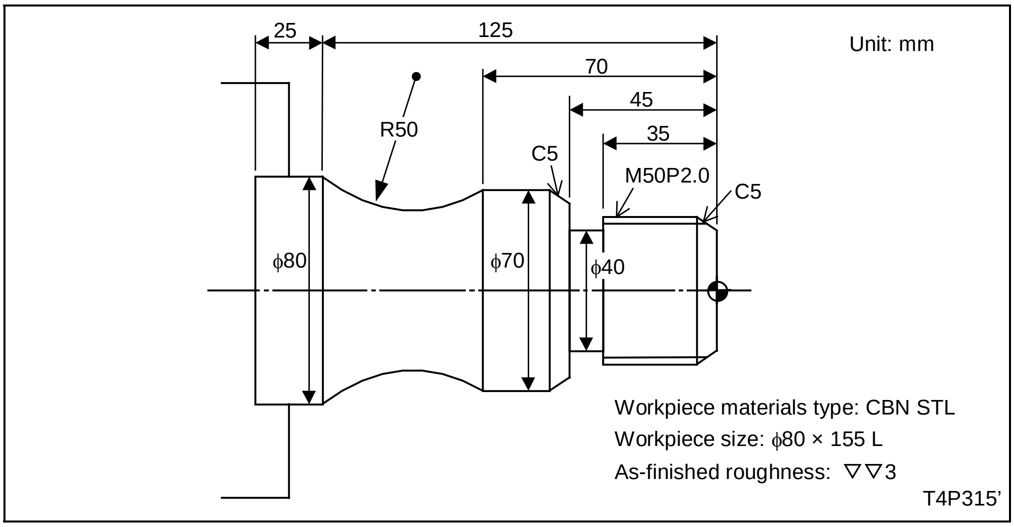
SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
F 1	GROOVE OUT	3.	A		◆	2.	◆	◆	◆	◆	120	0.08			

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANG	RGH
1		50.	20.	40.	20.			

UNo.	UNIT	CONTI.	REPEAT	SHIFT	NUMBER	ATC	RETURN	LOW RET.	WORK No.	EXECUTE
3	END	0	◆	◆	0	0	END	END		◆

Example 2: 2-axes machining (FACING, BAR, T. GROOVE, THREAD)

Machining drawing

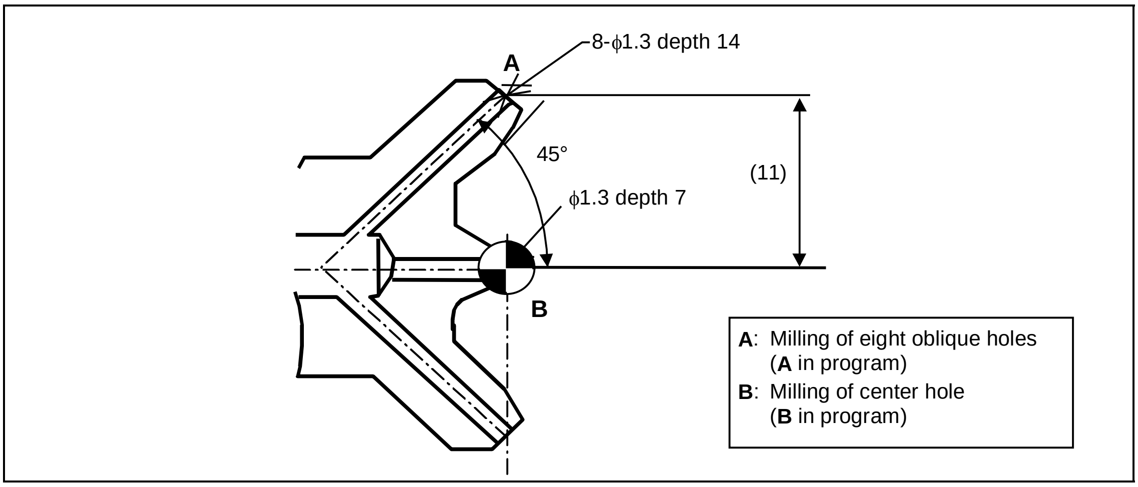


Program

```
UNo.  MAT.   OD-MAX ID-MIN LENGTH  WORK FACE   ATC MODE   RPM   LTUR DIA
 0  CBN STL  80.    0.    155.       5.         0      2000
UNo.  UNIT    PART     POS-B      FIN-Z
 1  FACING    FACE      0.        0.1
SNo.  TOOL        NOM.   No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M M M
 R 1 GENERAL EDGE  0.5  A ◤2      ◆     2.       ◆         ◆      ◆      ◆     120   0.3
 F 2 GENERAL EDGE  0.5  B         ◆     ◆        ◆         ◆      ◆      0.    160   0.1
 FIG              SPT-X    SPT-Z     FPT-X    FPT-Z                      RGH
 1                 80.      5.        0.       0.                        ▼▼3
UNo.  UNIT    PART     POS-B    CPT-X  CPT-Z   FIN-X   FIN-Z
 2    BAR     OUT       90.      80.    0.      0.2     0.1
SNo.  TOOL        NOM.   No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M M M
 R 1 GENERAL OUT  0.5  A          0    2.5       ◆         ◆      ◆      ◆     130   0.3
 F 2 GENERAL OUT  0.1  B          ◆     ◆        ◆         ◆      0.     0.    200   0.1
 FIG  PTN  S-CNR  SPT-X    SPT-Z     FPT-X    FPT-Z  F-CNR/$  R/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   POS-B   PAT.  No.  PITCH  WIDTH  FINISH
 3  T.GROOVE   OUT    90.     0    1    0.     10.     ◆
SNo.  TOOL        NOM.   No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M M M
 F 1  GROOVE OUT   3.  A          ◆     2.       ◆         ◆      ◆      ◆     120  0.08
 FIG     S-CNR     SPT-X    SPT-Z     FPT-X    FPT-Z  F-CNR   ANG     RGH
 1                 50.      45.       40.      45.
UNo.  UNIT    PART   POS-B  CHAMF LEAD   ANG    MULTI    HGT
 4  THREAD    OUT     90.     0    2.    55       1     1.299
SNo.  TOOL        NOM.   No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X  FIN-Z  C-SP   FR   M 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    CONTI.  REPEAT   SHIFT  NUMBER   ATC    RETURN  LOW RET.   WORK No.  EXECUTE
 5  END         0       ◆        ◆       0       0      END      END                 ◆
```

