

PROGRAMMING MANUAL

for

MAZATROL FUSION 640M Pro MAZATROL PROGRAMMING

MANUAL No. : H736PG0010E

Serial No. :

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

IMPORTANT NOTICE

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

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

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. The meanings of our safety precautions to DANGER, WARNING, and CAUTION are as follows:



DANGER

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



WARNING

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



CAUTION

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

Basics



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

Remarks on the cutting conditions recommended by the NC



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

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

Programming



- Fully check that the settings of the coordinate systems are correct. Even if the designated program data is correct, errors in the system settings may cause the machine to operate in unexpected places and the workpiece to abruptly move out from the machine in the event of contact with the tool.
- During surface velocity hold control, as the current workpiece coordinates of the surface velocity hold control axes approach zeroes, the spindle speed increases significantly. For the lathe, the workpiece may even come off if the chucking force decreases. Safety speed limits must therefore be observed when designating spindle speeds.
- Even after inch/metric system selection, the units of the programs, tool information, or parameters that have been registered until that time are not converted. Fully check these data units before operating the machine. If the machine is operated without checks being performed, even existing correct programs may cause the machine to operate differently from the way it did before.
- If a program is executed that includes the absolute data commands and relative data commands taken in the reverse of their original meaning, totally unexpected operation of the machine will result. Recheck the command scheme before executing programs.
- If an incorrect plane selection command is issued for a machine action such as arc interpolation or fixed-cycle machining, the tool may collide with the workpiece or part of the machine since the motions of the control axes assumed and those of actual ones will be interchanged. (This precaution applies only to NC units provided with EIA functions.)
- The mirror image, if made valid, changes subsequent machine actions significantly. Use the mirror image function only after thoroughly understanding the above. (This precaution applies only to NC units provided with EIA functions.)
- If machine coordinate system commands or reference position returning commands are issued with a correction function remaining made valid, correction may become invalid temporarily. If this is not thoroughly understood, the machine may appear as if it would operate against the expectations of the operator. Execute the above commands only after making the corresponding correction function invalid. (This precaution applies only to NC units provided with EIA functions.)
- The barrier function performs interference checks based on designated tool data. Enter the tool information that matches the tools to be actually used. Otherwise, the barrier function will not work correctly. (This precaution applies only to the M640T, M640MT and M640M Pro.)
- The system of G-code and M-code commands differs between the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060) and the machines equipped with M640MT/T (such as the INTGEREX non e-Series, the SQT Series and the MPX Series).
Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

Sample program	Machine with M640M Pro	Machine with M640MT/T
S1000M3	The milling spindle rotates at 1000 min ⁻¹ .	The turning spindle rotates at 1000 min ⁻¹ .
S1000M203	The turning spindle rotates at 1000 min ⁻¹ .	The milling spindle rotates at 1000 min ⁻¹ .

- For the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060), programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA program. However, for example, when the B-axis is rotated through 180 degrees around the Y-axis to implement machining with the turning spindle No. 2, the plus side of the X-axis in the programmed coordinate system faces downward and if the program is created ignoring this fact, the resulting movement of the tool to unexpected positions may incite collisions.

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



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.

Operations



WARNING

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



CAUTION

- During manual operation, fully check the directions and speeds of axial movement.
- For a machine that requires manual homing, perform manual homing operations after turning power on. Since the software-controlled stroke limits will remain ineffective until manual homing is completed, the machine will not stop even if it oversteps the limit area. As a result, serious machine damage will result.
- Do not designate an incorrect pulse multiplier when performing manual pulse handle feed operations. If the multiplier is set to 100 times and the handle operated inadvertently, axial movement will become faster than that expected.

OPERATIONAL WARRANTY FOR THE NC UNIT

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

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

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

Operating Environment

1. Ambient temperature

During machine operation: 5° to 40°C (41° to 104°F)

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

2. Relative humidity

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

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

- NOTE -

CONSTRUCTION

Introduction

Part 1 MAZATROL PROGRAMMING PROCEDURES

- Chapter 1 MAZATROL PROGRAM
- Chapter 2 SYSTEM OF COORDINATES
- Chapter 3 FUNCTION OF KEYS AND SWITCHES
- Chapter 4 PROCEDURE BEFORE PERFORMING THE MACHINING

Part 2 MAZATROL PROGRAM FUNCTIONS

- Chapter 1 CALLING UP AND THE END OF THE PROGRAM DISPLAY
- Chapter 2 EDITION OF DATA
- Chapter 3 HELP FUNCTIONS
- Chapter 4 PROGRAM CREATION
- Chapter 5 PRIORITY FUNCTION FOR THE SAME TOOL
- Chapter 6 COORDINATES MEASUREMENT FUNCTION
- Chapter 7 BARRIER INFORMATION DATA CREATION
- Chapter 8 TPC DATA CREATION
- Chapter 9 BACKGROUND PROGRAMMING
- Chapter 10 CASE OF APPEARANCE OF ALARM
- Chapter 11 THREE-DIGIT G-FORMAT

Appendix

- Appendix 1 PROGRAM EXAMPLES
- Appendix 2 WHAT TO DO IN SUCH A CASE?

- NOTE -

Introduction

This manual describes only programming based on the MAZATROL language of the MAZATROL FUSION 640M Pro system. The description given in this manual assumes that the readers have already read the relevant Operating Manual and thoroughly understood its contents.

Programming in the MAZATROL language uses an interactive method that allows the system to be operated in accordance with the messages displayed on the CRT monitor. Thus, even a user who is to operate the system for the first time can readily create and edit programs.

Carefully read this manual and the Operating Manual to correctly operate the MAZATROL FUSION 640M Pro system and use its capabilities to their maximum.

NOTE:

The MAZATROL FUSION 640M Pro controls a machine tool by digital calculation, but it is possible that the machining cannot be performed because of the processing of a calculation error. Before proceeding with automatic operation therefore, do not fail to inspect the path of the tool on the display in order to verify that the machining is being done correctly.

Organization of this manual

The following outlines the organization of this manual:

Part	Title	Upper row – Directions for using the chapter
		Lower row – Contents of the chapter
1	MAZATROL PROGRAMMING PROCEDURES	Those who are going to set up a MAZATROL program for the first time must first read this part. Actually operate the MAZATROL FUSION 640M Pro system from the beginning as directed in Chapter 4, "PROCEDURE BEFORE PERFORMING THE MACHINING." You will then be able to roughly understand the entire operating procedure from tool registration to creation of a MAZATROL program. Those who already have an experience in creating a program using the MAZATROL system will be able to understand the functions and usage of the M640M Pro system just by directly reading Part 2, "MAZATROL PROGRAM FUNCTIONS."
		The program architecture, coordinates systems, and operation keys are discussed here as the basic information required for program creation in MAZATROL language. In addition to the program creation procedures, successive processing procedures up to machining are described in order after the discussion of the basic information.
2	MAZATROL PROGRAM FUNCTIONS	Use the functions of the MAZATROL FUSION 640M Pro system only after understanding the MAZATROL program architecture and program creation procedures described in Part 1. Also, use this part as a reference whenever required, such as in cases where you have only a slight idea (or have forgotten) how to use a function, as you create a MAZATROL program.
		How to call up the PROGRAM display, how to edit data, how to use a function, and details of each function are described here. How to create programs in the background mode, and how to act in the event of alarms are also discussed.
APPENDIX		A sample programs, and a Q&A (Questions and Answers) list are contained. Read this appendix as appropriate.

How to use this manual

Next, how to use this manual is briefly described below.

1. In Part 2, "MAZATROL PROGRAM FUNCTIONS," where the operating procedures for carrying out the MAZATROL language functions are described, the order of selecting menu items is indicated at "Menu selection." Use the section as quick reference during menu selection.
2. In Part 2, "MAZATROL PROGRAM FUNCTIONS," where the operating procedures are described, the arrow "→" under the description of each step of the procedure is followed by a description of the results of the particular operation.

Example:

- (1) Press the **SEARCH** menu key.

→ The message **SEARCH DATA?** will be displayed and the menu will change to the following address menu:

The section underlined above denotes the results of the operation.

PART 1

MAZATROL PROGRAMMING PROCEDURES

Part 1 describes the architecture of MAZATROL programs, coordinate systems, the keys to be used for programming, and simplified programming procedures.

CONTENTS

	Page
1 MAZATROL PROGRAM.....	1-1
2 SYSTEM OF COORDINATES.....	2-1
2-1 Machine Coordinates System.....	2-2
2-2 Workpiece Coordinates System.....	2-3
2-3 Machine Coordinates System and Workpiece Coordinates System.....	2-4
2-4 Fundamental Coordinates	2-5
2-5 Auxiliary Coordinates.....	2-6
3 FUNCTION OF KEYS AND SWITCHES.....	3-1
4 PROCEDURE BEFORE PERFORMING THE MACHINING.....	4-1
4-1 Diagram of Preparation for the Machining	4-1
4-2 Part Drawings and Stock Materials.....	4-2
4-3 Registration of Tools.....	4-4
4-3-1 Calling up the TOOL FILE display	4-4
4-3-2 Registration of tools.....	4-5
4-4 Creation of Simple Programs.....	4-9
4-4-1 Program structure.....	4-9
4-4-2 Calling up the PROGRAM display	4-10
4-4-3 Assigning workpiece numbers	4-10
4-4-4 Creating a program.....	4-12
4-4-5 Creation of common unit	4-12
4-4-6 Creation of fundamental coordinates system unit	4-15

4-4-7	Index unit.....	4-17
4-4-8	Turning unit	4-19
4-4-9	C-axis point machining unit.....	4-26
4-4-10	Face machining unit	4-32
4-4-11	Fundamental coordinate shifting unit	4-38
4-4-12	Line machining unit.....	4-41
4-4-13	Point machining unit	4-46
4-4-14	End unit	4-50
4-4-15	End of the program.....	4-51
4-5	Registration of Tools in a Tool Data File	4-53
4-5-1	Pocket numbers and tool numbers	4-53
4-5-2	TOOL DATA display	4-54
4-5-3	TOOL LAYOUT display	4-55
4-5-4	Registration of tools.....	4-58
4-6	Check of the Machining Path.....	4-62

1 MAZATROL PROGRAM

The machining MAZATROL program of a workpiece consists in principle of the following four units:

1. Common unit

This concerns a unit which is obligatorily entered in the program head. It specifies the common data to a program assembly such as the material, the initial point, the machining of several workpieces, etc.

2. Basic coordinates system unit

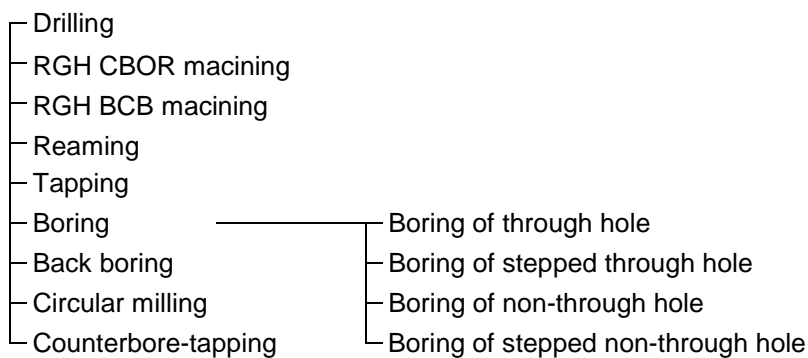
Use to specify the value of the coordinates (basic coordinates) of the workpiece zero point in the machine coordinates system.

3. Machining unit

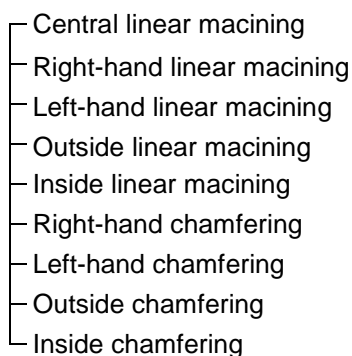
Use to specify the data concerning the machining method and the machining form.

The machining unit is available in the following four types:

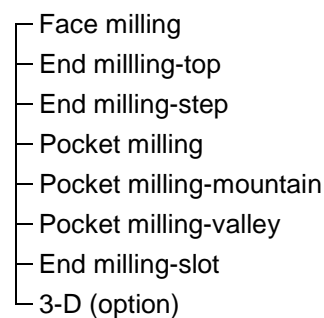
Point machining unit/C-axis point machining unit



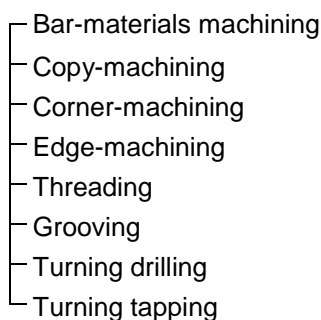
Line machining unit/C-axis line machining unit



Face machining unit



Turning unit



Moreover, the necessary data are specified in the following two sequences:

Tool sequence Use to specify the data concerning the name of the tool and the movement of the tool.

Shape sequence Use to specify the data related to the machining dimensions.

4. End unit

Unit created at the end of program.

Also, the following units are entered when necessary.

5. Workpiece shape unit

The shapes of cast blanks or forged blanks cannot be defined with the common unit alone. To machine these moldings, select the common unit first and then the workpiece shape unit before specifying the shape of the blank.

6. Auxiliary coordinates system unit

Use to specify the auxiliary coordinates system (OFFSET).

7. Fundamental coordinate shifting unit

Shifts the coordinates of the program origin (fundamental coordinates).

8. M-code unit

Use to output the M-code

9. Sub-program

Use to call up a sub-program

10. Pallet changing unit (*)

Use to change a pallet

11. Indexing unit

Use to specify the angle of the milling spindle, indexing table and chuck.

12. Process end unit

Delimits the useful scope of the priority function for the same tool.

13. Manual program mode unit

This unit is entered to establish a program corresponding to the EIA/ISO program using the G and M-codes which permits performing minute movement or a movement other than machining.

14. Coordinate measuring unit

Automatic measurement of a fundamental coordinates system (WPC).

15. Workpiece measuring unit

Measures the workpiece automatically.

16. Tool measuring unit

Measures the tool automatically.

17. Workpiece transfer unit (*)

For a machine having two turning spindles, specify the modification of the workpiece chucking position, the transfer of the workpiece, and the movement of the second turning spindle.

18. Head selection unit (*)

For a machine having two turning spindles, specify which spindle is to be used to machine the workpiece.

Note: * indicates that whether the unit can be used depends on the specifications of the machine.

- NOTE -

2 SYSTEM OF COORDINATES

In the preparation of the program, a system of coordinates is used for introducing the position of the machining and the form of the machining.

The system of coordinates consists of three axes of coordinates which each cross the reference zero-point at right angles.

An arbitrary point found in this system of coordinates can be defined by the value of the coordinates in the 3-axes (X, Y and Z).

There are two types of systems of coordinates:

- Machine coordinates system
- Workpiece coordinates system

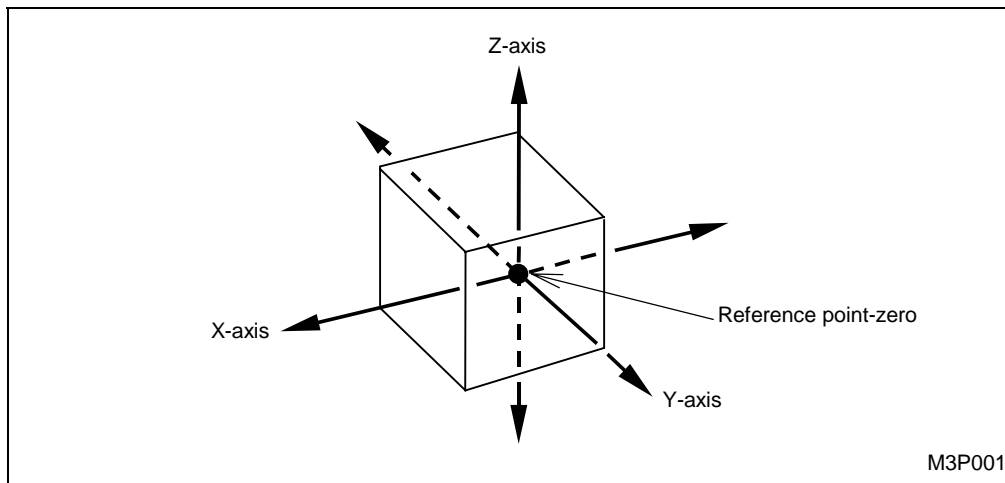


Fig. 2-1 Coordinates system

2-1 Machine Coordinates System

The machine actually moves in its own system of coordinates called the machine coordinates system. A point of reference in this system of coordinates is known as machine zero-point. Generally, the machine coordinates system has the machining zone on the side of the minus (negative) direction from the machine zero-point.

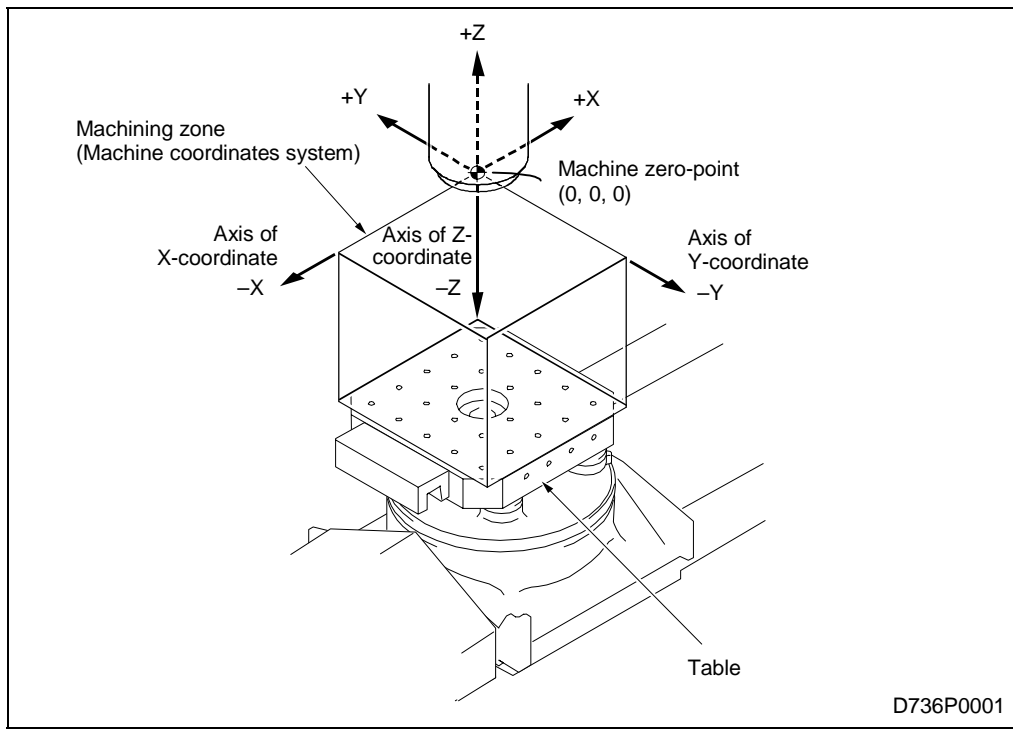


Fig. 2-2 Machine coordinates system (e1060V)

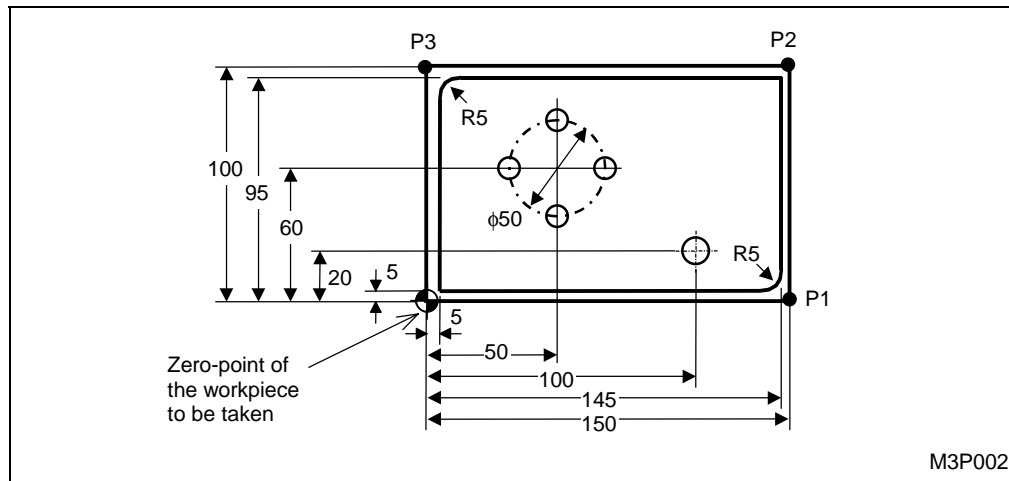
2-2 Workpiece Coordinates System

If the program is prepared on the basis of the machine coordinates system, the entering of the machining position and of the form of machining is very complex, tedious and inflexible.

Consequently, a temporary reference point is taken in the machine coordinates system for preparing the program.

The point thus taken is called the workpiece zero-point, and the system of coordinates taking this point as reference is called the workpiece coordinates system.

Example: Plan of the workpiece



- When the dimensions are entered of the configuration on the basis of the above plan of the workpiece, the bottom left hand corner is taken as the workpiece zero-point.
- In this case, the value of the coordinates of the configuration is the following:

$$\begin{aligned} \text{Workpiece zero-point} &= (0, 0, 0) \\ P1 &= (150, 0, 0) \\ P2 &= (150, 100, 0) \\ P3 &= (0, 100, 0) \end{aligned}$$

The adoption of the workpiece zero-point facilitates the entering of the machining dimensions and therefore the programming.

2-3 Machine Coordinates System and Workpiece Coordinates System

The relationship between the machine coordinates system and workpiece coordinates system when workpiece has been mounted on the table of a machine is shown below.

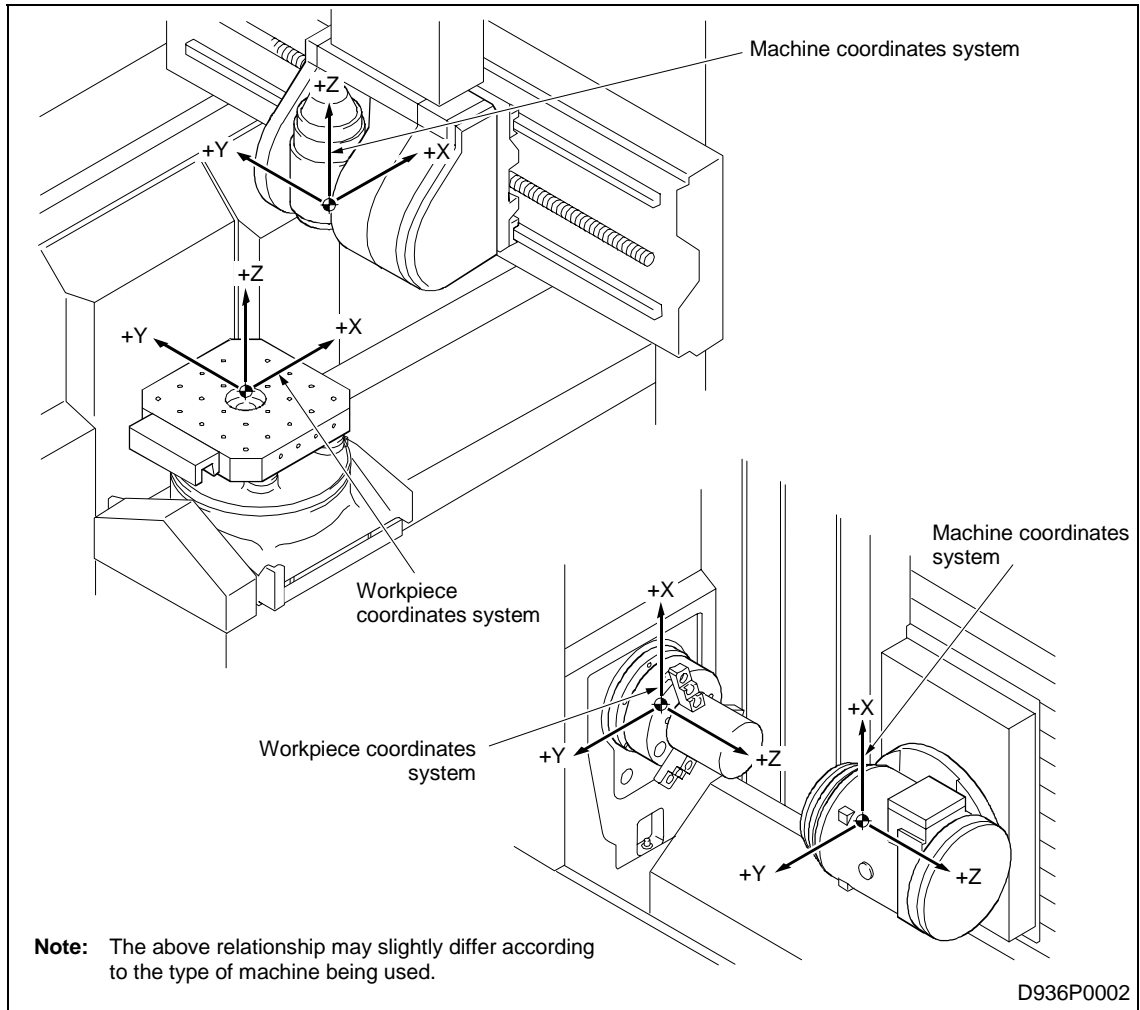


Fig. 2-3 Machine coordinates system and workpiece coordinates system

2-4 Fundamental Coordinates

The machine moves in the machine coordinates system whilst the program is prepared depending on the workpiece coordinates system.

It is necessary therefore to enter in the program, the position relation between the machine coordinates system and the workpiece coordinates system.

The unit of entry is called the fundamental coordinates system unit.

The fundamental coordinates are entered as values of the coordinates of the workpiece zero-point in the machine coordinates system.

The unit of the fundamental coordinates is entered by utilizing the coordinates measurement function after the workpiece is placed on the machine.

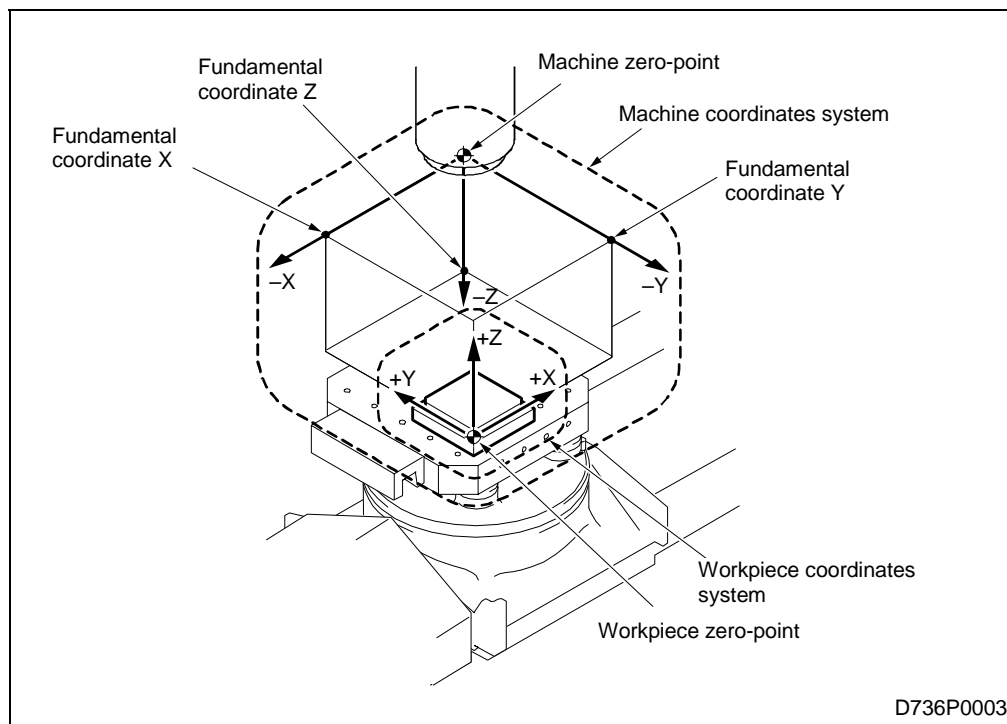


Fig. 2-4 Fundamental coordinates

Example of entering of the fundamental coordinates system unit:

UNO.	UNIT	ADD. WPC	X	Y	th	Z	C
1	WPC-0		-500.	-300.	0.	-200.	0.

Fundamental coordinates system unit
 No. of fundamental system coordinates
 Fundamental coordinate X (Example: -500)
 Fundamental coordinate Y (Example: -300)

Angle formed by the X- and Y-axes of the machine coordinates system and the X- and Y-axes of the workpiece coordinates system.

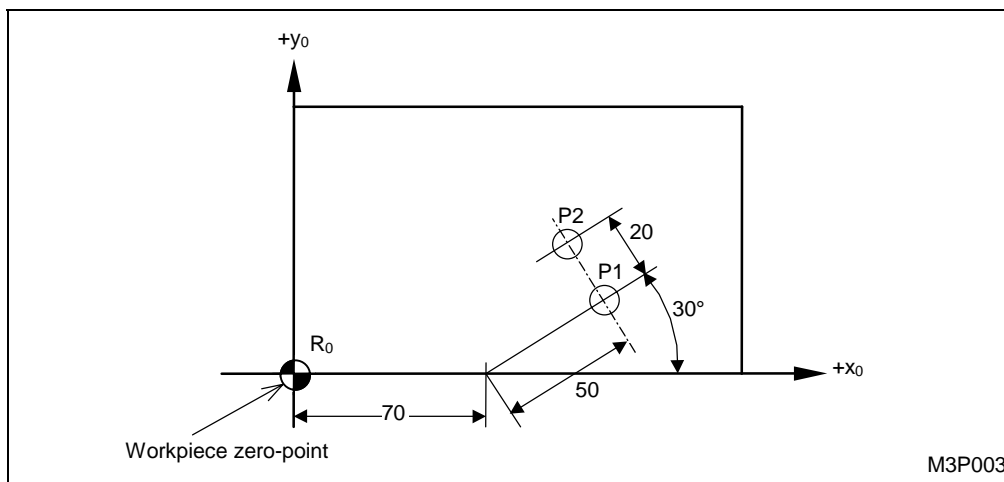
Fundamental coordinate Z (Example: -200)
 Coordinate or angle of 4th axis, if there is one.

2-5 Auxiliary Coordinates

The auxiliary coordinates are used for offsetting the workpiece zero-point to any position in order to further facilitate the preparation of the program.

The auxiliary coordinates unit (OFFSET) is entered as a value of offsetting in the workpiece zero-point.

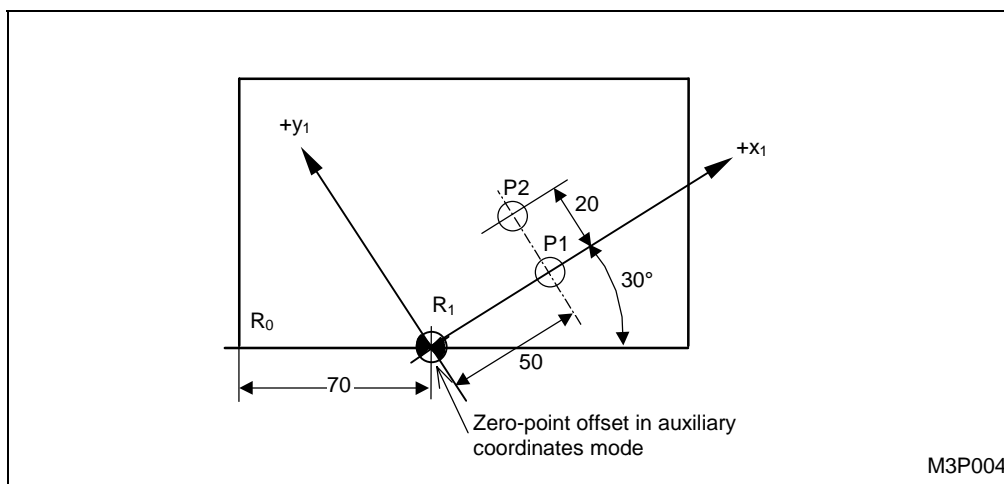
1. Example of entering of auxiliary coordinates



In this example, the entering of the position of hole P1 requires a very complicated calculation.

$$P1 = \left(70 + 50 \times \frac{\sqrt{3}}{2}, 50 \times \frac{1}{2} \right)$$

However, the use of auxiliary coordinates permits performing this entering easily.

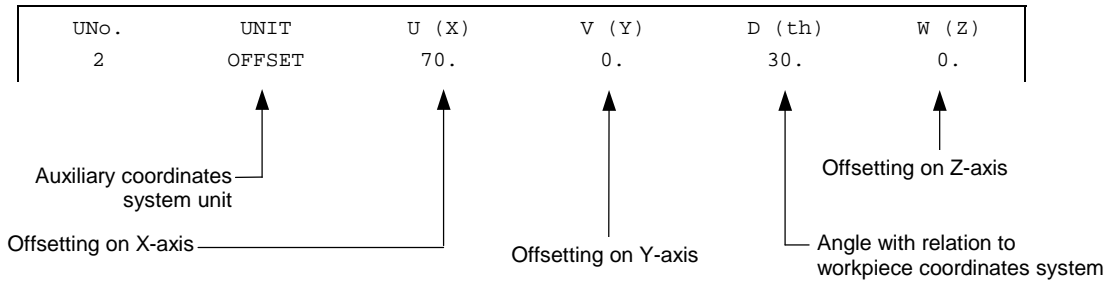


As the figure above shows, positions P1 and P2 are entered as follows by offsetting the workpiece zero-point to R1.

$$P1 = (50, 0)$$

$$P2 = (50, 20)$$

In this case, the auxiliary coordinates unit to be programmed is as follows:



2. Cancellation of auxiliary coordinates

The system of auxiliary coordinates is voided in the following cases:

- A. The system of auxiliary coordinates specified in a sub-program is voided at the time of the return on the main program. When the main program contains a system of auxiliary coordinates, the return is made to this system of auxiliary coordinates.
- B. The system of auxiliary coordinates is voided when a new system of fundamental coordinates is introduced. In this case, the state without a system of auxiliary coordinates is assumed. (When the system of fundamental coordinates was specified in the sub-program as shown in Figure 2-5, the return to the main program has the effect of voiding the system of auxiliary coordinates of the main program.)

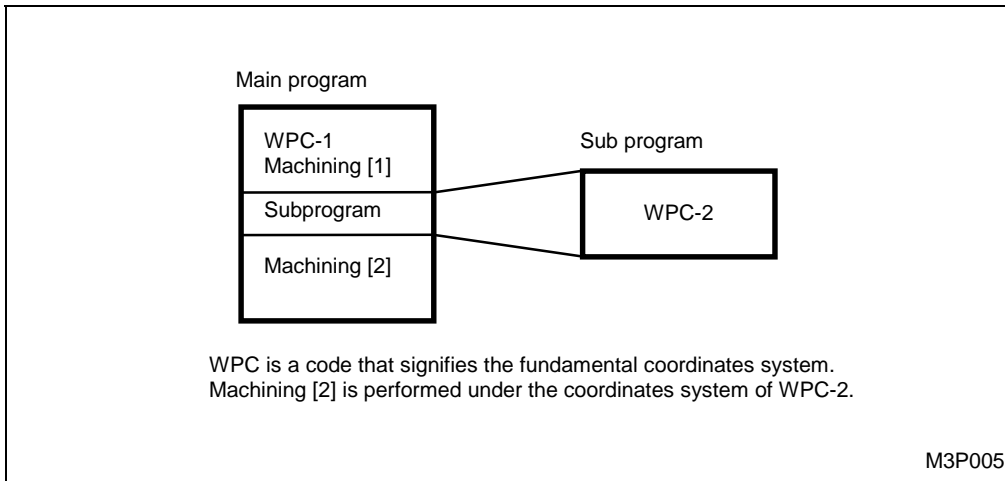


Fig. 2-5 Fundamental coordinates system after execution of subprogram

- NOTE -

3 FUNCTION OF KEYS AND SWITCHES

A MAZATROL program can be created using only the keys and switches indicated in the dotted portion on the operating panel shown below.

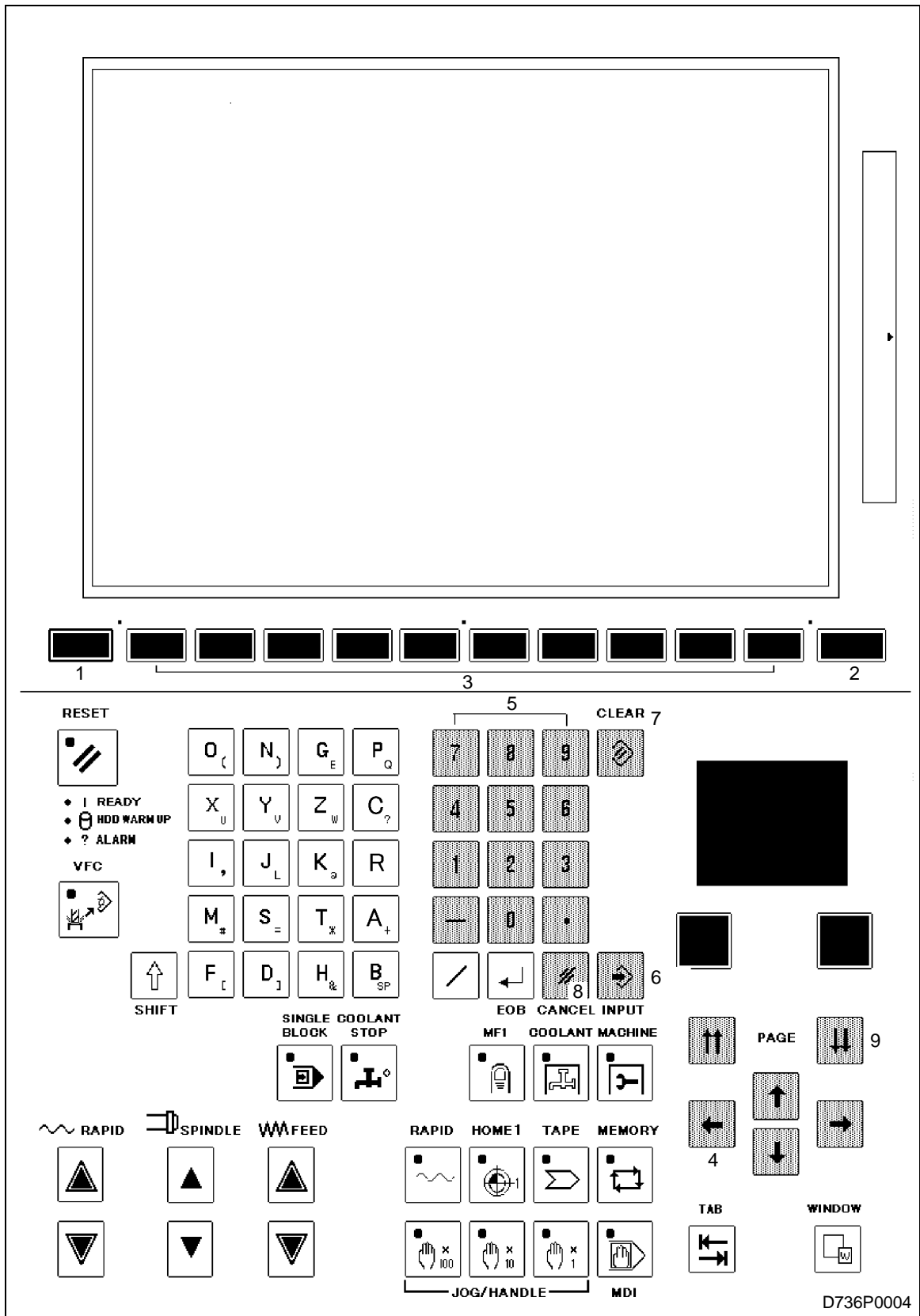


Fig. 3-1 Keys and switches used for creating a MAZATROL program (. portion)

Table 3-1 Functions of keys and switches

No.	Name	Description
1	Display selector key	Press this key to select between displays. Depression of this key causes a display selection menu to be displayed in the menu display area.
2	Menu selector key	Press this key to select menu that is displayed on the bottom of the screen.
3	Menu keys	A specific key is assigned to each of ten menu data that are displayed in the menu display area. Press the appropriate key to carry out the operation specified by the particular display or to input the data specified by the particular display.
4	Cursor keys	Press the appropriate key to move the cursor in the desired direction. If a cursor key is pressed and held down, the cursor will move continuously in the corresponding direction.
5	Numeric keys	Use these keys to key in the numerics 0 to 9, the minus sign (-), and/or the decimal point (.).
6	Input key	Press this key to register keyed-in data in the data display area.
7	Clear key	Press this key to cancel the data currently being displayed in the data selection area. This key is also used for erasure of the alarm display.
8	Data cancellation key	Press this key to cancel the data that has been displayed in the data display area. Depression of this key causes the data in the cursor position to be erased.
9	Page keys	Press the appropriate key to move the cursor unit by unit. Pressing and holding down the key will move the cursor unit by unit continuously.

4 PROCEDURE BEFORE PERFORMING THE MACHINING

4-1 Diagram of Preparation for the Machining

To machine a workpiece, it is essential to coordinate the programming and the preparations for the tool and the workpiece. The following diagram shows the general procedure to be followed before performing the machining.

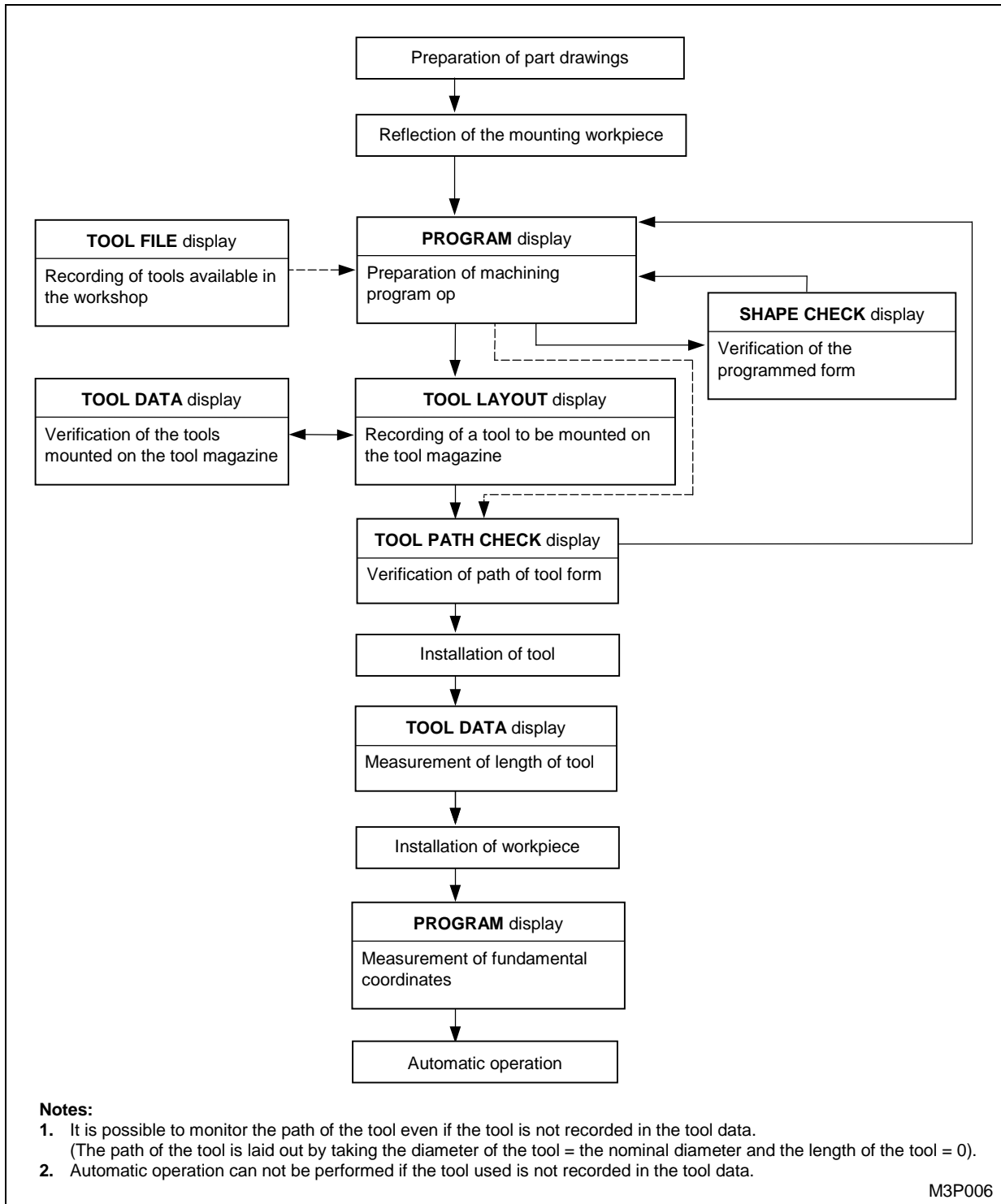


Fig. 4-1 Procedure before performing the machining

4-2 Part Drawings and Stock Materials

The following shows the part drawing for which a program is to be created in Section 4-4.