Example 3: Point milling

Machining drawing

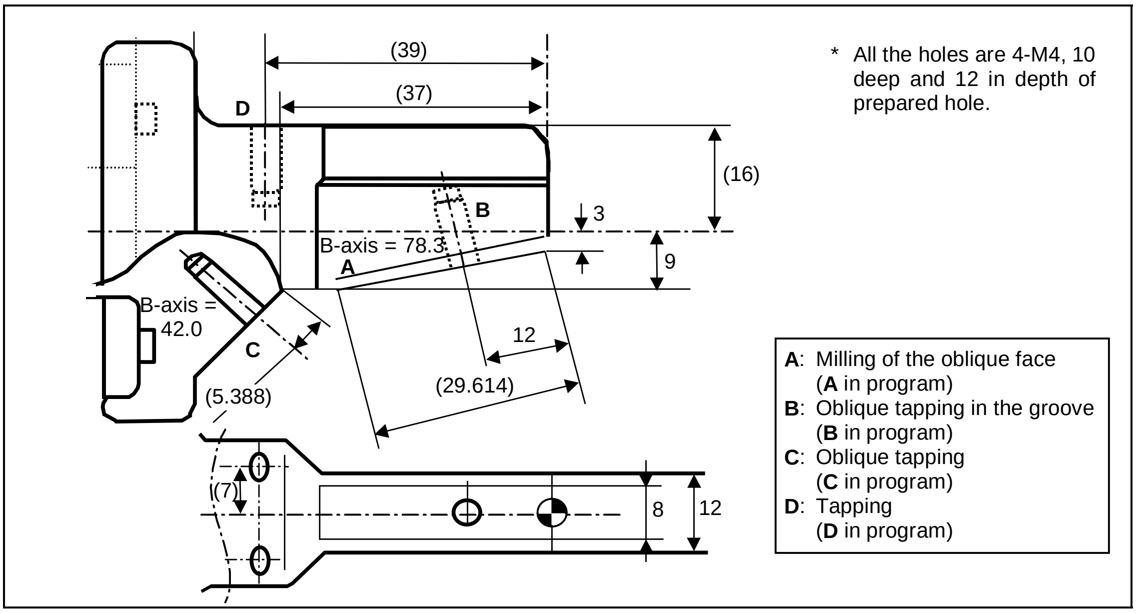


Program

```
   UNo.  MAT.  OD-MAX ID-MIN LENGTH  WORK FACE  ATC MODE  RPM  LTUR DIA
    0  FC      24.    0.     37.       0.         0    10000
   ---------------------------------------------------------------------------------------
   UNo.  UNIT    MODE   POS-B   POS-C    DIA    DEPTH   CHMF
    1  DRILLING  /C     45.       ◆      1.3     14.      0.
   SNo.  TOOL   NOM-ϕ No. # HOLE-ϕ  HOLE-DEP PRE-DIA PRE-DEP  RGH   DEPTH  C-SP   FR  M M M
A   1  CTR-DR    4.          1.3       ◆        ◆       ◆     90°   CTR-DR  10   0.1
    2  DRILL     1.3         1.3      14.       0.     100   PCK2 T  0.65    9   0.019
    FIG   PTN   SPT-R/x  SPT-C/y   SPT-Z    NUM.    ANG   Q  R
    1     ARC     11.       0.       0.       8      45   0  1
   ---------------------------------------------------------------------------------------
   UNo.  UNIT    MODE   POS-B   POS-C    DIA    DEPTH   CHMF
    2  DRILLING  XC      ◆        ◆      1.3     7.       0.
   SNo.   TOOL  NOM-ϕ No. # HOLE-ϕ HOLE-DEP PRE-DIA PRE-DEP  RGH   DEPTH  C-SP   FR  M M M
B   1  CTR-DR    4.          1.3       ◆        ◆       ◆     90°   CTR-DR  10   0.1
    2  DRILL     1.3         1.3       7.       0.     100   PCK2 T  0.65    9   0.019
    FIG   PTN   SPT-R/x  SPT-C/y   SPT-Z    NUM.    ANG   Q  R
    1     PT      0.        0.       0.       ◆       ◆    ◆  0
   ---------------------------------------------------------------------------------------
   UNo.  UNIT   CONTI.   REPEAT   SHIFT  NUMBER  ATC   RETURN   LOW RET.   WORK No.   EXECUTE
    3   END       0        ◆        ◆       0     0     END       END                   ◆
```

Example 4: Point/line milling

Machining drawing

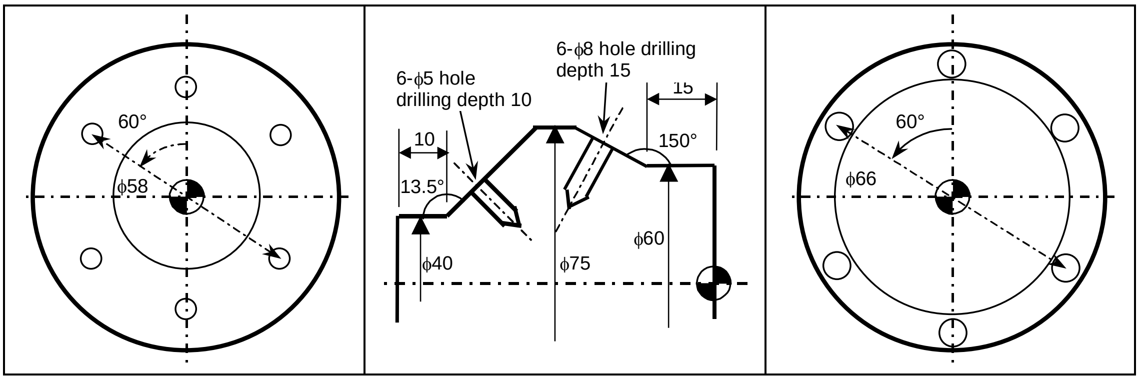


Program

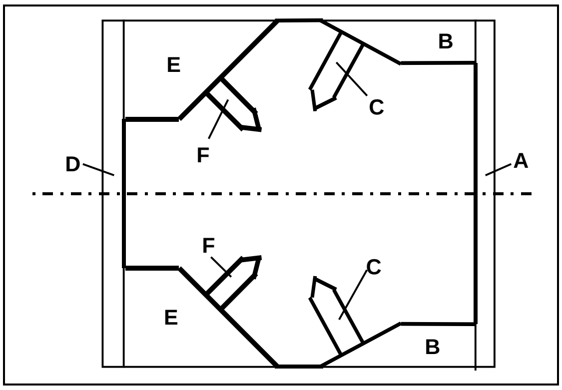
```
   UNo.  MAT.  OD-MAX ID-MIN LENGTH  WORK FACE ATC MODE   RPM   LTUR DIA
    0   FC      58.    0.    320.      0        0     10000
  ------------------------------------------------------------------------------------------
   UNo.  UNIT   MODE  POS-B  POS-C SRV-A  SRV-R   RGH   FIN-A          START   END
    6  LINE CTR  /Y    78.3   180.   2.     6.     1      0.           CLOSED  CLOSED
   SNo.  TOOL    NOM-φ  No. # APRCH-1  APRCH-2  TYPE  AFD   DEP-A   DEP-R  C-SP  FR  M M M
A   R 1 END MILL  20. A          ?        ?      ◆    G01    2.      ◆     100  0.2
   FIG    PTN   SHIFT-Z   SHIFT-R    X      Y     R/th   I    J    P   CNR  R-FEED   RGH
    1   LINE      0.       3.      0.      0.
    2   LINE      ◆        ◆     29.614   0.
  ------------------------------------------------------------------------------------------
   UNo.   UNIT    MODE  POS-B  POS-C   NOM.   MAJOR-φ  PITCH  TAP-DEP  CHMF
         TAPPING   /Y   78.3   180.   M 4.    4.      0.7     10.     1.
   SNo.  TOOL   NOM-φ No. # HOLE-φ HOLE-DEP PRE-DIA PRE-DEP  RGH  DEPTH C-SP  FR  M M M
B   1  CTR-DR    4.          10       ◆       ◆       ◆     90°  CTR-DR 10  0.1
    2  DRIL      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   PTN   SHIFT-Z  SHIFT-R   SPT-X  SPT-Y  CX/PX  CY/PY F   M   N   ANG   P  Q  R
    1   PT      0.       3.      12.     0.     ◆      ◆    ◆   ◆   ◆    ◆    0  ◆  0
  ------------------------------------------------------------------------------------------
   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 No. # HOLE-φ HOLE-DEP PRE-DIA PRE-DEP  RGH  DEPTH C-SP  FR  M M M
C   1  CTR-DR    4.          10       ◆       ◆       ◆     90°  CTR-DR 10  0.1
    2  DRIL      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   PTN   SHIFT-Z  SHIFT-R   SPT-X  SPT-Y  CX/PX  CY/PY F   M   N   ANG   P  Q  R
    1   LIN     37.      9.     5.388   -7.     14      ◆    ◆   2   ◆   90.   0  0  1
  ------------------------------------------------------------------------------------------
   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 No. # HOLE-φ HOLE-DEP PRE-DIA PRE-DEP  RGH  DEPTH C-SP  FR  M M M
D   1  CTR-DR    4.          10       ◆       ◆       ◆     90°  CTR-DR 10  0.1
    2  DRIL      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   PTN   SPT-R/x  SPT-C/y  SPT-Z  SPT-Y   NUM.   ANG   Q  R
    1   PT      16.       0.      39.    0.     ◆      ◆     ◆  0
  ------------------------------------------------------------------------------------------
   UNo.  UNIT  CONTI.  REPEAT  SHIFT  NUMBER  ATC   RETURN  LOW RET.  WORK No.  EXECUTE
    3   END     0        ◆       ◆      0      0     END      END                ◆
```

Example 5: Workpiece transfer

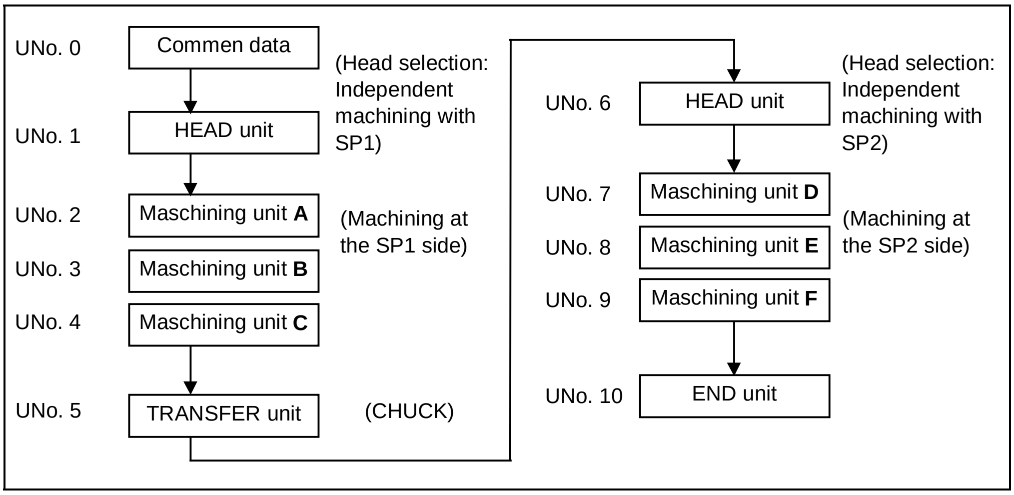
Machining drawing



Machining outline



Program outline

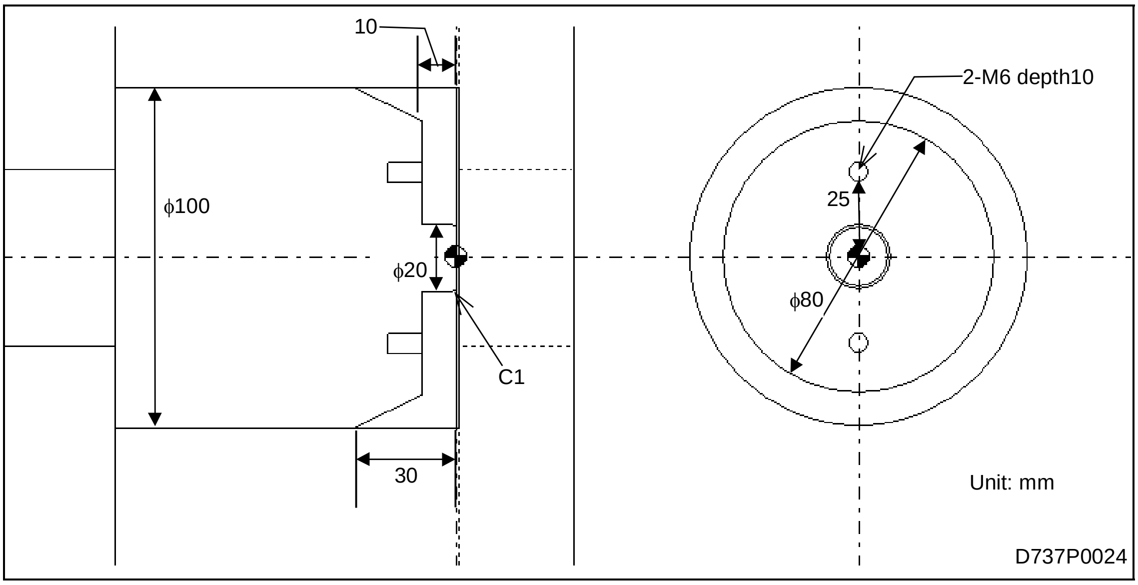


Program

```
UNo.  MAT.  OD-MAX POS-B  LENGTH  WORK FACE  ATC MODE  RPM   LTUR DIA
 0  CBN STL  75.    0.    62.       1.         0      1500
UNo.  UNIT    TYPE    HEAD    SPDL
 1  HEAD      SIN      1       ◆
UNo.  UNIT    PART   POS-B   FIN-Z
 2  FACING   FACE     0.      0.1
SNo.  TOOL        NOM.  No. # PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X FIN-Z C-SP   FR  M M M
 R 1 GENERAL EDGE  0.5  A ◤2      ◆     2.       ◆       ◆      ◆     ◆    120  0.25
 F 2 GENERAL EDGE  0.5  B         ◆     ◆        ◆       ◆      ◆     0.   160  0.2
 FIG                     SPT-X   SPT-Z   FPT-X   FPT-Z                 RGH
  1                       75.      1.      0.      0.                 ▼▼4
UNo.  UNIT    PART   POS-B   CPT-X   CPT-Z   FIN-X    FIN-Z
 3  BAR      OUT     90.     75.      0.      0.2      0.1
SNo.  TOOL       NOM.  No. #  PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X   FIN-X C-SP   FR   M M M
 R 1  GENERAL OUT 0.5 A         0    2.5       ◆        ◆       ◆      ◆    150   0.25
 F 2  GENERAL OUT 0.1 B         ◆     ◆        ◆        ◆       0.     0.   250   0.1
 FIG   PTN    S-CNR   SPT-X   SPT-Z   FPT-X   FPT-Z  F-CNR/$  R/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 No. # HOLE-φ HOLE-DEP PRE-DIA PRE-DEP  RGH   DEPTH  C-SP   FR   M M M
  1   CTR-DR   20.          10.      ◆        ◆       ◆      90°  CTR-DR  25   0.09
  2   DRILL     8.           8.     15.       0.     100    DRIL  T  4.   25   0.129
 FIG   PTN    SPT-R/x   SPT-C/y   SPT-Z     NUM.     ANG   Q  R
  1    ARC      33.        0.     20.196     6       60.   0  0
UNo.   UNIT     PAT.  HEAD  SPDL  PUSH CHUCK   W1   W2  Z-OFFSET C1  C2 C-OFFSET LTUR ESC TNo.
 5   TRANSFER  CHUCK 1→2   4     1      ◆    -1020  0     760    0. 0.    0.      -430.
UNo.  UNIT    TYPE    HEAD    SPDL
 6  HEAD      SIN      2       ◆
UNo.  UNIT    PART   POS-B   FIN-Z
 7  FACING   BACK    180.     0.
SNo.  TOOL        NOM.  No. # PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X   FIN-Z  C-SP   FR  M M M
 R 1  GENERAL EDGE 0.5 C        ◆     1.       ◆        ◆       ◆       ◆     150   0.3
 FIG                     SPT-X   SPT-Z   FPT-X   FPT-Z                 RGH
  1                       75.    -61.      0.     -60.                ▼▼4
UNo.  UNIT    PART   POS-B   CPT-X   CPT-Z   FIN-X    FIN-Z
 8  BAR      OUT     90.     75.     60.      0.2      0.1
SNo.  TOOL        NOM.  No. # PAT.  DEP-1 DEP-2/NUM. DEP-3  FIN-X   FIN-Z  C-SP   FR  M M M
 R 1  GENERAL OUT 0.5  C        0    1.5       ◆        ◆       ◆       ◆     150   0.3
 F 2  GENERAL OUT 0.1  D        ◆     ◆        ◆        ◆       0.      0.    196   0.
 FIG   PTN    S-CNR   SPT-X   SPT-Z   FPT-X   FPT-Z  F-CNR/$  R/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 No. # HOLE-φ HOLE-DEP PRE-DIA PRE-DEP  RGH   DEPTH  C-SP   FR   M M M
  1   CTR-DR   20.          5.6      ◆        ◆       ◆      90°  CTR-DR  25   0.09
  2   DRILL     5.           5.     10.       0.     100    DRIL  T 2.5   25   0.088
 FIG   PTN    SPT-R/x   SPT-C/y   SPT-Z     NUM.     ANG   Q  R
  1   ARC       29.        0.      41.       6       60.   0  0
UNo.  UNIT   CONTI.   REPEAT   SHIFT  NUMBER  ATC   RETURN   LOW RET.   WORK No.   EXECUTE