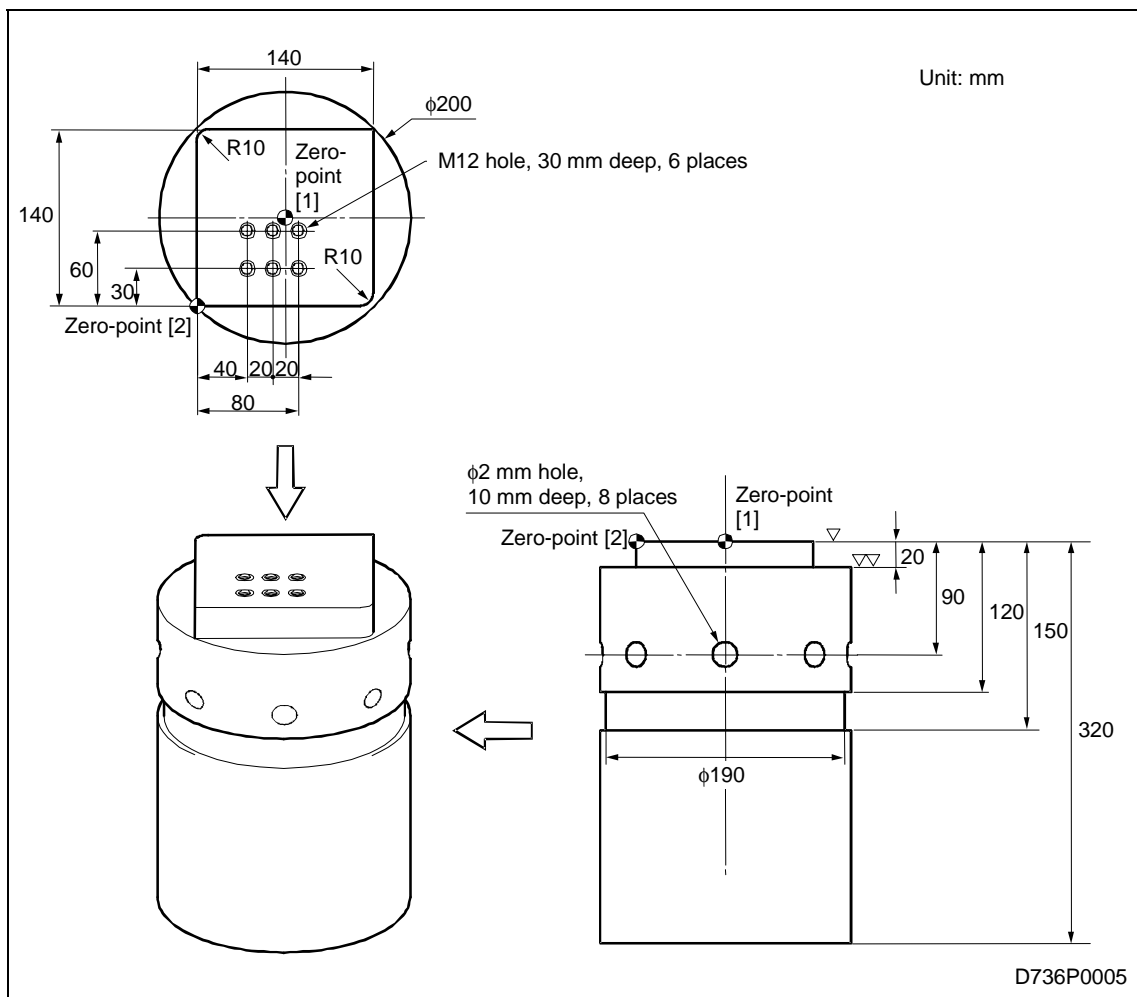
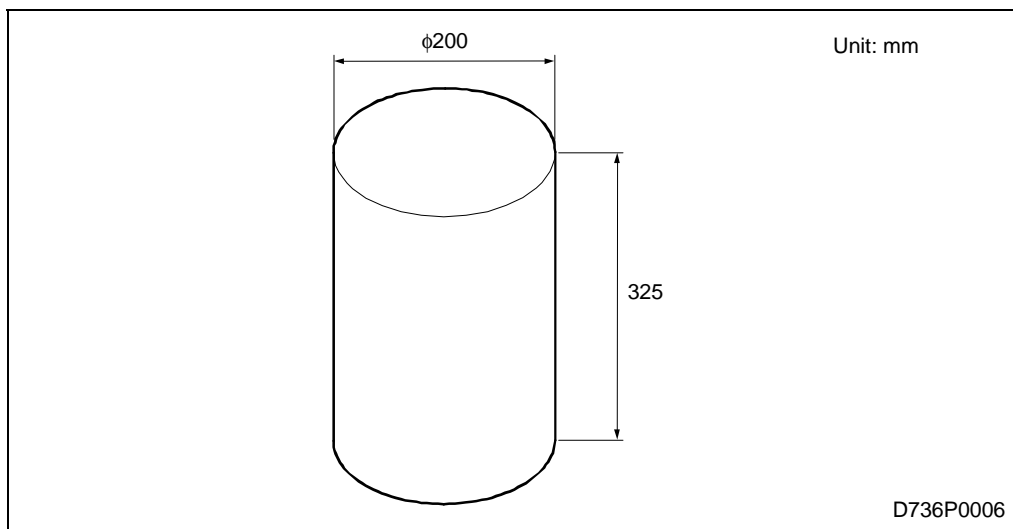
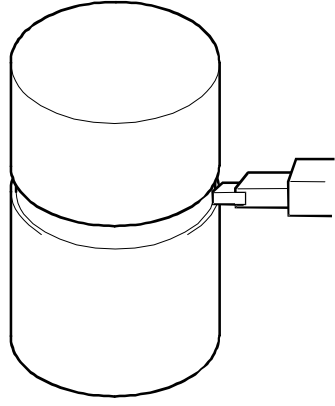
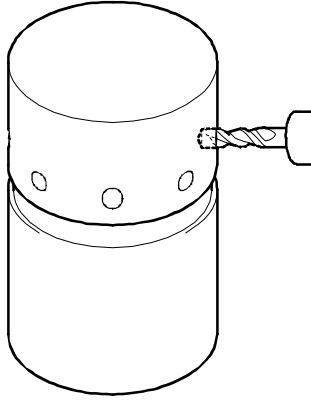
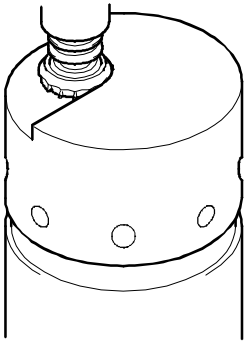
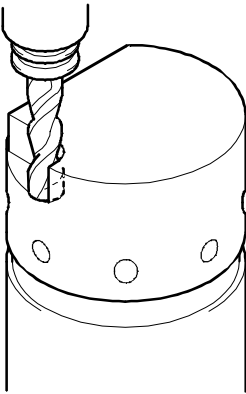
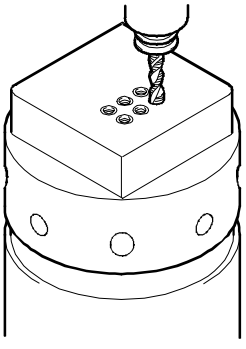


Fig. 4-2 Part drawing example

This machine part is made of a cylindrical material having the size shown in the figure below.



Machining of this part consists of the following five processes:

<p>[1] Grooving the cylindrical surface of the workpiece</p>	<p>[2] Drilling eight holes, 20 mm in diameter on the cylindrical surface of the workpiece</p>	
 <p style="text-align: right;">D736P0007</p>	 <p style="text-align: right;">D736P0008</p>	
<p>[3] Milling the top face of the workpiece.</p>	<p>[4] End-milling the edges of the workpiece.</p>	<p>[5] Drilling six M12 tapped holes.</p>
 <p style="text-align: right;">D736P0009</p>	 <p style="text-align: right;">D736P00010</p>	 <p style="text-align: right;">D736P00011</p>

4-3 Registration of Tools

First, turn on the power to return the axes to the zero-point.

Next, turn the reprogramming switch to the ENABLE position using the key.

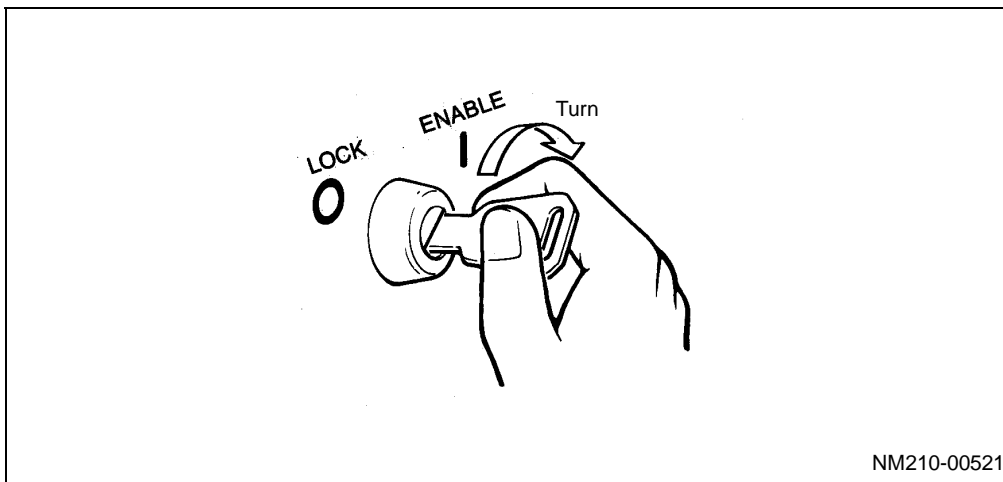
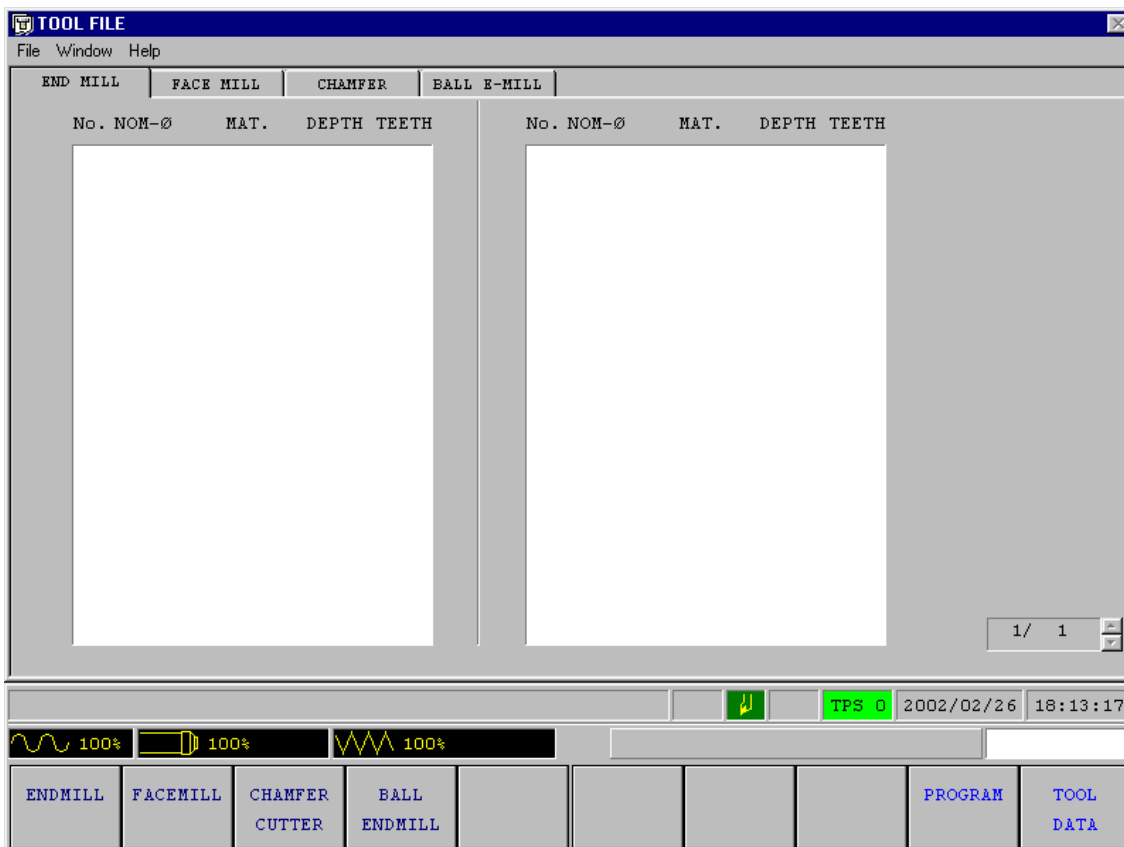


Fig. 4-3 Reprogramming switch

Before creating the program, information on the tools located in the workshop must be registered in a tool file. Unregistered tools cannot be used for programming or automatic run.


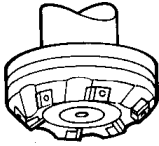
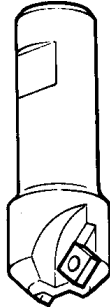

4-3-1 Calling up the TOOL FILE display

Call the **TOOL DATA** display by pressing firstly the display selector key and then the menu key **TOOL DATA**. Then press the menu key **TOOL FILE** to call the following **TOOL FILE** display.



D736P0012E

Register tools in the **TOOL FILE** display. The tools to be registered here are as follows:

End mill	Face mill	Chamfering cutter	Ball-end mill
			
NM210-00522	NM210-00523	NM210-00524	NM210-00525

Tools other than these four types do not need to be registered. Tool selection is automatically made by the NC unit during programming.

4-3-2 Registration of tools

In this chapter, programming is to be carried out by using the two types of tools shown below.

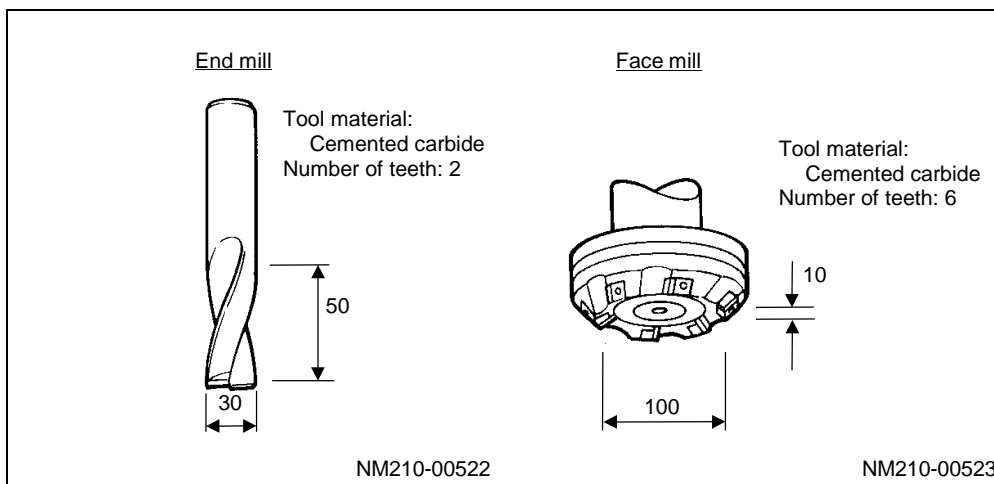



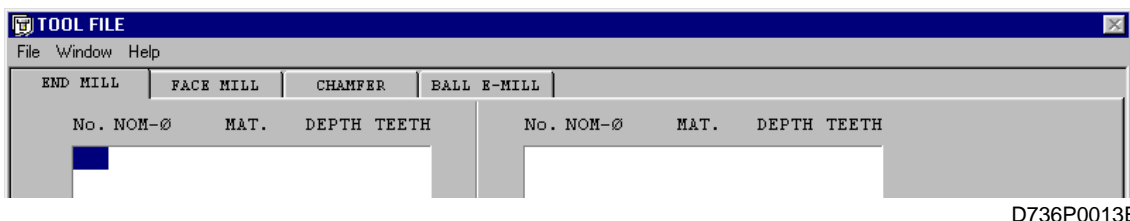
Fig. 4-4 Registered tools example

1. Registering the end mill

Using the page keys, search the display for the tool indicated as END MILL 30. A.

Up to 256 tools can be registered. If the tool is found, this indicates that the tool has been registered; therefore, proceed to the procedure described in “Registering the face mill.”

- (1) Press the cursor key  and position the cursor on an unregistered tool number.






D736P0013E

- ➔ The message “REGISTER TOOL <INPUT>?” will then be displayed in the message display area.

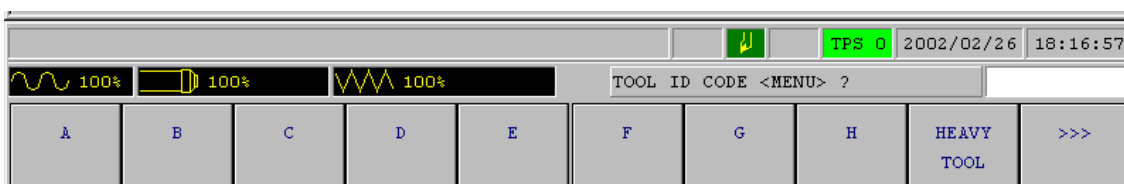
- (2) Press the input key  to register the tool.

- ➔ The message “NOMINAL DIAMETER?” will be displayed.

- (3) Input an approximate tool-diameter value (in this example, input 10 by pressing numeric keys  and  and then pressing the input key ).

- ➔ The message “TOOL ID CODE <MENU>?” will be displayed.

- (4) Input a specific code that identifies the particular tool (in this example, input A by pressing menu key A).






D736P0004E


- ➔ The message “TOOL MATERIAL <MENU>?” will be displayed.

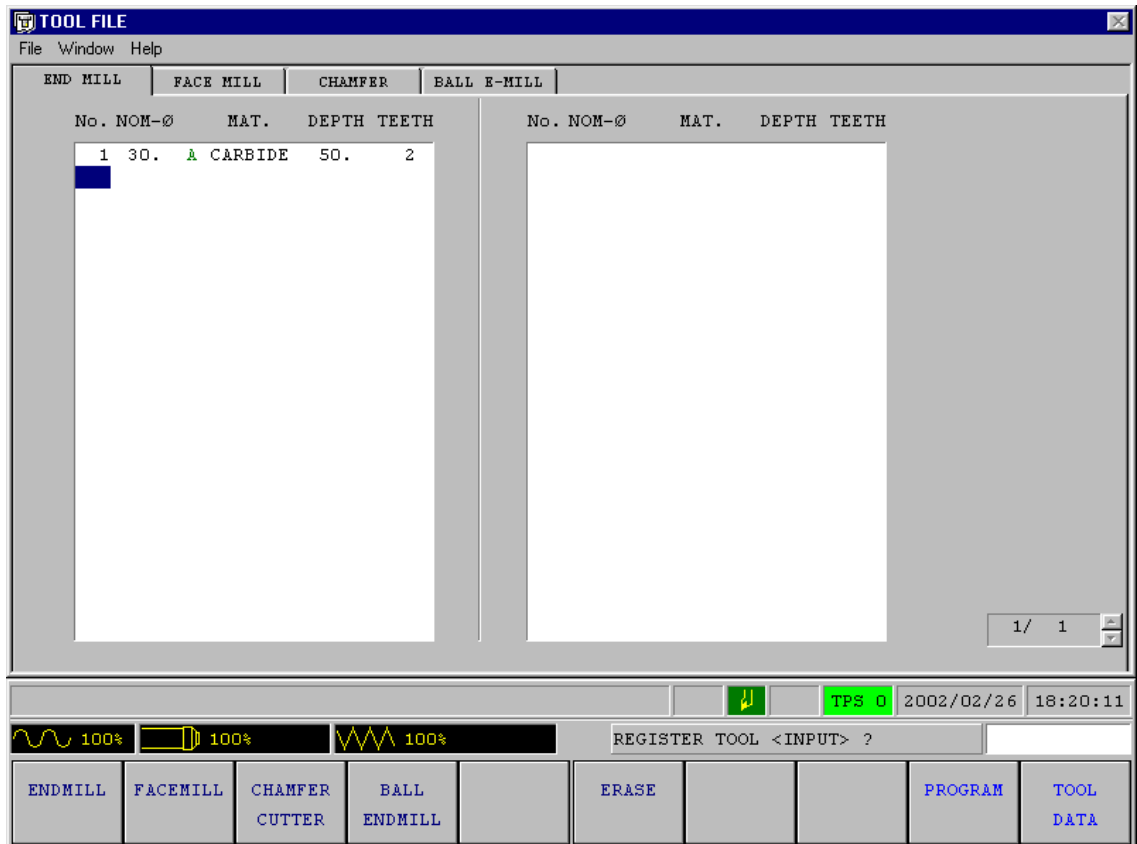
- (5) Select the tool material from the tool material menu. Material names registered in the **CUTTING CONDITION** display are displayed as menu (in this example, select cemented carbide by pressing menu key **CARBIDE**).

- ➔ The message “MAX DEPTH OF CUT” will be displayed.

- (6) Input the maximum depth to which the workpiece can be cut in the axial direction by one cutting operation (in this example, input 50 by pressing numeric keys  and  and then pressing the input key ).

- ➔ The message “NUMBER OF TEETH?” will be displayed.

- (7) Input the number of teeth of the tool (in this example, input 2 by pressing numeric key **2** and then pressing the input key ).
- Registration of the 30-mm diameter end mill will be completed as shown below.



D736P0014E




2. Registering the face mill

The operations to be carried out are similar to those required for registering the 30-mm diameter end mill. For brevity of description, the displayed message and key(s) to be pressed are noted to the right and left respectively as shown below.

- (1) Press the menu key **FACE MILL**.

Check if the tool indicated as 100. A is included in the **TOOL FILE (FACE MILL)** display. If the tool is found, then this indicates that the tool has been registered. Thus, the operations shown below are not required.

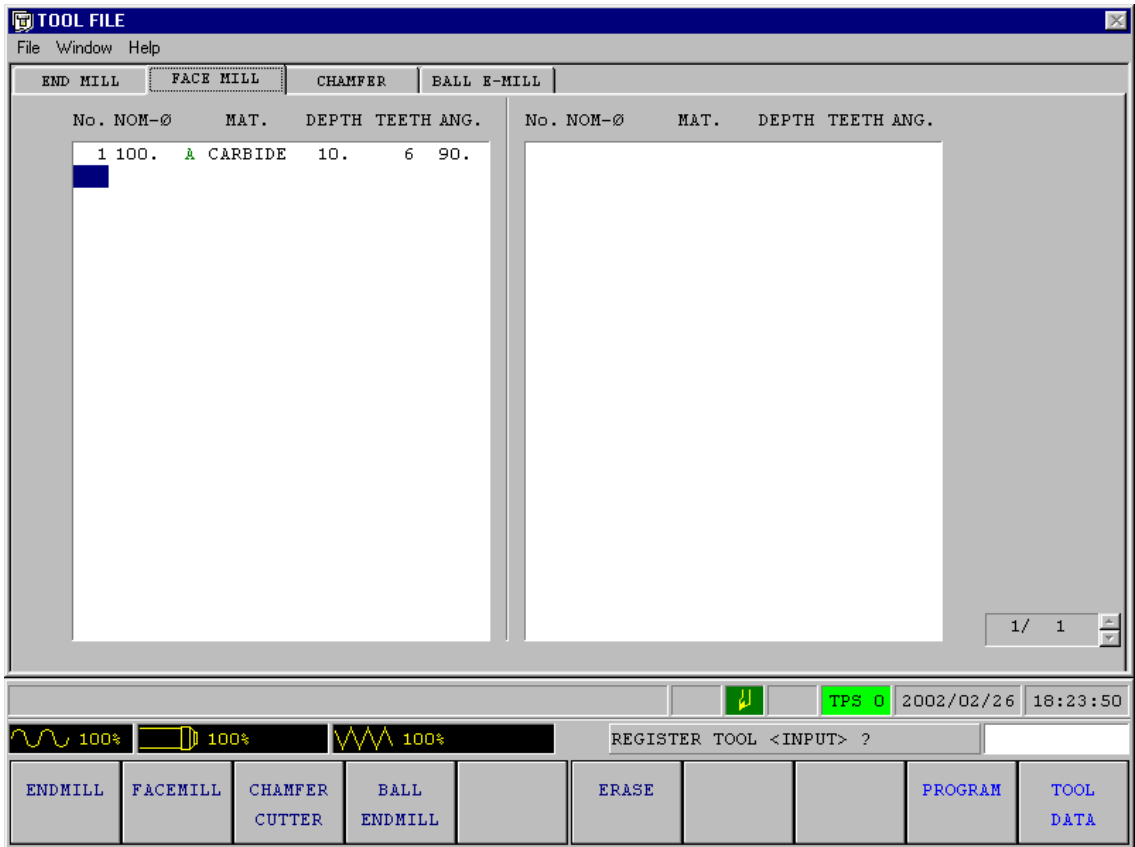
- (2) Position the cursor on an unregistered tool number.

- (3) REGISTER TOOL <INPUT>?..... 
- (4) NOMINAL DIAMETER?..... **1** **0** **0** 
- (5) TOOL ID CODE <MENU>? **A**
- (6) TOOL MATERIAL <MENU>? **CARBIDE**
- (7) MAX DEPTH OF CUT? **1** **0** 

4 PROCEDURE BEFORE PERFORMING THE MACHINING

- (8) NUMBER OF TEETH? **6**
- (9) ANGE OF CUT **9 0**

➔ This completes registration of the 100-mm diameter face mill as shown below.



D736P0015E

4-4 Creation of Simple Programs

4-4-1 Program structure

The structure of MAZATROL program is described before creating a program. Each MAZATROL program consists of units. Conversely, a combination of program units forms a program. The name of each part of a program is described below using part of the program to be created.

The screenshot shows the MAZATROL software interface with the following program structure:

UNo.	MAT.	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM				
0	CST IRN	MILL&TURN.	100.	0.	120.	0.	1	3000				
UNo.	UNIT	ADD. WPC	X	Y	th	Z	C					
1	WPC-	0	-201.23	-150.876	0.	-255.6	0.					
UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C						
2	INDEX				90.	45.						
UNo.	UNIT	DEPTH	SRV-Z	BTM	FIN-Z							
3	FCE MILL	0.	3.	2	0.							
SNo.	TOOL	NOM-Ø	No. APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R 1	FCE MILL	100.	A	?	XBI	3.	70.	120	0.173			
FIG PTN	P1X/CX	P1Y/CY	P3X/R	P3Y								
1	SQR	0.	0.	150.	100.			CN1	CN2	CN3	CN4	
UNo.	UNIT	CONTI. NUMBER	ATC	X	Y	Z	B	C				
4	END	0	0	0								

Below the screenshot, the following descriptions are provided for each unit:

- Common unit**
Basic data related to the entire program, such as the material of the workpiece, is to be input to this unit.
- Fundamental coordinates unit**
The coordinate values of the workpiece zero-point in the machine coordinates system are to be input to this unit.
- Index unit**
The index angle of the milling spindle head is to be specified.
- Machining units**
Various types of machining units are provided. Data related to selection of a machining method and to machining dimensions are to be input to these units.
- Tool sequence** consists of data inputs related to the operation of a tool.
- Shape sequence** consists of data inputs related to the machining shape on a drawing.
- End unit**
Denotes the end of a program (i.e., the completion of machining).

D736P0016E

Fig. 4-5 Program structure

4-4-2 Calling up the PROGRAM display

First, the **PROGRAM** display is to be presented as previously done when displaying the **TOOL FILE** display:

- (1) Press the display selector key.
 - (2) Press the menu key **PROGRAM**.
- ➔ The **PROGRAM** display will be displayed then as shown below.



D736P0017E

The above display status is referred to as the listing mode.
 The listing mode is a program-contents check mode.
 For details, see Part 2, Section 1-1, "Listing Mode and Creating Mode."

4-4-3 Assigning workpiece numbers

Number the individual programs to be created. These numbers identify the individual programs just as part number in a part drawing identifies the individual parts. These numbers are referred to as workpiece numbers, and the desired number from 1 to 99999999 can be selected for each program. Here, workpiece No. 1234 is to be set.

- (1) Press the menu key **WORK No.**
- (2) WORK No.?

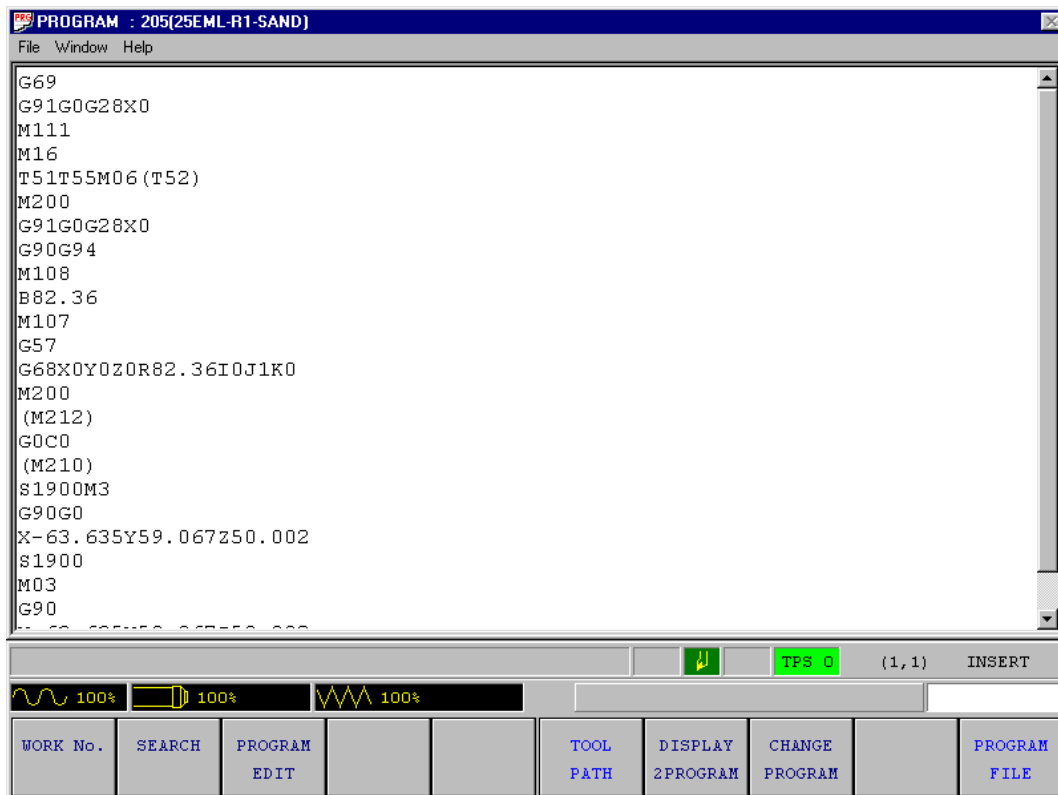
➔ The message "NEW PROGRAM <MENU>?" will be displayed.
 If either one of the two displays [1] and [2] shown below is displayed instead of this message, then this indicates that workpiece No. 1234 has already been used. In that case, input a different workpiece number.

[1] MAZATROL program



D736P0018E

[2] EIA/ISO program



D736P0019E

Note: In M640M Pro, two different programs can be created. The program [1] above is referred to as a MAZATROL program, and the one [2], as an EIA/ISO program.

4-4-4 Creating a program

Let us create the program for the machining of the workpiece shown in the figure 4-2 with the vertical tye e-Machine.

(1) After setting the workpiece number, press the menu key **MAZATROL PROGRAM**.

➔ The display shown in Figure 4-6 will be presented. The program creation will become possible. This status is referred to as the creating mode.

If the EIA/ISO programming function (option) is provided in the system, the following menu will be displayed.

WORK No.	EIA/ISO PROGRAM	MAZATROL PROGRAM							
----------	-----------------	------------------	--	--	--	--	--	--	--

(2) The following display will be presented. The program creation will become possible.



Fig. 4-6 PROGRAM display (edit mode)

D736P0020E

4-4-5 Creation of common unit

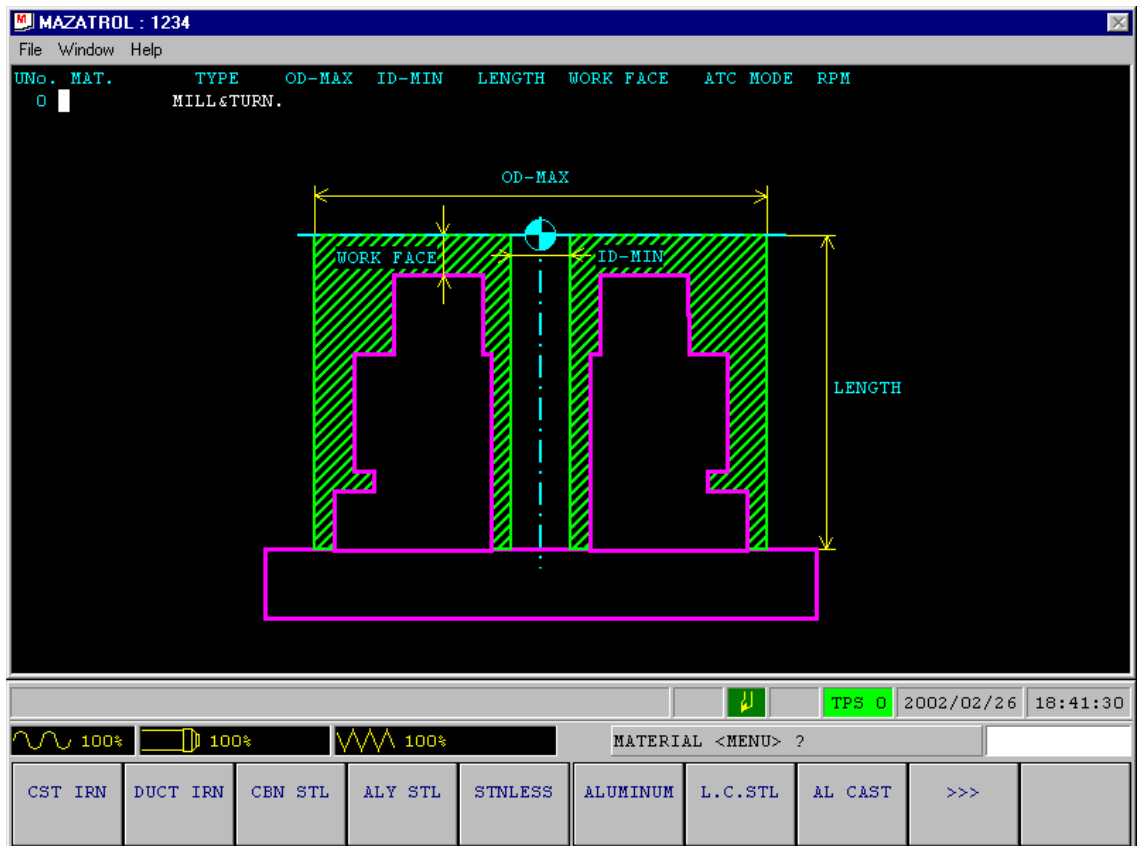
The common unit is the unit that must be created at the head of the program. In this unit, enter fundamental data relating to the entire program. Although the “workpiece scheme (MILLING & TURN.)” and the “initial-point scheme (ONLY MILLING)” are available as the entering methods, this chapter describes only examples of entry with the “initial-point scheme”.

Before describing the creation of the common unit, the MAZATROL Help function is first described below. When entering data in the unit, detailed explanatory information on data entry can be displayed. The display for calling up the information is referred to as the MAZATROL Help display.

Call up the MAZATROL Help display.

Details of the MAZATROL Help display differ according to the particular specifications of the machine.

- Press the menu key **HELP**.
The following MAZATROL Help display will be presented.



D736P0021E

Now, let's create the common unit.

MAT. Specify the material of the workpiece.

TYPE.....In this example, select the workpiece scheme.

OD-MAX.....Enter the outside diameter of the workpiece (blank).

ID-MIN.....Enter the inside diameter of the workpiece.

LENGTHEnter the length of the workpiece.

WORK FACEEnter the length of the workpiece edge.

ATC MODESpecify the mode of returning the tool to the automatic tool change position.

RPMSpecify the rotational speed limit for the turning spindle.

- MATERIAL <MENU>? **CBN STL**
- TYPE OF PROGRAM?..... **MILLING & TURN.**
- MAX. OUTER DIA. OF WORKPIECE ?
- MIN. INNER DIA. OF WORKPIECE ?
- WORKPIECE LENGTH ?
- STOCK REMOVAL OF WORK FACE ?
- ZERO RETURN <Z.X+Y:0, X+Y+Z:1> ?
- MAX. TURNING SPINDLE RPM LIMIT ?

4 PROCEDURE BEFORE PERFORMING THE MACHINING

The common unit will be created as shown below.

When the unit is created, the MAZATROL Help display will be closed and the program will await data input to the next unit.



D736P0022E

4-4-6 Creation of fundamental coordinates system unit

Let us set the fundamental coordinates (the coordinate values of the workpiece zero-point in the machine coordinate system). This data is to be input using the coordinates measuring function (refer to Part 2) after workpiece has been mounted on the machine. Here, input the data on the assumption that workpiece has been mounted in the position shown below.

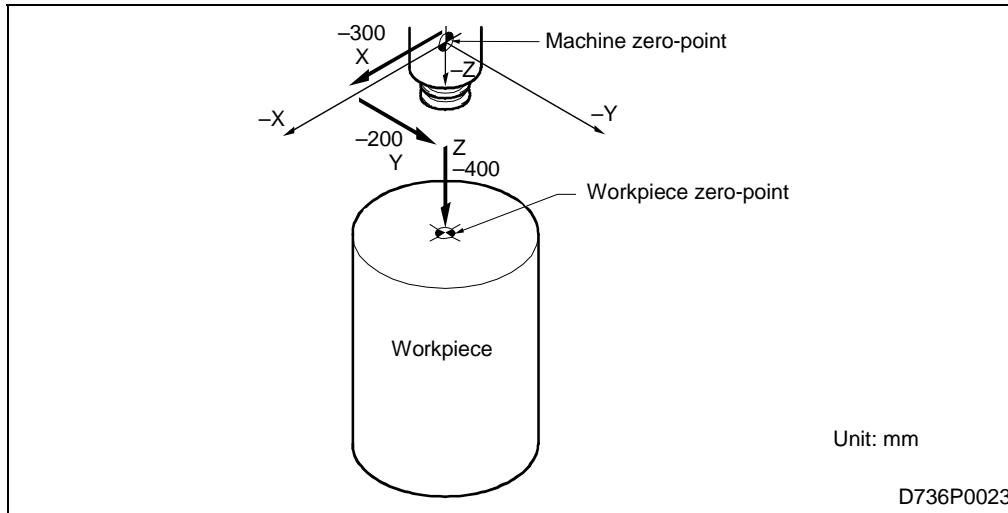
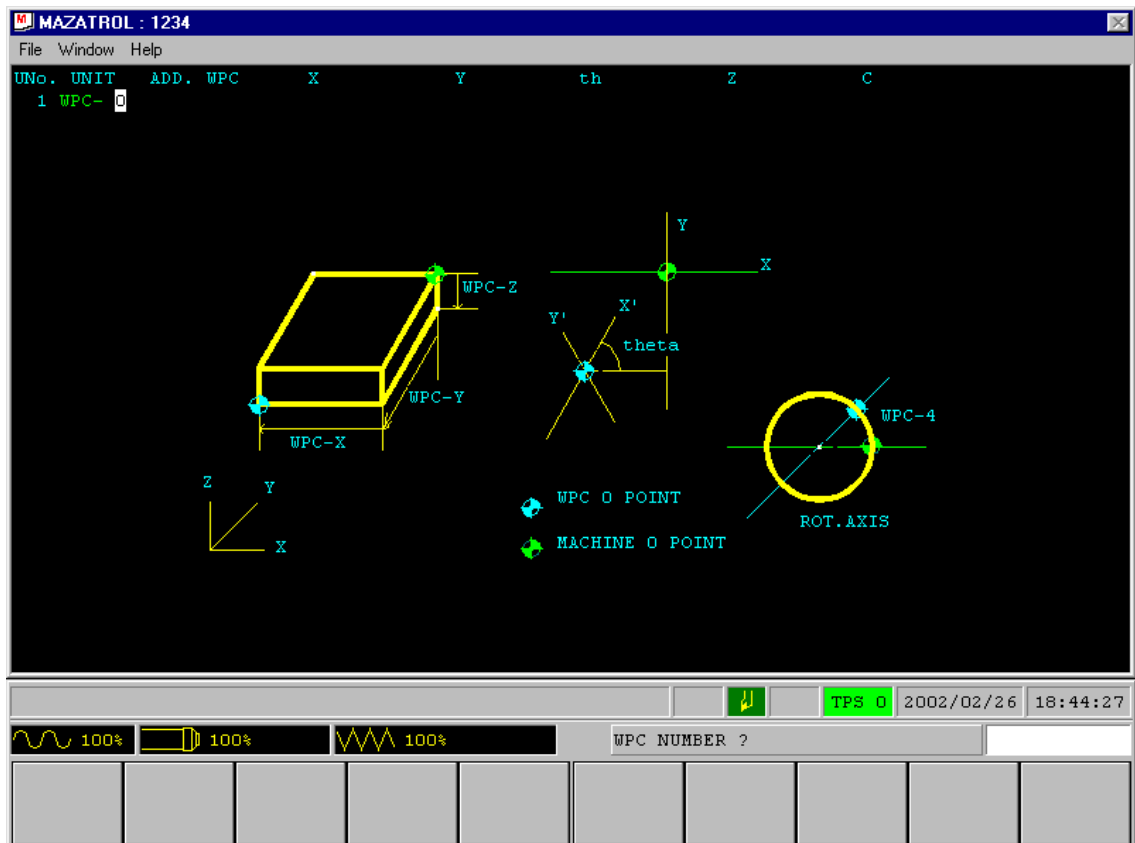


Fig. 4-7 Workpiece mounting position

- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **WPC**.
- (2) Press the menu key **HELP**.
 - ➔ The following MAZATROL Help display will be presented. "WPC" is the abbreviation of the fundamental coordinates.



D736P0024E

4 PROCEDURE BEFORE PERFORMING THE MACHINING

(3) WPC NUMBER? **1**

Next, although the coordinates of the workpiece zero-point in the machine coordinate system are to be set, if the workpiece zero-point is present in the turning center, the coordinates of the workpiece zero-point can likewise be specified by using the menu function **T.CENTER AUTO**. This menu key is used in the example here.

(4) WORKPIECE COORDINATE, WPC-X? **T. CENTER AUTO**

(5) WORKPIECE COORDINATE, WPC-Z? **- 4 0 0**

The fundamental coordinates system unit will be created as shown below.

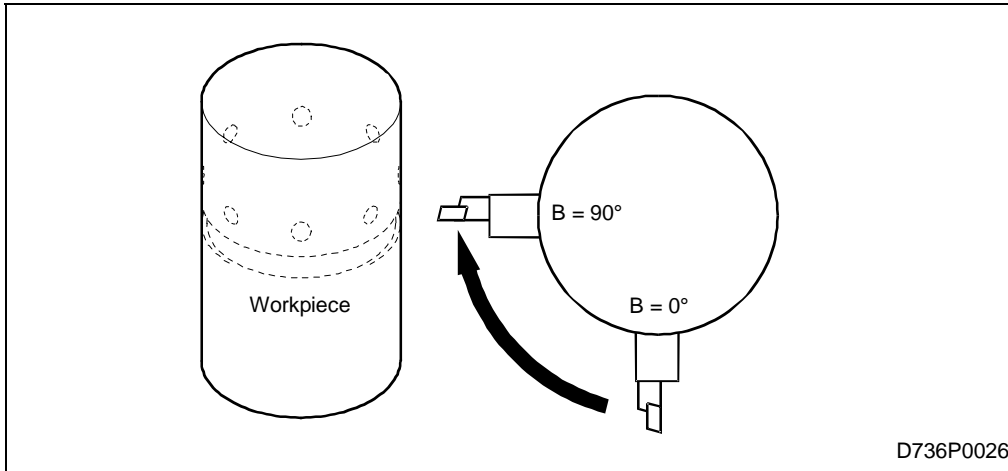
When the unit is created, the MAZATROL Help display will be closed and the program will await data input to the next unit.



D736P0025E

4-4-7 Index unit

The milling spindle head must be indexed to machine the cylindrical surface of the workpiece.



(1) While the message “MACHINING UNIT <MENU>?” is displayed, press the menu key >>>, and the menu key **INDEX**.

(2) B, C-AXIS INDEX POSITION-X?.....

(3) B, C-AXIS INDEX POSITION-Y?.....

(4) B, C-AXIS INDEX POSITION-Z?.....

To specify the index position, set the machine coordinates in steps (2), (3), and (4).

(5) ROTATION ANGLE FOR B AXIS?

(6) ROTATION ANGLE FOR C AXIS?.....

4 PROCEDURE BEFORE PERFORMING THE MACHINING

The index unit will be created as shown below.