 10  END        0        ◆        ◆       0      0     END       END                  ◆
```

Example 6: Machining with upper and lower turrets

Machining drawing



Program

UNo.	MAT.	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM	LTUR DIA
0	CST IRN	100.	0.	101.	1	0	1500	

UNo.	UNIT	PART	POS-B	FIN-Z
1	FACING	FACE	0.	0.1

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL EDGE	10.	◤		◆	1.5	◆	◆	◆	◆	150	0.15			
F 2	GENERAL EDGE	10.	B◤2		◆	◆	◆	◆	◆	0.	200	0.1			

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	RGH
1	100.	1.	0.	0.	▼▼3

UNo.	UNIT	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
2	BAR	OUT	90.	100.	0.	0.2	0.1

SNo.	TOOL	NOM.	No.	#	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	M
R 1	GENERAL OUT	5.	◤		1	1.5	◆	◆	◆	◆	150	0.17			
F 2	GENERAL OUT	10.	B◤2		◆	◆	◆	◆	0.	0.	200	0.1			

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/$	R/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	◆	◆	M6.	6.	1.	10.	1.5

SNo.	TOOL	NOM-ϕ	No.	#	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	M
1	CTR-DR	12.	◤		8.134	◆	◆	◆	90°	CTR-DR	59	0.09			
2	DRIL	5.1	D◤2		5.1	12.	0.	100	DRIL	T 2.55	25	0.08			
3	TAP	M6.	◤		6.	10.	TAP	◆	FIX	P1.	39	1.			

FIG	PTN	SPT-R/x	SPT-C/y	SPT-Z	NUM.	ANG	Q	R
1	ARC	25.	0.	10.	2	180.	0	0

UNo.	UNIT	CONTI.	REPEAT	SHIFT	NUMBER	ATC	RETURN	LOW RET.	WORK No.	EXECUTE
4	END	0	◆	◆	0	0	END	END		◆

- NOTE -

10 THREE-DIGIT G-FORMAT

10-1 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.

10-2 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.

10-3 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	–	2	5	1	)	EOB	*Description in 3-digit G-format*	EOB	EOR %

Identifier

(With program name)

EOB	*Program No.* O99999999	(	M	G	3	–	2	5	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 “(MG3-251)” succeeding the program number identifies a MAZATROL program described in the three-digit G-format of the MAZATROL MATRIX.
- 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 48 for naming a program stored in the NC memory.
  An excess in characters will be given away.

2. Special unit

A. Common unit G300

- Unit data

UNo.	Program type	MAT	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATCMODE	RPM	LTUR DIA
U	P	( )	X	I	Z	C	A	S	J
	1: MTPro								

B. End unit G301

- Unit data

UNo.	CONTI.	REPEAT	SHIFT	NUMBER	ATC	RETURN	LOW RET.	WORK No.	EXECUTE
U	B	K	I	C	A	D	L	( )	E
	0: No			0: No	0: None	0: None			0: No
	1: Yes			1: Yes	1: ATC → Axis feed	1: END			1: Yes
					2: Axis feed → ATC	2: FEXED PT			
						3: ARB PT			
						4: ESC 1			
						5: ESC 2			

- Sequence data (Setting retraction position)

SNo.	DATA-1	···	DATA-9
N	?	···	?

→ For example, set as follows: “X12345.6789”.
For axis name of 2 characters, the second characters 1 to 3 are converted to A to C respectively.
Ex. To set “123.456” for the X2-axis: “XB123.456”

C. Subprogram unit G303

- Unit data

UNo.	Measuring flag	WORK No.	$	NUM.	Turret	#
U	F	( )	V	L	K	Q
	0: Including measurement		0 to 9: Unit skip (0 to 9)			1 to 99: Simultaneous machining No.
	1: Not including measurement		–1 to –4: Multi-workpiece machining(A to D)			101: Lower turret retraction position 1
						102: Lower turret retraction position 2
						111: Lower turret retraction position 1
						112: Lower turret retraction position 2

- Sequence data

SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
N	?	?	?	?	?	?

→ Address and data are set as specified, e.g. “X123.456.”
(If macro variable has been specified, “X#100” is used.)

D. Manual program machining unit G382

- Unit data

UNo.	TOOL	Tool shape	NOM-ϕ	Suffix	Turret	No.	#
U	T	&T	D	S	K	P	Q

T:	Tool name
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

&T:	Tool shape
1	OUT
2	IN
3	EDGE
4	IN
5	EDGE
17	001
18	002
19	003
20	004
21	005
22	006
23	007
24	008
25	009

S:	Suffix
1	A
8	H
9	J
13	N
14	P
24	Z
-1	A
-8	H
-9	J
-13	N
-14	P
-24	Z

K:	Turret
0	Turret 1
1	Turret 2

Q:	Turret 2
1 to 99	Simultan. machining No.
101	Balanced cutting B2
110	Turret-2 retraction pos. 1
111	Turret-2 retraction pos. 2

P:	Priority No.
0	None
1 to 99	Priority machining
–1 to –99	Subsequent machining

- Sequence data

SNo.	G1	G2	DATA-1	.....	DATA-6	S	M/B
N	G	&G	?	.....	?	S	M/B

→ Address and data are set as specified, e.g. "X123.456."

E. M-code unit G302

- Unit data

UNo.	No. (Priority No.)	Turret	M1	.....	M12
U	P	K	MA	.....	ML
	0: None	0: Upper turret			
	1 to 99: Priority machining	1: Lower turret			
	–1 to –99: Subsequent machining				

F. Coordinates measuring unit G304

- Unit data

UNo.	TOOL	Tool shape	NOM-ϕ	Suffix	Turret	Priority No.	U. SKIP	$ (External unit skip)	# (Lower turret setting)
U	T	&T	D	S	R	P	K	V	W

G. Process end unit G385

- Unit data

UNo.
U

H. Materials shape unit G307

- Unit data

UNo.	Shape
U	E
	1: OUT
	3: IN

- Shape sequence data

SNo.	PTN	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
N	A	X	Z	&X	&Z	I
	1: LIN					
	2: TPR					
	3:					
	4:					

I. Workpiece measuring unit G308

- Unit data

UNo.	COMPENSATE	OFS-TOOL	OFS-TOOL Shape	OFS-TOOL Nom. dia./size	OFS-TOOL Suffix	OFS-TOOL Turret	COMP.DATA
U	H	T	&T	D	I	C	J
	0: Yes						0: Diameter
	1: No						1: Length

SNS-TOOL	SNS-TOOL Nom. dia	SNS-TOOL Suffix	SNS-TOOL Turret	Priority No.	# (Lower turret retraction)	INTERVAL	OUTPUT
K	R	S	E	P	W	L	Q
							0: No output
							1: HD
							2: Printer

- Sequence data (Internal measurement)

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z
N	A	X	Y	Z	&X	&Y	&Z
	1: OUTER X						0: Diameter
	2: OUTER Y						1: Length
	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						

T LIM+	T LIM–	BASE	Approach	Head angle
V	W	Q	L	R
		0: SPT	0: Z direction	
		1: FPT	1: X direction	

- Sequence data (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				

J. Workpiece transfer unit G309

- Unit data

UNo.	PAT.	HEAD	SPDL	PUSH	CHUCK
U	P	H	S	J	K
	1: CHUCK	<Case of CHUCK>	0: Spindle stop	0: With pushing	0: Chuck open
	2: BAR	1: 1→2	1: Spindle forward	1: Without pushing	1: Chuck close
	3: MOVE	2: 2→1	2: Spindle reverse		
		<Case of BAR>	3: Spindle orient		
		1: HEAD1	4: C-axis positioning		
		2: HEAD2	5: Spindle mov. keep		

W1/Z1	W2/Z2	Z-OFFSET	C1	C2	C-OFFSET	LTUR ESCZ X	LTUR ESC Z	TNo.
ZA	ZB	L	CA	CB	P	X	Z	R

K. Head selection unit G310

- Unit data

UNo.	TYPE	HEAD	SPDL	Turret
U	P	H	L	K
	1: SIN	1: HEAD1	0: THE OTHER SPDL SYNCRO	0: TR1
	2: SYNC	2: HEAD2	1: THE OTHER SPDL STOP	1: TR2
	3: CROSS			

L. Tool measuring unit G311

- Unit data

UNo.	COMPENSATE	OFS-TOOL	Tool shape	Nom.dia./size	Suffix	Turret	Priority No.
U	H	T	&T	D	S	K	P
	0: Yes						
	1: No						

Lower turret retraction	INTERVAL	OUTPUT
W	L	Q

- Sequence data

SNo.	PTN	T-LIM-X	T-LIM-Z	TOOL EYE	DIR
N	A	V	W	Q	R
	1: Laser			0: Retract	←:0
	2: TOOL EYE #1			1: Not retract	↓:90
	3: TOOL EYE #2				→:180
	4: TOOL EYE #3				
	5: TOOL EYE #4				

M. Simultaneous machining unit G312

- Unit data

UNo.	Priority No.	SIMUL.No.	RPM
U	P	L	S

N. 2-workpiece machining unit G313

- Unit data

UNo.	PAT.	SP1/SP2
U	A	B
	0: START	0: HI/LOW
	1: END	1: LOW/HI

3. Turning units

A. Bar-materials machining unit G320

- Unit data

UNo.	PART	POS-B	CPT-X	CPT-Z	FIN-X	FIN-Z
U	E	B	X	Z	&X	&Z
	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)					

B. Copy-machining unit G321

- Unit data

UNo.	PART	POS-B	CPT-X	CPT-Z	SRV-X	SRV-X	FIN-X	FIN-Z
U	E	B	X	Z	I	J	&X	&Z

Refer to "Bar-materials machining unit".

C. Corner-machining unit G322

- Unit data

UNo.	PART	POS-B	FIN-X	FIN-Z
U	E	B	&X	&Z

Refer to "Bar-materials machining unit".

D. Facing unit G323

- Unit data

UNo.	PART	POS-B	FIN-Z
U	E	B	&Z

Refer to "Bar-materials machining unit".

E. Threading unit G324

- Unit data

UNo.	PART	POS-B	CHAMF	LEAD	ANG	MULTI	HGT
U	E	B	C	K	D	R	H
			1: 45°				
			2: 60°				

PART (E): Refer to “Bar-materials machining unit”.

F. Grooving unit G325

- Unit data

UNo.	PART	POS-B	PAT.	No.	PITCH	WIDTH	FINISH/ Overshoot
U	E	B	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-corner cut-off)				

PART (E): Refer to “Bar-materials machining unit”.

G. Turning drilling unit G326

- Unit data

UNo.	PART	POS-B	DIA
U	E	B	D

PART (E): Refer to “Bar-materials machining unit”.

H. Turning tapping unit G327

- Unit data

UNo.	PART	POS-B	NOM.	PITCH
U	E	B	*	F

NOM. (*): Refer to “Tapping unit”.

PART (E): Refer to “Bar-materials machining unit”.

I. Bar-materials machining/Copy-machining/Corner-machining/Facing unit

- Sequence data (turning tool)

SNo.	TOOL	Tool shape	Nom. dia./ size	Suffix	Turret	Priority No.	#	PAT.
N	T	&T	D	S	K	P	L	Q

L
1 to 99: Simultan. machining No.
100: Balanced cut (cut × 2)
101: Balanced cut (feed × 2)
110: Lower turret retraction pos. 1
111: Lower turret retraction pos. 2

DEP-1	DEP-2/ NUM.	DEP-3	No. of cut passes	FIN-X	FIN-Z	C-SP	FR	M	M	M	R/F
RA	RB	RC	E	X	Y	I	J	MA	MB	MC	F

F
R: 0
F: 1

PAT.