The program will await data input to the next unit.

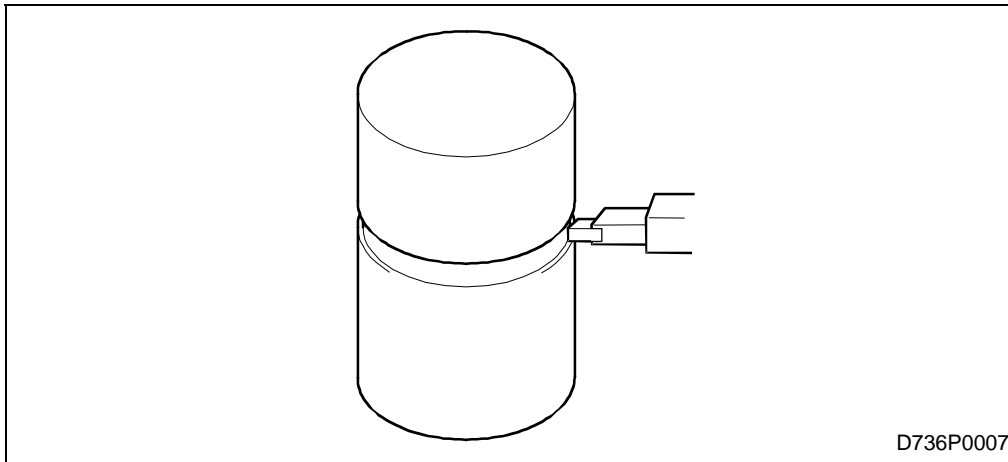


D736P0027E

The next operation required is to create a turning unit.

4-4-8 Turning unit

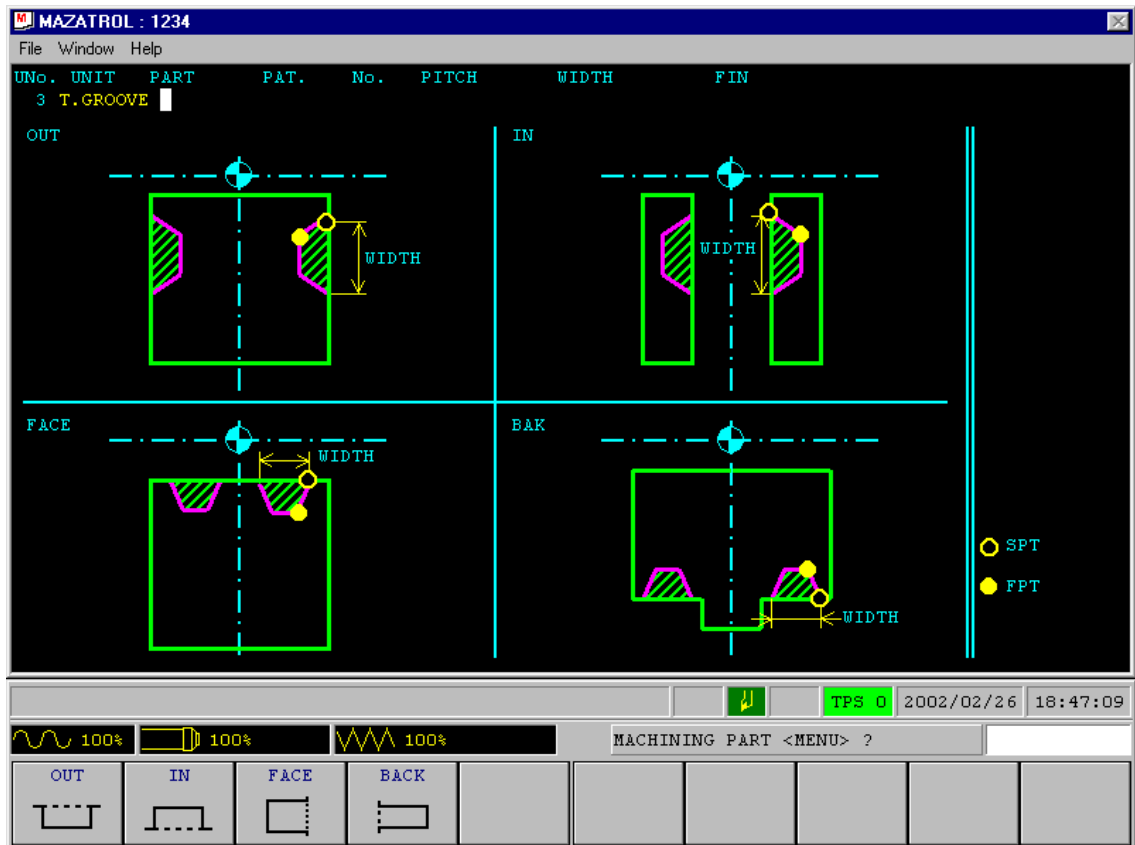
Create the unit for performing grooving operations on the cylindrical surface of the workpiece. This turning unit is referred to as the grooving unit.



D736P0007

1. Creation of a turning unit

- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **TURNING**.
- (2) MACHINING UNIT <MENU>? **T. GROOVE**
- (3) Press the menu key **HELP**.
 - ➔ The following MAZATROL Help display is presented.

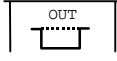
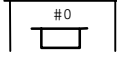





D736P0028E

4 PROCEDURE BEFORE PERFORMING THE MACHINING

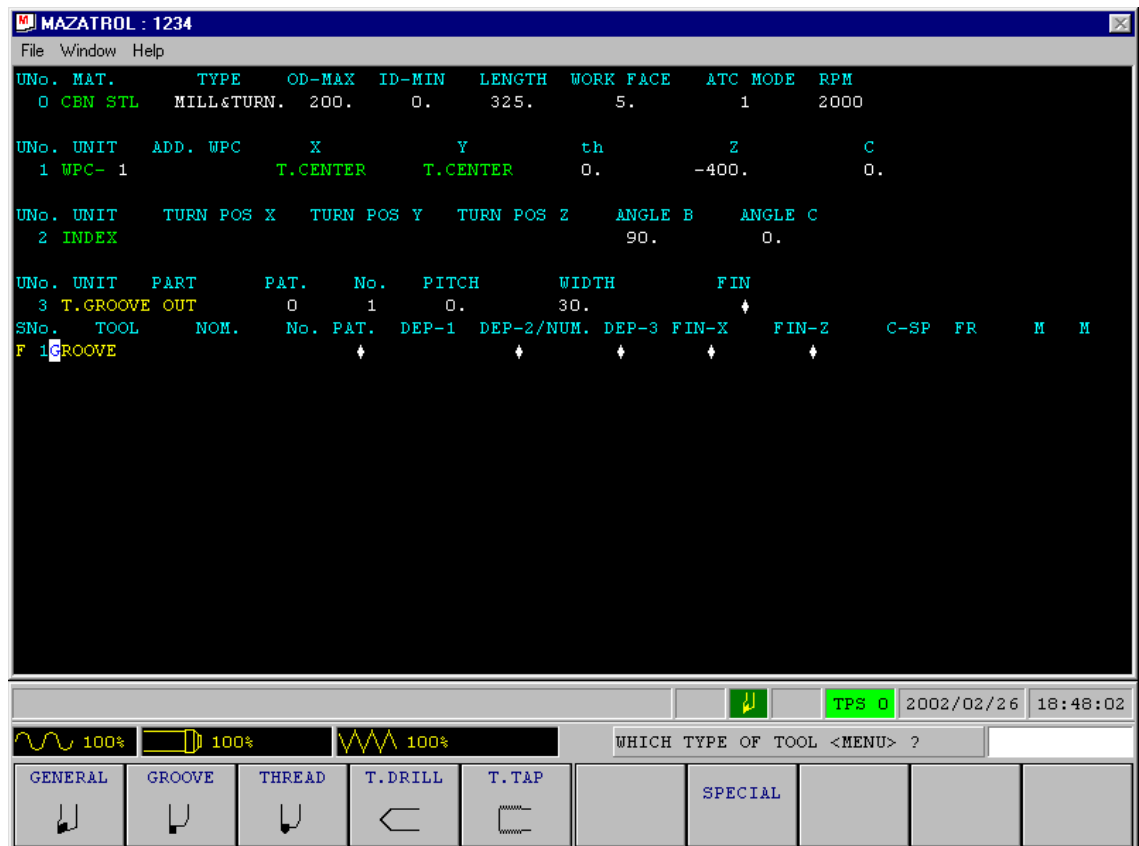
Now, let's create the grooving unit. For this unit, enter data for the item PART, PAT., No., PITCH, WIDTH, and FIN.

- PART Specify the surface to be machined.
- PAT..... Specify the shape pattern for grooving.
- No. Specify the number of grooves to be machined.
- PITCH..... Specify the grooving pitch when machining multiple grooves.
- WIDTH..... Specify the groove width.
- FIN..... Specify the thickness of the section to be chipped during the finishing process.
When the roughness level of the bottom is specified using either finishing symbol listed in the menu, FIN-Z will be set automatically.

- (4) MACHINING UNIT <MENU>? 
- (5) GROOVE CUTTING PATTERN <MENU>? 
- (6) NUMBER OF GROOVES? 
- (7) GROOVING PITCH? 
- (8) WIDTH OF GROOVE? 

(There is no need to specify the finishing allowance for a groove pattern of "0".)








The turning unit shown below has been created. The tool to be used in the tool sequence is automatically selected in accordance with the data of the turning unit. In the following example, a grooving tool is selected:



D736P0029E

2. Creation of a tool sequence

Now, let's create the tool sequence.

- (1) While the message "WHICH TYPE OF TOOL <MENU>?" is displayed, press the cursor key .
- (2) MACHINING PART OF TOOL <MENU>? **OUT OUTER DIAMETER**
- (3) NOMINAL OF TOOL?..... 
 (This set value is the identification number for the tool. Use this number according to the particular needs of the user.)
- (4) TOOL ID CODE <MENU>? **A**
- (5) MACHINING PRIORITY No.?..... 
 (Skip to the next item since the priority function for the same tool is not to be used here. See Part 2, Chapter 5.)
- (6) MAX DEPTH OF CUT <AUTO→MENU>? **CARBIDEL AUTO**
 (This example assumes the use of a cemented carbide grooving tool.)
- (7) CUTTING SPEED, AUTO→<MENU>? 
- (8) FEEDRATE, AUTO→<MENU>? 
 (The surface velocity and the feed rate are automatically determined at the same time the maximum cutting stroke is auto-set.)
- (9) M CODE?..... 
- (10) M CODE?..... 

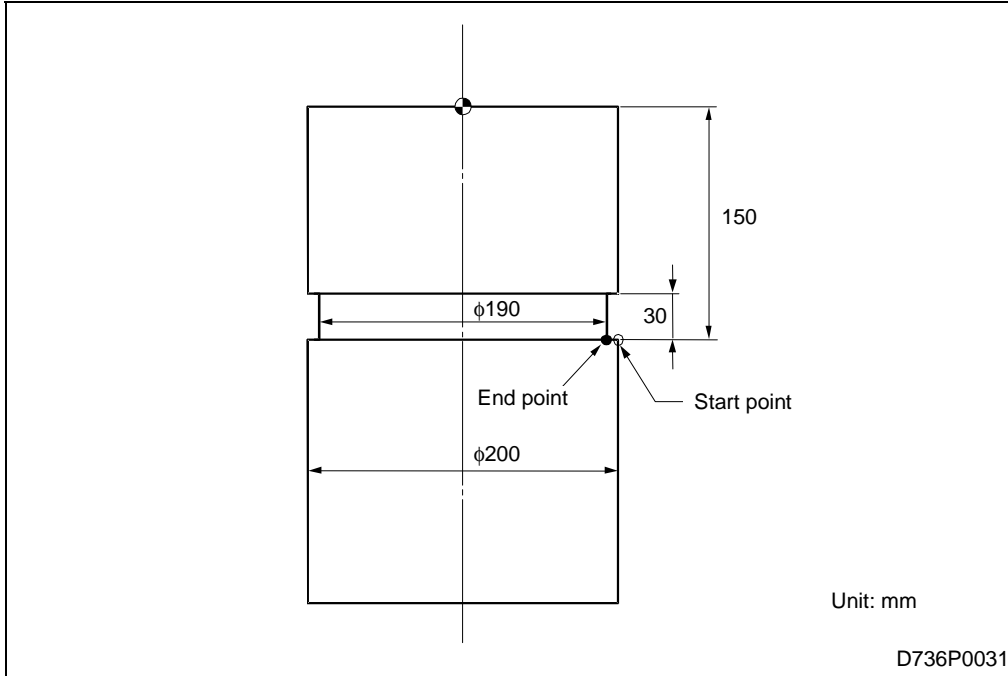
The tool sequence shown below has been created.



3. Creating a shape sequence

Next, create a shape sequence.

Read the desired machining area from the drawing, and enter the necessary coordinates for the shape of the section to be machined. For grooving, enter the dimensions of such a shape as shown in the following figure:



- (1) STARTING POINT-X<AUTO→MENU>?.....
- (2) STARTING POINT-Z<AUTO→MENU>?.....
- (3) FINAL POINT-X <AUTO → MENU>?.....
- (4) FINAL POINT-Z <AUTO → MENU>?.....

The shape sequence shown below has been created.

MAZATROL : 1234

File Window Help

UNo.	MAT.	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
0	CBN STL	MILL&TURN.	200.	0.	325.	5.	1	2000

UNo.	UNIT	ADD. WPC	X	Y	th	Z	C
1	WPC- 1		T.CENTER	T.CENTER	0.	-400.	0.

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C
2	INDEX				90.	0.

UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FIN
3	T.GROOVE	OUT	0	1	0.	30.	

SMo.	TOOL	NOM.	No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
F 1	GROOVE	OUT	30.	A	2.					108	0.1		

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANG	RGH
1		200.	150.	190.	150.			

UNo.	UNIT
4	

TPS 0 2002/02/26 18:49:51

MACHINING UNIT <MENU> ?

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

D736P0032E

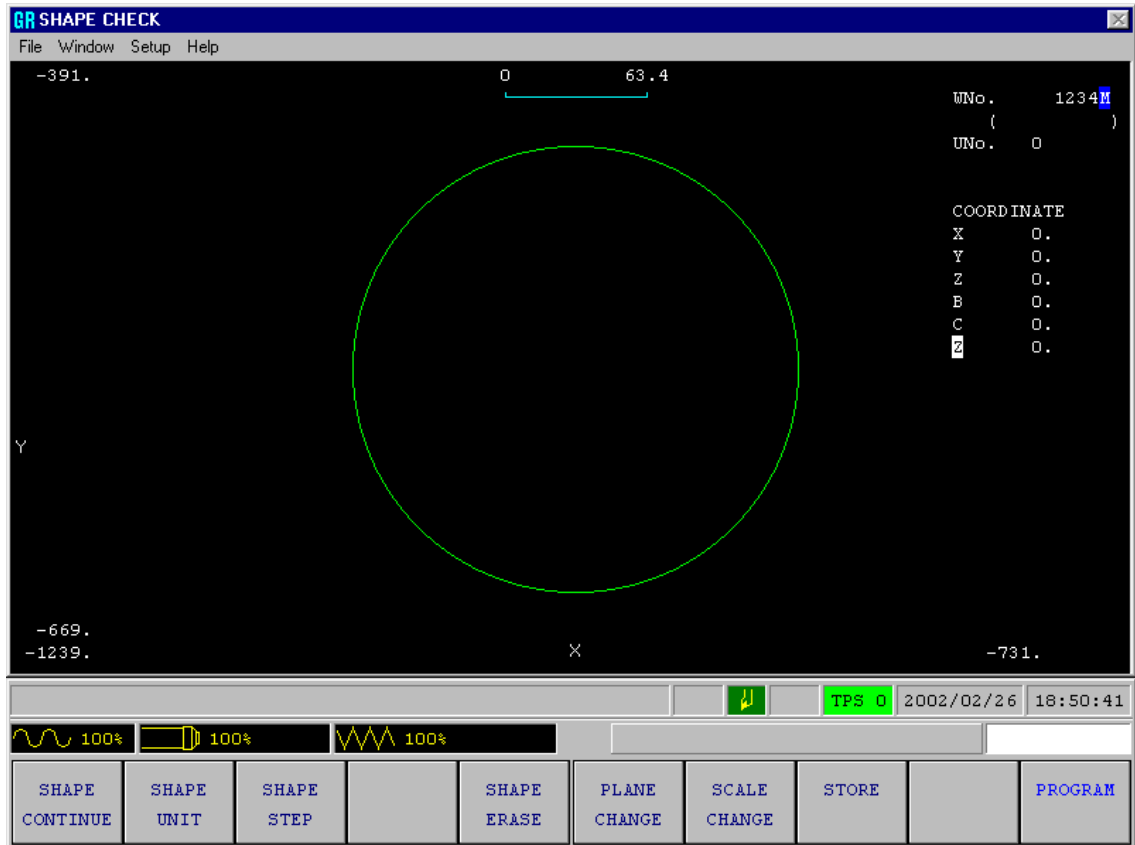
4. Shape check

Check the **SHAPE CHECK** display to ensure that no errors are included in the input data.

Of the various functions available with the **SHAPE CHECK** display, only those for continuous display of shapes, for erasure of shapes, and for step display of shapes are described here. See Section 6-3, "SHAPE CHECK Display" of the Operating Manual for details of other functions.

(1) Press the menu key **SHAPE CHECK**.

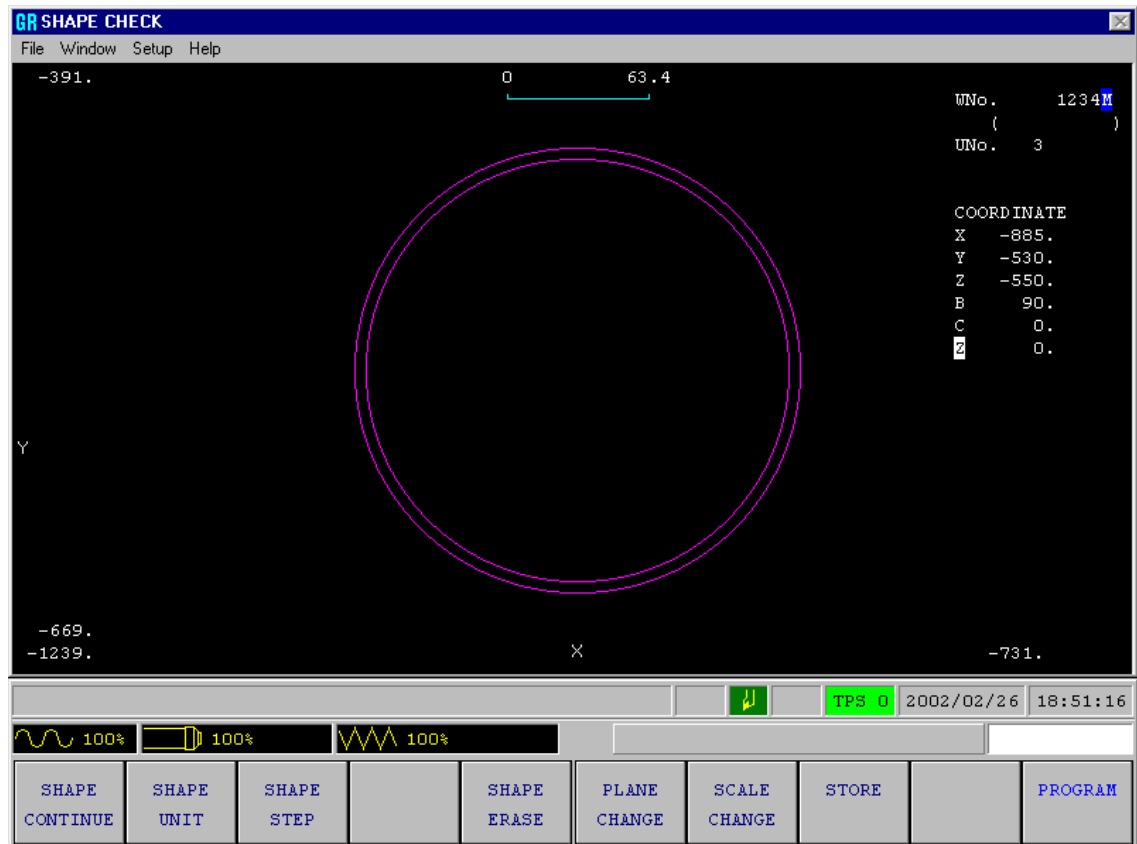
➔ The following **SHAPE CHECK** display will be displayed.



D736P0033E

(2) Press the menu key **SHAPE CONTINUE** to display shapes continuously.

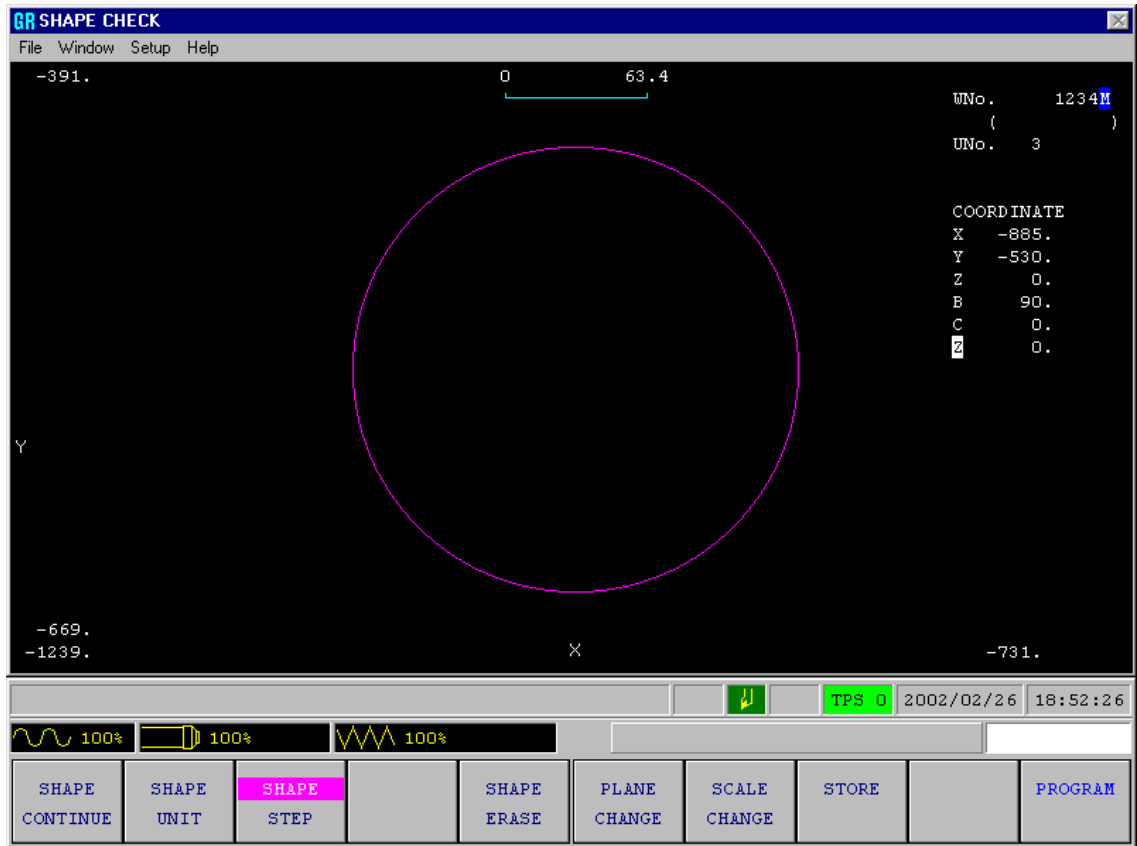
→ Shapes will be displayed continuously. The shapes displayed on the screen will be those of the portions which are chamfered by machining.



D736P0034E

(3) Press the menu key **SHAPE ERASE** to erase the shape.

- (4) Repeatedly press the menu key **SHAPE STEP** several times to call shapes step by step on the screen.
 - The reverse display state of the menu will be cleared when the final group of shapes is displayed.



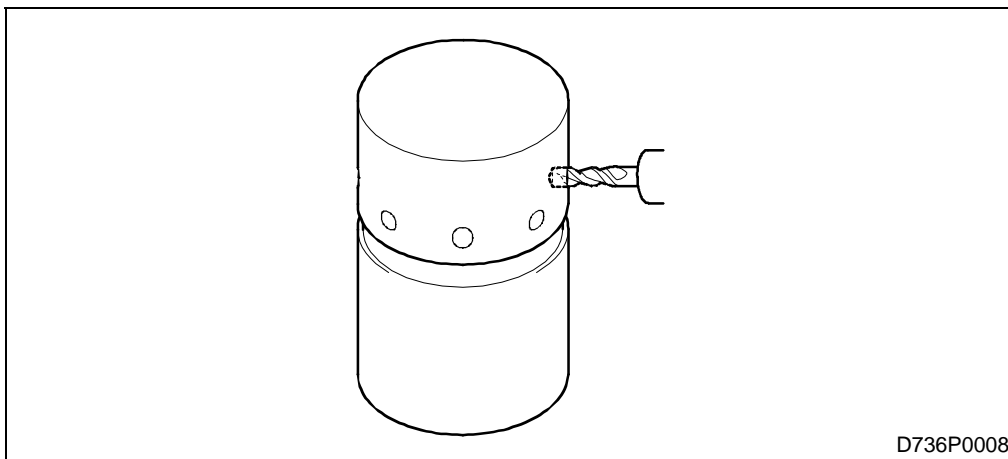
D736P0035E

- (5) Press the menu key **PROGRAM** to resume the **PROGRAM** display.

This completes a grooving unit in turning mode. The next operation required is to create a C-axis point machining unit.

4-4-9 C-axis point machining unit


The creation of the unit for drilling eight 20mm-diameter holes along the cylindrical surface of the workpiece is described below. This machining unit is referred to as the C-axis drilling unit.



D736P0008

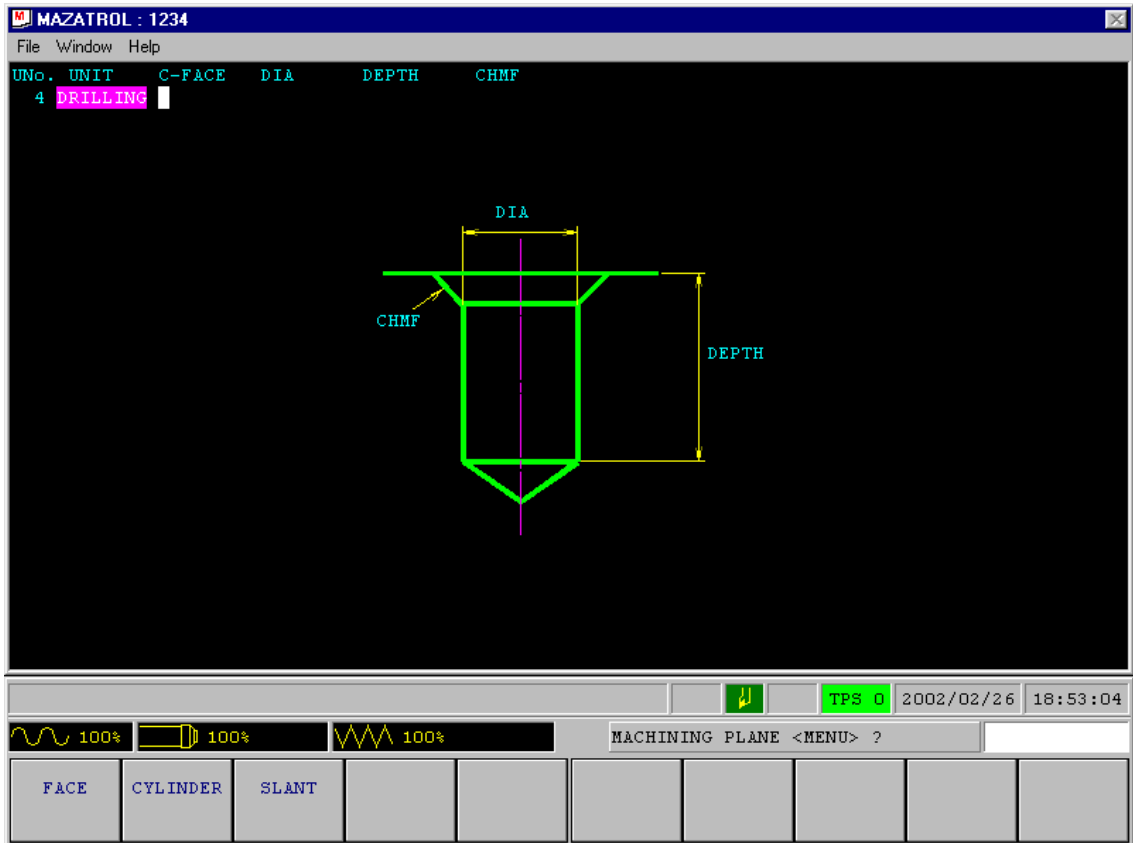
1. Creation of a C-axis point machining unit

(1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key >>>, and the menu key **C-POINT MACH-ING**.

(2) MACHINING UNIT <MENU>? 

(3) Press the menu key **HELP**.

➔ The following MAZAROL Help display will be displayed.



D736P0036E

Now, let's create the C-axis point machining unit. For this unit, enter data for the item C-FACE, DIA, DEPTH, and CHMF.

(4) MACHINING PLANE <MENU>?..... **CYLINDER**
Specify the surface to be machined. In the example here, holes are drilled along the cylindrical surface.

(5) HOLE DIAMETER?.....

(6) HOLE DEPTH?

(7) CHAMFER WIDTH?

4 PROCEDURE BEFORE PERFORMING THE MACHINING

A C-axis point machining unit such as that shown below is now completed. This shows that a spot and a drill have been automatically selected for the tool sequence that is to be programmed.



D736P0037E

2. Creation of a tool sequence

For the point machining unit, it is necessary just to set the cutting parameters (circumferential speed and feedrate) since most of the tool sequence data are set automatically.

- (1) While the message "WHICH TYPE OF TOOL <MENU>?" is displayed, move the cursor to the position under the item C-SP by pressing the cursor key several times in succession.
- (2) CUTTING SPEED, AUTO→<MENU>? **HSS AUTO**
- (3) FEEDRATE, AUTO→<MENU>? **HSS AUTO**
- (4) M CODE?.....
- (5) M CODE?.....
- (6) Press the cursor key several times in succession to move the cursor to the position under the item C-SP.
- (7) CUTTING SPEED, AUTO→<MENU>? **HSS AUTO**
- (8) FEEDRATE, AUTO→<MENU>? **HSS AUTO**
- (9) M CODE?.....
- (10) M CODE?.....

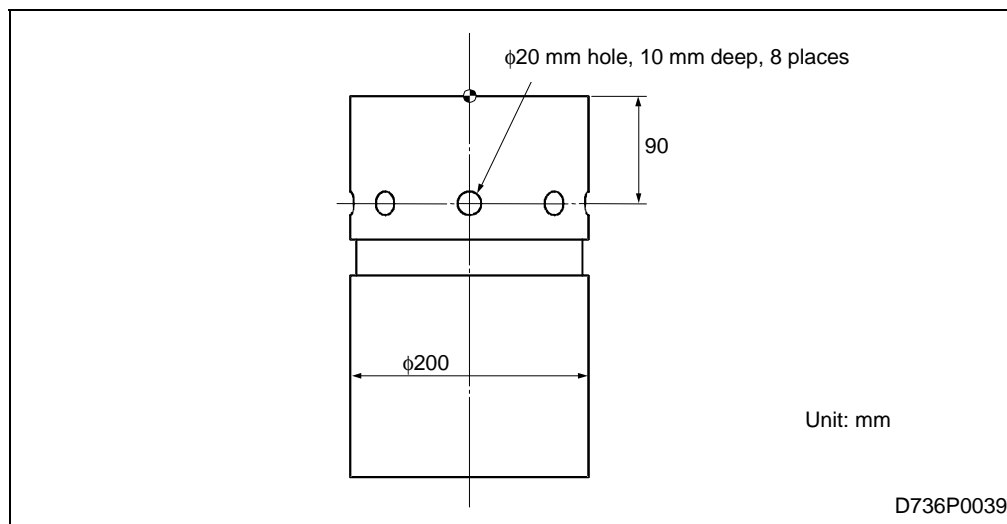
The tool sequence shown below has been created.



D736P0038E

3. Creation of shape sequence

Next, create a shape sequence. For drilling, enter the coordinates of the hole center as shown below.



D736P0039


(1) While the message "POINT CUTTING PATTERN <MENU>?" is displayed, press the menu key **CIRCLE**.

(2) STARTING POINT-R?


(3) STARTING POINT-theta?

(4) STARTING POINT-X?

4 PROCEDURE BEFORE PERFORMING THE MACHINING

(5) NUMBER OF HOLES? **8** 

Specify the number of holes to be drilled along the cylindrical surface.

(6) RETURN POSITION <INIT:0, R:1>? **1** 

Specify the height of the tool path to the next hole. For the details, see the descriptions about the positions of the initial point and referential point defined in the tool path in Section 4-6, Part 2.

(7) POINT CUTTING PATTERN <MENU>? **SHAPE END**

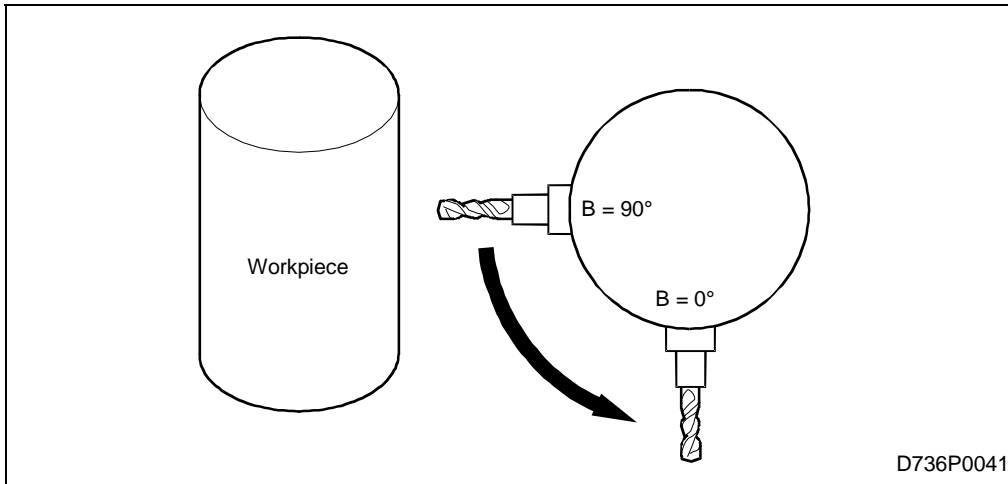
The shape sequence shown below has been created.



D736P0040E

Next, let's create an index unit.

Description of machining up to the above phase assumes a head index angle of 90 degrees. Description of subsequent processes, however, assumes a head index angle of 0 degrees. Description of the unit creating method is omitted since it has already been explained.



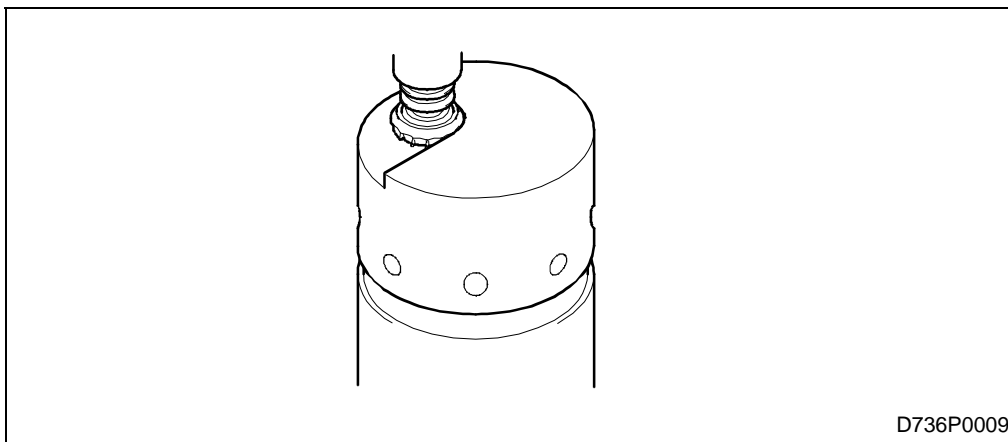
Create the index unit as follows to set the head index angle to 0 degrees:



D736P0042E

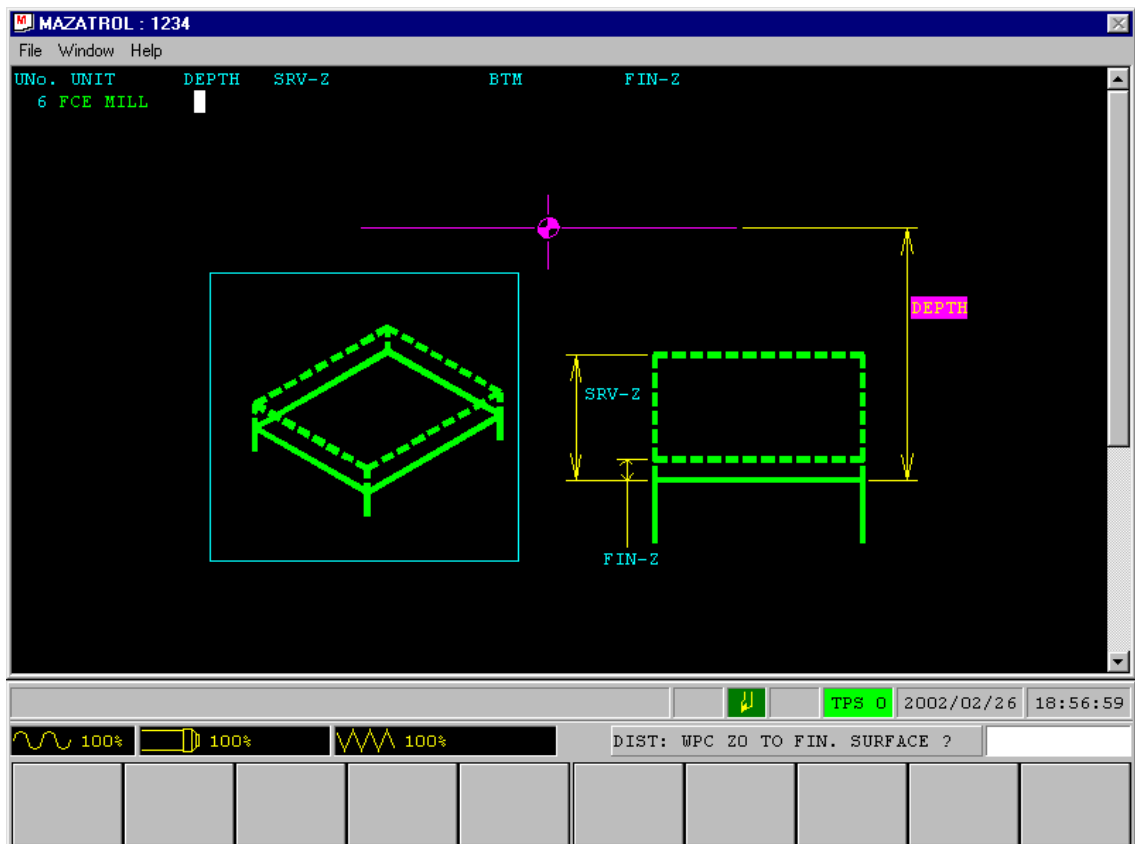
4-4-10 Face machining unit

Let us program a unit that mills the top face of a particular material. This machining unit is referred to as a face milling unit.



1. Creation of face machining unit

- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **FACE MACH-ING**.
- (2) MACHINING UNIT <MENU>? **FACE MIL**
- (3) Press the menu key **HELP**.
 - ➔ The following help window is presented.



D736P0043E

Now, let's create the face machining unit. For this unit, enter data for the item DEPTH, SRV-Z, BTM, and FIN-Z.

DEPTH..... Input the distance from the workpiece zero-point of the Z-axis to the surface to be finished.

Here, the depth is 0 because the workpiece zero-point is on the finishing surface. (See the figure below.)

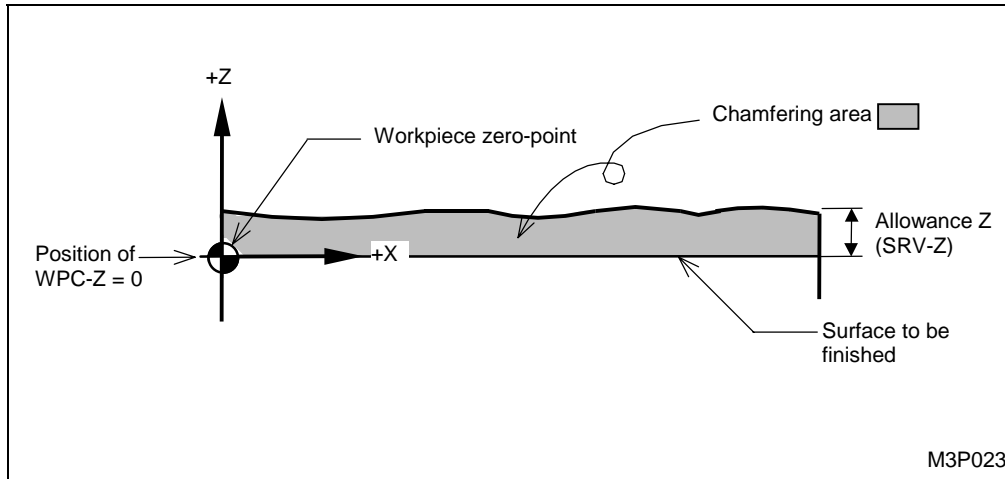


Fig. 4-8 DEPTH and SRV-Z

SRV-Z Specify the thickness of the section to be chipped. An approximate value may be input because the workpiece surface is not plane. (See the figure above.)

BTM Input an appropriate finishing code that designates the roughness of the workpiece surface to be finished.

FIN-Z..... Specify the thickness of the section to be chipped during the finishing process. This data will be automatically set if the roughness of the bottom is specified using a finishing code of the menu.

- (4) DIST: WPC Z0 TO FIN. SURFACE? 0 [↔]
- (5) Z AXIS STOCK REMOVAL? 5 [↔]
- (6) BOTTOM FACE ROUGHNESS <MENU>? [▼ 2]
- (7) FINISH ALLOWANCE Z? [↓]

(When the bottom roughness is input, the FIN-Z value will be set automatically. Thus, the operator can proceed directly to the next data item using the appropriate cursor key.)


4 PROCEDURE BEFORE PERFORMING THE MACHINING

A face machining unit such as that shown below is now completed. Usually, a tool appropriate for the particular tool sequence is automatically selected on the basis of the data of the face machining unit. Here, a face mill has been selected.



D736P0044E

2. Creation of a tool sequence

- (1) While the message "WHICH TYPE OF TOOL <MENU>?" is displayed, press the cursor key .
- (2) Press the menu key **TOOL.F WINDOW** to call the **TOOL FILE** window on the screen, and check that the previously registered tool is included in the display.
 - ➔ The following **TOOL FILE** window shows that the 100 mm diameter face mill is registered.



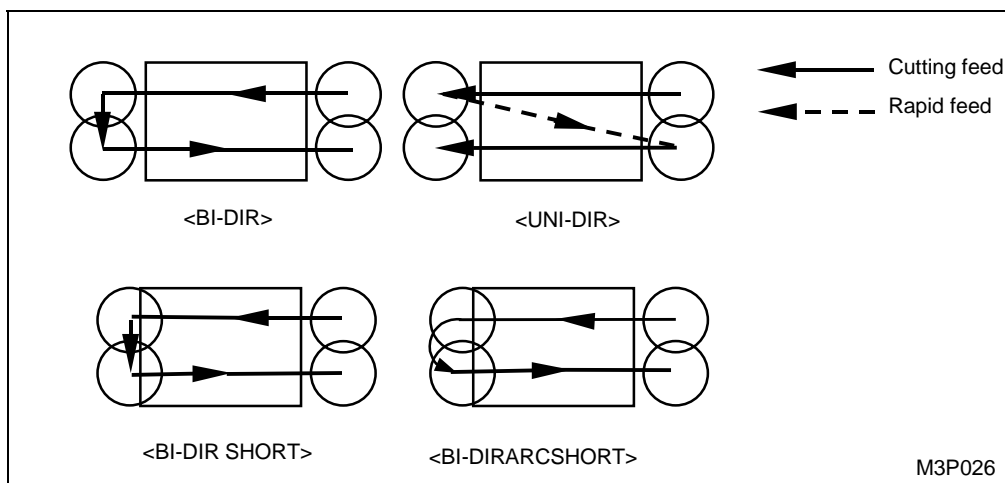
D736P0045E

- (3) NOMINAL DIAMETER?
- (4) TOOL FILE CODE? **A**
- (5) MACHINING PRIORITY No.?



(Skip to the next item since the identical-tool priority function is not used here. Refer to Part 2, Chapter 5.)