Q	BAR and CORNER units	THREAD unit	T.DRILL unit
0	Normal cycle	#0 STANDARD (Standard pattern)	Drilling cycle (blind hole)
1	High-speed rough-machining cycle	#1 CONST. DEPTH (Constant-threading pattern)	Deep-hole drilling cycle (blind hole)
2	Inside diameter enlarging cycle	#2 CONST. AREA (Area-constant pattern)	High-speed deep-hole drilling cycle (blind hole)
3	Normal + Chip cutting cycle	#0 STANDARD (zigzag threading)	Reaming cycle (blind hole)
4	High-speed + Chip cutting cycle	#1 CONST. DEPTH (zigzag threading)	Ultra-deep-hole drilling cycle (blind hole)
5		#2 CONST. AREA (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)

J. Bar-materials machining/Copy-machining unit

- Sequence data (Shape)

FIG	PTN	S-CNR	SPT-X	SPT-Z	SPT INTER PT	F-CNR/$	FPT-X	FPT-Z	FIN INTER PT
N	A	C/R	X	Z	K	&C/&R/Q	&X	&Z	L
	1: LIN				1: UP				1: UP
	2: TPR				2: DOWN				2: DOWN
	3:				3: LEFT				3: LEFT
	4:				4: RGT				4: RGT
	5: CTR								

R/th	FGH/ Feedrate set flag	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?

K. Corner-machining unit

- Sequence data (Shape)

FIG	SPT-X	SPT-Z	F-CNR/$	FPT-X	FPT-Z	RGH code	Feedrate
N	X	Z	&C/&R/Q	&X	&Z	E	F

L. Facing unit

- Sequence data (Shape)

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	RGH code	Feedrate
N	X	Z	&X	&Z	E	F

M. Threading unit

- Sequence data (Shape)

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z
N	X	Z	&X	&Z

N. Grooving unit

- Sequence data (Shape)

FIG	S-CNR	SPT-X	SPT-Z	F-CNR	FPT-X	FPT-Z	ANGLE	RGH code	Feedrate	INTER PT flag
N	C/R	X	Z	&C/&R	&X	&Z	J	E	F	H

O. Turning drilling/Turning tapping unit

- Sequence data (Shape)

FIG	FPT-X	FPT-Z
N	&X	&Z

4. Milling units

A. Point machining unit

- 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

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- Tapping unit G354

UNo.	MODE	POS-B	POS-C	NOM.	MAJOR-ϕ	PITCH	TAP-DEP	CHMF
U	Q	Y	W	*	E	P	H	C

Refer to “Drilling unit”.

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 dia. D:

Nominal dia. 2 V:

Example:

M10.	A1D10.
UNn 1-2	A2D1V2
UN 1H-2	A3D1V2B1
PT 2Q	A4D2B2

- Through hole boring unit G358

UNo.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	WAL
U	Q	Y	W	D	H	C	J

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- 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

Refer to “Drilling unit”.

- Circular milling unit G356

UNo.	TORNA.	MODE	POS-B	POS-C	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2
U	K	Q	Y	W	D	H	C	I	&D	&C	E	F

TORNA. (K):

0: CIRCUL
1: TORNADE
2: HIGH AC.

Refer to “Drilling unit”.

- Counterbore-tapping unit G357

UNo.	MODE	POS-B	POS-C	NOM.	MAJOR-ϕ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM	CHP
U	Q	Y	W	*	E	P	H	C	&D	&H	&C	I	K

Refer to “Tapping unit”.

Refer to “Drilling unit”.

B. Line machining unit

- Central linear machining unit G362

UNo.	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	Start/End pt. attribute
U	Q	Y	W	Z	R	F	&Z	A
								bit 0: ON = Start pt. CLOSED
								bit 1: ON = End pt. CLOSED

→ Refer to "Drilling unit".

- Right-hand/Left-hand linear machining unit G363/G364

UNo.	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R	Start/End pt. attribute
U	Q	Y	W	Z	R	F	&Z	&R	A

→ Refer to "Drilling unit".

- Outside/Inside linear machining unit G365/G366

UNo.	MODE	POS-B	POS-C	SRV-A	SRV-R	RGH	FIN-A	FIN-R	Start/End pt. attribute
U	Q	Y	W	Z	R	F	&Z	&R	A

→ Refer to "Drilling unit".

- Right-hand/Left-hand chamfering unit G367/G368

UNo.	MODE	POS-B	POS-C	INTER-Z	INTER-R	CHMF	Start/End pt. attribute
U	Q	Y	W	I	J	C	A

→ Refer to "Drilling unit".

- Outside/Inside chamfering unit G369/G370

UNo.	MODE	POS-B	POS-C	INTER-Z	INTER-R	CHMF	R-chamfering flag	Start/End pt. attribute
U	Q	Y	W	I	J	C	R	A
							0: Chamfering	
							1: Rounding	

→ Refer to "Drilling unit".

C. Face machining unit

- Face milling unit G371

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	FIN-A
U	Q	Y	W	Z	I	&Z

→ Refer to "Drilling unit".

- End milling-top unit G372

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	FIN-A
U	Q	Y	W	Z	I	&Z

→ Refer to "Drilling unit".

- End milling-step unit G373

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R
U	Q	Y	W	Z	I	J	&Z	&R

→ Refer to “Drilling unit”.

- Pocket milling unit G374

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R	INTER-R	CHMF	R-chamfering flag
U	Q	Y	W	Z	I	J	&Z	&R	K	C	R
											0: Chamfering
											1: Rounding

→ Refer to “Drilling unit”.

- Pocket milling-mountain unit G375

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R
U	Q	Y	W	Z	I	J	&Z	&R

→ Refer to “Drilling unit”.

- Pocket milling-valley unit G376

UNo.	MODE	POS-B	POS-C	SRV-A	BTM	WAL	FIN-A	FIN-R
U	Q	Y	W	Z	I	J	&Z	&R

→ Refer to “Drilling unit”.

- End milling-slot unit G377

UNo.	MODE	POS-B	POS-C	SRV-A	SLOT-WID	BTM	WAL	FIN-A	FIN-R
U	Q	Y	W	Z	D	I	J	&Z	&R

→ Refer to “Drilling unit”.

D. Tool sequence

- Point machining tool sequence

SNo.	TOOL	NOM-ϕ	Suffix	Turret	Priority No.	Lower turret retraction	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP
N	T	D	S	K	P	L	E	H	&E	&H

RGH	DEPTH	C-SP	FR	M	M	M
F	Z	I	J	MA	MB	MC

T:	Tool name
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

&T:	Tool shape
1	OUT
2	IN
3	EDGE
4	IN (BACK)
5	EDGE (BACK)
17	001
18	002
19	003
20	004
21	005
22	006
23	007
24	008
25	009

K:	Turret
0	TR1
1	TR2

S:	Suffix
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:	Priority No.
0	None
1 to 99	Priority machining
–1 to –99	Subsequent machining

L:	Lower turret retraction pos.
110:	Position 1
111:	Position 2

- Line/Face machining tool sequence

SNo.	TOOL	NOM-ϕ	Suffix	Turret	Priority No.	#	APRCH-1	APRCH-2	TYPE	ZFD	TYPE	CPT-Z	DEP-R
N	T	D	S	K	P	L	X	Y	Q	F	H	Z	R

C-SP	FR	M	M	M	R/F
I	J	MA	MB	MC	B
					0: R
					1: F

Refer to “Point machining tool sequence”.

X, Y:	APRCH-1, 2
?	? for auto-set
&123.456	Auto-set
123.456	Normal input

Q:	TYPE
1	1BI
2	2BI
3	1UN
4	2UN
5	1BI-S
6	2BI-S
7	CW
8	CCW

F:	ZFD
–1	G01
0	G00
0.1 to 9.9	

H:	Approach type
1	Bidirectional X
2	Bidirectional Y
3	Unidirectional X
4	Unidirectional Y
5	Bidirectional, arc X
6	Bidirectiona, arc Y
16	CW
17	CCW

E. Shape sequence

- Linear/face (arbitrary) machining shape sequence ZC

FIG	PTN	Z	C	SPT-R	R/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	R feedrate
F	S	B (1: arbitrary)	Q	H	&F

- Linear/face (arbitrary) machining shape sequence XC

FIG	PTN	Control flag	R/x	C/y	SHIFT-Z	R/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	R feedrate
F	S	B	Q	H	&F

- Linear/face (arbitrary) machining shape sequence XY

FIG	PTN	Control flag	R/x	C/y	SHIFT-Z	R/th	I	J	P	CNR
N	A	P	I/X	J/Y	Z	&R/&C	V/&X	W/&Y	L	R/C/K

RGH/ Feedrate set flag	RGH code/ Feedrate	SPT	Type of shape	? mark	AUTO SET	R feedrate	Wall
E	F	S	B	Q	H	&F	&W

- Linear/face (arbitrary) machining shape sequence ZY

FIG	PTN	Z	Y	SHIFT-Z	R/th	I	J	P	CNR
N	A	Z	Y	I	&R/&C	V	W	L	R/C/K

RGH/ Feedrate set flag	RGH code/ Feedrate	SPT	Type of shape	? mark	AUTO SET	R feedrate	Wall
E	F	S	B	Q	H	&F	&W

- Linear/face (arbitrary) machining shape sequence /Y

FIG	PTN	SHIFT-Z	SHIFT-R	X	Y	R/th	I	J	P	CNR
N	A	Z	I	X	Y	&R/&C	V	W	L	R/C/K

RGH/ Feedrate set flag	RGH code/ Feedrate	SPT	Type of shape	? mark	AUTO SET	R feedrate	Wall
E	F	S	B	Q	H	&F	&W

Feedrate, R-feedrate

[RGH code input]

0: Surface roughness 0
1: Surface roughness 1
2: Surface roughness 2
3: Surface roughness 3
4: Surface roughness 4
5: Surface roughness 5
6: Surface roughness 6
7: Surface roughness 7
8: Surface roughness 8
9: Surface roughness 9

[Feedrate input]

Unit: mm/10000 inch/100000

Input range: 0 to 99.999

[Rapid feed specified]

–1

Data format for the other shape sequence is same as for conventional type.

5. TPC data

A. Parameter

	A	B	C	D	E	F	H	I	J	K	L	M	P	Q	R	S	Y	Z	&C	&D	&E	&X	&Y	&Z
DRILLING	D1	D3	D16	D17									D41	D42	D91		D45	D46	TC37	TC38	TC39	TC40		
RGH CBOR	D1	D3	D16	D17		D19	D23						D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
RGH BCB	D1	D3	D16	D17	D5	D19		D40					D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
REAMING	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
TAPPING	D1	D3	D16	D17	D22			D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
BK-CBORE	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D33	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
CIRC. MIL	D1		D16	D17		D19	D23						D41	D42	D91	D92			TC37	TC38	TC39	TC40		
CBOR-TAP	D1	D3	D16	D17	D22	D19	D23	D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
BORE T1 S1 T2 S2	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40		
LINE CTR		E2		E7	E9		E17							E95					TC37	TC38	TC39	TC40		
LINE RGT, LFT		E2		E7	E9		E17			E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40		
LINE OUT, IN	E1	E2	E5	E7	E9		E17		E21	E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40		
CHMF RGT, LFT		E2		E8	E9	E11	E17							E95					TC37	TC38	TC39	TC40		
CHMF OUT, IN	E1	E2		E8	E9	E11	E17		E21					E95					TC37	TC38	TC39	TC40		
FCE MILL					E9	E12	E15												TC37	TC38	TC39	TC40		
TOP EMIL				E7	E9	E13	E17							E97					TC37	TC38	TC39	TC40		
STEP	E1	E2	E5	E7	E9	E16	E17		E21	E22	E23	E24	E25	E91	E98				TC37	TC38	TC39	TC40		
POCKET	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E92					TC37	TC38	TC39	TC40		
PCKT MT	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E93					TC37	TC38	TC39	TC40		
PCKT VLY	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E94	E98				TC37	TC38	TC39	TC40		
SLOT				E7	E9		E17		E21					E96					TC37	TC38	TC39	TC40		
ANG. FACE					E9	E12													TC37	TC38	TC39	TC40		
MMS	L1	L2	K21	K22															TC37	TC38	TC39	TC40		TC62
WORK MES	K17	K18	K19	K23															TC37	TC38	TC39	TC40		TC62
TOOL MES	K17	K18	K20																TC37	TC38	TC39	TC40		TC62
TRANSFER		TC44	TC57	TC58	TC59																			
BAR	TC67	TC68	TC1	TC5	TC6														TC37	TC38	TC39	TC40		TC62
COPY	TC7																		TC37	TC38	TC39	TC40		TC62
CORNER	TC67	TC68	TC1																TC37	TC38	TC39	TC40		TC62
FACING			TC56	TC76	TC1														TC37	TC38	TC39	TC40		TC62
THREAD		TC41	TC77	TC78			TC82												TC37	TC38	TC39	TC40		TC62
T. GROOVE	TC52	TC42	TC43	TC73	TC74		TC75												TC37	TC38	TC39	TC40		TC62
T. DRILL	TC47	TC20	TC11	TC12															TC37	TC38	TC39	TC40		TC62
T-TAP	TC21	TC22																	TC37	TC38	TC39	TC40		TC62

B. Common data for unit

Common/Relay point	Turret	Rough/Finish	Tool rotational position	Rotation clearance X	Rotation clearance Z	Fixed point X	Fixed point Z
C	K	F	P	XA	ZA	XB	ZB
0: Relay point section	0: UPPER	0: ROUGH					
1: Common data for unit	1: LOWER	1: FINISH					

C. Relay point

Common/Relay point	Turret	Rough/ Finish	Relay point type	Relay point setting	X1	Y1	Z1	X2	Y2	Z2	X3	Y3	Z3
C	K	F	A	B	XA	YA	ZA	XB	YB	ZB	XC	YC	ZC
0: Relay point section	0: UPPER	0: ROUGH	0: APPROACH	0: MANU									
1: Common data for unit	1: LOWER	1: FINISH	1: ESCAPE	1: AUTO									