- (6) APPROACH POINT X, AUTO → <MENU>? **AUTO SET**
- (7) APPROACH POINT Y, AUTO → <MENU>? **AUTO SET**
- (8) CUTTING DIRECTION <MENU>? **X BI-DIR**

The possible directions of cutting are shown below.



4 PROCEDURE BEFORE PERFORMING THE MACHINING

- (9) DEPTH OF CUT? **AUTO SET**
- (10) WIDTH OF CUT?..... **AUTO SET**
- (11) CUTTING SPEED, AUTO → <MENU>? **AUTO SET**
- (12) FEEDRATE, AUTO → <MENU>? **AUTO SET**
- (13) M CODE? 
- (14) M CODE?..... 

A tool sequence such as that shown below is now completed. For most of the data items, the optimal data is set by depressing the menu key **AUTO SET**. The data of the item APRCH-X, APRCH-Y, however, maintains ? since the necessary data is automatically set upon completion of the tool path check.



D736P0046E

3. Cutting parameters

The cutting parameters that have been automatically set in procedural step (12) and (13) of creation of tool sequence above are described here.

The speed at which a tool revolves (circumferential speed) and the rate at which the tool is fed (feed rate) are referred to as cutting parameters.

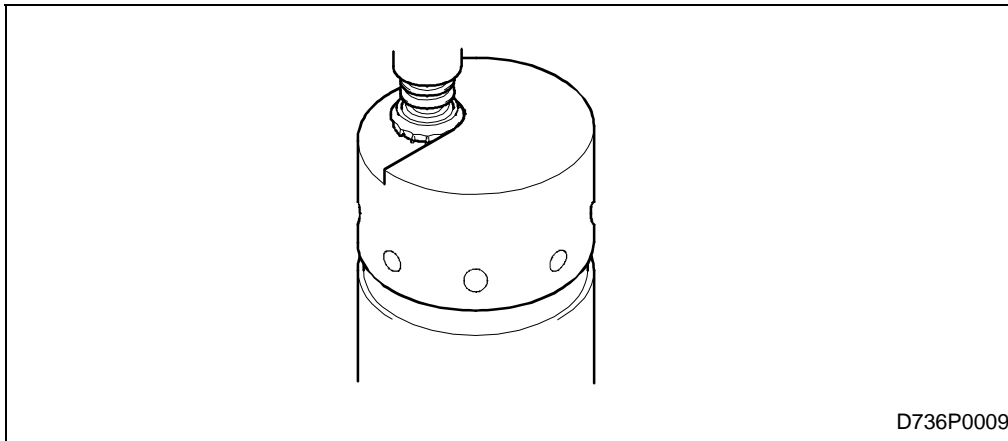
Circumferential speed (C-SP):

Speed (m/min) at which a tool moves on the outer circumference. The relationship between the revolutions per minute and circumferential speed is given by the following equation.

$$\text{Circumferential speed (m/min)} \times \frac{\text{Tool diameter (mm)} \times \text{Revolutions per minute (rpm)} \times 3.14}{1000}$$

Feedrate (FR):

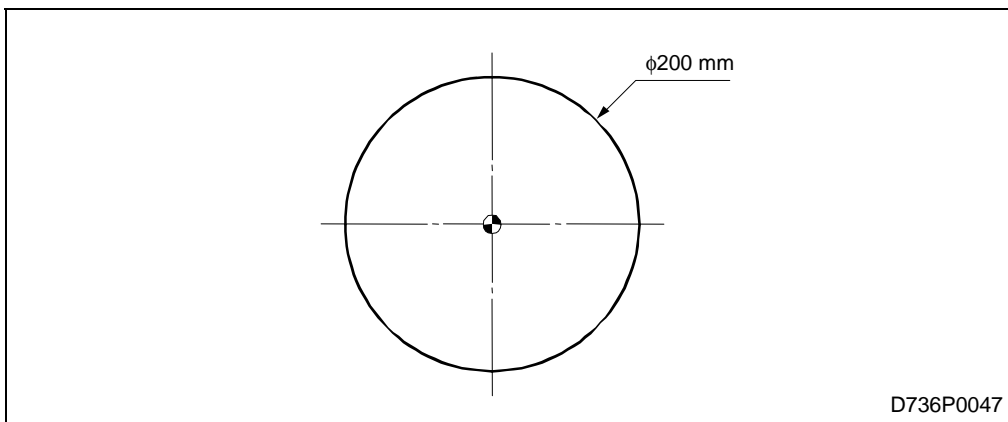
The distance (mm/rev) through which workpiece is cut during one revolution of the tool. This feedrate is called synchronous feedrate.



4. Creation of a shape sequence

Next, let us create a shape sequence.

Read the desired machining area from the drawing, and enter the necessary coordinates for the shape of the section to be machined. For face milling, enter the dimensions of such a shape as shown in the following figure:

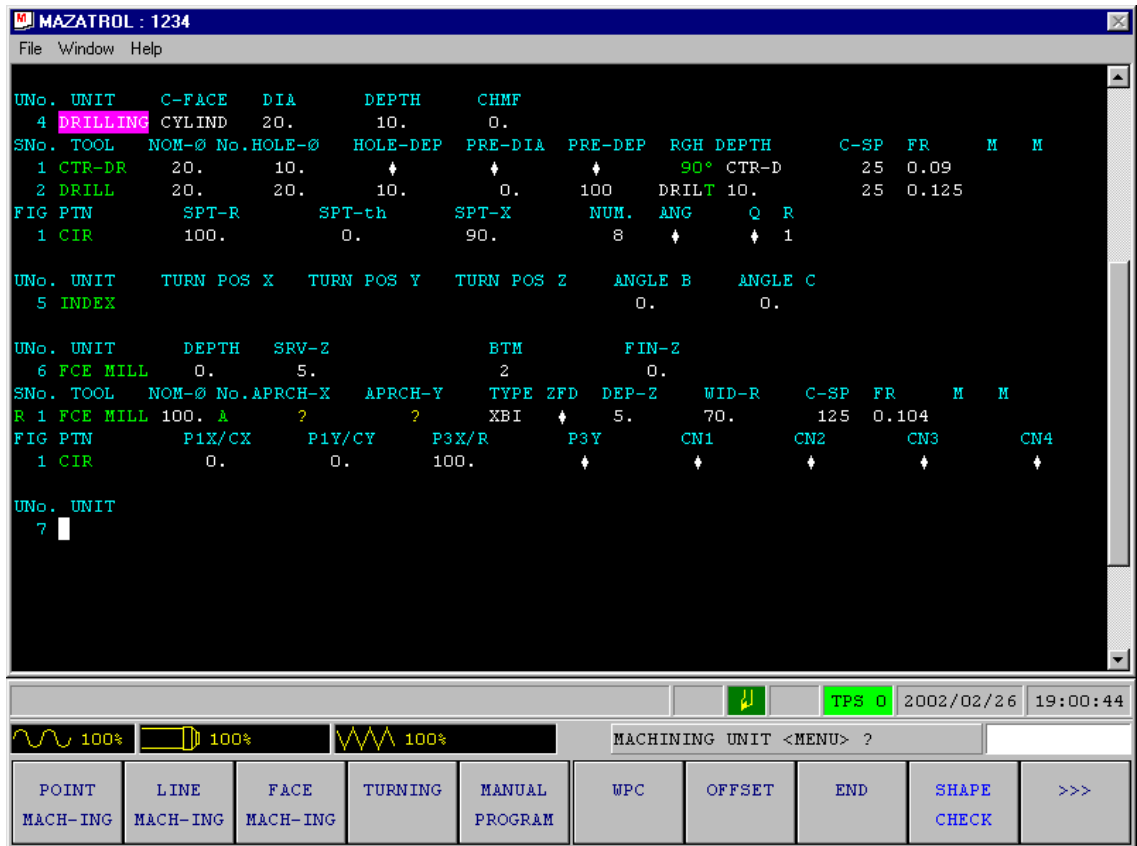


(1) While the message "PATTERN OF FIGURE <MENU>?" is displayed, press the menu key **CIRCLE**.

(2) CIRCLE CENTER X?

- (3) CIRCLE CENTER Y?.....
- (4) CIRCLE RADIUS R?.....

The shape sequence shown below has been created.



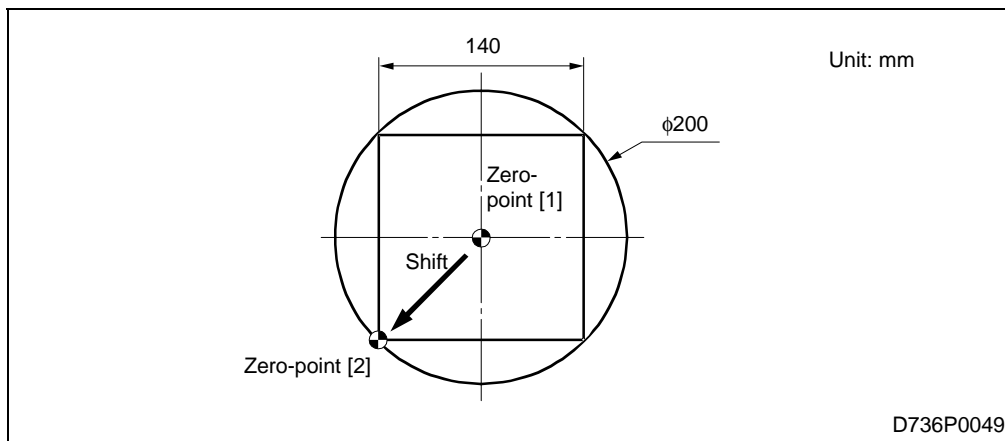
D736P0048E

This completes a face milling unit for use of the face machining. The next operation required is to create a fundamental coordinate shifting unit.

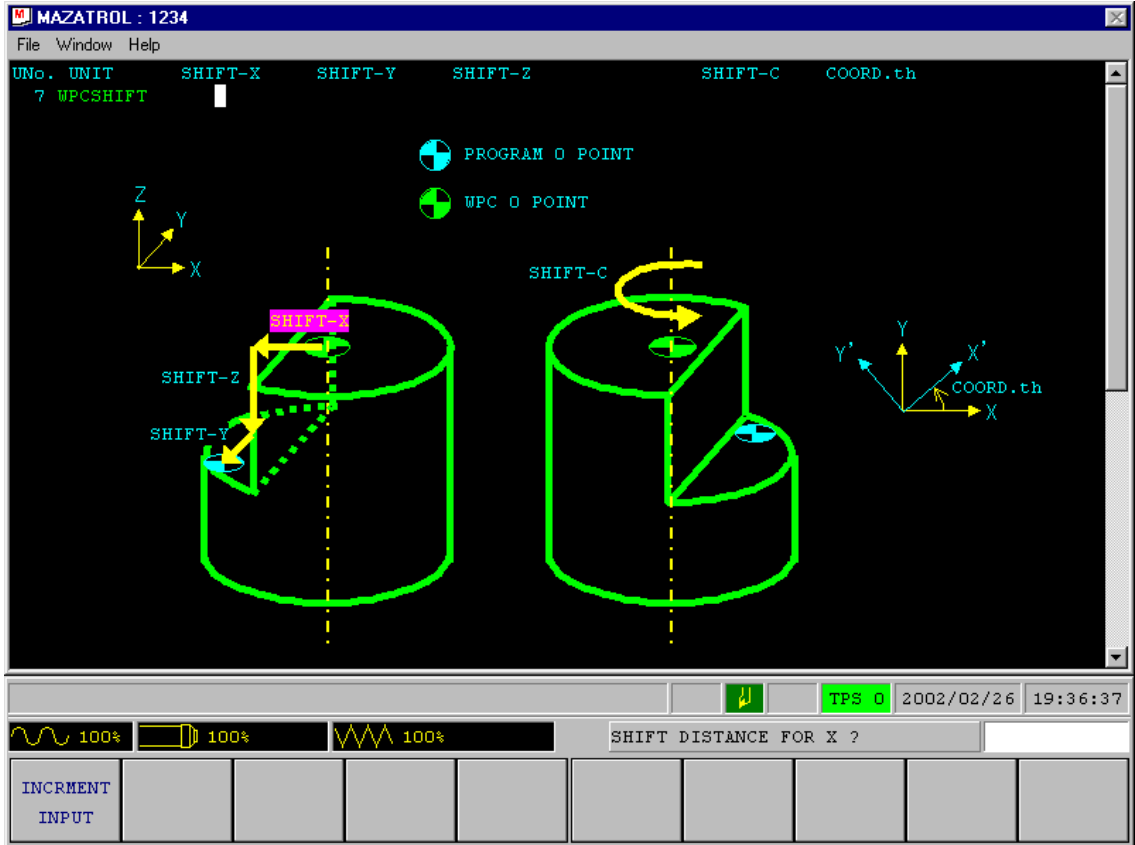
4-4-11 Fundamental coordinate shifting unit

The fundamental coordinate shifting unit shifts the program zero-point.

In the example here, the program Y zero-point is shifted to the position indicated in the figure below.



- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key >>> two times and the menu key **WPC SHIFT**.
- (2) Press the menu key **HELP**.
 - ➔ The following MAZATROL Help display appears. The abbreviation code "WPC SHIFT" on the display denotes fundamental coordinates shifting.



D736P0050E

- (3) SHIFT DISTANCE FOR X? - 7 0 [↔]
- (4) SHIFT DISTANCE FOR Y? - 7 0 [↔]
- (5) SHIFT DISTANCE FOR Z? 0 [↔]
- (6) SHIFT ANGLE? 0 [↔]
- (7) COORDINATE, theta? 0 [↔]

The fundamental coordinate shifting unit will be created as shown below.

4 PROCEDURE BEFORE PERFORMING THE MACHINING

When the unit is created, the MAZATROL Help display will be closed and the program will await data input to the next unit.

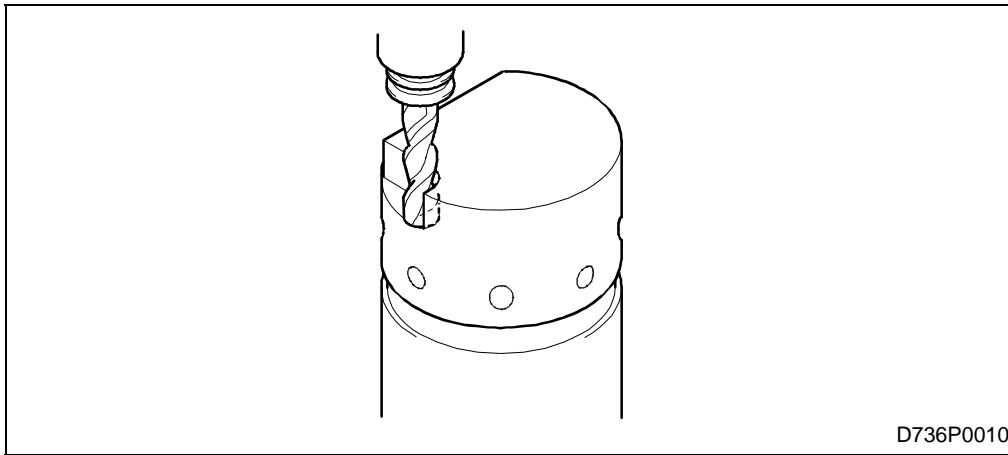


D736P0051E

The next operation required is to create a line machining unit.

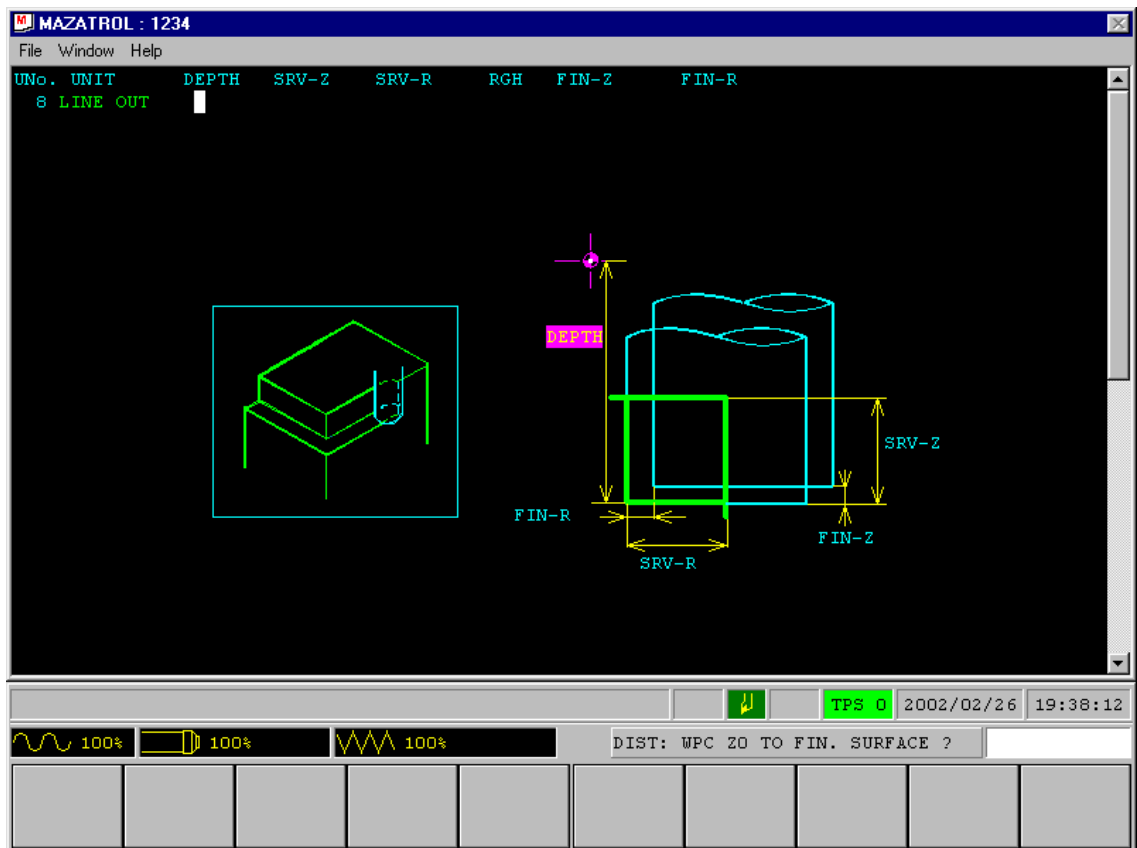
4-4-12 Line machining unit

Let us create a unit that steps the edges of a material by contour machining. This machining unit is referred to as an outside linear machining.



1. Creation of line machining unit

- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **LINE MACH-ING**.
- (2) MACHINING UNIT <MENU>? **LINE OUT**
- (3) Press the menu key **HELP**.
 - The MAZATROL Help display will be displayed.



D736P0052E

4 PROCEDURE BEFORE PERFORMING THE MACHINING

Now, let's create the line machining unit. For this unit, enter data for the item DEPTH, SRV-Z, SRV-R and RGH.

DEPTH..... Enter a depth value of 20.

SRV-Z Enter the wall thickness of the workpiece through which it is to be chamfered in the Z axial direction from the face milled surface (see figure below).

SRV-R..... Enter the wall thickness of the workpiece through which it is to be chamfered in the radial direction.


- (4) DIST: WPC Z0 TO FIN. SURFACE?
- (5) Z AXIS STOCK REMOVAL?
- (6) X/Y AXIS STOCK REMOVAL?
- (7) SURFACE ROUGHNESS <MENU>?.....
- (8) FINISH ALLOWANCE Z?
- (9) FINISH ALLOWANCE R?

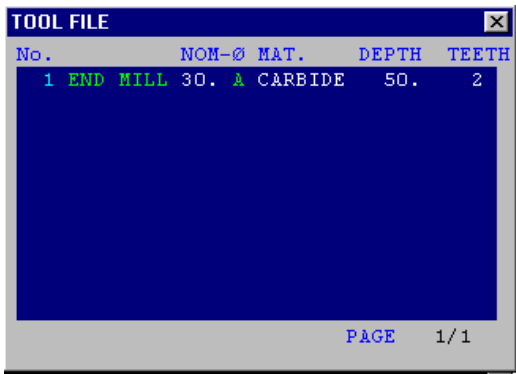
A line machining unit such as that shown below is now completed. This display shows that an end mill has been automatically selected for the particular tool sequence that is to be programmed.







D736P0053E

2. Creation of a tool sequence




- (1) While the message "WHICH TYPE OF TOOL <MENU>?" is displayed, press the cursor key .
- (2) Press the menu key **TOOL F. WINDOW** to call the **TOOL FILE** display on the screen, and check that the previously registered tools are included in the display.
 - ➔ The following **TOOL FILE** window shows that the 30-mm diameter end mill is registered.



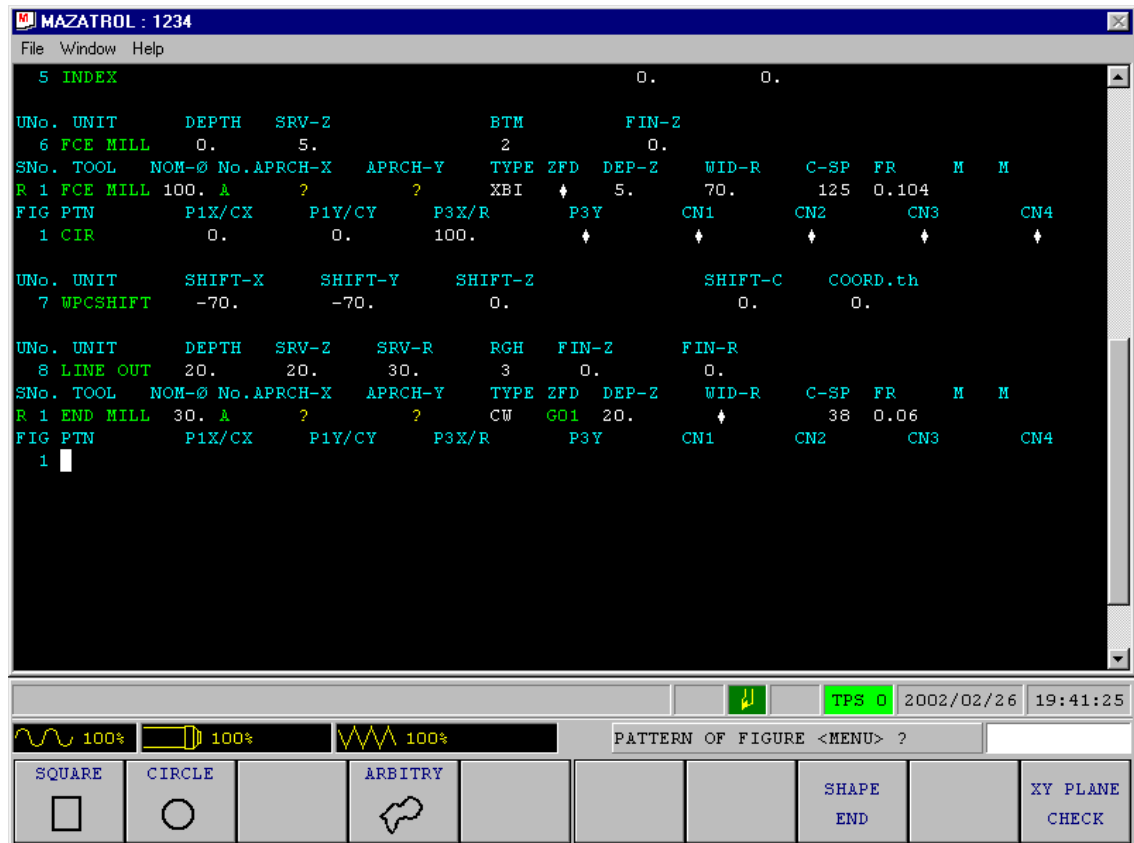
D736P0054E

- (3) NOMINAL DIAMETER?   
- (4) TOOL FILE CODE <MENU>? **A**
- (5) MACHINING PRIORITY No.? 

(Skip to the next item since the priority function for the same tool is not to be used here. See Part 2, Chapter 5.)
- (6) APPROACH POINT X, AUTO → <MENU>? **AUTO SET**
- (7) APPROACH POINT Y, AUTO → <MENU>? **AUTO SET**
- (8) CUTTING DIRECTION <MENU>? **CW CUT**

(Specify the direction in which the circumference is to be cut. The possible directions of cutting are clockwise (CW) and counterclockwise (CCW).)
- (9) FEEDRATE Z, <MENU>/DATA <INPUT>? 
- (10) DEPTH OF CUT? **AUTO SET**
- (11) CUTTING SPEED, AUTO → <MENU>? **AUTO SET**
- (12) FEED RATE, AUTO → <MENU>? **AUTO SET**
- (13) M CODE? 
- (14) M CODE? 

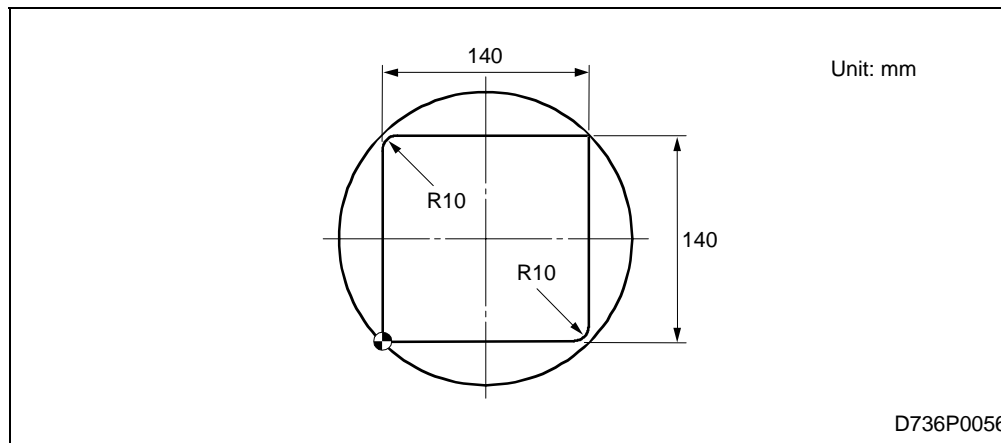
A tool sequence such as that shown below is now completed.



D736P0055E

3. Creation of a shape sequence

Next, let us program a shape sequence. For outside linear machining, the edges of a material are to be machined. Set the dimensions of a shape such as that shown below.



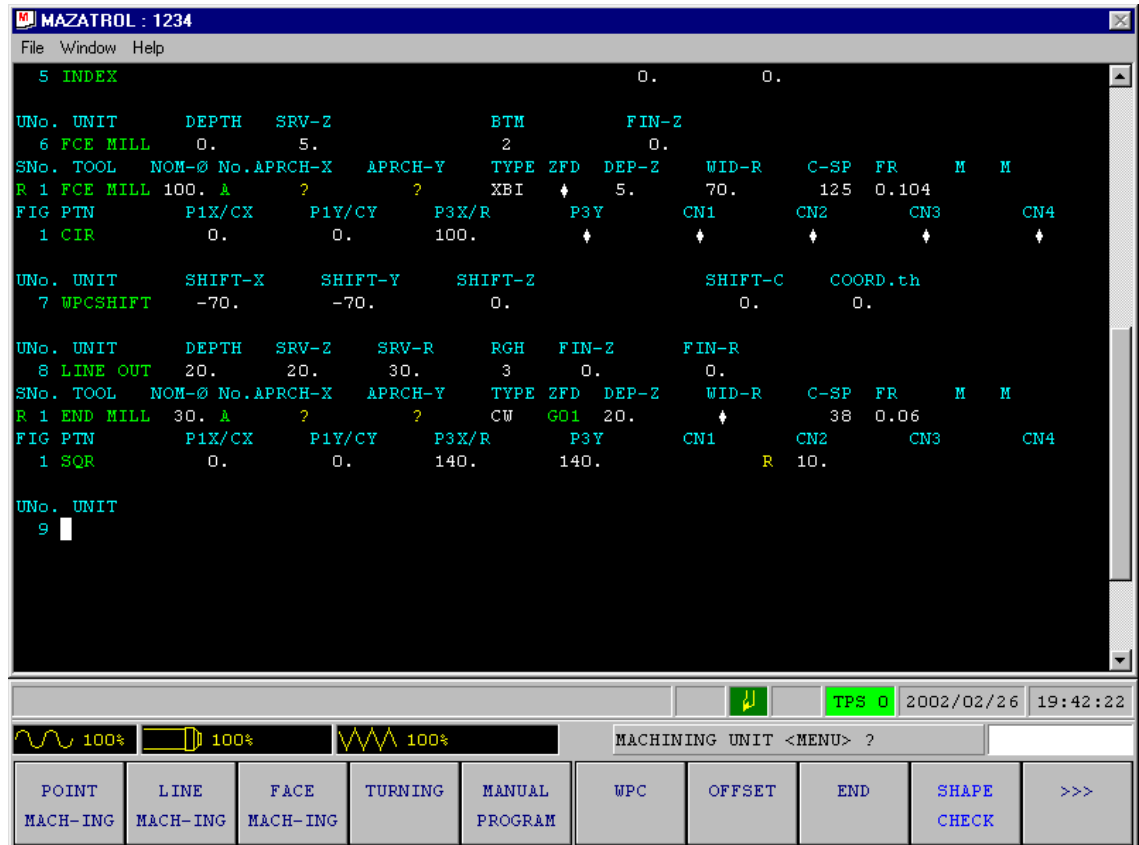
D736P0056

(1) While the message "PATTERN OF FIGURE <MENU>?" is displayed, press the menu key **SQUARE**.

- (2) STARTING POINT X?
- (3) STARTING POINT Y?
- (4) CORNER 3 COORDINATE X?
- (5) CORNER 3 COORDINATE Y?

- (6) CORNER 1 CHAMFER?
- (7) CORNER 2 CHAMFER?
- (8) CORNER 3 CHAMFER?
- (9) CORNER 4 CHAMFER?
- (10) PATTERN OF FIGURE <MENU>?..... **SHAPE END**

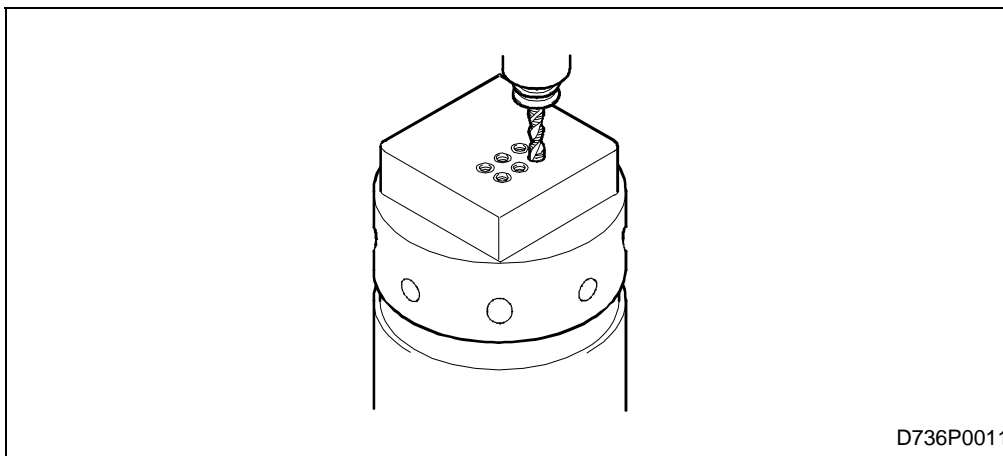
A shape sequence such as that shown below is now completed.



D736P0057E

4-4-13 Point machining unit

Let us create a unit that is used to drill six M12 tapped through-holes. This machining program unit is referred to as a tapping unit.

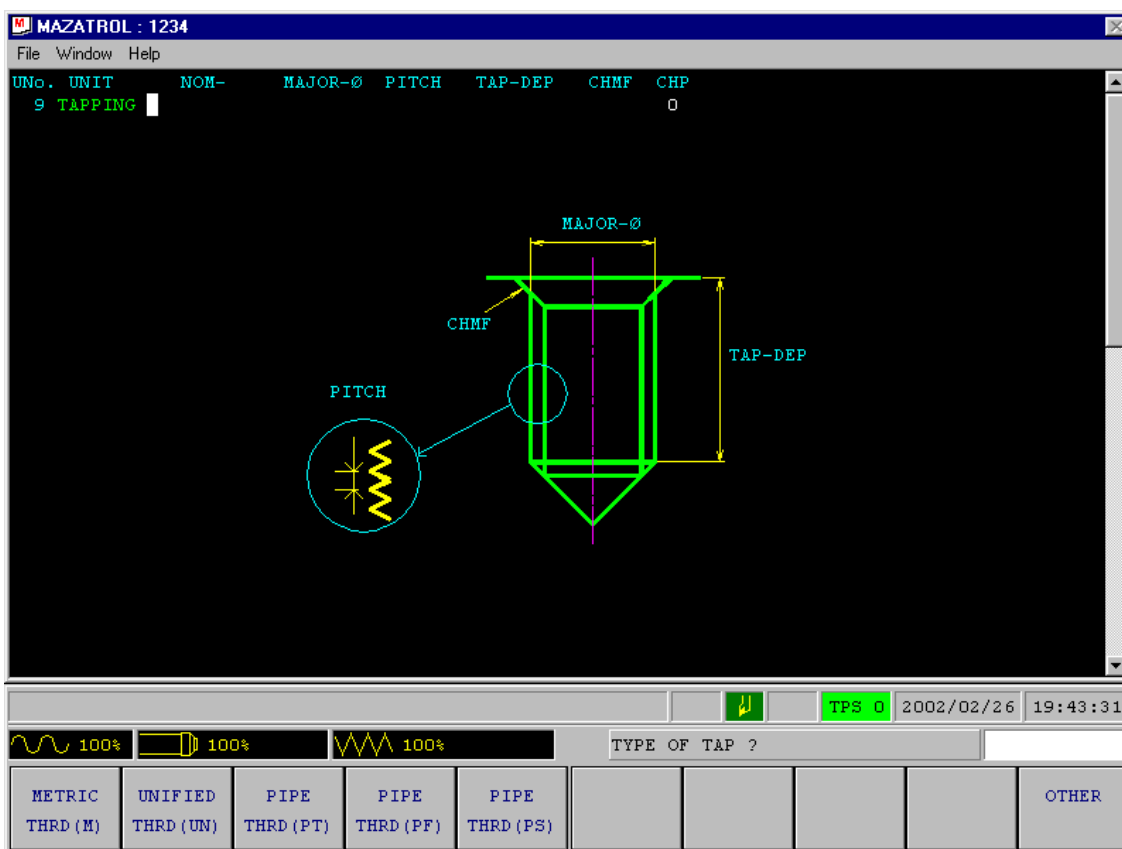


1. Creation of a point machining unit

(1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **POINT MACH-ING**.

(2) MACHINING UNIT <MENU>? 

(3) Press the menu key **HELP**.



D736P0058E

Now, let's create the point machining unit. For this unit, enter data for the item NOM-φ, TAP-DEP.

- (4) TYPE OF TAP? **METRIC THRD (M)**
- (5) NOMINAL DIAMETER?
- (6) TAPPING DEPTH?
- (7) CHAMFER WIDTH?
- (8) CHIP VAC. CLEANER <Y:1, N:0>?

A point machining unit such as that shown below is now completed. This shows that after the data input for the point machining unit, a spot, a drill and a tap have been automatically selected for the tool sequence that is to be programmed.









D736P0059E

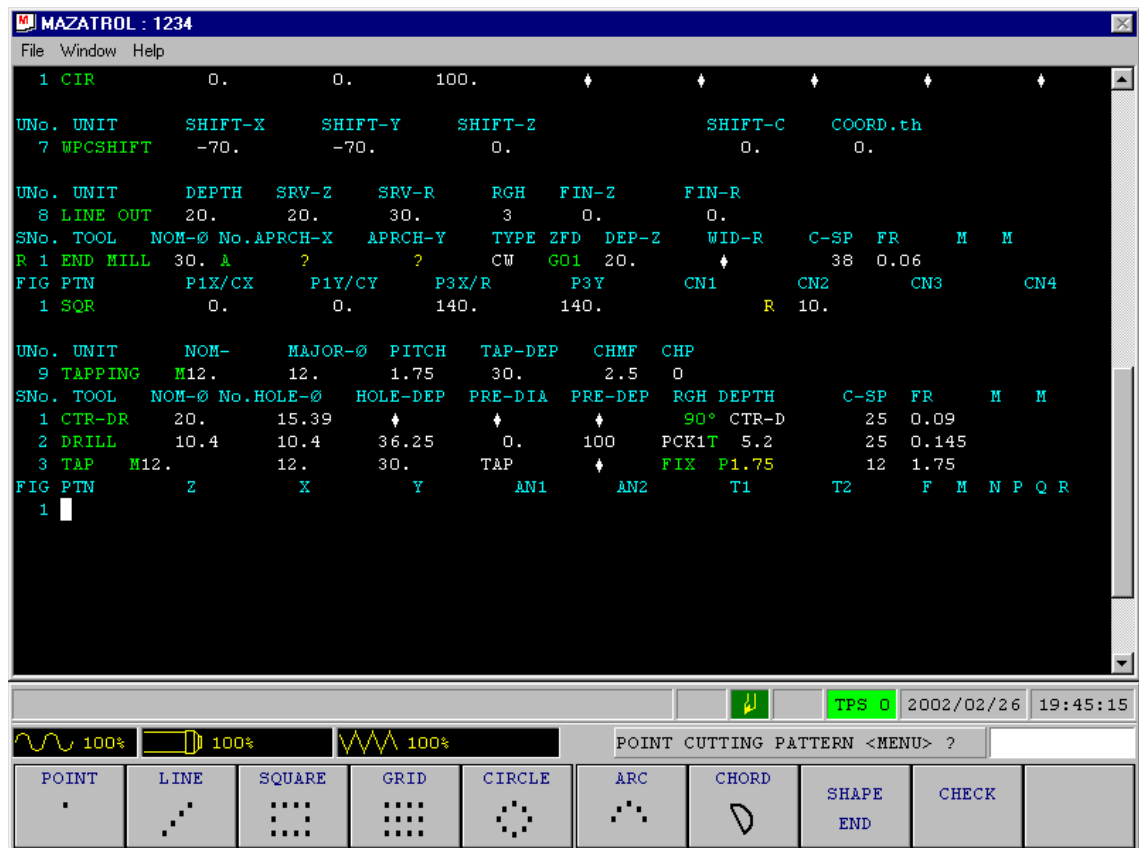
2. Creation of a tool sequence

- (1) Press the cursor key several times in succession to move the cursor to the position under the item C-SP.
- (2) CUTTING SPEED, AUTO → <MENU>? **HSS AUTO**
- (3) FEEDRATE, AUTO → <MENU>? **HSS AUTO**
- (4) M CODE?
- (5) M CODE?

4 PROCEDURE BEFORE PERFORMING THE MACHINING

- (6) Press the cursor key  several times in succession to move the cursor to the position under the item C-SP.
- (7) CUTTING SPEED, AUTO → <MENU>? **HSS AUTO**
- (8) FEEDRATE, AUTO → <MENU>? **HSS AUTO**
- (9) M CODE?..... 
- (10) M CODE?..... 
- (11) Press the cursor key  several times in succession to move the cursor to the position under the item C-SP.
- (12) CUTTING SPEED, AUTO → <MENU>? **HSS AUTO**
- (13) FEEDRATE, AUTO → <MENU>? **HSS AUTO**
- (14) M CODE?..... 
- (15) M CODE?..... 

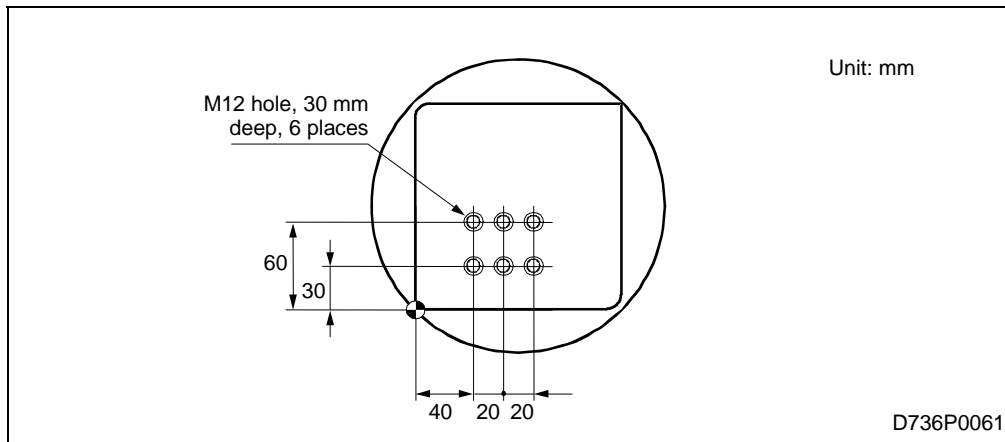
A tool sequence such as that shown below is now completed.



D736P0060E

3. Creation of a shape sequence

Next, create a shape sequence. For tapping, six holes are to be drilled. These holes are to be set as one shape, as shown below.



- (1) While the message "POINT CUTTING PATTERN <MENU>?" is displayed, press the menu key **SQUARE**.
- (2) Z VALUE OF WORK SURFACE?
- (3) STARTING POINT X?
- (4) STARTING POINT Y?
- (5) START LINE ANGLE FROM X AXIS?.....
- (6) ANGLE BETWEEN THE TWO LINES?.....
- (7) PITCH/LENGTH OF PATTERN AN1?.....
- (8) PITCH/LENGTH OF PATTERN AN2?.....
- (9) T1&T2→<PITCH:0,LINE LENG.:1>?.....
- (10) NUMBER OF HOLES IN LINE AN1?
- (11) NUMBER OF HOLES IN LINE AN2?
- (12) OMIT 4 CNR EX SPT <Y:1, N:0>?
- (13) OMIT SPT MACHINING <Y:1, N:0>?
- (14) RETURN POSITION <INIT:0, R:1>?
- (15) POINT CUTTING PATTERN <MENU>? **SHAPE END**

A shape sequence such as that shown below is now completed.



D736P0062E

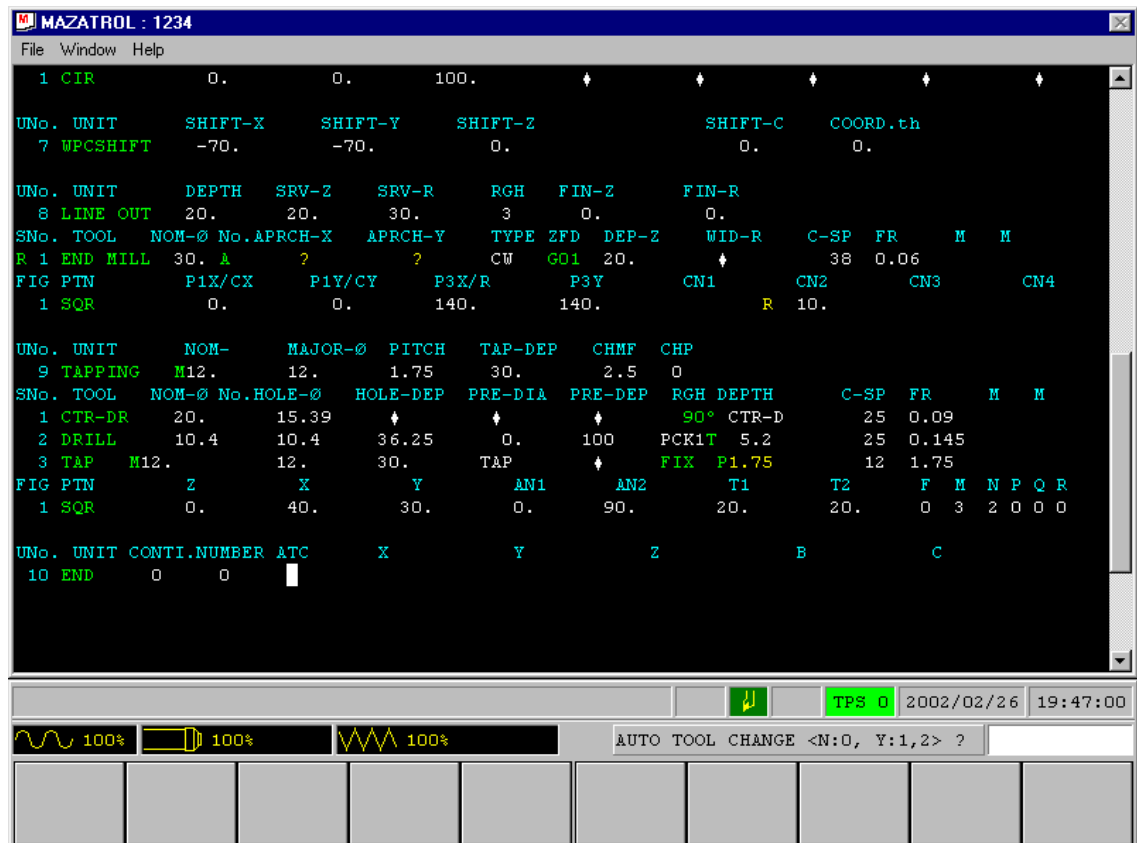
This completes the creation of a point machining (tapping) unit.

4-4-14 End unit

Let us create an end unit that indicates the end of the program.