D. Barrier information

- Barrier information for turning spindle

Type	Jaw No.	Jaw code/Name	Gripping dia.	Parts count	Z offset	C offset
A	B	C/( )	E	D	F	J
0: Turning spindle 1						
1: Turning spindle 2						

- Barrier information for tailstock

Type	Taistock used/ not used	Tailstock barrier	Projection length	End coordinate Z	Tailstock pos. 1	Tailstock pos. 2
A	H	I	L	Z	P	Q
1: (Fixed)	0: Not used	0: Invalid				
	1: Used	1: Valid				

10-4 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

G10L40T_H_C_&C_D_I_S_P_A_E_

TNo.	Turret	TOOL	PART	NOM-ϕ	INTERFER.	ID CODE	PKNo.	LENGTH	ACT-ϕ
T	H	C	&C	D	I	S	P	A	E

- Turning tool (GENERAL, GROOVE, THREAD and OTHER)

G10L40T_H_C_&C_D_I_S_P_A_E_B_

TNo.	Turret	TOOL	PART	NOM-ϕ	INTERFER.	ID CODE	PKNo.	LENGTH A	NOSE-R	LENGTH B
T	H	C	&C	D	I	S	P	A	E	B

- Turning tool (T-DRILL and T-TAP)

G10L40T_H_C_&C_D_I_S_P_A_E_B_

TNo.	Turret	TOOL	PART	NOM-ϕ	INTERFER.	ID CODE	PKNo.	LENGTH	ACT-ϕ
T	H	C	&C	D	I	S	P	A	B

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
2	IN
3	EDGE
4	IN (BACK)
5	EDGE (BACK)
17	001
18	002
19	003
20	004
21	005
22	006
23	007
24	008
25	009

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:	CUT DIR.
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

- Milling tool

G10L41T_H_Y_C_P_R_F_D_S_(_)Q_I_V_M_B_

TNo.	Turret	TAP TYPE	LENG COMP.	THRUST F.	REC. FEED	EDG-ANG	HORSE PW	MAX. ROT	MAT.	HOLDER	BORDER	TIP POS.	CORNER R	THEETH
T	H	Y	C	P	R	F	D	S	( )	Q	I	V	M	B

- Turning tool

G10L41T_H_K_A_F_E_(_)Q_J_V_Z_

TNo.	Turret	CUT DIR.	CUT ANGLE/ GRV DEPTH	EDG-ANG/ TIP-WID	WIDTH	MAT.	HOLDER.	HLD. TYPE	INDEX ANG.	LBB No.
T	H	K	A	F	E	( )	Q	J	V	Z

C. Tool data 3

G10L42T_H_M_N_P_Q_F_(_)D_E_V_W_X_Y_Z_&X_&Y_&Z_F_A_B_C_I_J_K_

TNo.	Turret	LIFE TIME	CUT TIME	LIFE NUM.	USED NUM.	GROUP No.	IDNo.	ACT-ϕ CO.	ACT-ϕ CO. No.	LENG. CO.	LENG. CO. No.	WEAR COMP X/Y/Z	MAX WEAR &X/&Y/&Z	TOOL PROJ	EASY COMP X/Y/Z	CONS. COMP
T	H	M	N	P	Q	F	( )	D	E	V	W	X/Y/Z	&X/&Y/&Z	R	A/B/C	I/J/K

2. Tool offset

A. Type A

G10L10H_P_R_

B. Type B

Geometric compensation for tool length ...... G10L10H_P_R_
Wear compensation for tool length .... G10L11H_P_R_
Geometric compensation for tool radius ...... G10L12H_P_R_
Wear compensation for tool radius .... G10L13H_P_R_

C. Type C

Geometric compensation Z .... G10L10H_P_R_
Wear compensation Z.................................. G10L11H_P_R_
Geometric compensation for tool radius .... G10L12H_P_R_
Wear compensation for tool radius .... G10L13H_P_R_
Geometric compensation X .... G10L14H_P_R_
Wear compensation X.................................. G10L15H_P_R_
Geometric compensation Y.......................... G10L16H_P_R_
Wear compensation Y.................................. G10L17H_P_R_
Direction .... G10L18H_P_R_

D. Type D

Offset Z... G10L10H_P_R_
Tool radius .... G10L11H_P_R_
Offset X .... G10L12H_P_R_
Offset Y .... G10L13H_P_R_
Direction .... G10L14H_P_R_

E. Without EIA option

Offset Z... G10L10H_P_R_
Offset X .... G10L11H_P_R_
Offset Y .... G10L12H_P_R_

Turret	Offset No.	OFFSET
H	P	R

3. Tool file

A. End mill and ball-end mill

G10L49P_C_D_S_(_)R_H_

Tool file No.	TOOL	NOM-ϕ	ID code	MAT.	DEPTH	No.
P	C	D	S	( )	R	H

Refer to “Tool data.”

B. Face mill

G10L49P_C_D_S_(_)R_H_F_

Tool file No.	TOOL	NOM-ϕ	ID code	MAT.	DEPTH	No.	ANG
P	C	D	S	( )	R	H	F

Refer to “Tool data.”

C. Chamfering cutter

G10L49P_C_D_S_(_)E_H_M_F_

Tool file No.	TOOL	NOM-ϕ	ID code	MAT.	MIN-ϕ	No.	R-chamfering flag	ANG
P	C	D	S	( )	E	H	M	F
							0: Chamfering	
							1: Rounding	

Refer to “Tool data.”

4. 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

G10L68A_Z_ Cutting condition parameter A1 to A108
G10L68B_Z_ Cutting condition parameter B1 to B108
G10L68C_Z_ Cutting condition parameter C1 to C108

Parameter address	Setting
A/B/C	Z

5. Workpiece offset

A. Standard

G10L2N_P_

N1 (System 1) to N4 (System 4)

Coordinate sytem	Coordinate sytem shifting	G54	G55	G56	G57	G58	G59
P	0	1	2	3	4	5	6

B. Additional workpiece coordinate system

G10L20N_P_

N1 (System 1) to N4 (System 4)
P: Axis No.

Axis No.	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
P	X	Y	Z	A	B	C	D	E	F	H	I	J	K	M	N	O

6. Parameter

A. User parameter

G10L50 User parameter setting start
D1Z1 1 is set to D1 (example).
D2Z2
:
G11 User parameter setting end

Parameter	D:	E:	TC:	IO:	I:	SU:	SD:	F:
G10 format	D1 to D144	E1 to E144	T1 to T154	H1 to H456 57*8	I1 to I384 24*16	A1 to A672 168*4	B1 to B168	F1t o F168

B. Machine parameter

```
G10L51    Machine parameter setting start
J1Z1      1 is set to J1 (example).
J2Z1
 ⋮
G11       Machine parameter setting end
```

Parameter	J:	K:	L:	M:	N:	S:	SA:	SP:	SV:	BA:	R register
G10 format	J1 to J144	K1 to K144	L1 to L144	M1 to M768 48*16	N1 to N768 48*16	S1 to S768 48*16	W1 to W1152 144*8	P1 to P2048 256*8	V1 to V6144 384*16	X1 to X528 132*4	R2100 to R2527 R10500 to R11199 R16176 to R16383

7. 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).

8. Pallet management

G10L46P_U_R_W_S_J_N_K_M_A_B_C_D_Q_X_Y_Z_E_

9. 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	( )

- NOTE -