- (1) While the message "MACHINING UNIT <MENU>?" is displayed, press the menu key **END**.
- (2) CONTINUE <Y:1, N:0>?
 (Specify whether or not the machining program is to be repeated. If 0, the program ends with the particular step. If 1, the program returns to its beginning and returns to machine the next workpiece.)
- (3) PARTS COUNTER <Y:1, N:0>?
 (Specify whether or not the number of times that the machining operation has been repeated is to be counted. If 0, counting does not occur. If 1, counting occurs and the results are displayed on the **POSITION** display of the screen.)

An end unit such as that shown below is now completed, which means that the creation of the program is to be ended.







D736P0063E

4-4-15 End of the program

- (1) Press the menu selector key.
- (2) Press the menu key **PROGRAM COMPLETE**.

Display of the created program

Let us trace the created program back to its beginning. Repeated depression of the cursor key  causes the screen to scroll up continuously to display the part of the program that precedes the existing part. Depressing of the cursor key  causes the program to advance.

In addition, the program can also be checked using the page keys  . In this case, however, the cursor moves on a unit basis.

4 PROCEDURE BEFORE PERFORMING THE MACHINING

The following shows the entire program (actuate the cursor keys to check the program):

UNO.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM				
0	CBN STL	MILL&TURN.	200.	0.	325.	5.	1	2000				
UNO.	UNIT	ADD. WPC	X	Y	th	Z	C					
1	WPC- 1		T.CENTER	T.CENTER	0.	-400.	0.					
UNO.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C						
2	INDEX				90.	0.						
UNO.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FIN					
3	T.GROOVE	OUT	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
F 1	GROOVE	OUT	30.	A	◆	2.	◆	◆	◆	108	0.1	
FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	RADIUS/th	RGH				
1		200.	150.	190.	150.							
UNO.	UNIT	C-FACE	DIA	DEPTH	CHMF							
4	DRILLING	CYLIND	20.	10.	0.							
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M M
1	CTR-DR	20.		10.	◆	◆	◆	90°	CTR-D	25	0.09	
2	DRILL	20.		20.	10.	0.	100	DRILT	10.	25	0.125	
FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R				
1	CIR	100.	0.	90.	8	◆	◆	1				
UNO.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C						
5	INDEX				0.	0.						
UNO.	UNIT	DEPTH	SRV-Z	BTM	FIN-Z							
6	FCE MILL	0.	5.	2	0.							
SNO.	TOOL	HOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R 1	FCE MILL	100.	A	?	?	XB X	◆	5.	70.	125	0.104	
FIG	PTN	PIX/CX	PIY/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4			
1	CIR	0.	0.	100.	◆	◆	◆	◆	◆			
UNO.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	COORD.th						
7	WPCSHIFT	-70.	-70.	0.	0.	0.						
UNO.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R					
8	LINE OUT	20.	20.	30.	3	0.	0.					
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
A	END MILL	30.		?	?	CW	G01	20.	◆	38	0.06	
FIG	PTN	PIX/CX	PIY/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4			
1	SQR	0.	0.	140.	140.			R 10.			R 10.	
UNO.	UNIT	NOM-φ	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP					
9	TAPPING	M12.	12.	1.75	30.	2.5	0					
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M M
1	CTR-DR	20.		15.39	◆	◆	◆	90°	CTR-D	25	0.09	
2	DRILL	10.4		10.4	36.25	0.	100	PCK1T	5.2	25	0.145	
3	TAP	M12.		12.	30.	TAP	◆	FIX	P1.75	12	1.75	
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F M N P Q R			
1	SQR	0.	40.	30.	0.	90.	20.	30.	0 3 2 0 0 0			
UNO.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	B	C			
10	END	0	0									

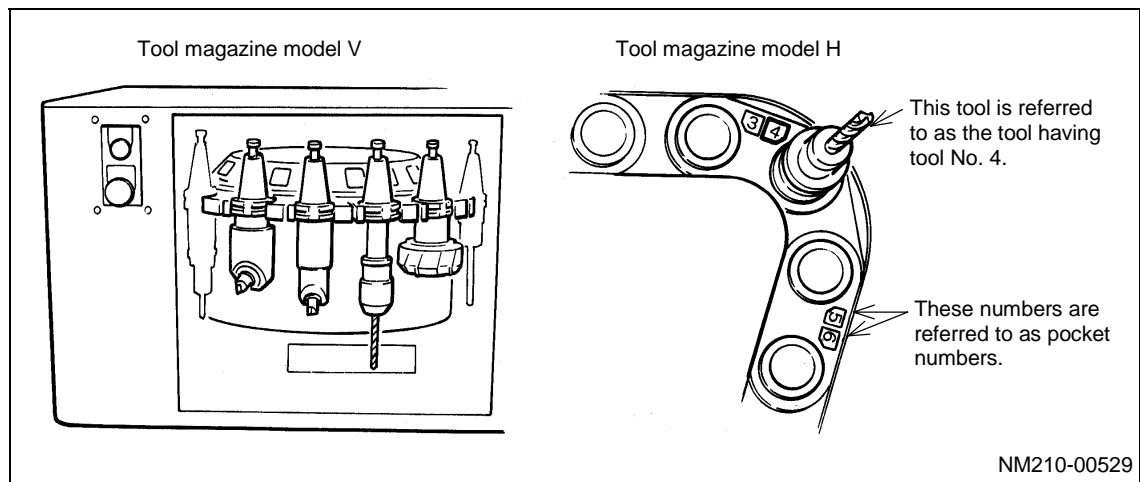
Fig. 4-9 Sample program

4-5 Registration of Tools in a Tool Data File

During creation of a program, the names and nominal diameters of the tools required for machining should have already been entered. To machine a workpiece by automatic run, those tools must be mounted in a tool magazine in advance. In addition, the tools to be mounted in a tool magazine must be registered in a tool data file. Registration of tools in a tool data file is referred to as assignment of pocket numbers to the tools.

4-5-1 Pocket numbers and tool numbers

The pocket number refers to the number inscribed on the side face of the pocket of a tool magazine. The tool that has been mounted in the pocket is specified by the pocket number. The identification number of the pocket which accommodates a tool is called tool number.



Note: The design of the tool magazine slightly differs according to the type of machine to be used.

4-5-2 TOOL DATA display

Carry out the following key operations to call the **TOOL DATA** display:

- (1) Press the display selector key.
- (2) Press the menu key **TOOL DATA**.
 - ➔ The **TOOL DATA** display will then be displayed as shown below.

Function of **TOOL DATA** display

The **TOOL DATA** display shows the types of tools mounted in the tool magazine. The types of tools are displayed in order of pocket number; the pocket numbers being displayed on the left end of the screen.

If no tool types are being displayed on the screen, this indicates that no tools are mounted in the tool magazine. The pocket numbers on the **TOOL DATA** display correspond to the pocket numbers of the tool magazine. The relationship between these two types of pocket numbers can be represented as shown below.

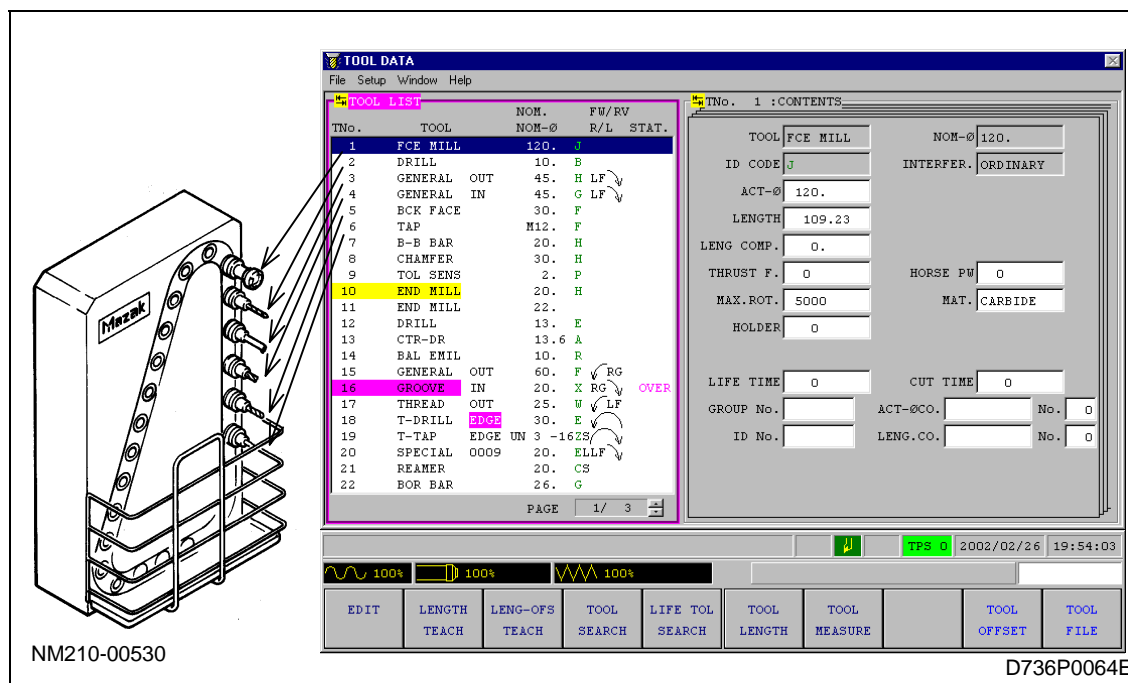


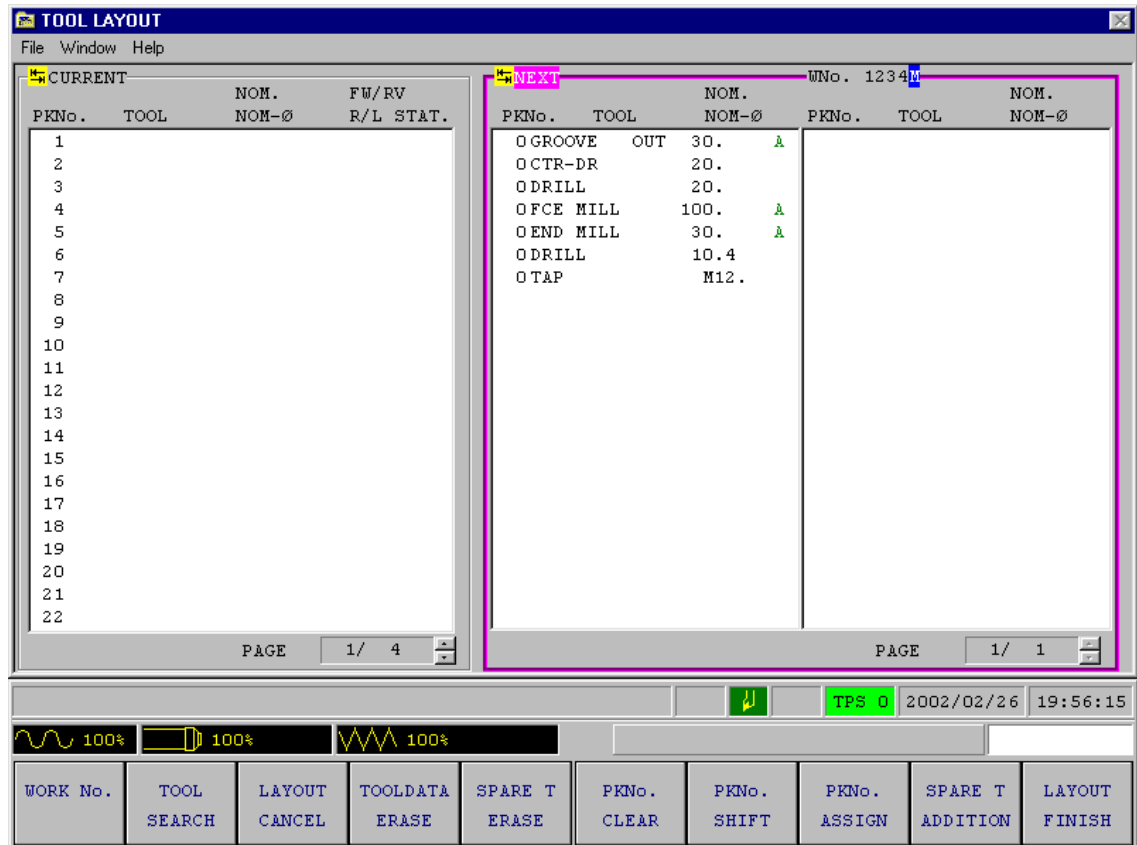
Fig. 4-10 Correspondence to the pocket numbers

4-5-3 TOOL LAYOUT display

Carry out the following key operation to call the **TOOL LAYOUT** display:

- (1) Press the display selector key.
- (2) Press the menu key **TOOL LAYOUT**.

➔ The following **TOOL LAYOUT** display will be displayed.



D736P0065E

Function of **TOOL LAYOUT** display

The **TOOL LAYOUT** display is the display on which the tools that have been set on the program are to be registered in a tool data file. This display consists of two sections:

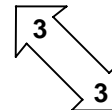
NEXT display on the right side, and CURRENT display on the left side. On the CURRENT display, tool names and pocket numbers are indicated in the same state as that of the **TOOL DATA** display. The NEXT display is used to carry out setup operations such as display of the tools that have been set on the program and assignment of pocket numbers to them. When the setup operations are completed, the tools on the NEXT display will be registered on the CURRENT display (tool data).

The following figure shows the relationship between the **PROGRAM** display, **TOOL DATA** display and **TOOL LAYOUT** display:

4 PROCEDURE BEFORE PERFORMING THE MACHINING

TNo.	TOOL		NOM. NOM-Ø	FW/RV R/L STAT.
1	GROOVE	OUT	30.	A
2	CTR-DR		20.	
3	DRILL		20.	
4	FCE MILL		100.	A
5	END MILL		30.	A
6	DRILL		10.4	
7	TAP		M12.	
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				
19				
20				
21				
22				

The CURRENT display and the **TOOL DATA** display always possess the same contents. Thus, the tool data also changes when new tools have been registered on the CURRENT display.



D736P0066E

PKNo.	TOOL		NOM. NOM-Ø	FW/RV R/L STAT.
1	GROOVE	OUT	30.	A
2	CTR-DR		20.	
3	DRILL		20.	
4	FCE MILL		100.	A
5	END MILL		30.	A
6	DRILL		10.4	
7	TAP		M12.	
8				
9				
10				
11				
12				
13				
14				
15				
16				
17				
18				
19				
20				
21				
22				

D736P0067E

UNo.	MAT.	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM					
0	CBN STL	MILL&TURN.	200.	0.	325.	5.	1	2000					
UNo.	UNIT	ADD.	WPC	X	Y	th	Z	C					
1	WPC-	1		T.CENTER	T.CENTER	0.	-400.	0.					
UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C							
2	INDEX				90.	0.							
UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FIN						
3	T.GROOVE	OUT	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
F	1GROOVE	OUT	30.	A	2.					108	0.1		
FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANG	RGH					
1		200.	150.	190.	150.								
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF								
4	DRILLING	CYLIND	20.	10.	0.								
SNo.	TOOL	NOM-Ø	No.	HOLE-Ø	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20.	10.					90°	CTR-D	25	0.09		
2	DRILL	20.	20.	10.	0.	100	DRILT	10.		25	0.125		
FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R					
1	CIR	100.	0.	90.	8			1					
UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C							
5	INDEX				0.	0.							

WORK No. SEARCH PROGRAM BARRIER WPC MSR TOOL PROCESS PROGRAM HELP PROGRAM
EDIT INFORM. PATH CONTROL LAYOUT FILE

D736P0068E

2 Registration of new tools

1 The tools that have been set on the program are displayed.

PKNo.	TOOL	NOM.	NOM-Ø
1	GROOVE	OUT	30. A
2	CTR-DR		20.
3	DRILL		20.
4	FCE MILL		100. A
5	END MILL		30. A
6	DRILL		10.4
7	TAP		M12.

PAGE 1/1

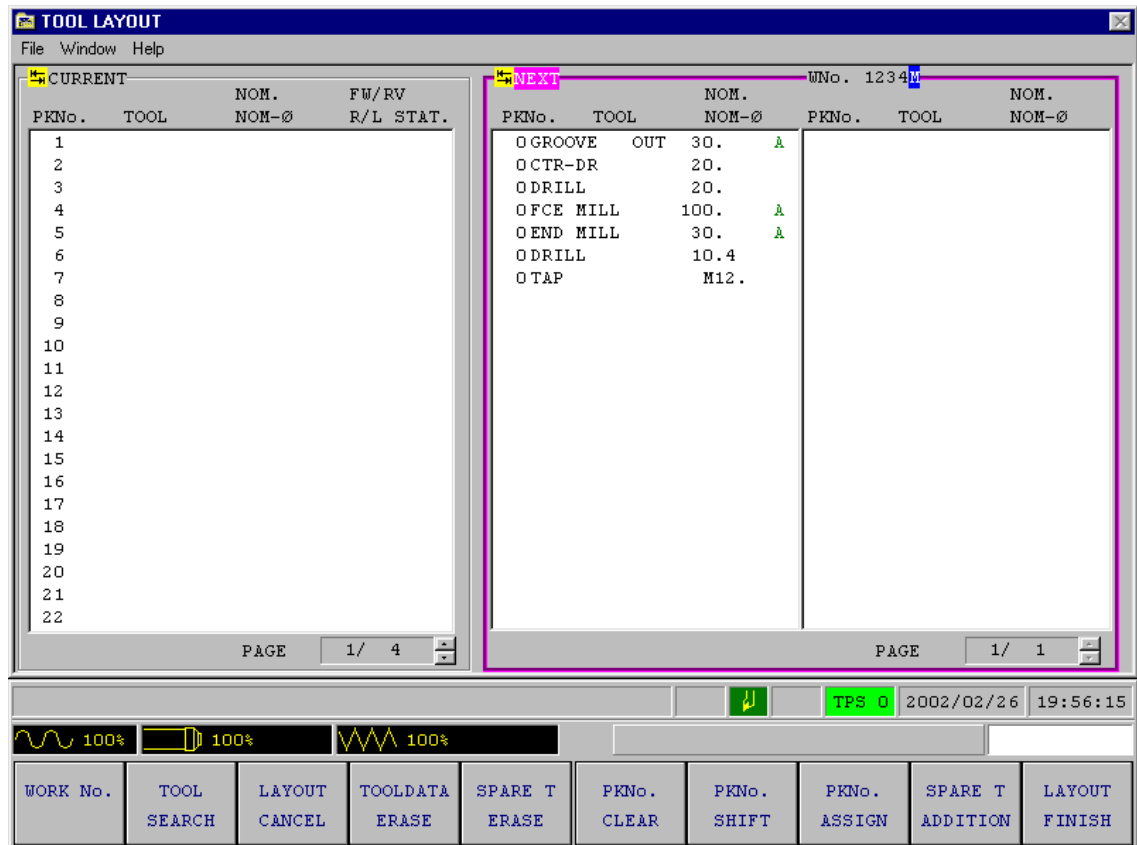
D736P0069E

4-5-4 Registration of tools

The display of menu data **WORK No.** is reversed, and the message “WORKPIECE PROGRAM NUMBER?” is displayed on the **TOOL LAYOUT** display. Enter the work number of the program which has been previously created in Section 4-4.

See Section 5-1, “TOOL LAYOUT Display” of the Operating Manual for details of registration of tools.

(1) WORKPIECE PROGRAM NUMBER?..... 1 2 3 4 ↔



D736P0070E

The tools that have been set on the program having work No. 1234 will then be displayed as shown on the NEXT display. Let us assign pocket numbers to the displayed tools.

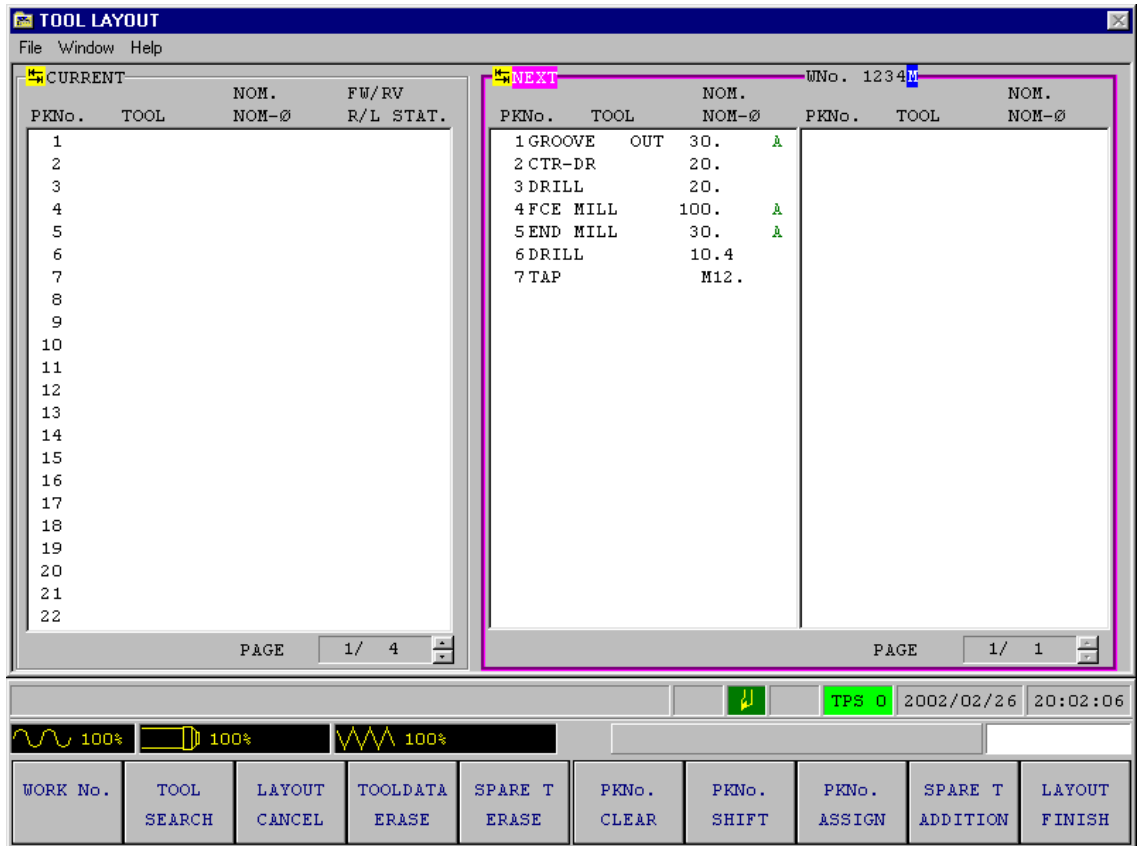
Three modes are available for assigning pocket numbers: automatic, manual, and shift. Automatic assignment, the simplest of these three modes, is described below.

Note: If automatic assignment is performed, data of the registered tools will be erased entirely. To prevent this data erasure, either the manual assignment mode or the shift mode must be used.

(2) PKNo. SHIFT OR ASSIGN <MENU>? **PKNo. ASSIGN**

(3) POCKET NUMBER ASSIGN <INPUT>?..... 

➔ Pocket numbers will then be assigned to the tools as shown below.



The screenshot shows the 'TOOL LAYOUT' window with two main panels: 'CURRENT' and 'NEXT'. The 'CURRENT' panel lists 22 pocket numbers (1-22) in a column. The 'NEXT' panel shows a table of tool assignments for workpiece WNo. 1234. The table has columns for PKNo., TOOL, and NOM. NOM-Ø. The assignments are:

PKNo.	TOOL	NOM. NOM-Ø
1	GROOVE OUT	30. A
2	CTR-DR	20.
3	DRILL	20.
4	FCE MILL	100. A
5	END MILL	30. A
6	DRILL	10.4
7	TAP	M12.

At the bottom of the window, there is a status bar with 'TPS 0', date '2002/02/26', and time '20:02:06'. Below the status bar are several icons and a row of function buttons: WORK No., TOOL SEARCH, LAYOUT CANCEL, TOOLDATA ERASE, SPARE T ERASE, PKNo. CLEAR, PKNo. SHIFT, PKNo. ASSIGN, SPARE T ADDITION, and LAYOUT FINISH.

D736P0071E

4 PROCEDURE BEFORE PERFORMING THE MACHINING

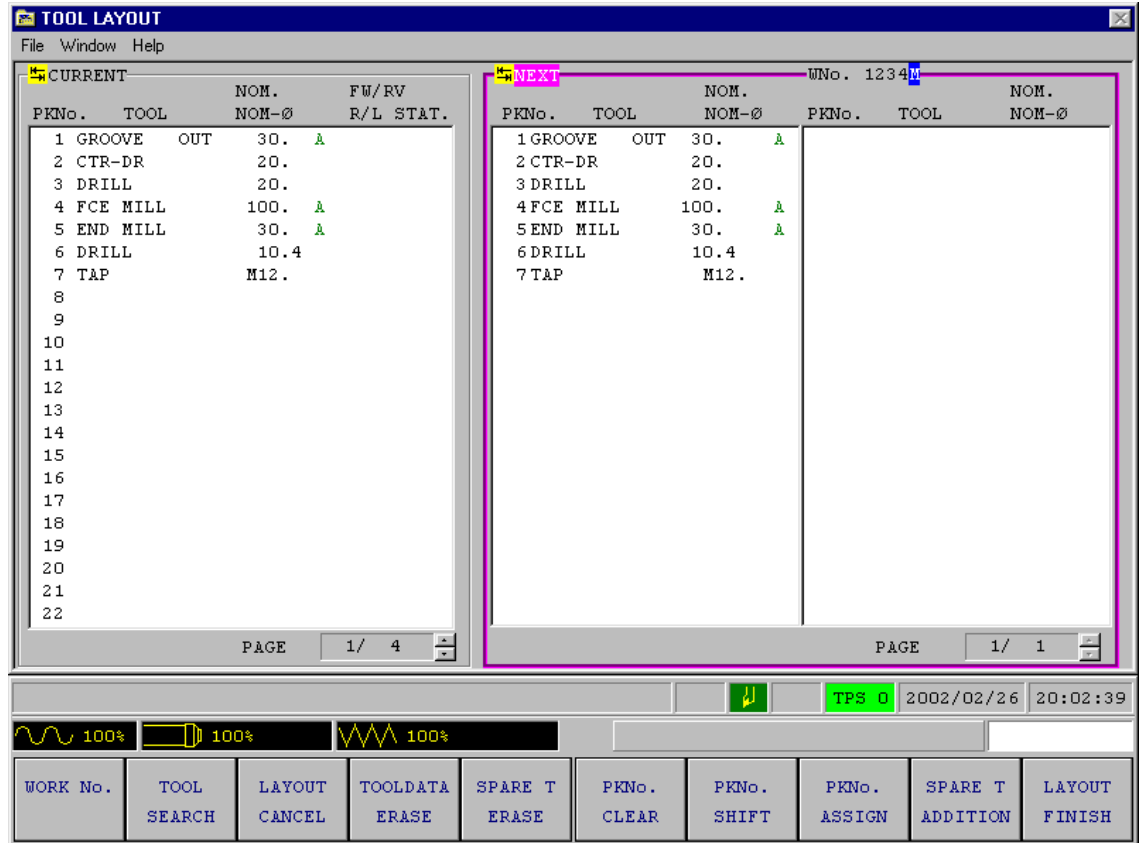
Next, carry out the following key operations to register the tools on the NEXT display onto the CURRENT display.

(4) Press the menu key **LAYOUT FINISH**.

(5) LAYOUT FINISH <INPUT>?..... 

(6) LAYOUT FINISH

➔ The tools will be registered on the CURRENT display as shown below.



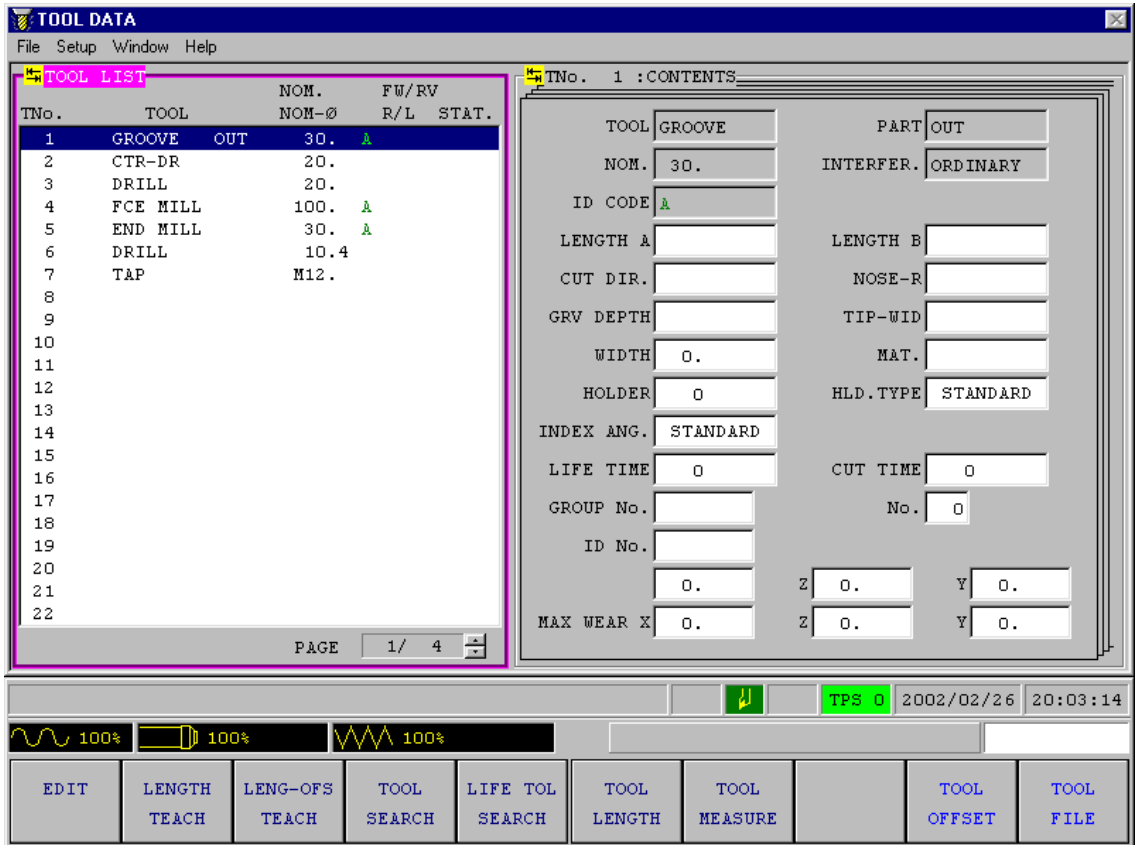
D736P0072E

Let us recall the **TOOL DATA** display on the screen.

(7) Press the display selector key.

(8) Press the menu key **TOOL DATA**.

➔ The tools that have been registered in the **TOOL LAYOUT** display will be presented in the **TOOL DATA** display, as shown below.



D736P0073E

4-6 Check of the Machining Path

Let us return the **PROGRAM** display.

(1) Press the display selector key.

(2) Press the menu key **PROGRAM**.

➔ The program with work No. 1234 will be displayed as shown below.

The screenshot displays the MAZATROL 1234 software interface. The main window shows a list of program parameters and tool settings. The 'DRILLING' tool is highlighted in pink. Below the main window, there are several status indicators and a menu bar.

UNo.	MAT.	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
0	CBN STL	MILL&TURN.	200.	0.	325.	5.	1	2000

UNo.	UNIT	ADD.	WPC	X	Y	th	Z	C
1	WPC-1			T.CENTER	T.CENTER	0.	-400.	0.

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C
2	INDEX				90.	0.

UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FIN
3	T.GROOVE	OUT	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
F 1	GROOVE	OUT	30.	A		2.				108	0.1		

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANG	RGH
1			200.	150.	190.	150.			

UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF
4	DRILLING	CYLIND	20.	10.	0.

SNo.	TOOL	NOM-Ø	No.	HOLE-Ø	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	20.		10.				90°	CTR-D	25	0.09		
2	DRILL	20.		20.	10.		100	DRILT	10.	25	0.125		

FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R
1	CIR	100.	0.	90.	8			1

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C
5	INDEX				0.	0.

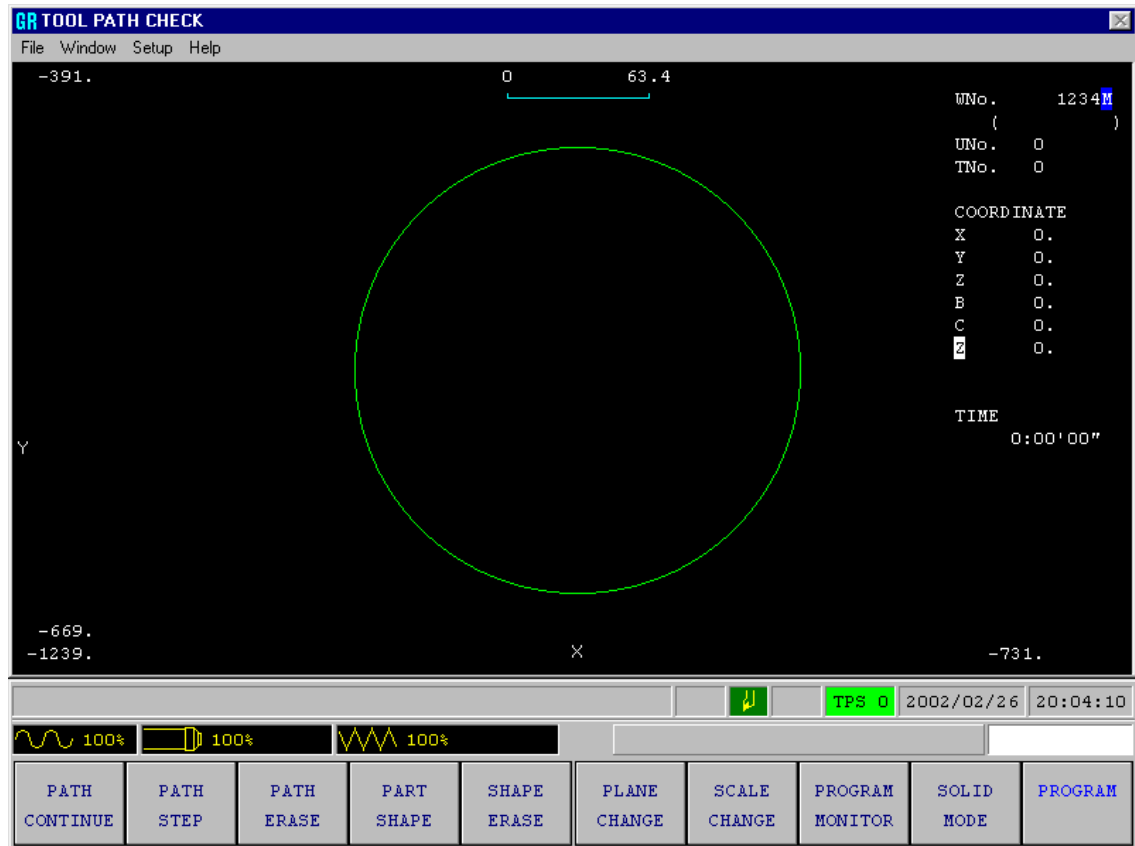
At the bottom of the interface, there are several status indicators and a menu bar:

- Waveform icons: 100% (yellow), 100% (green), 100% (red)
- Buttons: WORK No., SEARCH, PROGRAM EDIT, BARRIER INFORM., WPC MSR, TOOL PATH, PROCESS CONTROL, PROGRAM LAYOUT, HELP, PROGRAM FILE
- TPS indicator: TPS D
- Date and Time: 2002/02/26 20:03:47

D736P0074E

(3) Press the menu key **TOOL PATH**.

→ The following **TOOL PATH** display will be displayed.

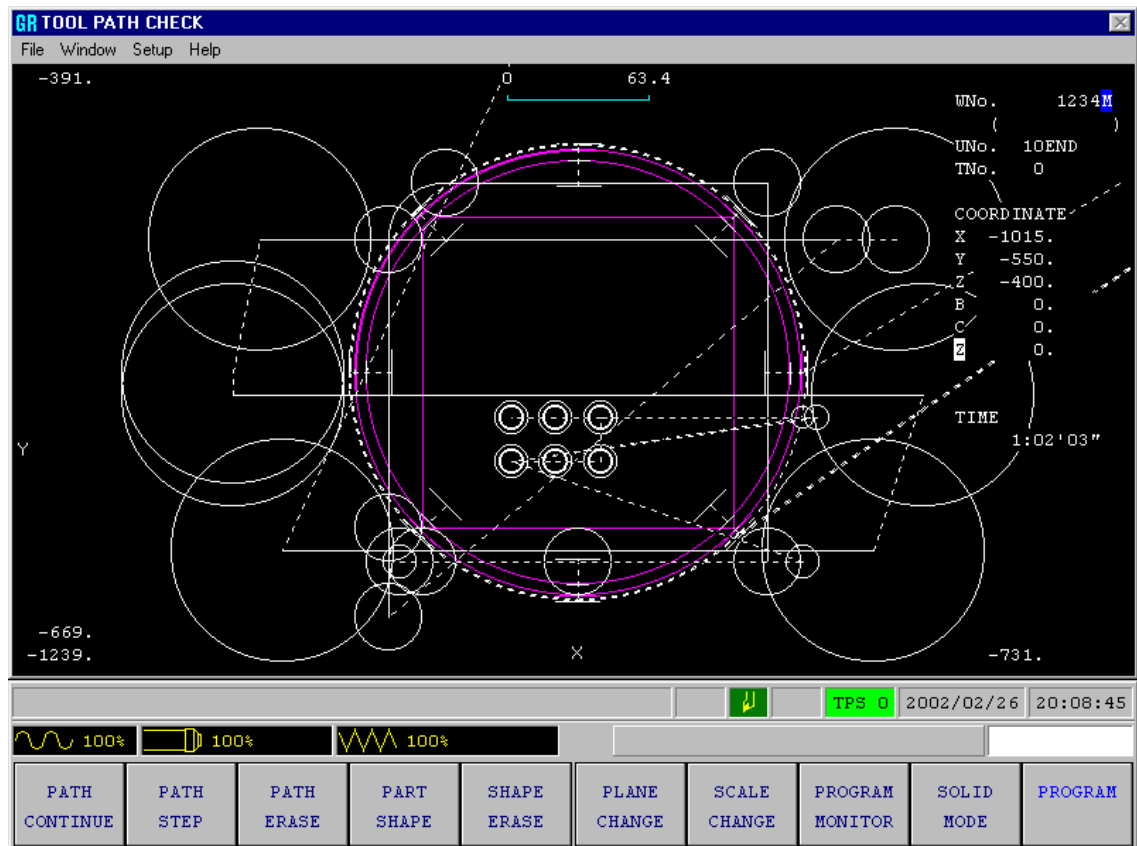


D736P0075E

(4) Press the menu key **PART SHAPE** to display shapes continuously.

4 PROCEDURE BEFORE PERFORMING THE MACHINING

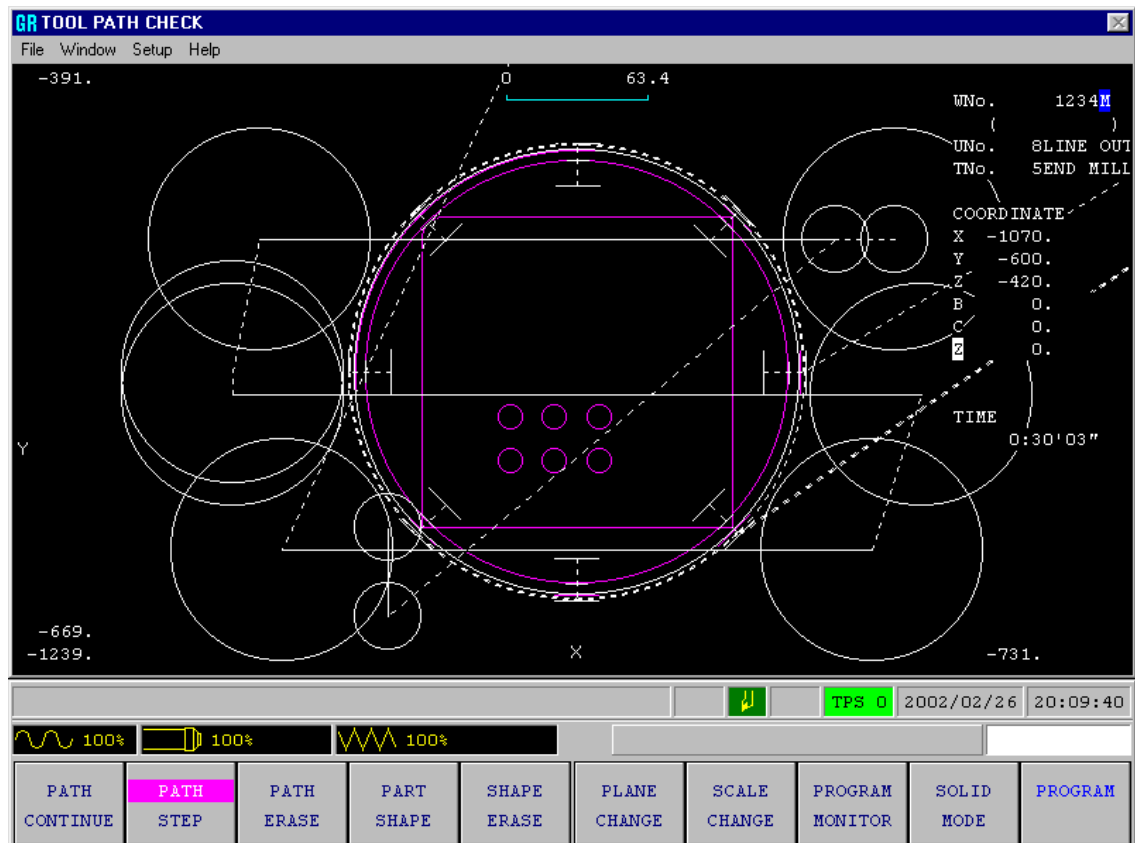
- (5) Next, press the menu key **PATH CONTINUE** to display the machining path of the tool through to the end.



D736P0076E

- (6) Press the menu key **PATH ERASE** to erase the machining path of the tool.

- (7) Press the menu key **PATH STEP** several times to display the machining path in steps. The reverse-display state of the menu continues until the entire machining path appeared.



D736P0077E

- (8) To check the machining path in further details, use the PLANE CHANGE function or the SCALE CHANGE function.
- (9) When the all operation has been completed, press the display selector key and the menu key **POSITION** to resume the **POSITION** display.

4

PROCEDURE BEFORE PERFORMING THE MACHINING

- NOTE -

PART 2

MAZATROL PROGRAM FUNCTIONS

Part 2 describes all the functions of the MAZATROL program.

CONTENTS

Page

1	CALLING UP AND THE END OF THE PROGRAM DISPLAY	1-1
1-1	Listing Mode and Creating Mode	1-1
1-2	Calling Up the PROGRAM Display (Listing Mode)	1-1
1-3	Calling Up the PROGRAM Display (Creating Mode)	1-2
1-4	End of the Program Creation	1-2
2	EDITION OF DATA.....	2-1
2-1	Cursor Movement	2-1
2-1-1	Case of listing mode	2-1
2-1-2	Case of creating mode	2-1
2-2	Editing.....	2-2
2-2-1	Editing functions and menus.....	2-2
2-2-2	Search.....	2-2
2-2-3	Insertion	2-7
2-2-4	Deletion	2-10
2-2-5	Copy.....	2-14
2-2-6	End of program.....	2-20
3	HELP FUNCTIONS	3-1
3-1	MAZATROL Help.....	3-1
3-2	TOOL FILE Window.....	3-2
3-3	TOOL DATA Window.....	3-3
3-4	Tap Nominal Diameter Window	3-4

4	PROGRAM CREATION.....	4-1
4-1	Common Unit.....	4-1
4-2	Fundamental Coordinates System Unit (WPC).....	4-6
4-3	Index Unit (INDEX)	4-7
4-4	Fundamental Coordinate Shifting Unit (WPC SHIFT)	4-10
4-5	Auxiliary Coordinates System Unit (OFFSET)	4-12
4-6	Types of the Machining Unit	4-13
4-7	Point Machining Unit.....	4-15
4-7-1	Types of point machining units	4-15
4-7-2	Procedure for selecting point machining unit	4-16
4-7-3	Unit data and automatic tool development.....	4-17
4-7-4	Automatic tool development for cemented carbide drill.....	4-44
4-7-5	New tapping auto-setting scheme.....	4-45
4-7-6	Tool sequence data.....	4-50
4-7-7	Tool path	4-56
4-7-8	Shape sequence	4-101
4-8	Line Machining Units	4-118
4-8-1	Types of line machining units	4-118
4-8-2	Procedure for selecting line machining unit	4-119
4-8-3	Unit data, automatic tool development and tool path	4-119
4-8-4	Line machining tool sequence data	4-157
4-8-5	Line machining shape sequence data	4-159
4-8-6	Precautions in line machining	4-160
4-8-7	Automatic corner override.....	4-164

4-9	Face Machining Units	4-166
4-9-1	Types of face machining units	4-166
4-9-2	Procedure for selecting face machining unit	4-167
4-9-3	Unit data, automatic tool development and tool path	4-167
4-9-4	Face machining tool sequence data	4-213
4-9-5	Precautions in face machining	4-217
4-9-6	Override in case of the overall width cutting	4-223
4-9-7	Definitions of forms in line machining and face machining units.....	4-225
4-10	End Unit (END)	4-246
4-11	M-Code Unit (M-CODE).....	4-248
4-12	Sub-Program Unit (SUB PRO).....	4-249
4-13	Pallet Changing Unit (PALT CHG).....	4-250
4-14	Process End Unit (PROC END).....	4-251
4-15	Manual Program Mode Unit (MANL PRG).....	4-253
4-15-1	Menu selection	4-253
4-15-2	Unit data setting	4-253
4-15-3	Sequence data setting.....	4-254
4-16	Materials Shape Unit (MATERIAL)	4-256
4-16-1	Setting unit data	4-256
4-16-2	Setting sequence data.....	4-256
4-17	Head Selection Unit (HEAD).....	4-259
4-17-1	Setting unit data	4-259
4-18	Workpiece Transfer Unit (TRANSFER)	4-261
4-18-1	Setting unit data	4-261

4-19	C-Axis Point Machining Unit	4-267
4-19-1	Types of C-axis point machining units	4-267
4-19-2	Procedure for selecting point machining unit	4-268
4-19-3	C-axis point machining unit data and automatic tool development.....	4-269
4-19-4	C-axis point machining tool sequence data	4-289
4-19-5	Tool path	4-289
4-19-6	C-axis point machining shape sequence	4-289
4-20	C-Axis Line Machining Unit.....	4-295
4-20-1	Types of C-axis line machining units	4-295
4-20-2	Procedure for selecting C-axis line machining unit.....	4-296
4-20-3	C-axis line machining unit data and automatic tool development	4-296
4-8-1	C-axis line machining tool sequence data.....	4-306
4-20-5	Definitions of forms in C-axis line machining and face machining units.....	4-306
4-20-6	Precautions in C-axis line machining	4-314
4-21	Turning Units	4-315
4-21-1	Types of turning units	4-315
4-21-2	Procedure for selecting turning unit	4-315
4-21-3	Bar-materials machining unit (BAR).....	4-316
4-21-4	Copy-machining unit (CPY)	4-329
4-21-5	Corner-machining unit (CORNER).....	4-333
4-21-6	Edge-machining unit (FACING)	4-337
4-21-7	Threading unit (THREAD).....	4-341
4-21-8	Grooving unit (T. GROOVE)	4-348
4-21-9	Turning drilling unit (T. DRILL).....	4-357
4-21-10	Turning tapping unit (T. TAP)	4-364

4-22	Coordinate Measuring Unit (MMS)	4-369
4-22-1	Procedure for calling up the MMS unit	4-369
4-22-2	Composition of the MMS unit.....	4-369
4-22-3	Composition of the MMS sequence	4-370
4-22-4	Measurement of length of feeler	4-371
4-22-5	Feeler calibration measurement	4-373
4-22-6	Type of measurement.....	4-375
4-23	Workpiece Measuring Unit (WORK MES)	4-394
4-23-1	Procedure for selecting workpiece measuring unit.....	4-394
4-23-2	Setting the unit data in workpiece measuring unit.....	4-394
4-23-3	Setting the sequence data in workpiece measuring unit	4-395
4-23-4	Selection of a measurement type	4-396
4-23-5	Offset value and the direction of offset.....	4-403
4-23-6	Correction judgment	4-406
4-23-7	Retry function and an alarm.....	4-406
4-24	Tool Measuring Unit (TOOL MES).....	4-407
4-24-1	Procedure for selecting tool measuring unit.....	4-407
4-24-2	Setting the unit data in tool measuring unit	4-407
4-24-3	Setting the sequence data in tool measuring unit	4-408
4-24-4	Measuring patterns.....	4-409
5	PRIORITY FUNCTION FOR THE SAME TOOL	5-1
5-1	Priority Machining Order	5-1
5-2	Priority Machining Zone	5-3
5-3	Editing Function and Input Method of Priority Numbers.....	5-5

5-3-1	Input of priority numbers	5-5
5-3-2	Assignment of priority numbers	5-6
5-3-3	Change of priority numbers	5-6
5-3-4	Deletion of all the priority numbers	5-7
5-3-5	How to use the SUB PROG PROC END function	5-8
5-4	Relation between the Sub-program Unit and the Priority Machining Function	5-10
5-5	Relation between the Index Unit and Priority Machining Function	5-11
5-6	Relation between the M-Code Unit and the Priority Machining Function	5-12
5-7	Relation between Multi-workpiece Machining and the Priority Machining Function	5-13
6	COORDINATES MEASUREMENT FUNCTION	6-1
6-1	Method of Measurement of Coordinates by TEACH Function	6-1
6-2	Method of Measurement of Coordinates by MDI-MMS	6-5
7	BARRIER INFORMATION DATA CREATION	7-1
7-1	BARRIER INFORMATION Display	7-1
7-2	Barrier Information Data Registration	7-4
8	TPC DATA CREATION	8-1
8-1	Setting Tool Path Control (TPC) Data	8-1
8-2	Description of Each TPC Data Item	8-4
9	BACKGROUND PROGRAMMING	9-1
10	CASE OF APPEARANCE OF ALARM	10-1
11	THREE-DIGIT G-FORMAT	11-1

11-1 Outline 11-1

11-2 Detailed Description..... 11-1

11-3 Three-digit G-format of MAZATROL Program 11-2

11-4 Various Data Description Using G10 11-23

- NOTE -

1 CALLING UP AND THE END OF THE PROGRAM DISPLAY

1-1 Listing Mode and Creating Mode

The **PROGRAM** display has following two modes:

- Listing mode
This mode serves to display the contents of a program.
- Creating mode
This mode serves to create or edit a program.

Listing mode	Creating mode
<ul style="list-style-type: none"> - Selecting of program to be displayed - Changeover to creating mode - Changeover to TOOL PATH display - Changeover to PROGRAM FILE display - Coordinate measuring function - Checking of program during automatic operation 	<ul style="list-style-type: none"> - Program creation - Program editing (INSERT, ERASE, COPY) - Shape check function - Changeover to listing mode <p style="text-align: right;">See the Notes below.</p>
<ul style="list-style-type: none"> - Help function - Searching function 	

Note 1: During automatic operation, it is impossible to edit the program and the sub-program concerned.

Note 2: During input/output processing (loading, saving, comparing, etc.), it is impossible to edit the program involved.

1-2 Calling Up the PROGRAM Display (Listing Mode)

In order to display the contents of a program (listing mode), perform the following procedure to select the **PROGRAM** display.

(1) Press the display selector key (key located to the left of the menu keys).

➔ The following menu will be displayed.

POSITION	TOOL LAYOUT	PROGRAM	TOOL DATA	C-COND.	PARAM	DIAGNOS	DATA I/O		DISPLAY MAP
----------	-------------	---------	-----------	---------	-------	---------	----------	--	-------------

(2) Press the menu key **PROGRAM**.

➔ This causes the change of the menu and the transition to listing mode.

WORK No.	SEARCH	PROGRAM EDIT	TPC	WPC MSR	TOOL PATH	PROCESS CONTROL	PROGRAM LAYOUT	HELP	PROGRAM FILE
----------	--------	--------------	-----	---------	-----------	-----------------	----------------	------	--------------

1-3 Calling Up the PROGRAM Display (Creating Mode)

In order to edit a program (creating mode), perform the following procedure.

- (1) Press the menu key **WORK No.** in listing mode.

WORK No.	SEARCH	PROGRAM EDIT	TPC	WPC MSR	TOOL PATH	PROCESS CONTROL	PROGRAM LAYOUT	HELP	PROGRAM FILE
----------	--------	-----------------	-----	---------	--------------	--------------------	-------------------	------	-----------------

➔ The display of WORK No. is reversed and the window for work No. selection is displayed on the screen.

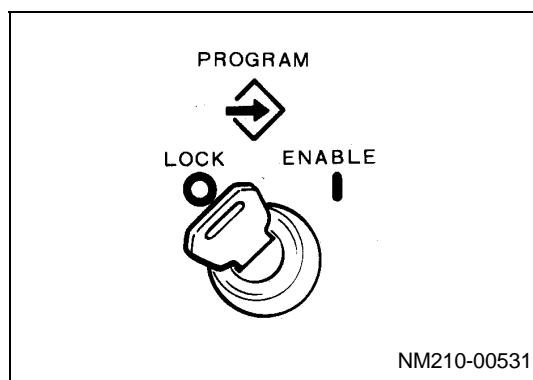
- (2) Enter the workpiece program number by means of numeric keys.

Example: Work No. 1000

Press the keys in this order.

➔ The program is displayed on the screen. At the time of the creation of a new program, nothing is displayed on the screen.

- (3) Place the reprogramming switch in position I ENABLE.



- (4) Change to creating mode.

Press the menu key **PROGRAM EDIT**.

➔ This causes the transition from listing mode to creating mode.

1-4 End of the Program Creation

The following shows the operation to complete the program creation.

The end of the program creation is mentioned also in Part 1, Subsection 4-4-15, "End of the program."

- (1) Press the menu selector key (key located to the right of the menu keys).

➔ The following menu will be displayed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

- (2) Press the menu key **PROGRAM COMPLETE**.

➔ This completes the program creation.

2 EDITION OF DATA

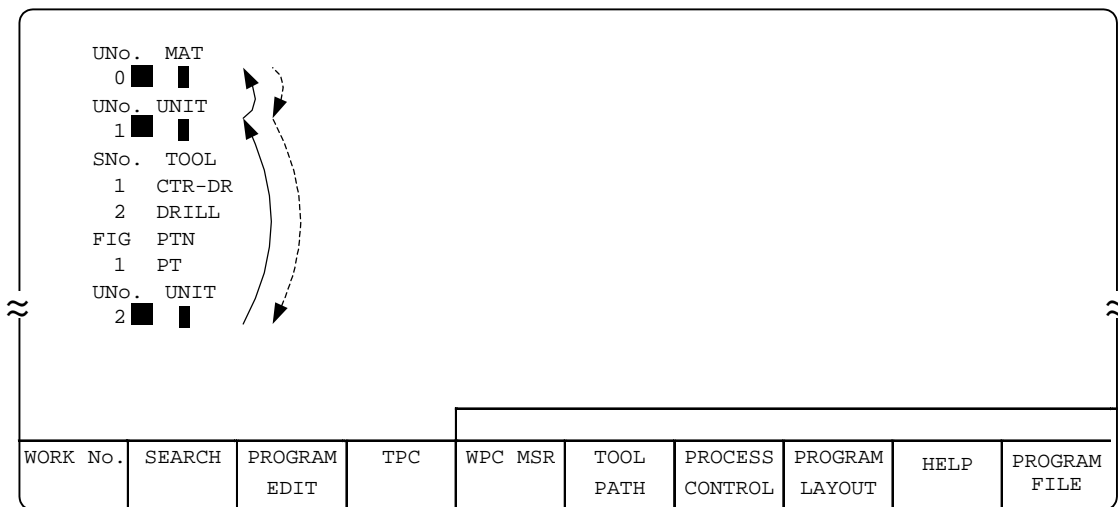
2-1 Cursor Movement

The listing mode and the creating mode offer the different cursor movement methods. Therefore, move the cursor depending on the method appropriate for each mode.

2-1-1 Case of listing mode

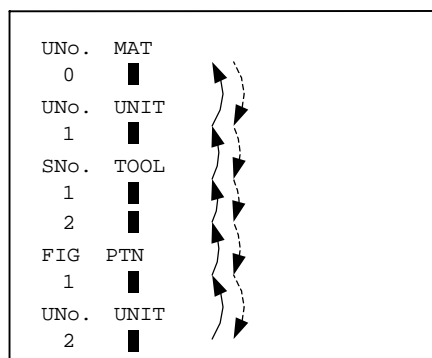
1. Pressing the page key (↑↑, ↓↓)

The cursor moves unit by unit in the direction of the arrows.



2. Pressing the desired cursor key (←, →, ↑, ↓)

The cursor moves line by line in the direction of the arrow.



Note: In listing mode, the cursor can only move up or down. Consequently, even if the ← or → key is depressed, the cursor only moves vertically.

2-1-2 Case of creating mode

In creating mode, the cursor movement is done only by pressing the desired cursor key.

The cursor can be moved freely in the direction of the arrow.

In this case, there is a change of menu in order to display the menu of the article to which the cursor is moved.

2-2 Editing

2-2-1 Editing functions and menus

1. Editing functions

The following 6 editing functions are available:

- SearchUsed to display the intended unit or tool sequence
- InsertionUsed to insert a unit, a tool sequence or a shape sequence
- DeletionUsed to erase the unit, tool sequence or shape sequence
- CopyUsed to copy the program, unit or shape
- HelpUnit data is displayed in the help window graphically.
(See Chapter 3, "HELP FUNCTIONS.")
- End of programUsed to return from creating mode to listing mode

Note: Only search and help functions can be used in the listing mode.

2. Editing menus

In order to use these functions, first of all go to creating mode. Then press the menu selector key in order to display the following menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
------------------	--------	----------	-----	--------	-------	------------	-----------	--------------	------

In listing mode, select **SEARCH** in the menu displayed after pressing the menu key **WORK No.** and entering of the program work No.

WORK No.	SEARCH	PROGRAM EDIT	TPC	WPC MSR	TOOL PATH	PROCESS CONTROL	PROGRAM LAYOUT	HELP	PROGRAM FILE
----------	--------	--------------	-----	---------	-----------	-----------------	----------------	------	--------------

Note: This following subsections describe the functions centered on the creating mode. The listing mode requires a few operations different from those for the creating mode. Refer to the description in the following notes.

2-2-2 Search

This **SEARCH** function is used for displaying on the screen, a unit or a tool sequence which one wishes to check or modify in the program.

There are the following four 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

1. Search for a unit number

This function serves to display the necessary unit from the unit number.

[Menu selection] **SEARCH** → **UNIT No. SEARCH**

- (1) Display the search menu.

Press the menu selector key in creating mode.

→ Pressing of the menu selector key displays the menu for editing.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

Note: In listing mode, this operation can be omitted.

- (2) Press the menu key **SEARCH**.

→ This causes the display of the search menu.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH					
--	--------------------	----------------	----------------	----------------	--	--	--	--	--

- (3) Press the menu key **UNIT No. SEARCH**.

→ 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:

→ 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 number of the entered 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.

[Menu selection] **SEARCH** → **LAST SEARCH**

- (1) Display the search menu.

In creating mode, press the menu selector key.

→ The menu for editing is displayed when the menu selector key is pressed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

Note: This operation is not necessary in listing mode.

- (2) Press the menu key **SEARCH**.
 → The search menu is then displayed.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH					
--	--------------------	----------------	----------------	----------------	--	--	--	--	--

- (3) Press the menu key **LAST SEARCH**.
 → This causes the display of **LAST SEARCH** to reverse and the screen displays the message "LAST SEARCH <INPUT>?".


- (4) Press the input key .
 → 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.

[Menu selection] **SEARCH** → **UNIT SEARCH** → **POINT MACH-ING/LINE MACH-ING/FACE MACH-ING/TURNING** ··· → Unit name

- (1) Display the search menu.
 In creating mode, press the menu selector key.

When the menu selector key is pressed, the menu for editing is displayed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

Note: The listing mode does not require this operation.

- (2) Press the menu key **SEARCH**.
 → The search menu is then displayed.

	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH					
--	--------------------	----------------	----------------	----------------	--	--	--	--	--










- (3) Press the menu key **UNIT SEARCH**.
 → The unit menu is displayed and the message "UNIT NAME SEARCH <INPUT>?" is displayed on the screen.

					UNIT NAME SEARCH <INPUT>?				
POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	TURNING	MANUAL PROGRAM	WPC	OFFSET	END	SHAPE CHK	>>>

(4) Select the name of the unit to be found.


Example: Search under the name of the **RGH CBOR** machining unit.

(a) Pressing of the menu key **POINT MACH-ING** causes the point machining unit menu to display.

UNIT NAME SEARCH <INPUT>? ()									
DRILLING 	RGH CBOR 	RGH BCB 	REAMING 	TAPPING 	BORING 	BK CBOR 	CIRC MIL 	CBOR TAP 	HI SPD. DRL.USE


1) Press the menu key **RGH CBOR**.

➔ The display of RGH CBOR is then reversed.

2) Press the input key .

The cursor then goes to the line of the entered unit and the unit is displayed on the screen.

UNo.	UNIT	
10	RGH CBOR	← The cursor moves here.
SNo.	TOOL	
1	CTR-DR	
2	DRILL	

3) Another pressing of the input key  results in the searching for the following same unit name.

UNo.	UNIT	
24	RGH CBOR	← The cursor moves to the following same unit name.
SNo.	TOOL	
1	CTR-DR	
2	DRILL	

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 **TOOL SEARCH** function, serves to display the sequence line of the required tool on the basis of the name of the tool.

[Menu selection] **SEARCH** → **TOOL SEARCH** → Tool name

(1) Display the search menu.

In creating mode, press the menu selector key.

➔ The menu for editing is displayed when the menu selector key is pressed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

Note: The listing mode does not require this operation.

(2) Press the menu key **SEARCH**.

➔ The search menu is then displayed.






	UNIT No. SEARCH	LAST SEARCH	UNIT SEARCH	TOOL SEARCH					
--	--------------------	----------------	----------------	----------------	--	--	--	--	--

(3) Press the menu key **TOOL SEARCH**.

➔ 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			>>>


- Pressing the menu key >>> permits changing the menu

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR. BAR	CHIP VACUUM		>>>
GENERAL 	GROOVE 	THREAD 	T. DRILL 	T. TAP 		SPECIAL			>>>

Example: Search under the tool name: DRILL

1) Press the menu key **DRILL**.


This will cause the display of **DRILL** to reverse.

2) Press the input key .

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  results in finding the following same tool name.

SNo.	TOOL
1	CTR-DR
2	DRILL
3	DRILL
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.

2-2-3 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

[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 located here.

- (2) In creating mode, press the menu selector key.

The menu for editing is displayed when the menu selector key is pressed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
------------------	--------	----------	-----	--------	-------	------------	-----------	--------------	------

- (3) Press the menu key **INSERT**.

➔ The display of INSERT is reversed and the screen displays the message "LINE INSERT <INPUT>?".

- (4) Press the input key.

➔ This causes the insertion of the empty unit.

```

UNo.   MAT
  0   CBN STL
UNo.   UNIT
  1   DRILLING
SNo.   TOOL
  1   CTR-DR
  2   DRILL
    
```

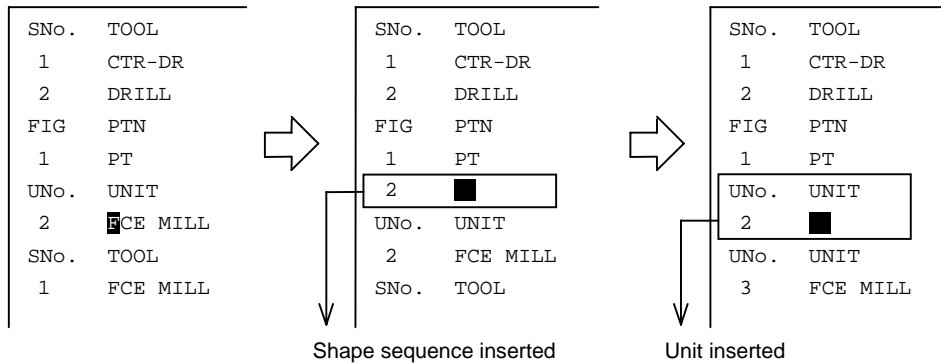
➔

```

UNo.   MAT
  0   CBN STL
UNo.   UNIT
  1   █
UNo.   UNIT
  2   DRILLING
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 menu key **SHAPE END** results in the insertion of the empty unit.



Note 2: When the insertion operation is done in UNo. 0 (common unit), the alarm “409 ILLEGAL INSERTION” is displayed.

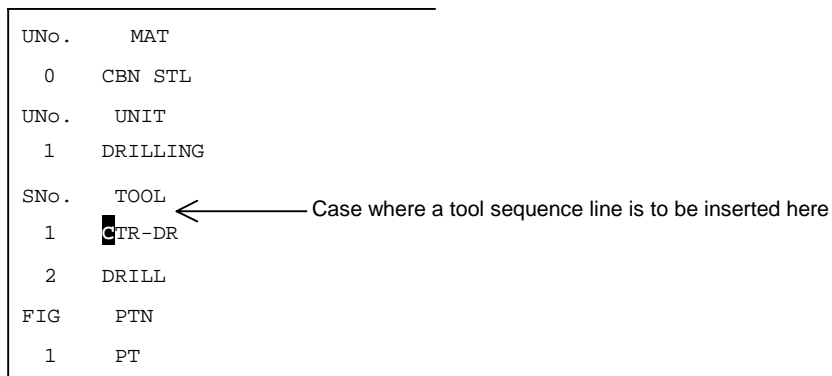
- (5) Enter the data. Refer to the Chapter 4, “PROGRAM CREATION” for the selection of each unit and data setting.

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

[Menu selection] **INSERT**

- (1) Bring the cursor to the next line of the tool sequence to be inserted.



- (2) In creating mode, press the menu selector key.

The menu for editing is displayed when the menu selector key is pressed.

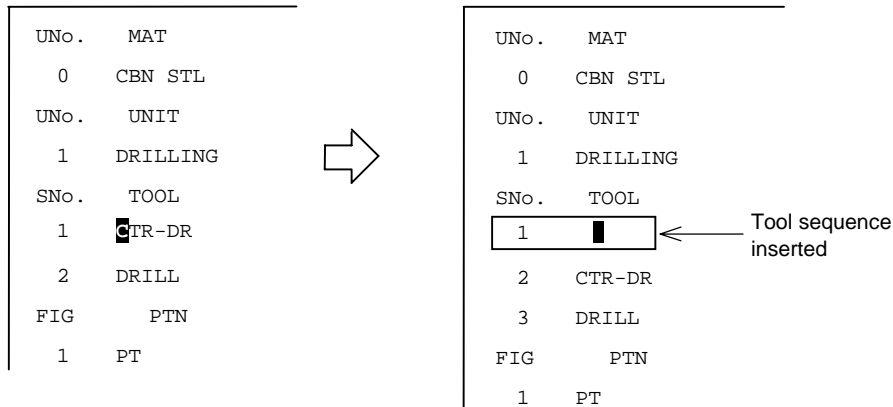
PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
------------------	--------	----------	-----	--------	-------	------------	-----------	--------------	------

- (3) Press the menu key **INSERT**.

➔ The display of **INSERT** is reversed and the screen displays the message “LINE INSERT <INPUT>?”.

(4) Press the input key.

➔ This causes the insertion of the empty tool sequence line.



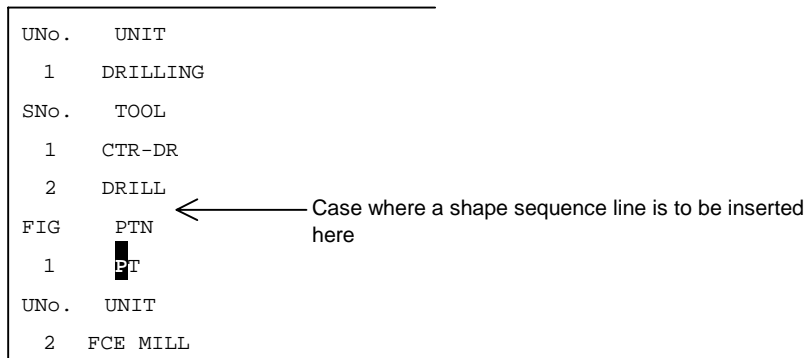
(5) Enter the data. Refer to the Chapter 4, “PROGRAM CREATION” for the selection of each unit and data setting.

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

[Menu selection] **INSERT**

(1) Bring the cursor to the next line of the shape sequence to be inserted.




(2) In creating mode, press the menu selector key.

The menu for editing is displayed when the menu selector key is pressed.

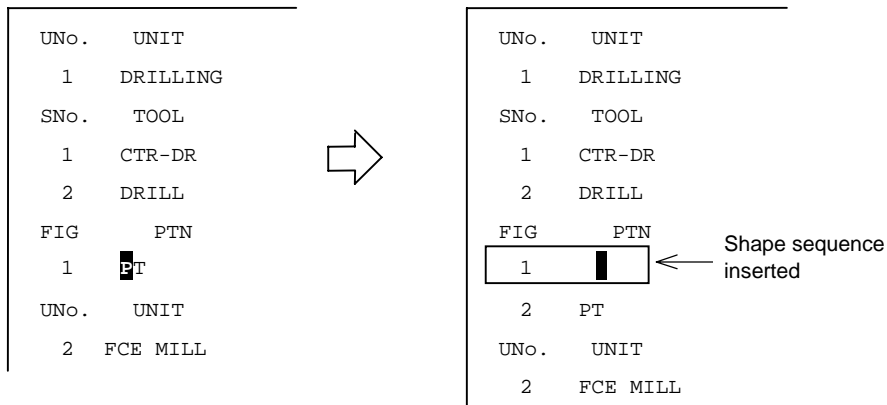
PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

(3) Press the menu key **INSERT**.

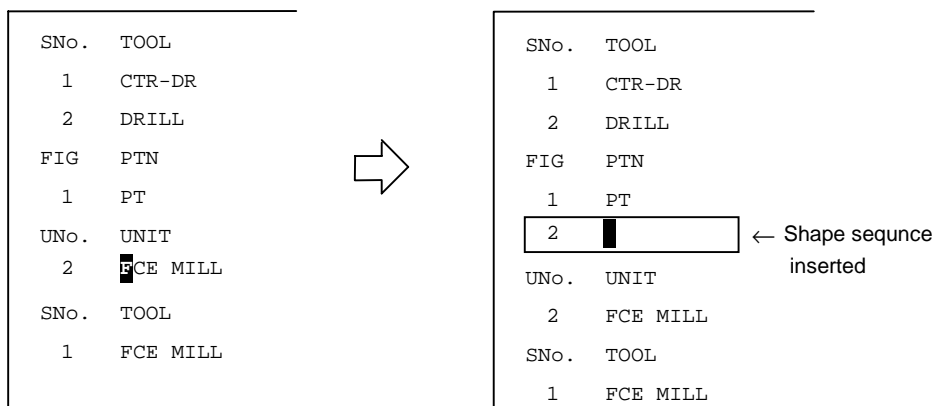
➔ The display of INSERT is reversed and the screen displays the message “LINE INSERT <INPUT>?”.

(4) Press the input key .

→ This causes the insertion of the empty shape sequence line.



Note: When the cursor is located on the line of the unit which follows the line of the shape sequence, the empty shape sequence is inserted as follows.



(5) Enter the data. Refer to the Chapter 4, "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.

2-2-4 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

The line to be deleted is determined by the cursor position.

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 line is to be deleted
SNo.  TOOL
    1  CTR-DR
    2  DRILL
    
```


- (2) In creating mode, press the menu selector key.

➔ When the menu selector key is pressed, the menu for editing is displayed.

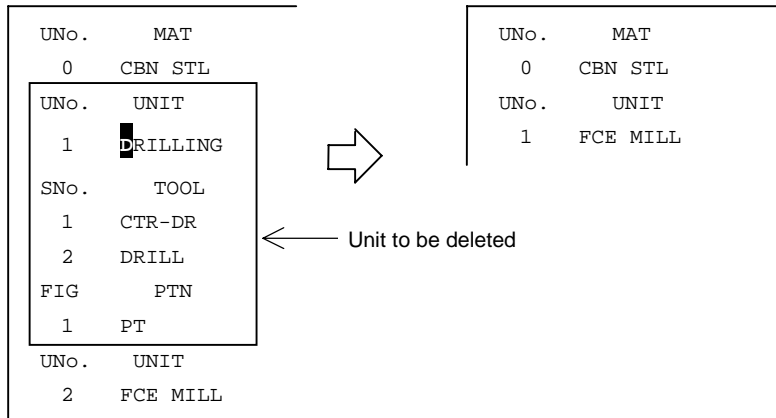
PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

- (3) Press the menu key **ERASE**.

➔ The display of **ERASE** is reversed and the screen displays the message "ERASE <INPUT>?".

- (4) Press the input key .

➔ The unit on which the cursor is located is then deleted. The tool sequence and the shape sequence in this unit will equally be deleted.



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 sequence to be deleted.

```

UNo.  MAT
    0  CBN STL
UNo.  UNIT
    1  DRILLING
SNo.  TOOL
    1  DRILL ← Case where this line is to be deleted
UNo.  UNIT
    2  FCE MILL
    
```


(2) In creating mode, press the menu selector key.

➔ When the menu selector key is pressed, the menu for editing is displayed.

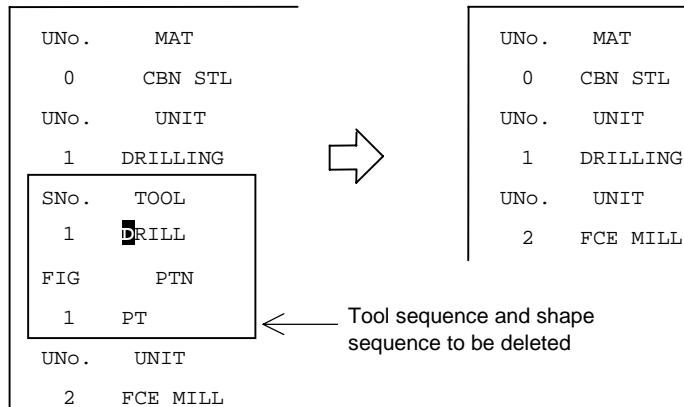
PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

(3) Press the menu key **ERASE**.

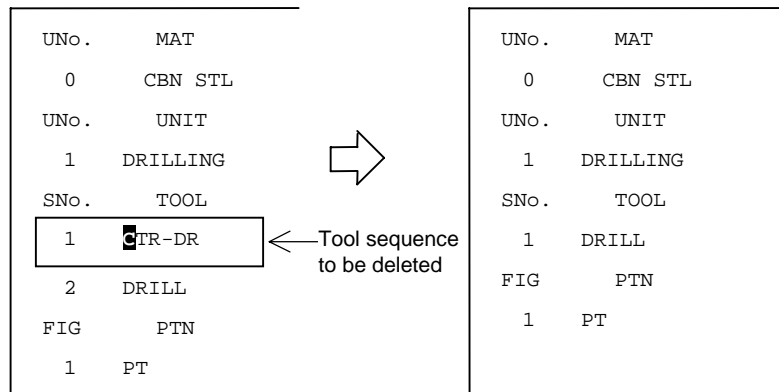
➔ The display of **ERASE** is reversed and the screen displays the message "ERASE <INPUT>?".

(4) Press the input key .

- If there is only one tool sequence line, the tool sequence line in which the cursor is located and the shape sequence which follows will be deleted at the same time.



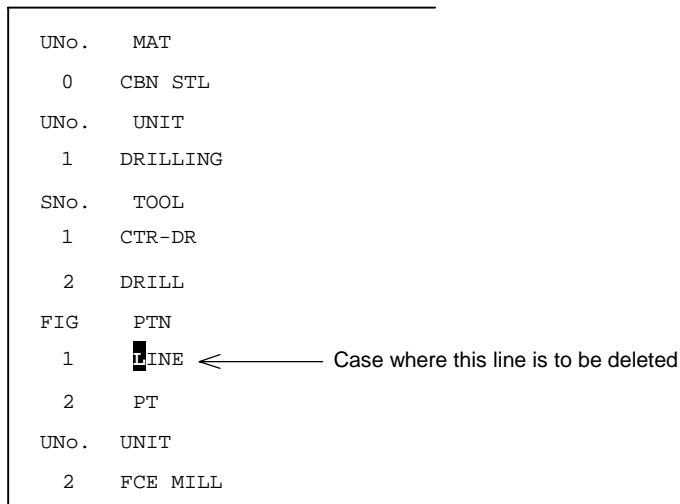
- If there is more than 1 tool sequence line, the tool sequence line on which the cursor is located will be deleted.



3. Deletion of the shape sequence

[Menu selection] **ERASE**

- (1) Place the cursor on the shape sequence to be deleted.




- (2) In creating mode, press the menu selector key.

➔ When the menu selector key is pressed, the menu for editing is displayed.

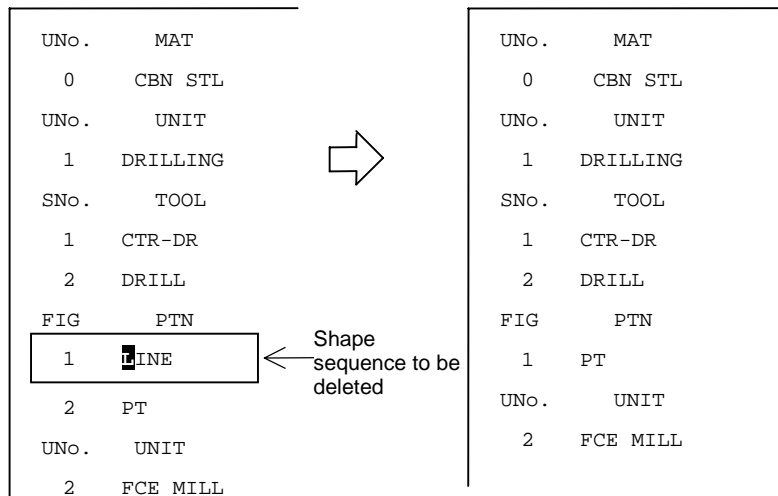
PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

- (3) Press the menu key **ERASE**.

➔ The display of ERASE is reversed and the screen displays the message "ERASE <INPUT>?".

(4) Press the input key .

→ The shape sequence on which the cursor is located is then deleted.



2-2-5 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

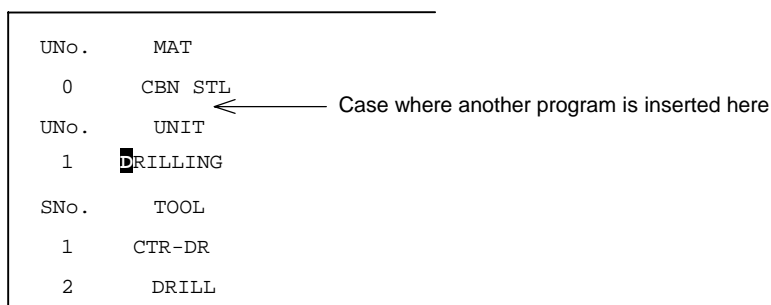
1. Copying of a program

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.



Note: When the cursor is not located on the unit line or 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) Press the menu selector key to display the menu for editing.


PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
------------------	--------	----------	-----	--------	-------	------------	-----------	--------------	------

(3) Press the menu key **PROGRAM COPY**.

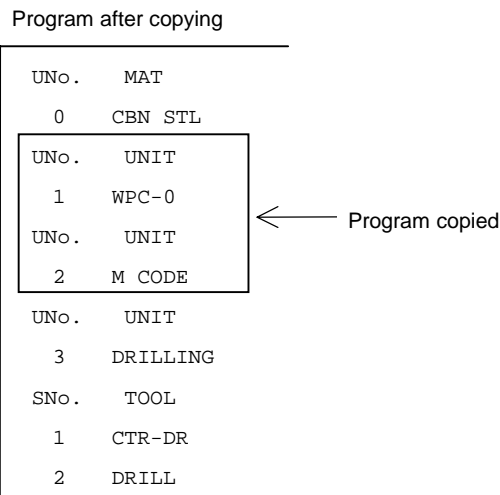
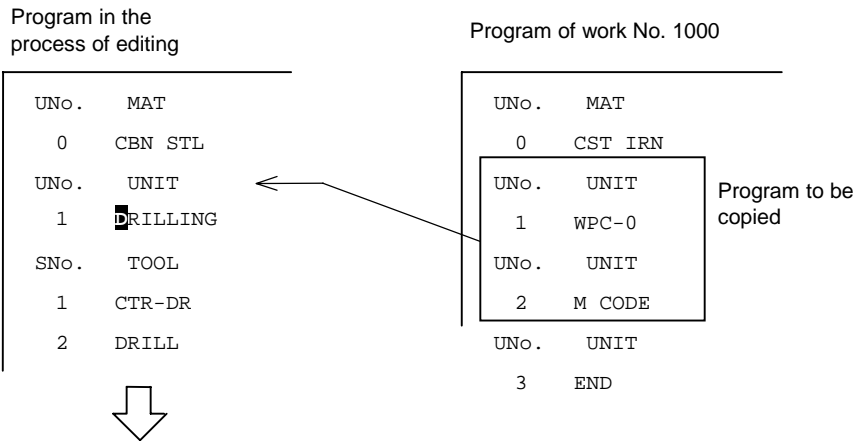
➔ The display of **PROGRAM COPY** is then reversed and the screen displays the window for work No. selection.

(4) Enter the program workpiece number to be copied.

Example: Work No. 1000

Press the following keys: **1** **0** **0** **0** 

➔ The program work No.1000 is then copied.



Note 1: A program in the process of editing cannot be copied. Any attempt to make such a copy will cause the alarm “455 SAME PROGRAM No. DESIGNATED” to be displayed.

Note 2: When a program work No. which is not recorded is entered, the alarm “405 PROGRAM No. NOT FOUND” is displayed.

Note 3: When an EIA/ISO program work No 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	
UNo.	UNIT	← Case where another unit is inserted here
1	DRILLING	
SNo.	TOOL	
1	CTR-DR	
2	DRILL	

Note: When the cursor is not located on the unit line or the cursor is located on the common unit, the alarm "454 CURSOR POSITION INCORRECT" will be displayed when selecting the menu key **UNIT COPY**.

- (2) Press the menu selector key in creating mode.

➔ When the menu selector key is pressed, the menu for editing is displayed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

- (3) Press the menu key **UNIT COPY**.

➔ The display of **UNIT COPY** is then reversed and the screen displays the window for selection of a work No.

- (4) Enter the program workpiece number containing the unit to be copied.

Example: Work No. 1000

Press the following keys:

➔ When the program work No. 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 ↔

➔ Unit number 1 in the program of workpiece No. 1000 is then copied.

Program in the process of editing

```

UNo.  MAT
  0  CBN STL
UNo.  UNIT
  1  DRILLING
SNo.  TOOL
  1  CTR-DR
  2  DRILL
  
```



Program after copying

```

UNo.  MAT
  0  CBN STL
UNo.  UNIT
  1  FCE MILL
SNo.  TOOL
  1  FCE MILL
FIG.  PTN
  1  SQR
UNo.  UNIT
  2  DRILLING
SNo.  TOOL
  1  CTR-DR
  2  DRILL
  
```

Program of work No. 1000

```

UNo.  MAT
  0  CST IRN
UNo.  UNIT
  1  FCE MILL
SNo.  TOOL
  1  FCE MILL
FIG.  PTN
  1  SQR
UNo.  UNIT
  2  RGH CBOR
  
```

Unit to be copied

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 workNo. is entered, the alarm "405 PROGRAM No. NOT FOUND" is displayed.

Note 3: When a work No. 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.

```

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) In creating mode, press the menu selector key.
 - ➔ When the menu selector key is pressed, the menu for editing is displayed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

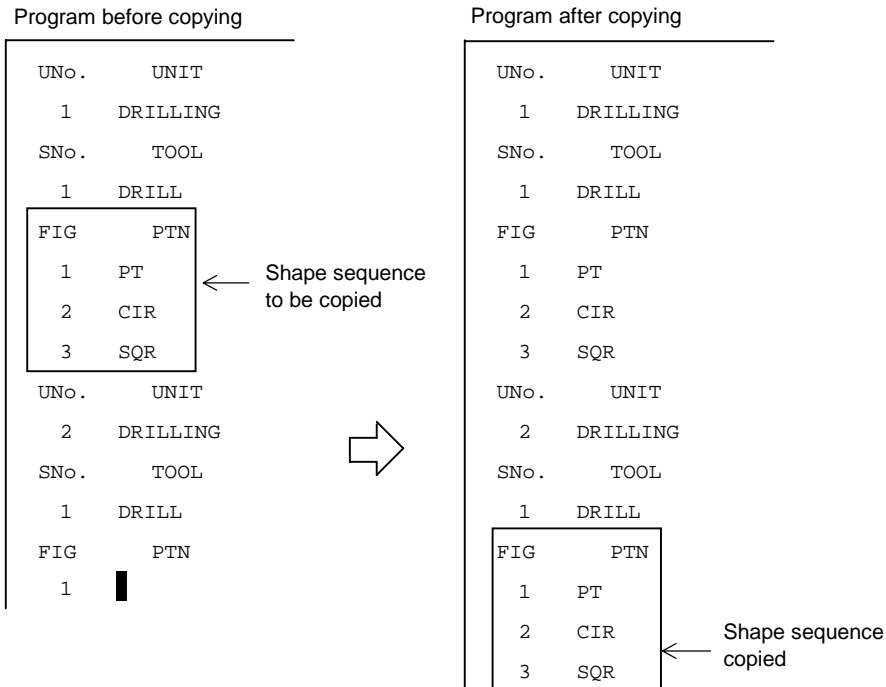
- (3) Press the menu key **SHAPE COPY**.
 - ➔ The display of **SHAPE COPY** is then reversed and the screen displays the message "UNIT NUMBER <INPUT>?".

(4) Enter the number of the unit containing the shape sequence to be copied.

Example: Unit number 1

Press the following keys: 1 ↔

➔ The shape sequence under unit No. 1 is then 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.

2-2-6 End of program

This **PROGRAM COMPLETE** function is used to go from creating mode to listing mode.

[Menu selection] **PROGRAM COMPLETE**

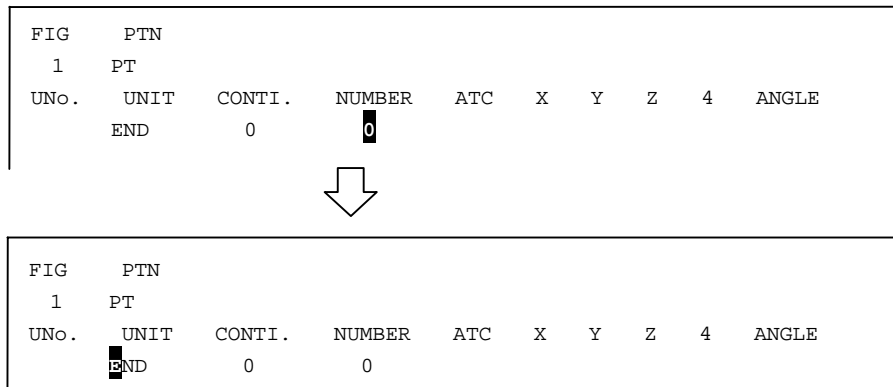
(1) In creating mode, press the menu selector key.

➔ When the menu selector key is pressed, the menu for editing is displayed.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
------------------	--------	----------	-----	--------	-------	------------	-----------	--------------	------

(2) Press the menu key **PROGRAM COMPLETE**.

➔ This causes the cursor to move to the left end and the transmission from creating mode to listing mode.



3 HELP FUNCTIONS

<Functions>

The functions for aiding operator entry using the windows shown below are available to enter data when creating and editing programs.

These windows can be displayed by pressing the corresponding menu keys.

3-1 MAZATROL Help

The MAZATROL help display can be displayed by pressing the menu key **HELP** while the cursor remains set to a unit.

When the window is displayed, the unit line to be set will be displayed at the top of the screen and the data item corresponding to the data type specified by the cursor will be displayed in reverse form.

The help window can be displayed for the following units:

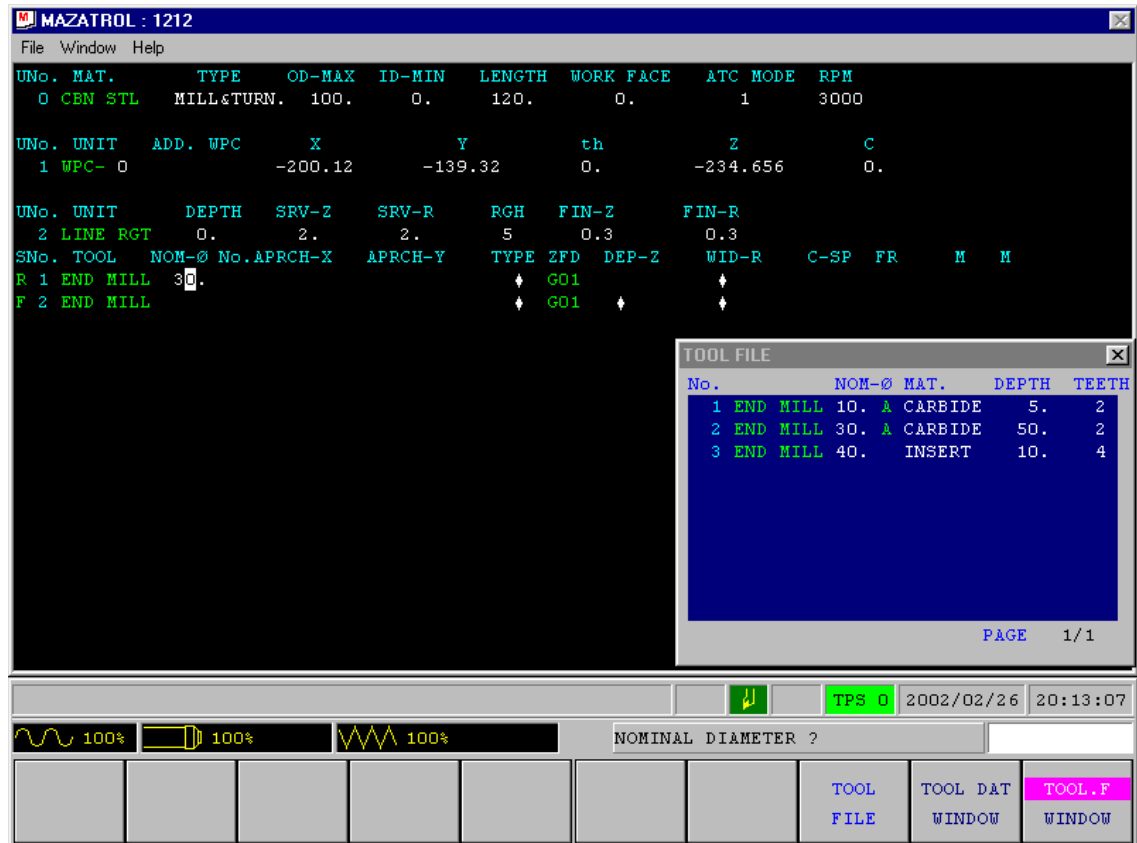
- Common unit
- Machining unit
- Fundamental coordinates system unit (WPC)
- Auxiliary coordinates system unit (OFFSET)
- Fundamental coordinate shifting unit
- Workpiece transfer unit
- Head selection unit
- Shape sequence of the MMS unit (except for measurement of workpiece inclination and feeler calibration measurement)
- Shape sequence of the workpiece measuring unit (except for external milling measurement and external turning measurement)
- Shape sequence of the tool measuring unit
- Shape sequence of the bar-materials machining, copy-machining, and grooving units

3-2 TOOL FILE Window

The TOOL FILE window can be displayed by pressing the menu key **TOOL. F WINDOW** while the cursor remains set to tool sequence data item “NOM-φ” of a line/face machining unit, or an end mill or chamfering cutter data item “NOM-φ” of the tool sequence line of a point-machining unit.

The tool data items corresponding to that sequence will be displayed in the window at that time. Press the page key to display the next page.

Example:



D736P0078E

3-3 TOOL DATA Window

The TOOL DATA window can be displayed by pressing the menu key **TOOL DAT WINDOW** while the cursor remains set to either “C-SP” or “FR” of the tool sequence line.

The tool data items corresponding to that sequence will be displayed in the window at that time. Press the page key to display the next page.

Example:

The screenshot shows the MAZATROL 1212 interface. The main display area contains the following tool data:

UNo.	MAT.	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
0	CBN STL	MILL&TURN.	100.	0.	120.	0.	1	3000

UNo.	UNIT	ADD. WPC	X	Y	th	Z	C
1	WPC-0		-200.12	-139.32	0.	-234.656	0.

UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
2	LINE RGT	0.	2.	2.	5	0.3	0.3

SMo.	TOOL	NOM-Ø	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R 1	END MILL	30.	A	?	?	GO1	1.7						
F 2	END MILL					GO1							

The 'TOOL DATA' window displays the following information:

TNo.		NOM. NOM-Ø	MAT.	STAT.
10	END MILL	20.	H	HSS
11	END MILL	22.		COAT.HSS
23	END MILL	30.	A	CARBIDE

The control panel at the bottom shows a status bar with 'TPS 0', date '2002/02/26', and time '20:15:40'. Below the status bar are three waveform indicators (100%), a 'CUTTING SPEED, AUTO-><MENU> ?' display, and a row of function keys including 'TOOL DAT WINDOW'.

D736P0079E

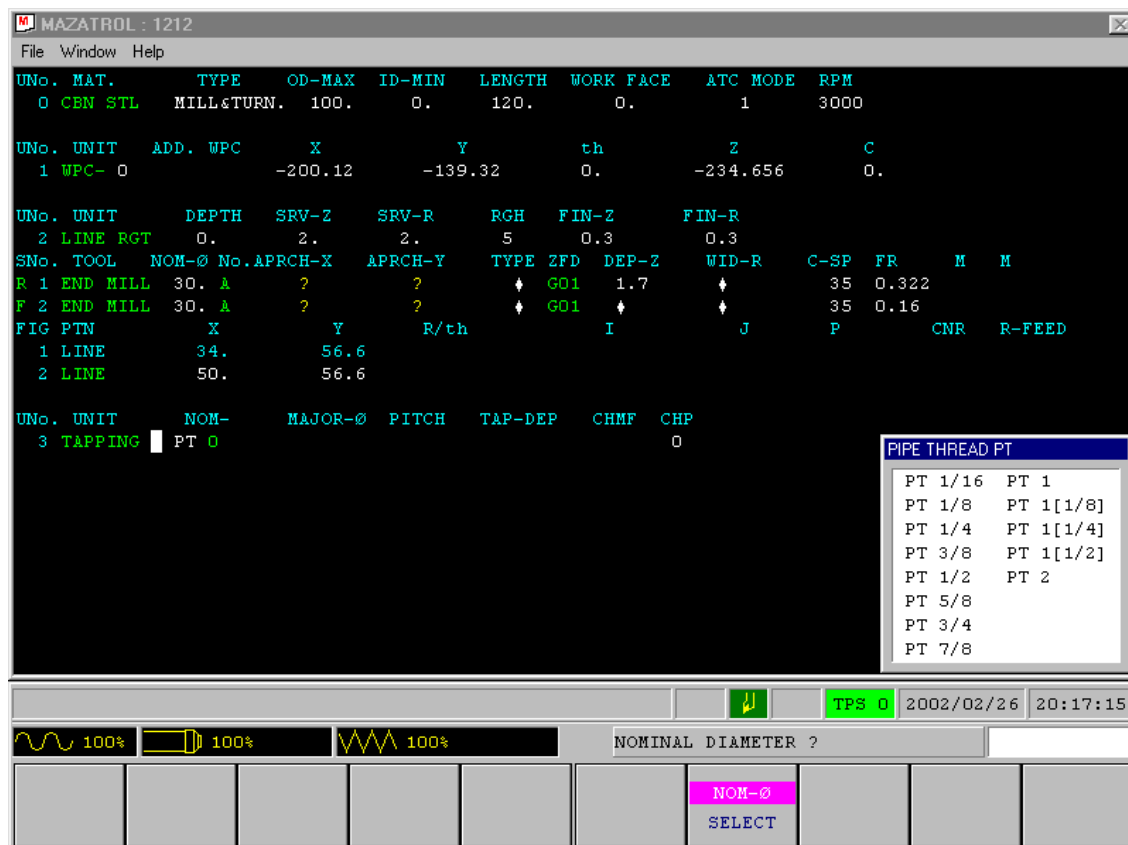
3-4 Tap Nominal Diameter Window

The tap nominal diameter window can be displayed by pressing the menu key **NOM-φ SELECT** while the cursor remains set to “NOM” of the tapping or counterbore-tapping unit and unified screw or pipe screw has been selected.

Select the appropriate tap diameter in the window using the cursor key to make the display to be reversed. The tap diameter will be automatically set in the program by pressing the input key.

Press the page key to display the next page.

Example:



D736P0080E

4 PROGRAM CREATION

This chapter only describes the functions of the various units to be created using the MAZATROL program creation procedures which are discussed in Part 1; data-setting procedures are not described here.

As discussed previously in Part 1, Section 4-4, "Creation of Simple Programs," programming will use an interactive method in which you are to enter and set necessary data with the numeric keys or menu keys in accordance with the messages displayed on the CRT screen. Carefully read the description of functions that is given here. You will then be able to carry out operations easily. If you find part of the description unclear, then use the window functions that will give you detailed information on operations, or see the description of operations given in Part 1.

4-1 Common Unit

This concerns the unit which is always set at the head of the program at the time of the programming.

Data for the common unit are set on the **PROGRAM** display in creating mode.

When creating a new program, only the common unit line shown below is displayed on the upper part of the screen.

1. Data setting in the common unit

Two modes for creating the common unit are provided: the "workpiece scheme" and the "initial-point scheme."

By selecting either creating mode, the tool path between units, such as for "from face milling to drilling", is determined automatically according to the data entered in the common unit.

The data items to be set differ according to the selected creating mode.

Workpiece scheme (MILLING & TURN.)

Since the shape of the workpiece is to be specified in cylindrical form, the connection path between units moves along the outside of the workpiece.

Select the workpiece scheme to machine more than one face of the workpiece or to perform the machining operations that include turning.

UNo.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
0	<input type="checkbox"/>	MILL&TURN.	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

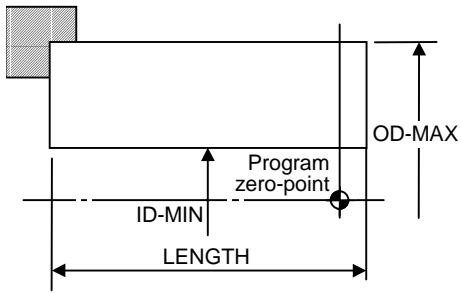
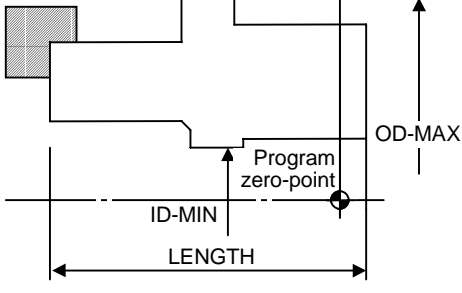
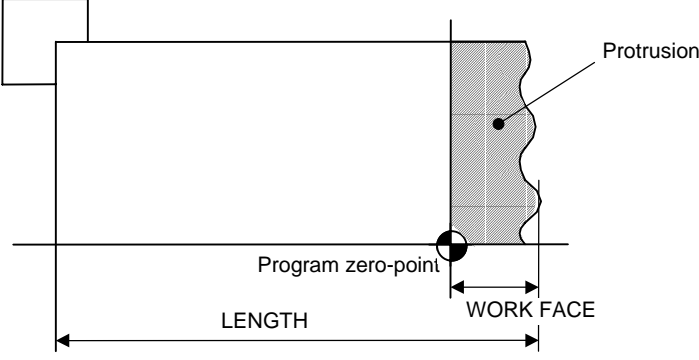
Initial-point scheme (ONLY MILLING)

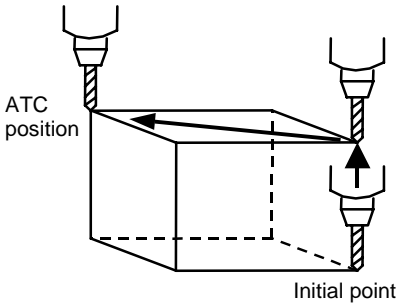
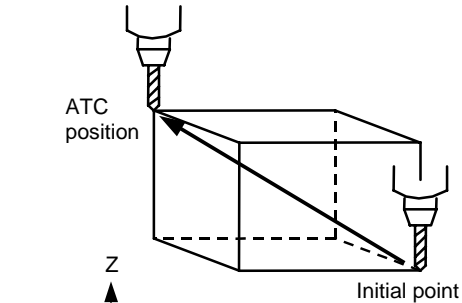
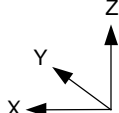
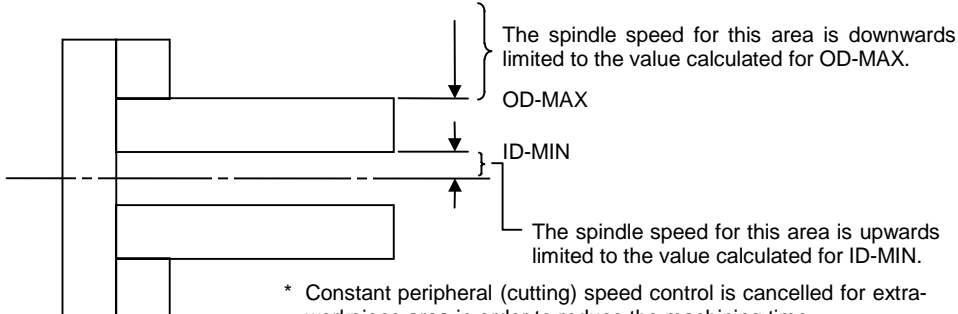
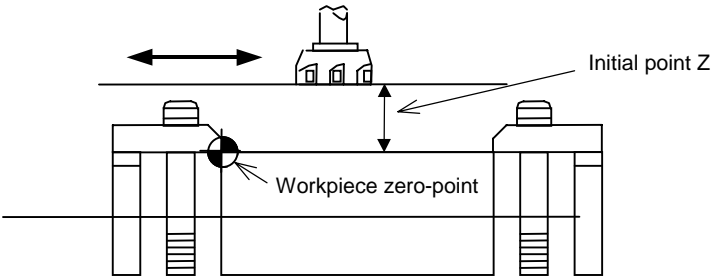
Since a Z-axial position at which the tool does not interfere with the workpiece or jigs is to be specified as an initial point, the connection path between units moves away to that position.

The initial-point scheme is suitable for milling one face of the workpiece.

This scheme allows multiple workpieces of the same type to be machined repeatedly.

UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	<input type="checkbox"/>	ONLY MILL.	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Entering										
MAT	<p>Specify the material of the workpiece using the menu key to perform the automatic determination of the cutting conditions.</p> <p>The names of workpiece materials in the menu are the same as those which are listed on the CUTTING CONDITION (W.-MAT./T.-MAT.) display.</p> <p>Also, these names of workpiece materials are already registered in the system and they are MAZAK-recommended ones.</p> <p>To register new workpiece material names, refer to Section 8-1, "CUTTING CONDITION (W.-MAT./T.-MAT.) Display," of the relevant Operating Manual.</p> <p>Example:</p> <table border="1" data-bbox="454 571 1390 638"> <tr> <td>CST IRN</td> <td>DUCT IRN</td> <td>CBN STL</td> <td>ALY STL</td> <td>STNLESS</td> <td>ALUMINUM</td> <td>L.C.STL</td> <td>AL CAST</td> <td>>>></td> <td></td> </tr> </table>	CST IRN	DUCT IRN	CBN STL	ALY STL	STNLESS	ALUMINUM	L.C.STL	AL CAST	>>>	
CST IRN	DUCT IRN	CBN STL	ALY STL	STNLESS	ALUMINUM	L.C.STL	AL CAST	>>>			
TYPE	<p>Select the mode for creating the common unit either "workpiece scheme(MILLING & TURN.)" or the "initial-point scheme (ONLY MILLING)."</p> <p>When creating a new program, MILLING & TURN. mode is selected automatically.</p>										
OD-MAX, ID-MIN, LENGTH	<p>Set the maximum outside diameter, minimum inside diameter, and maximum length, respectively, of the workpiece.</p> <div style="display: flex; justify-content: space-around;"> <div data-bbox="438 907 901 1243">  <p>Round bar materials</p> </div> <div data-bbox="933 907 1396 1243">  <p>Molded materials</p> </div> </div> <p style="text-align: right;">T4P017</p> <p>Set the workpiece length, including the edge protrusion (edge section to be cut), in item [4].</p>										
WORK FACE	<p>Set the length of the workpiece edge protrusion in the Z-axis direction.</p> <div style="text-align: center;">  </div> <p style="text-align: right;">T4P019</p> <p>The workpiece edge protrusion refers to a section to be cut during an edge-machining unit (FACING). For units other than edge-machining unit, 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 edge-machining unit must be selected before selecting an unit involving other machining operations. Either 0 or a plus value must always be set for this item.</p>										

Cursor position	Entering
<p>ATC MODE</p>	<p>Specify the mode of movement from the initial point to the ATC position at the time of ATC.</p> <p><Independent movement on each axis: 0> <Simultaneous movement on all axes: 1></p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;">  <p>ATC position</p> <p>Initial point</p> <p>Movement on Z axis only, then movement on X and Y axes</p> </div> <div style="text-align: center;">  <p>ATC position</p> <p>Initial point</p> <p>Simultaneous movement on X, Y and Z axes</p> </div> </div> <div style="text-align: center; margin-top: 10px;">  </div> <p style="text-align: right;">M3P078</p> <p>The data of the ATC MODE are only valid in the related program. Consequently, during the execution of the sub-program, the movement is done in conformity with the data entered in the ATC MODE in the sub-program.</p> <p>Note: In the case of designation of 1 (simultaneous movement), check that the tool does not interfere with the workpiece or with the fastening device of the workpiece.</p>
<p>RPM</p>	<p>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.</p> <p>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.</p> <div style="text-align: center;">  <p style="text-align: right;">The spindle speed for this area is downwards limited to the value calculated for OD-MAX.</p> <p style="text-align: right;">OD-MAX</p> <p style="text-align: right;">ID-MIN</p> <p style="text-align: right;">The spindle speed for this area is upwards limited to the value calculated for ID-MIN.</p> <p>* Constant peripheral (cutting) speed control is cancelled for extra-workpiece area in order to reduce the machining time.</p> </div>
<p>INITIAL-Z</p>	<p>Specify the position on the Z-axis (Z plane) as an absolute value from the workpiece zero-point in order to prevent interference of the tool edge with the workpiece or a fastening device in case of movement on the X- and Y-axes. Without specification, an alarm will be displayed.</p> <p>These data will be used in the same way in the case of the use of the auxiliary coordinates system. In automatic operating mode, this height is taken into consideration for the positioning of the tool.</p> <div style="text-align: center;">  <p style="text-align: right;">Initial point Z</p> <p style="text-align: center;">Workpiece zero-point</p> </div> <p style="text-align: right;">M3P077</p>

Cursor position	Entering																																																																									
MULTI MODE	<p>Specify the multi-workpiece machining mode.</p> <table border="1" style="width: 100%; border-collapse: collapse; margin: 10px 0;"> <tr> <td style="width: 10%;">MULTI OFF</td> <td style="width: 10%;">MULTI 5*2</td> <td style="width: 10%;">OFFSET TYPE</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> </tr> </table> <p>- MULTI OFF: Ordinary machining (machining of one workpiece) - MULTI 5 * 2: Machining of several workpieces (for machining with reference to jig) - OFFSET TYPE: Machining of several workpieces deviated arbitrarily</p> <p>(1) MULTI 5*2 When pressing the menu key MULTI 5 * 2, enter data in the columns MULTI FLAG, PITCH-X and PITCH-Y with numeric keys and press the input key.</p> <p>(2) OFFSET TYPE By fixing the amount of X, Y offset for each workpiece, it is possible to machine the workpieces arranged in arbitrary positions. Therefore, unlike the case of MULTI 5 * 2, there are no restrictions that the workpieces must be arranged in an equidistant manner or in 2 rows/5 rows. Press the menu key OFFSET TYPE and enter each coordinate as offset data of the common unit, as an amount of offset with relation to the programmed workpiece zero-point.</p> <table border="1" style="width: 100%; border-collapse: collapse; margin: 10px 0;"> <thead> <tr> <th>UNo.</th> <th>MAT</th> <th>TYPE</th> <th>INITIAL-Z</th> <th>ATC MODE</th> <th>MULTI MODE</th> <th>MULTI FLAG</th> <th>PITCH-X</th> <th>PITCH-Y</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>CBN</td> <td>STL ONLY</td> <td>MILL</td> <td>50</td> <td>0</td> <td>OFFSET TYPE</td> <td>◆</td> <td>◆</td> </tr> <tr> <td>OFS</td> <td>X</td> <td>Y</td> <td>th</td> <td>Z</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>1</td> <td>X₁</td> <td>Y₁</td> <td>th₁</td> <td>Z₁</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>X₂</td> <td>Y₂</td> <td>th₂</td> <td>Z₂</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>3</td> <td>X₃</td> <td>Y₃</td> <td>th₃</td> <td>Z₃</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>↓</td> <td>↓</td> <td>↓</td> <td>↓</td> <td>↓</td> <td></td> <td></td> <td></td> <td></td> </tr> </tbody> </table> <p>It is possible to set a maximum of 10 OFS numbers. The relation between the position of the workpiece and the coordinates is as follows:</p> <p>- Case of th = 0</p> <p style="text-align: right;">M3P079</p> <p>- Case of designation of th</p> <p style="text-align: right;">M3P080</p>	MULTI OFF	MULTI 5*2	OFFSET TYPE								UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y	0	CBN	STL ONLY	MILL	50	0	OFFSET TYPE	◆	◆	OFS	X	Y	th	Z					1	X ₁	Y ₁	th ₁	Z ₁					2	X ₂	Y ₂	th ₂	Z ₂					3	X ₃	Y ₃	th ₃	Z ₃					↓	↓	↓	↓	↓				
MULTI OFF	MULTI 5*2	OFFSET TYPE																																																																								
UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y																																																																		
0	CBN	STL ONLY	MILL	50	0	OFFSET TYPE	◆	◆																																																																		
OFS	X	Y	th	Z																																																																						
1	X ₁	Y ₁	th ₁	Z ₁																																																																						
2	X ₂	Y ₂	th ₂	Z ₂																																																																						
3	X ₃	Y ₃	th ₃	Z ₃																																																																						
↓	↓	↓	↓	↓																																																																						

Cursor position	Entering																				
MULTI FLAG	<p>Specify the execution or non-execution of the machining for each workpiece in case of MULTI 5*2.</p> <p>Non-execution 0 Execution 1</p> <p>A maximum of 10 identical workpieces can be machined with only one program.</p> <p>The relation of the value of 10 figures in the MULTI FLAG column and the workpiece positions is as follows:</p> <div style="text-align: center;"> <table border="1"> <tr> <td>MULTI MODE</td> <td>MULTI FLAG</td> <td>PITCH-X</td> <td>PITCH-Y</td> </tr> <tr> <td>5 * 2</td> <td>0 0 0 0 0 0 0 0 0 0</td> <td></td> <td></td> </tr> <tr> <td></td> <td>↑ ↑ ↑ ↑ ↑ ↑ ↑ ↑</td> <td></td> <td></td> </tr> <tr> <td></td> <td>10 9 8 7 6 5 4 3 2 1</td> <td></td> <td></td> </tr> </table> </div> <p>Example of entering:</p> <div style="display: flex; align-items: center;"> <div style="border: 1px solid black; padding: 5px; margin-right: 20px;"> <p style="text-align: center;">Position of workpieces</p> </div> <div style="text-align: center;"> <table border="1"> <tr> <td>MULTI FLAG</td> </tr> <tr> <td>1 0 1 0 1 0 1 0 1 0</td> </tr> <tr> <td>↑ ↑ ↑ ↑</td> </tr> <tr> <td>10 8 6 4 2</td> </tr> </table> </div> </div>	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y	5 * 2	0 0 0 0 0 0 0 0 0 0				↑ ↑ ↑ ↑ ↑ ↑ ↑ ↑				10 9 8 7 6 5 4 3 2 1			MULTI FLAG	1 0 1 0 1 0 1 0 1 0	↑ ↑ ↑ ↑	10 8 6 4 2
MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y																		
5 * 2	0 0 0 0 0 0 0 0 0 0																				
	↑ ↑ ↑ ↑ ↑ ↑ ↑ ↑																				
	10 9 8 7 6 5 4 3 2 1																				
MULTI FLAG																					
1 0 1 0 1 0 1 0 1 0																					
↑ ↑ ↑ ↑																					
10 8 6 4 2																					
PITCH-X PITCH-Y	<p>Set the pitch in the directions X and Y between the located workpieces in case of MULTI 5*2.</p> <div style="text-align: center;"> </div> <p>The workpieces are arranged with an equal distance of the values PITCH-X, PITCH-Y from the position of the reference workpiece.</p> <p>Note: Minus data cannot be entered in the items PITCH-X, -Y.</p>																				

- Note 1:** The multi-workpiece machining with the manual program mode unit requires the absolute position command of the three axes in the first sequence.
- Note 2:** The M-code specified at the end of the tool sequence is executed in case of the machining of each workpiece which undergoes multi-workpiece machining. The M-code entered in the M-code unit is only executed once.
- Note 3:** In the program containing the multi-workpiece machining, priority function for the same tool is valid.

4-2 Fundamental Coordinates System Unit (WPC)

This unit serves to specify the distance from the machine zero-point to the workpiece zero-point. When data setting for the common unit has been finished, the following menu is displayed for the next unit setting. If this menu is not displayed, press the menu selector key (key located at the right of the menu keys) in creating mode.

1. Menu selection

Press the menu key **WPC**.

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

2. Data setting in fundamental coordinates system unit

UNo.	UNIT	ADD. WPC	X	Y	th	Z	4	5
1	WPC-	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Note: Instead of “4” or “5”, the axis name is indicated on the NC display.

Cursor position	Entering
UNIT	Several fundamental coordinates systems (WPC) can be specified in one program. They are differentiated each other by their numbers. Permissible input range (0 to 99)
ADD. WPC (option)	Set data in this item only if you are to use additional fundamental coordinate data (coordinates A to J) and workpiece offset data (G54 to G59), instead of using the normal WPC data. If data is set, the modal status display of the POSITION display will remain set to G54.1 P0 and external workpiece offsets will not be displayed exactly.
X	Enter the X coordinate of the workpiece zero-point in the machine coordinates system. Permissible input range (0 to ±99999.999)
Y	Enter the Y coordinate of the workpiece zero-point in the machine coordinates system. Permissible input range (0 to ±99999.999)
th	Enter the angle formed by the machine coordinates system and the workpiece coordinates system. Permissible input range (0 to ±999.999)
Z	Enter the Z coordinate of the workpiece zero-point in the machine coordinates system. Permissible input range (0 to ±99999.999)
4	Enter the 4th axis coordinate of the workpiece zero-point in the machine coordinates system. For the rotational axis enter the angle of the 4th axis formed by the machine coordinates system and the workpiece coordinates system. Permissible input range (0 to ±99999.999) Permissible input range for the rotational axis (0 to ±9999.999)
5	Enter 5th axis coordinate of the workpiece zero-point in the machine coordinates system. For the rotational axis enter the angle of the 5th axis formed by the machine coordinates system and the workpiece coordinates system. Permissible input range (0 to ±99999.999) Permissible input range for the rotational axis (0 to ±9999.999)

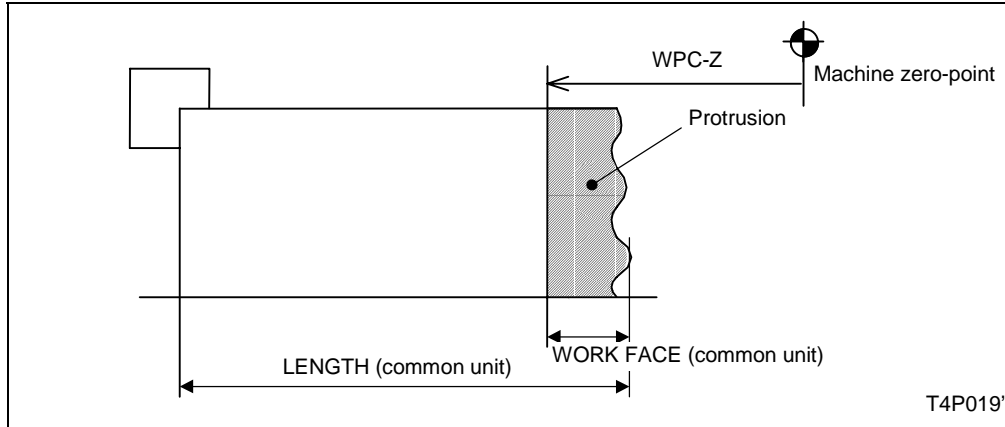
When the cursor is positioned at coordinates, the WPC zero-point can be set to the turning center by pressing the menu key **T.CENTER AUTO**.

UNo.	UNIT	ADD. WPC	X	Y	th	Z	c
1	WPC-99		T.CENTER	T.CENTER	0		0

Note 1: The modal display value on the **POSITION** display during the start of the MAZATROL program is G54.1P0.

Note 2: When the “workpiece scheme (MILL& TURN)” is selected in the common unit, set “WPC-Z” (Z position of the zero-point of the fundamental coordinates system) to the edge of the workpiece.

The above setting operation is required for NC to identify the Z-axial position of the workpiece. Thus, the path is created so that the tool does not interfere with the workpiece.



4-3 Index Unit (INDEX)

Select the index unit to control the angle of the desired surface when the machine has a head index mechanism and an NC rotary table, or a C-axis function.

When the machining priority function for the same tool is to be performed over each face, provide angle control using the index unit.

1. Menu selection

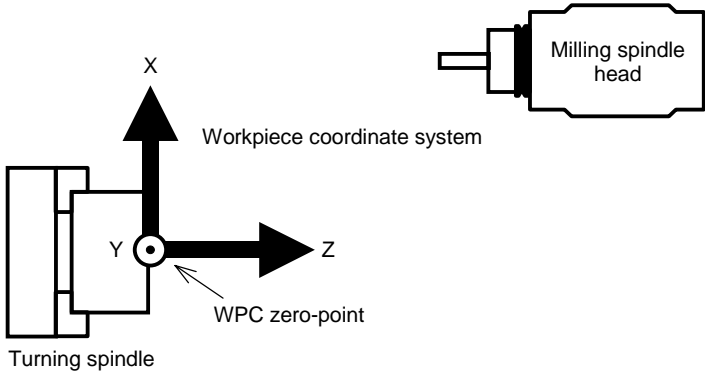
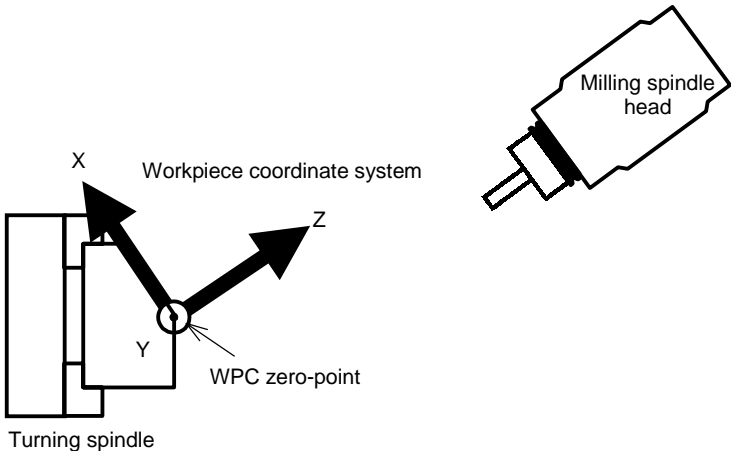
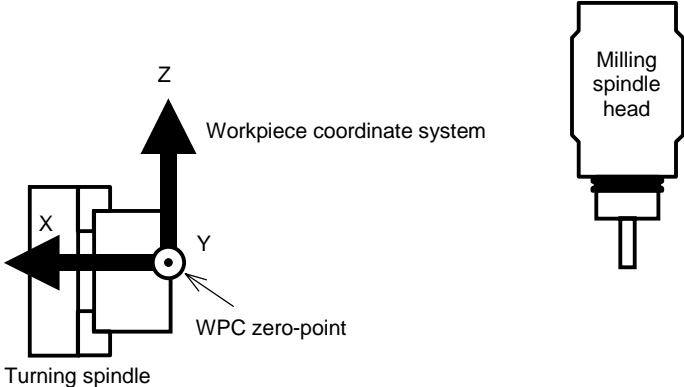
Press the menu key **INDEX**.

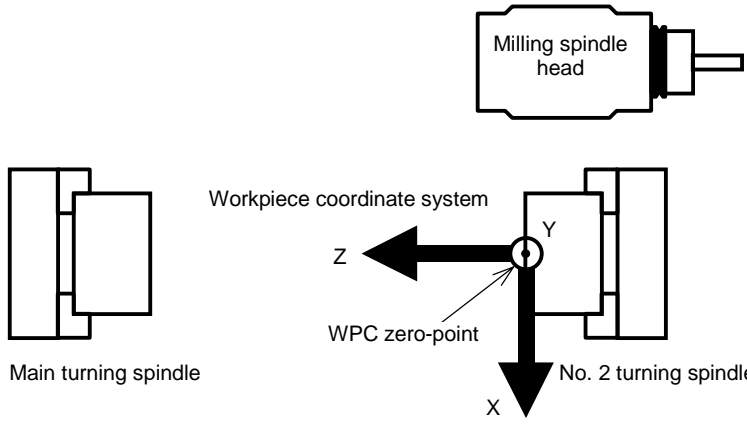
C-POINT MACH-ING	C-LINE MACH-ING	INDEX	M CODE	SUB PROGRAM	MMS	WORKPICE MEASURE		WORKPICE SHAPE	>>>
---------------------	--------------------	-------	--------	----------------	-----	---------------------	--	-------------------	-----

2. Data setting in indexing unit

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C
1	INDEX	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
TURN POS X	Enter the X coordinate of the rotating center position on the basis of the machine coordinates system by means of numeric keys. Example: Setting 100 for X coordinate..... <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="text" value="↔"/> Without this entry, axis movement does not take place. Permissible input range (0 to ±99999.999)
TURN POS Y	Enter the Y coordinate of the rotating center position on the basis of the machine coordinates system by means of numeric keys. Without this entry, axis movement does not take place. Permissible input range (0 to ±99999.999)
TURN POS Z	Enter the Z coordinate of the rotating center position on the basis of the machine coordinates system by means of numeric keys. Without this entry, axis movement does not take place. Permissible input range (0 to ±99999.999)

Cursor position	Description
<p>ANGLE B</p>	<p>Specify the index angle for the milling spindle head.</p> <p>Permissible input range (0 to ± 99999.999)</p> <p>When head index angle B is entered and the milling spindle head is tilted, the workpiece coordinate system for milling (point, line, and/or surface machining) will also incline correspondingly. If the spindle head is tilted using the index unit, the coordinates in the coordinate system which has inclined are to be set in subsequent milling program data.</p> <p>For turning, the workpiece coordinate system will not incline, even when the spindle head is tilted. Regardless of head index angle B, the X-axis and the Z-axis denote the diametral direction and longitudinal direction, respectively, of the workpiece.</p> <p>Example: For INTEGREX H (horizontal type)</p> <p>If head index angle B = 0 deg</p>  <p style="text-align: right;">D736P0081</p>
	<p>If head index angle B is oblique</p>  <p style="text-align: right;">D736P0082</p>
	<p>If head index angle B = 90 deg</p>  <p style="text-align: right;">D736P0083</p>

Cursor position	Description
ANGLE B	<p>If head index angle B = 180 deg (Only for machine specifications with the No. 2 turning spindle)</p>  <p style="text-align: right;">D736P0084</p>
ANGLE C	<p>Enter the indexing angle of the C-axis.</p> <p>Without this entry, axis movement does not take place.</p> <p>Permissible input range (0 to ± 9999.999)</p> <p>Note: Enter the angle as an absolute value on the basis of the workpiece coordinates system.</p>

Note: When either the end unit or the machining unit using the tool follows, the index unit is executed immediately before the end unit or the machining unit is executed. The index unit is the unit required for modal data entry for the surface that is to be machined. If the indexing unit is programmed to be followed by other unit than that mentioned above, such as an M-code unit, then that unit will be actually executed first.

3. Simultaneous movement of table rotation indexing and of ATC

By entering 1 or 2 to parameter **L41**, it is possible to perform the simultaneous rotation of the table and the ATC.

With the parameter **L41** = 1, movement to the indexing position, rotation of the table and ATC are executed simultaneously. With parameter **L41** = 2, rotation of the table and ATC are executed simultaneously after movement to the indexing position.

Note 1: In the case of the simultaneous execution of the rotation of the table and the ATC, it is possible that the table pivots to a position different from the rotation position specified by the index unit.

Note 2: With parameter **L41** = 1, the entering of data is only possible in the item TURN POS X of the index unit.

The entering of data for the Y-axis and the Z-axis will cause an alarm.

4-4 Fundamental Coordinate Shifting Unit (WPC SHIFT)

In the process of programming, the workpiece zero-point can be offset to any position in order to facilitate the entering of data. In order to do this, it is necessary to select this fundamental coordinate shifting unit.

1. Menu selection

Press the menu key **WPC SHIFT**.

SELECT HEAD	TRANSFER WORKPIECE	PROCESS END	PALLET CHANGE	WPC SHIFT					>>>
----------------	-----------------------	----------------	------------------	--------------	--	--	--	--	-----

2. Data setting in fundamental coordinate shifting unit

UNO.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	COORD.th	MIRROR
1	WPCSHIFT	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
SHIFT-X	Enter the amount of shift from the reference zero-point (WPC) in the X-axial direction to the program zero-point. Permissible input range (0 to ±99999.999)
SHIFT-Y	Enter the amount of shift from the reference zero-point (WPC) in the Y-axial direction to the program zero-point. Permissible input range (0 to ±99999.999)
SHIFT-Z	Enter the amount of shift from the reference zero-point (WPC) in the Z-axial direction to the program zero-point. Permissible input range (0 to ±99999.999)
SHIFT-C	Specify the rotational angle for the C-axis in the workpiece coordinates system. Permissible input range (0 to ±9999.999)
COORD.th	Specify the angle through which the workpiece coordinates system on the X-Y plane after the plane to be machined has been indexed is to be rotated for returning the coordinate system to the required direction for machining. Permissible input range (0 to ±999.999)
MIRROR	Specify the the mirror image of the X-axis existing on the X-Y plane is valid or invalid after the surface to be machined has been indexed. This value must be either 0 or 1 (0: X-axis mirror image invalid, 1: Valid). To mill the edge of the workpiece by operating the No. 2 turning spindle, the head index angle must be set to 180 degrees. In this case, since the X-axis faces downward, it can be directed upward using the X-axis mirror image. For further details, see the section on the index unit.

Note 1: Specify each shifting stroke in the axial direction of the machine coordinate system.
(When the spindle head is tilted using the index unit, although the workpiece coordinate system will also incline, each shifting stroke in the axial direction of the machine coordinate system is to be specified, irrespective of the axial direction in which the workpiece coordinate system has inclined.)

Note 2: When the fundamental coordinates system unit (WPC) is not assigned, the amount of shift from the machine zero-point will be used.

Note 3: If the fundamental coordinate shifting unit alone is specified in the MAZATROL subprogram, the fundamental coordinate shift data within the subprogram will not be taken over to the main program, even after control has been returned to the main program. Therefore, specify the fundamental coordinates system unit in the subprogram as well.

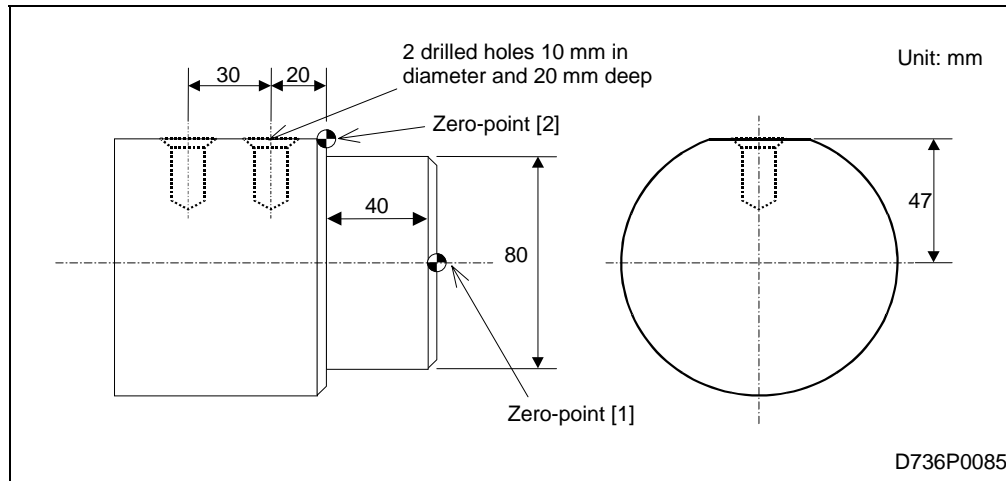
Note 4: In an EIA/ISO subprogram, the rotational angle value “COORD.th” on the fundamental coordinate shifting unit specified in the main program is invalid, as with “th” of the fundamental coordinates system unit.

Specifying a coordinate system rotate command (G92.5) in the EIA/ISO subprogram is recommended in that case. For further details, refer to the Programming Manual for EIA/ISO Programming.

3. Sample program

A sample machining program using the fundamental coordinate shifting unit is shown below.

Workpiece: Material stock $\phi 100 \times L200$



UNO.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM											
0	CBN STL	MILL&TURN.	100.		200.	0.	0												
UNO.	UNIT	ADD.WPC	X	Y	th	Z													
1	FRM- 0		T.CENTER	T.CENTER	0.	-500.													
UNO.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C													
2	INDEX	0.			90.	0.													
UNO.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z													
3	BAR	OUT	100.	0.	0.3	0.1													
SNO.	TOOL	NOM. No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M							
R1	GENERAL OUT	80. A	0	5.	◆	◆	◆	◆	200	0.3									
F2	GENERAL OUT	60. B	◆	◆	◆	◆	0.	0.	300	0.1									
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANG	RGH										
1	LIN	C 2.	◆	◆	80.	40.		◆	▼▼▼5										
2	LIN	C 2.	◆	◆	100.	42.		◆	▼▼▼5										
UNO.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z			SHIFT-C	COORD.th	MIRROR										
4	WPCSHIFT	47.	0.	-40			0.	0.	0										
UNO.	UNIT	DIA	DEPTH	CHMP															
5	DRILLING	10.	20.	2.															
SNO.	TOOL	NOM-φ No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M							
1	CTR-DR	20. A	14.	◆	◆	◆	90°	CTR-DR	60	0.1									
2	DRILL	10. A	10.	20.	0.	100	DRL T	5.	60	0.1									
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R					
1	LIN	0.	20.	0.	0.	◆	30.	◆	0	2	◆	◆	0	0					
UNO.	UNIT	CONTI. NUMBER	ATC	X	Y	Z	B	ANGLE											
6	END																		

4-5 Auxiliary Coordinates System Unit (OFFSET)

In the process of programming, the workpiece zero-point can be offset to any position in order to facilitate the entering of data. In order to do this, it is necessary to select this auxiliary coordinates system unit.

1. Menu selection

Press the menu key **OFFSET**.

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

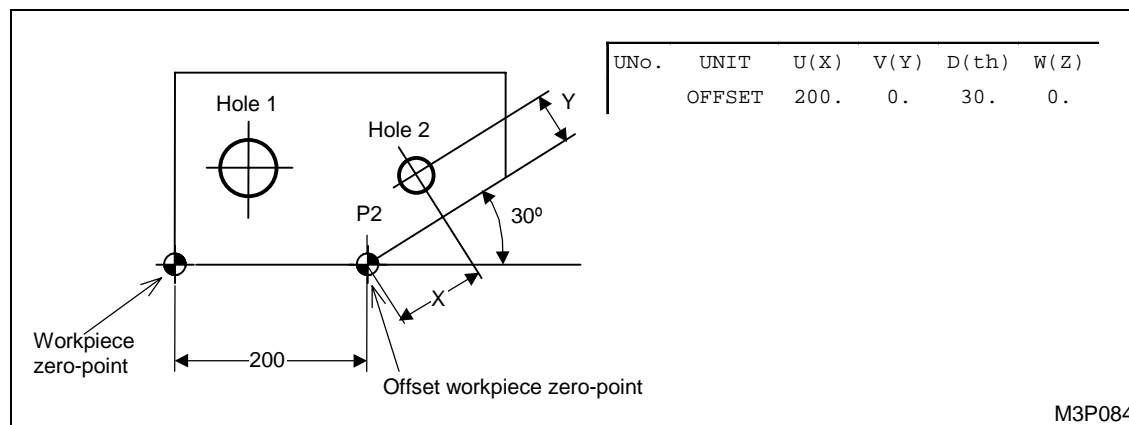
2. Data setting in auxiliary coordinates system unit

UNo.	UNIT	U(X)	V(Y)	D(th)	W(Z)
1	OFFSET	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Entering
U(X)	Enter the distance of offset from the workpiece zero-point on the X-axis. Permissible input range (0 to ±99999.999)
V(Y)	Enter the distance of offset from the workpiece zero-point on the Y-axis. Permissible input range (0 to ±99999.999)
D(th)	Enter the angle formed by the workpiece coordinates system and the auxiliary coordinates system. Permissible input range (0 to ±999.999)
W(Z)	Enter the distance of offset from the workpiece zero-point on the Z-axis. Permissible input range (0 to ±99999.999)

Note: Specify each shifting stroke in the axial direction of the workpiece coordinate system. (When the spindle head is tilted using the index unit, the workpiece coordinate system will also incline, each shifting stroke in the axial direction of the workpiece coordinate system is to be specified.)

Example:



The entering of the position of hole 2 with relation to the workpiece zero-point is very complicated. Offsetting the workpiece zero-point to point P2 makes the programming easy. In order to return to the workpiece zero-point, enter 0 in the columns U (X), V (Y), D (th), and W (Z).

4-6 Types of the Machining Unit

The machining unit is available in the following types:

- Point machining unit/ C-axis point machining unit
..... used for drilling of holes (Section 4-7/4-19)
- Line machining unit/C-axis line machining unit
..... used for a contour machining (Section 4-8/4-20)
- Face machining unit
..... used for machining an area and machining form (Section 4-9)
- Turning unit
..... used for turning (Section 4-21)

Each machining unit includes tool sequence and shape sequence.

Positions of the initial point and reference point (R)

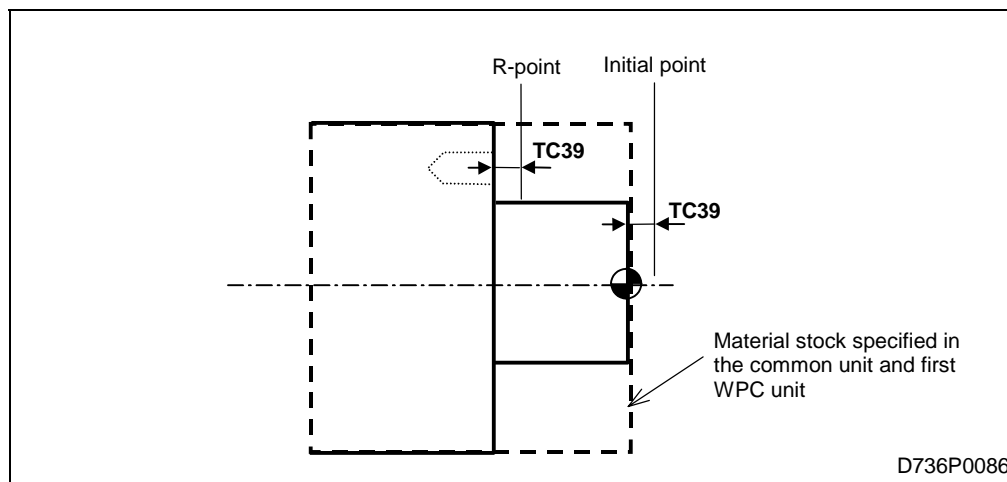
The positions of the initial point and reference point (R) defined in the tool path data of each machining unit differ as follows according to the selected mode for the common unit:

Creating mode specified in the common unit	Initial point	Reference point/(Safety) clearance
Workpiece scheme (MILL&TURN.)	Differs according to the specified head index angle. See the figure below.	
Initial-point scheme (ONLY MILL.)	Initial point specified in the common unit	Point machining unit Parameter D41 Line/face machining unit Parameter E9 When pre-machining is specified, D41/E9 may be switched to other parameters. For further details, see the sections on each unit.

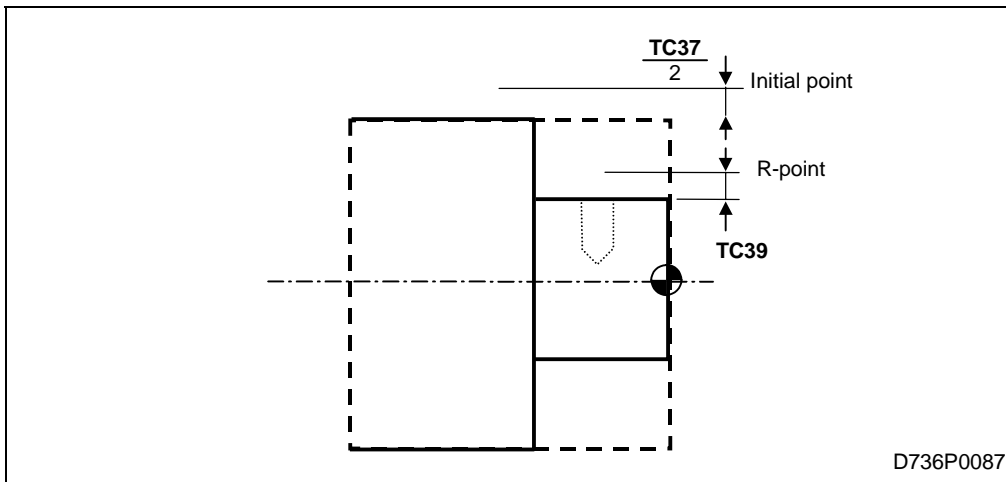
The figure below shows the initial point and reference point existing when the selected creating mode specified in the common unit is the workpiece scheme.

The points are shown below using INTEGREGX H (horizontal type) as an example.

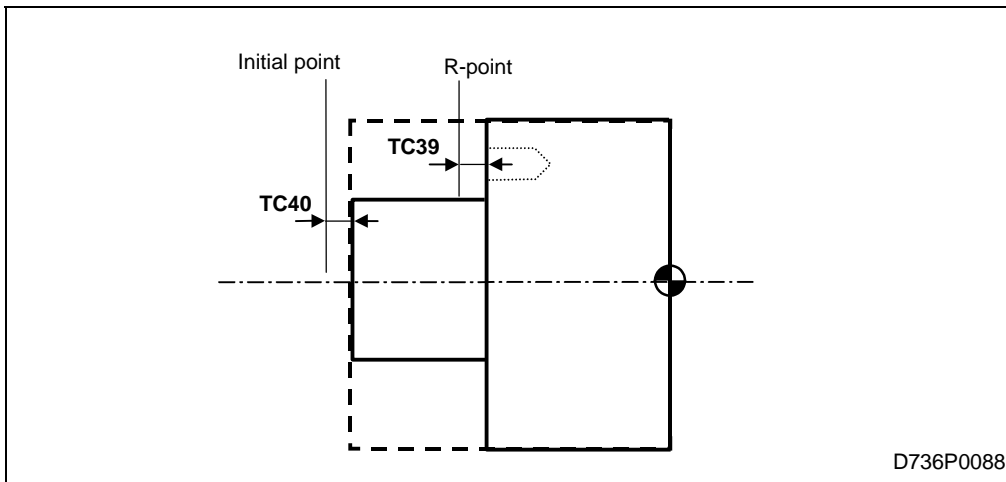
- If head index angle B = 0 deg



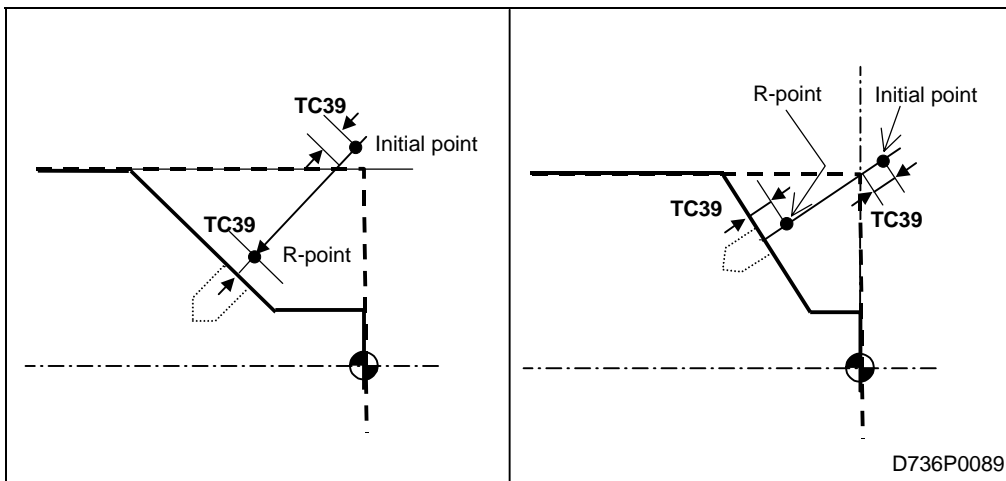
- If head index angle B = 90 deg



If head index angle B = 180 deg (Only for machine specifications with the No. 2 turning spindle)



If head index angle B is oblique



4-7 Point Machining Unit

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.

4-7-1 Types of point machining units

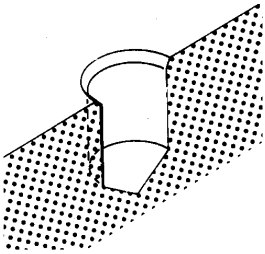
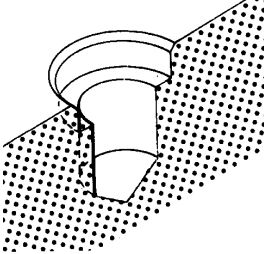
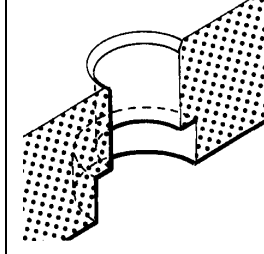
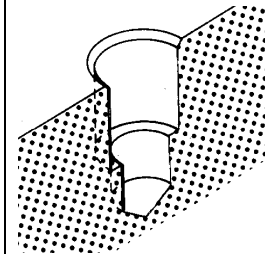
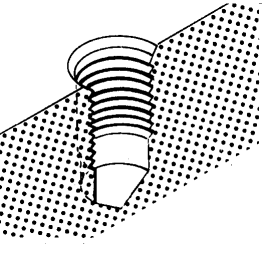
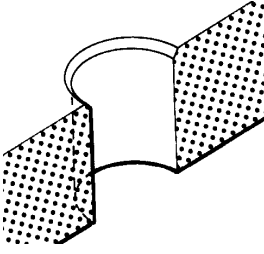
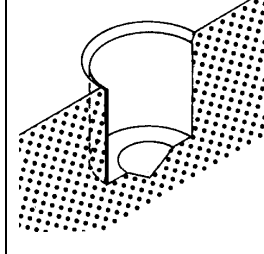
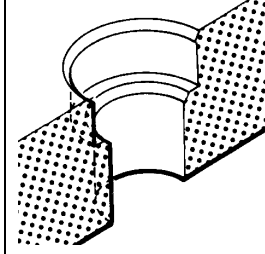
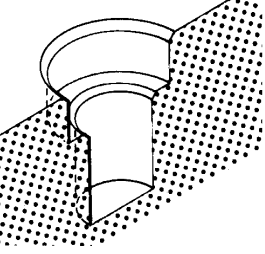
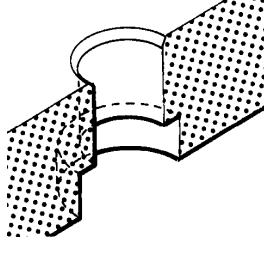
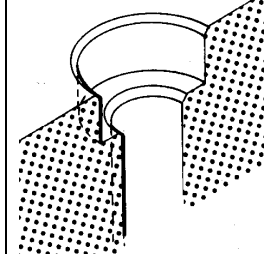
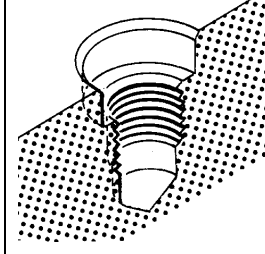
1. Drilling  NM210-00532	2. RGH CBOR machining  NM210-00533	3. RGH BCB machining  NM210-00534	4. Reaming  NM210-00535
5. Tapping  NM210-00536	6-(1) Boring of through hole  NM210-00537	6-(2) Boring of non-through hole  NM210-00538	6-(3) Boring of stepped through hole  NM210-00539
6-(4) Boring of stepped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










Fig. 4-1 Types of point machining units

4-7-2 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	WPC	OFFSET	END	SHAPE CHECK	>>>
-------------------	------------------	------------------	---------	-------------------	-----	--------	-----	----------------	-----

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

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

- (3) Press the appropriate menu key of the desired machining unit.
 - When the menu key **BORING** is pressed, the menu of the four following machining sub-units is displayed.

				()					
BORING 	BORING 	BORING 	BORING 						

Note: For menu item **HI SPD DRL. USE** refer to Subsection 4-7-4 “Automatic tool development for cemented carbide drill.”

4-7-3 Unit data and automatic tool development

1. Drilling unit (DRILLING)

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

A. Data setting

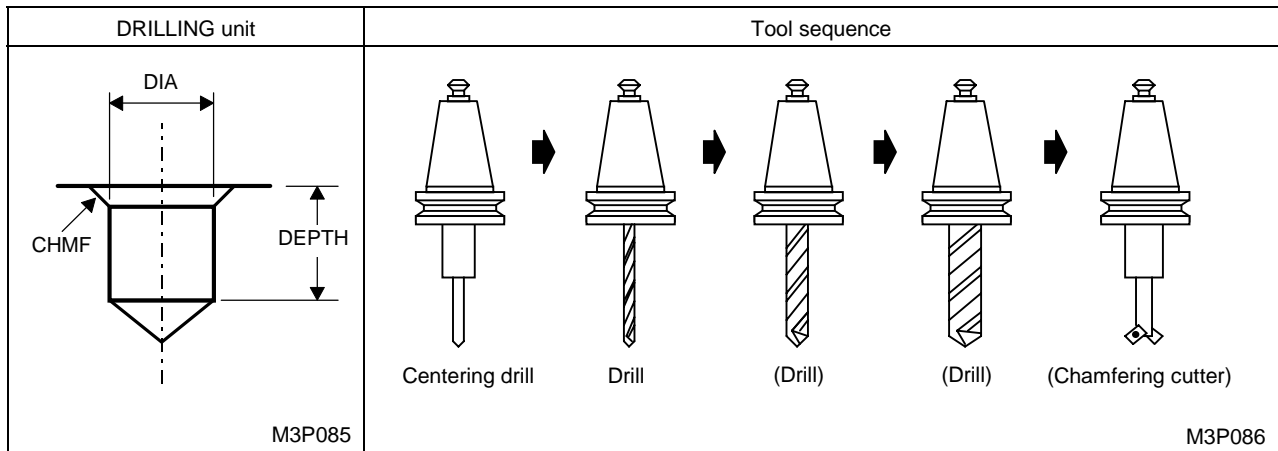
UNo.	UNIT	DIA	DEPTH	CHMF									
2	DRILLING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum entry value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are selected depending on the diameter of the hole. $0.05 \leq \text{DIA} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} \leq \mathbf{D10}$: Three drills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ $\text{CHMF} = 0$

The bold codes represent parameter addresses.

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

- DEPTH < CHMF
- DIA < 0.05
- **D10** < DIA

2. RGH CBOR machining unit (RGH CBOR)

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

A. Data setting

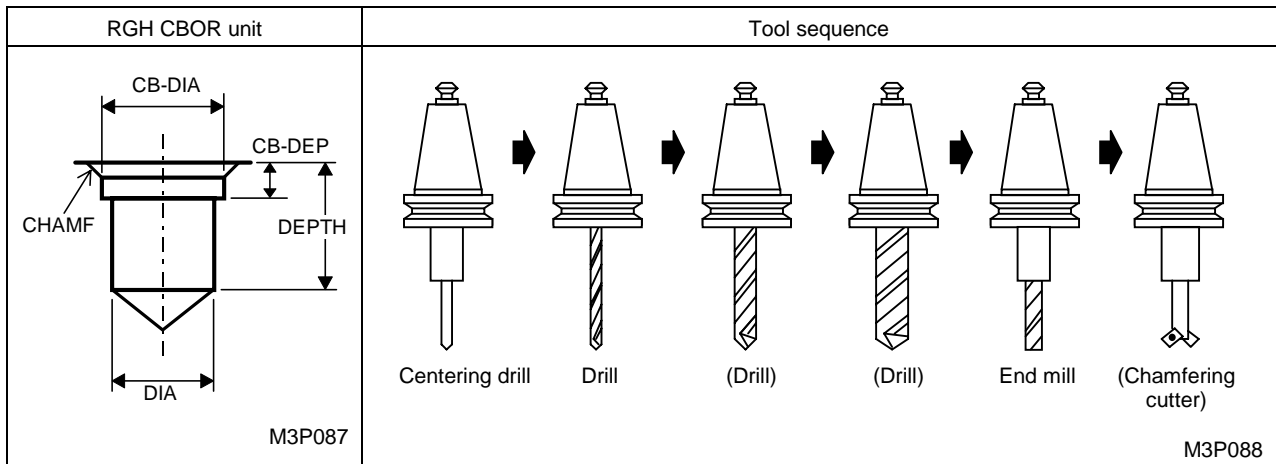
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	BTM	DIA	DEPTH						
2	RGH CBOR	999.999	999.999	99.9		999.999	999.999						
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○	○	◆	◆	◆	90°	CTR-D					
2	DRILL	○	○	○	○	○	○	○					
3	DRILL	○	○	○	○	○	○	○					
4	DRILL	○	○	○	○	○	○	○					
5	END MILL		○	○	○	○	◆	○					
6	CHAMFER		○	○	○	○	◆	○					

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are selected depending on the diameter of the hole. $0.05 \leq \text{DIA} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} \leq \mathbf{D10}$: Three drills are selected.
End mill	End mill is always selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{CHMF} = 0$ $\text{DIA} + (\text{CB-DEP} \times 2) \geq \text{CB-DIA} + (\text{CHMF} \times 2) < \mathbf{D13}$

The bold codes represent parameter addresses.

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

- $\text{CB-DIA} < \text{DIA}$
- $\text{DEPTH} < \text{CB-DEP}$
- $\text{DEPTH} < \text{CHMF}$
- $\mathbf{D10} < \text{DIA}$

3. RGH BCB machining unit (RGH BCB)

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

A. Data setting

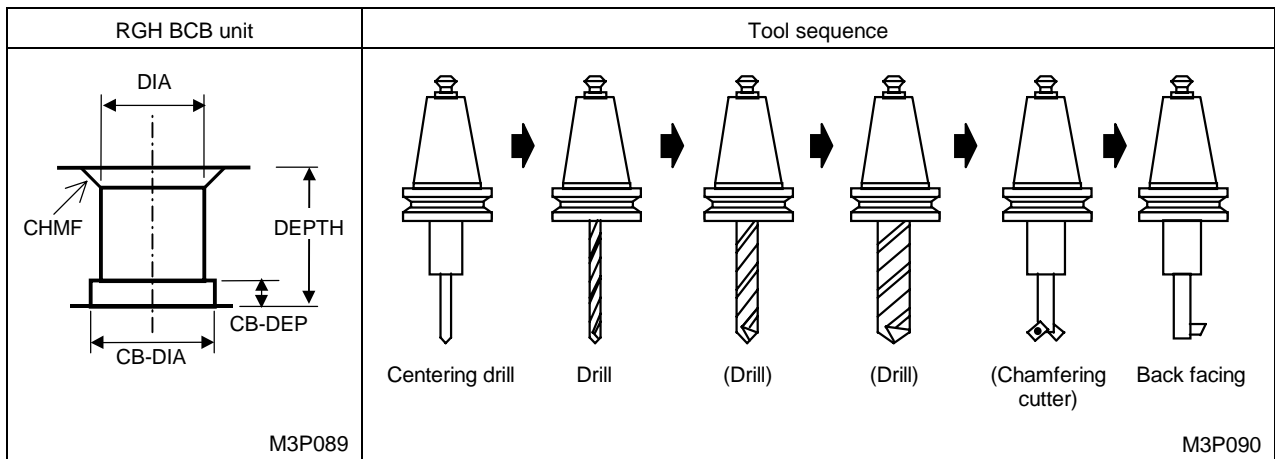
UNo.	UNIT	CB-DIA	CB-DEP	DIA	DEPTH	CHMF							
2	RGH BCB	999.999	999.999	999.999	999.999	99.9							
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆		CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER	○		○	○	○	○	◆	○				
6	BCK FACE	○		○	○	◆	○	◆	◆				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are selected depending on the diameter of the hole. $0.05 \leq \text{DIA} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} \leq \mathbf{D10}$: Three drills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D13}$ $\text{CHMF} = 0$
Back facing tool	Back facing tool is always selected.

The bold codes represent parameter addresses.

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

- $\text{CB-DIA} < \text{DIA}$
- $\text{DEPTH} < \text{CB-DEP}$
- $\text{DEPTH} < \text{CHMF}$
- $\mathbf{D10} < \text{DIA}$

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.

Case of preceding process = drilling

A. Data setting

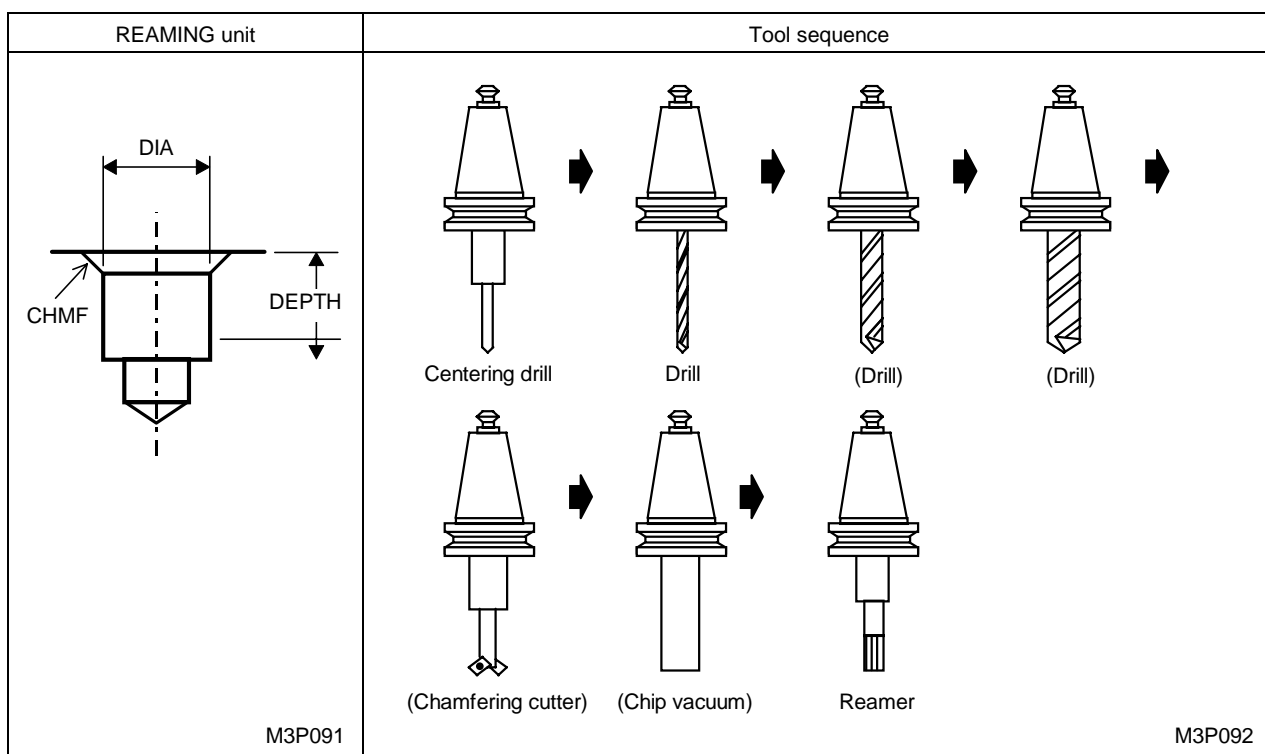
UNo.	UNIT	DIA	DEPTH	CHMF	PRE-REAM	CHP							
2	REAMING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				
6	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
7	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are developed depending on the diameter of the hole. $0.05 \leq \text{DIA} - \mathbf{D35} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} - \mathbf{D35} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} - \mathbf{D35} \leq \mathbf{D10}$: Three drills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ CHMF = 0
Chip vacuum	Chip vacuum is not selected if chip removal is not required.
Reamer	Reamer is always selected.

The bold codes represent parameter addresses.

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

Case of preceding process = boring

A. Data setting

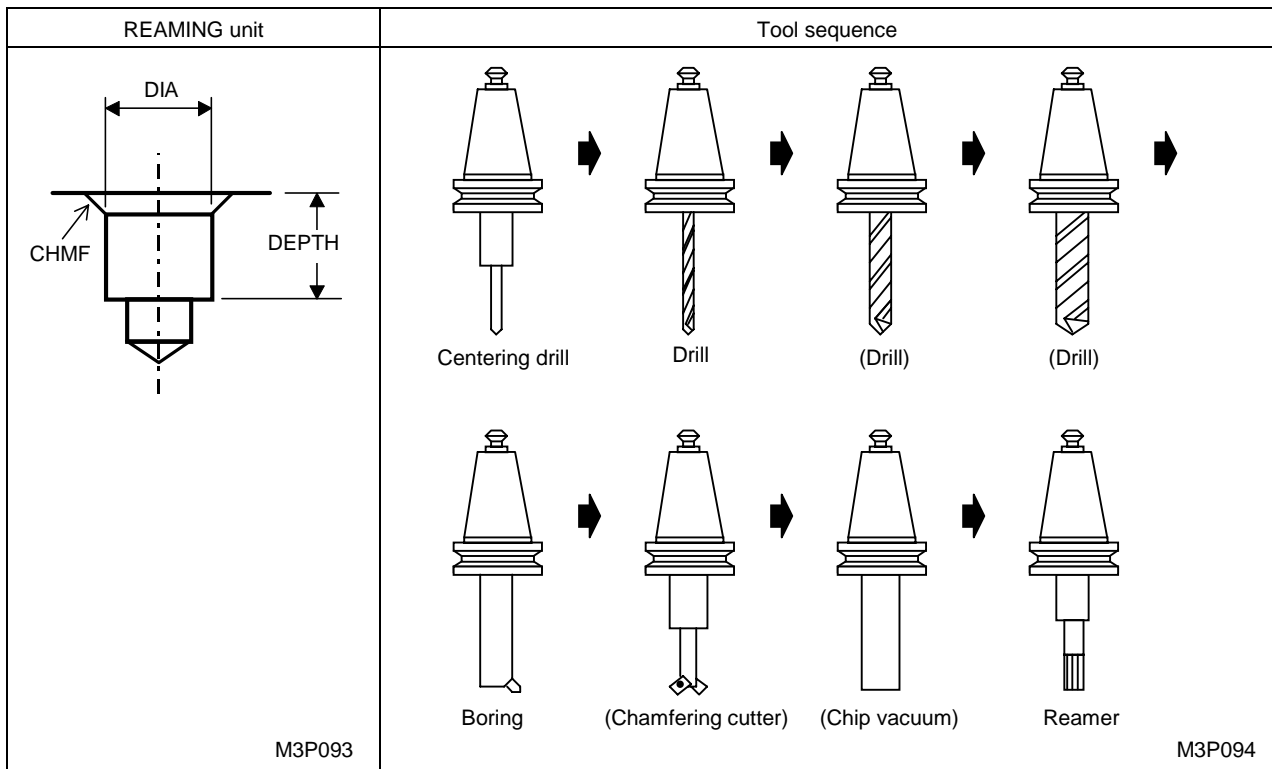
UNo.	UNIT	DIA	DEPTH	CHMF	PRE-REAM	CHP							
2	REAMING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
8	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are selected depending on the diameter of the hole. $0.05 \leq \text{DIA} - \mathbf{D36} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} - \mathbf{D36} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} - \mathbf{D36} \leq \mathbf{D10}$: Three drills are selected.
Boring tool	Boring tool is always selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ $\text{CHMF} = 0$
Chip vacuum	Chip vacuum is not selected if the chip removal is not required.
Reamer	Reamer is always selected.

The bold codes represent the parameter addresses.

Note: In the following case the alarm “416 AUTO PROCESS IMPOSSIBLE” will be displayed.
 - $\text{DEPTH} < \text{CHMF}$

Case of preceding process = end mill

A. Data setting

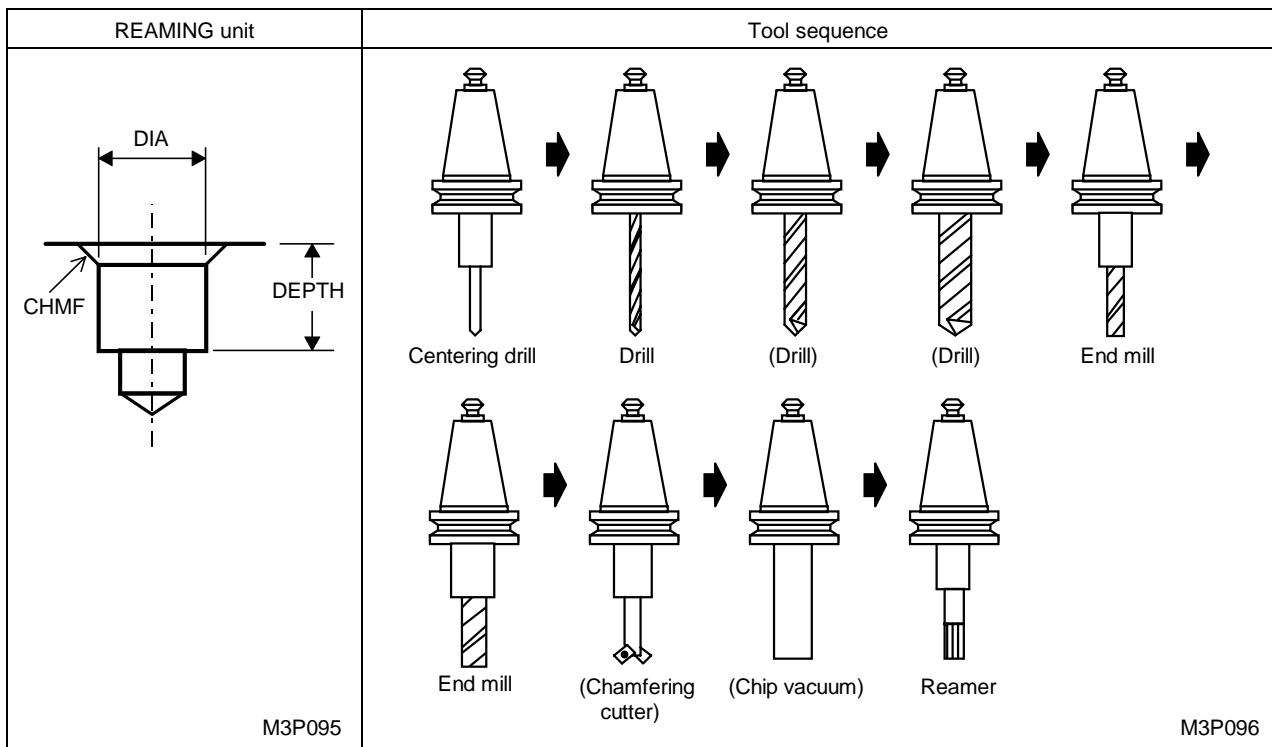
UNo.	UNIT	DIA	DEPTH	CHMF	PRE-REAM	CHP							
2	REAMING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	END MILL			○	○	○	○	○	○				
6	END MILL			○	○	○	○	○	○				
7	CHAMFER			○	○	○	○	◆	○				
8	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
9	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are developed depending on the diameter of the hole. $0.05 \leq \text{DIA} - \mathbf{D37} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} - \mathbf{D37} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} - \mathbf{D37} \leq \mathbf{D10}$: Three drills are selected.
End mill	Two end mills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{DIA} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ $\text{CHMF} = 0$
Chip vacuum	Chip vacuum is not selected if the chip removal is not required.
Reamer	Reamer is always selected.

The bold codes represent the parameter addresses.

Note: In the following case the alarm “416 AUTO PROCESS IMPOSSIBLE” will be displayed.
 - $\text{DEPTH} < \text{CHMF}$

5. Tapping unit (TAPPING)

Select this unit for performing tapping.

A. Data setting

UNo.	UNIT	NOM-	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP						
2	TAPPING				999.999	99.9							
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				
6	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
7	TAP	○		○	○	○	◆	○	○			○	

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.

Note 3: If TAPPING CYCLE menu item is selected for PRE-DIA, there is no need to set data in PRE-DEP.

<Setting the nominal diameter of unified screws>

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

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

<Setting the nominal diameter of pipe screws>

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

For PF 1 screws: Press the keys **1** and **↕** in this order.

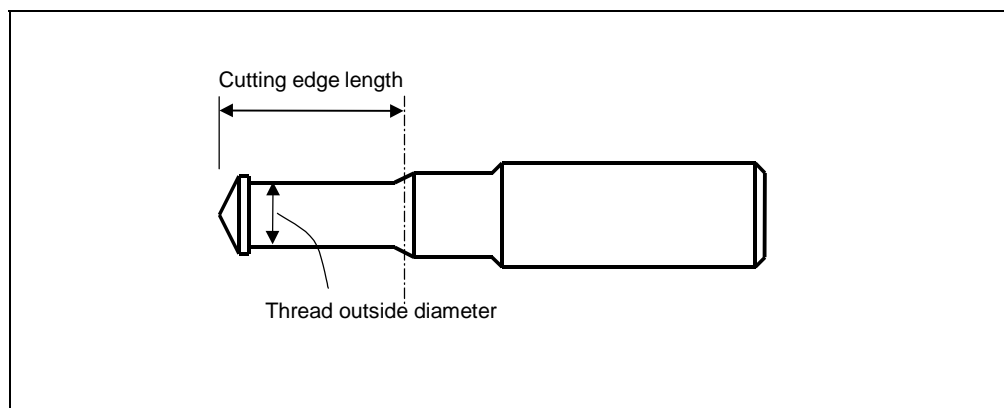
Note 1: The thread depths of PT screws or PS screws are set automatically according to MAZAK specifications.

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

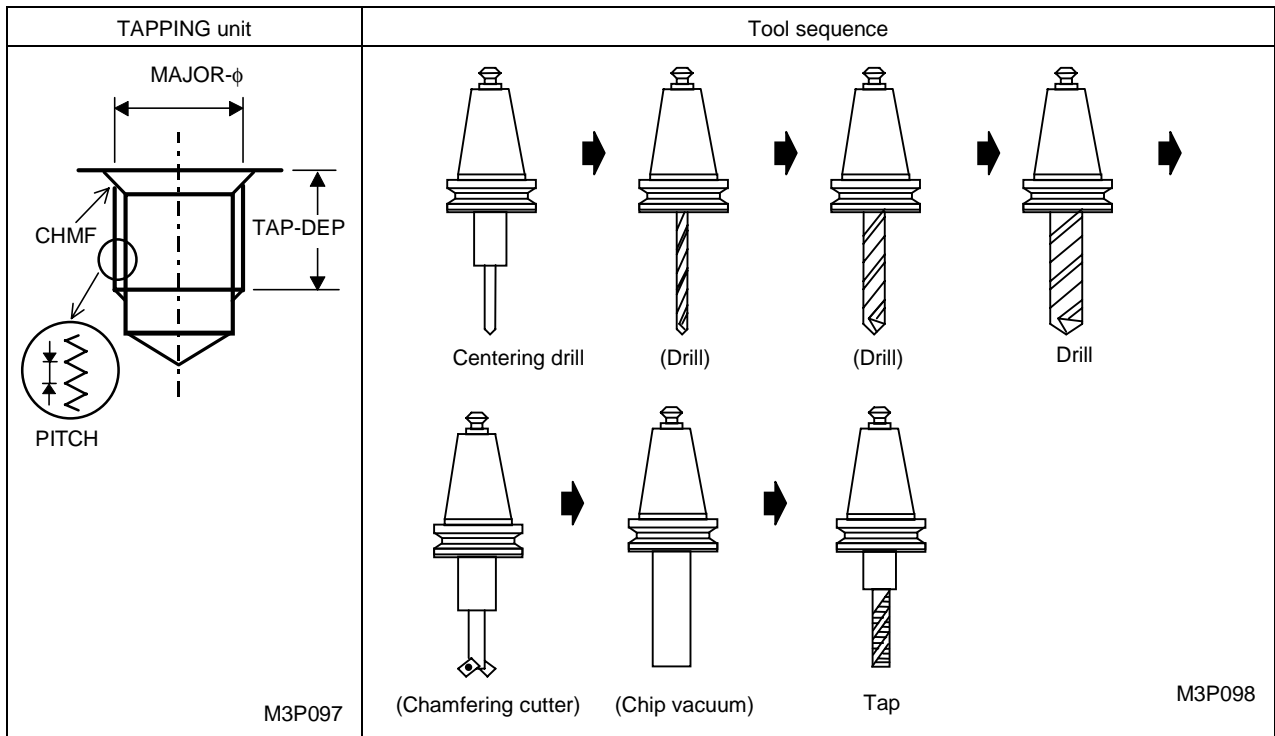
For TAP-DEP, enter the cutting edge length specified in the tool catalogue.

Also, enter the tool-catalogued data for the following data items:

- Enter the catalogued nominal diameter in the tool data item ACT- ϕ .
- Enter the catalogued thread outside diameter in the tool data item ACT- ϕ .
- Press the **FLOATING TAP** menu key for selecting "synchronous/asynchronous" as the type of tapping in the tool data item AUXIL.
- For data set in the tool data item "LENG COMP.", the NC unit does not function.



Remark: Any given value for tapping can be specified as an auto-set value by editing the required text file within the hard disk. For details refer to "4-7-5 New tapping auto-setting scheme."



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three tools are selected depending on the diameter of the hole. $0.05 \leq \text{Diameter of pre-hole drilling} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{Diameter of pre-hole drilling} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{Diameter of pre-hole drilling} \leq \mathbf{D10}$: Three drills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: $\text{Diameter of hole} + (\text{CHMF} \times 2) \leq \mathbf{D2} - \mathbf{D4}$ CHMF = 0
Chip vacuum	Chip vacuum is not selected if the chip removal is not required.
Tap	Tap is always selected.

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.

Through hole boring unit (BORING T1)

Select this unit for performing through-hole boring.

A. Data setting

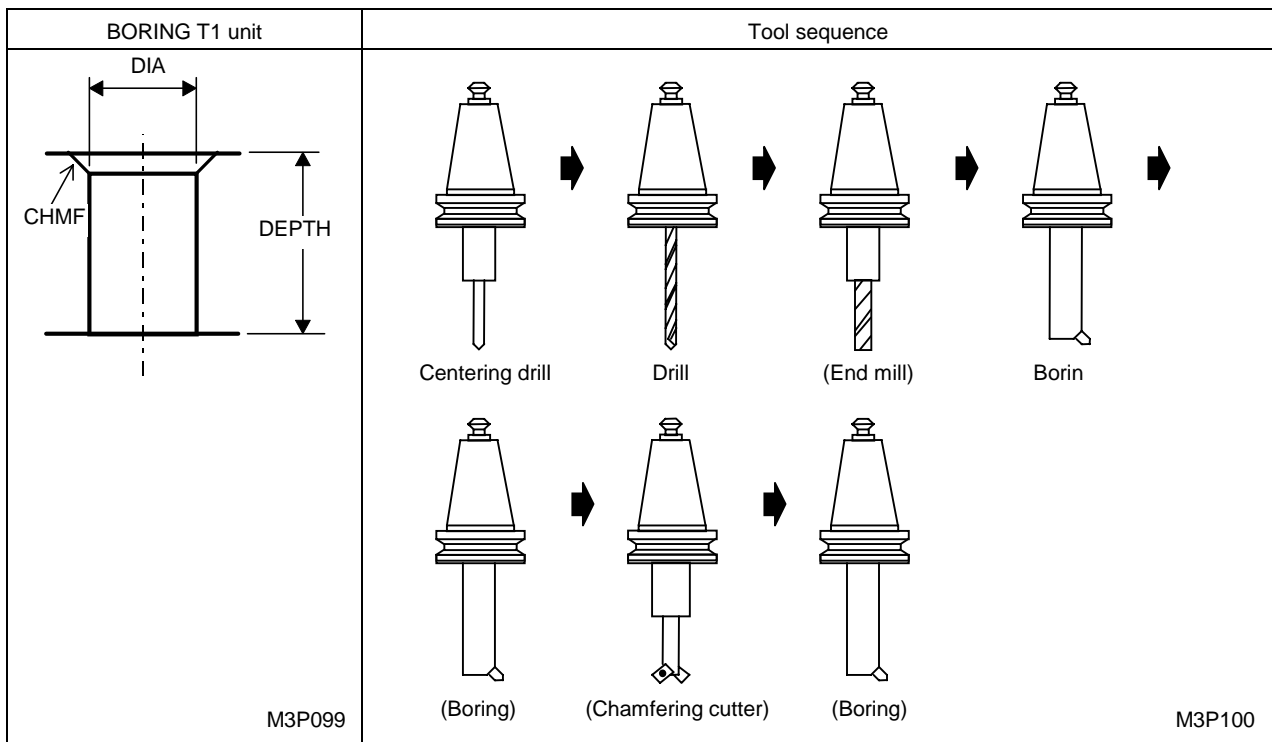
UNo.	UNIT	DIA	DEPTH	CHMF	WAL								
2	BORE T1	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	○	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	Drill is always selected.
End mill	End mill is not selected in the following case: DIA – 6.0 < D8
Boring tool	A maximum of up to three boring tools is selected depending on the wall roughness. Wall roughness = 1, 2 : One boring tool is selected. Wall roughness = 3, 4 : Two boring tools are selected. Wall roughness = 5, 6, 7, 8, 9 : Three boring tools are selected.
Chamfering cutter	Chamfering cutter is not selected 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
- DIA ≤ 6.0

Non-through hole boring unit (BORING S1)

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

A. Data setting

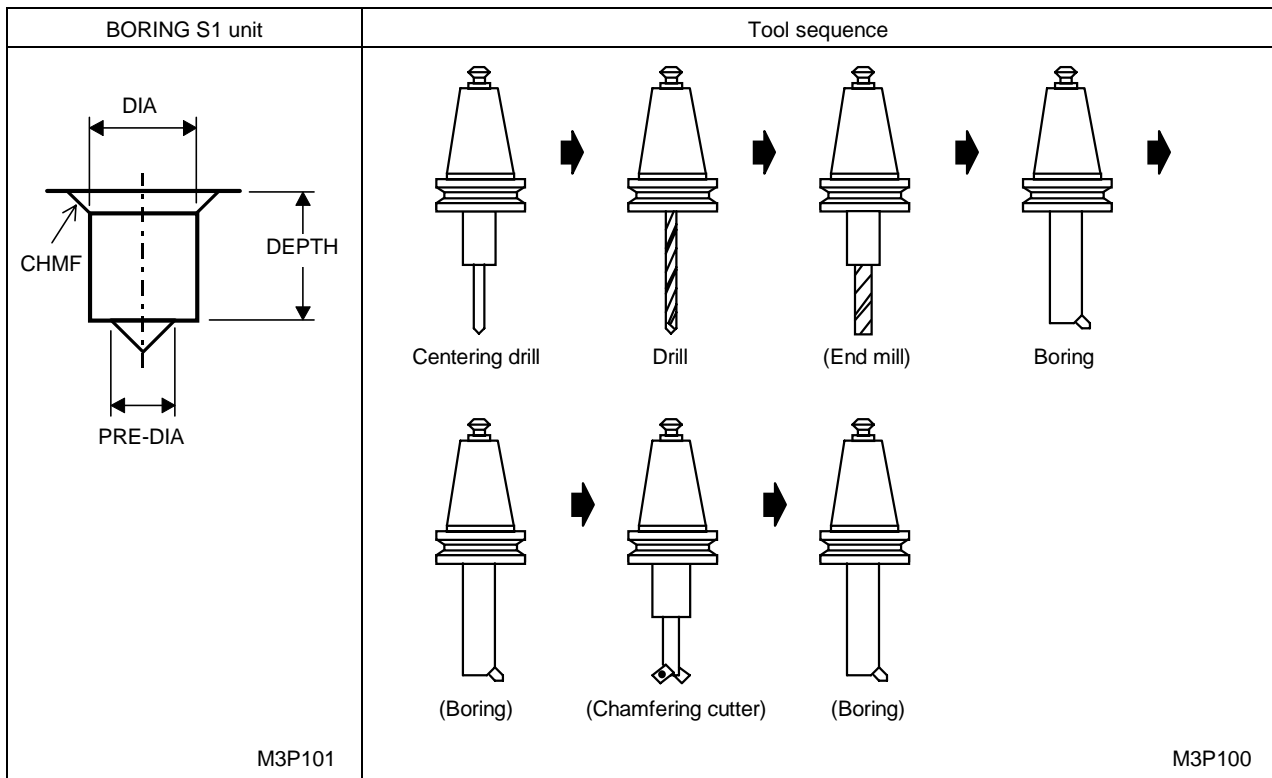
UNo.	UNIT	DIA	DEPTH	CHMF	BTM	WAL	PRE-DIA						
2	BORE S1	999.999	999.999	99.9			999.999						
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	○	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	Drill is always selected.
End mill	End mill is not selected if the following three conditions are fulfilled: $DIA - 6.0 < \mathbf{D8}$ $10.0 < PRE-DIA$ $DIA - PRE-DIA \leq 6.0$
Boring tool	A maximum of up to three boring tools are selected according to the wall roughness. Wall roughness = 1, 2 : One boring tool is selected. Wall roughness = 3, 4 : Two boring tools are selected. Wall roughness = 5, 6, 7, 8, 9 : Three boring tools are selected.
Chamfering cutter	Chamfering cutter is not selected 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:

- $DIA < PRE-DIA$
- $DIA \leq 6.0$
- $DEPTH < CHMF$
- $PRE-DIA = 0 \rightarrow \text{Depth of hole} < (A/3.328558 + \mathbf{D12})$
- $PRE-DIA \neq 0 \rightarrow \text{Depth of hole} < (A - PRE-DIA)/3.328558$

A	$DIA - 6.0$ (in case of $DIA - 6.0 < \mathbf{D8}$)
	$\mathbf{D8}$ (in case of $\mathbf{D8} \leq DIA - 6.0$)

Stepped through hole boring unit (BORING T2)

Select this unit for performing stepped through hole boring.

A. Data setting

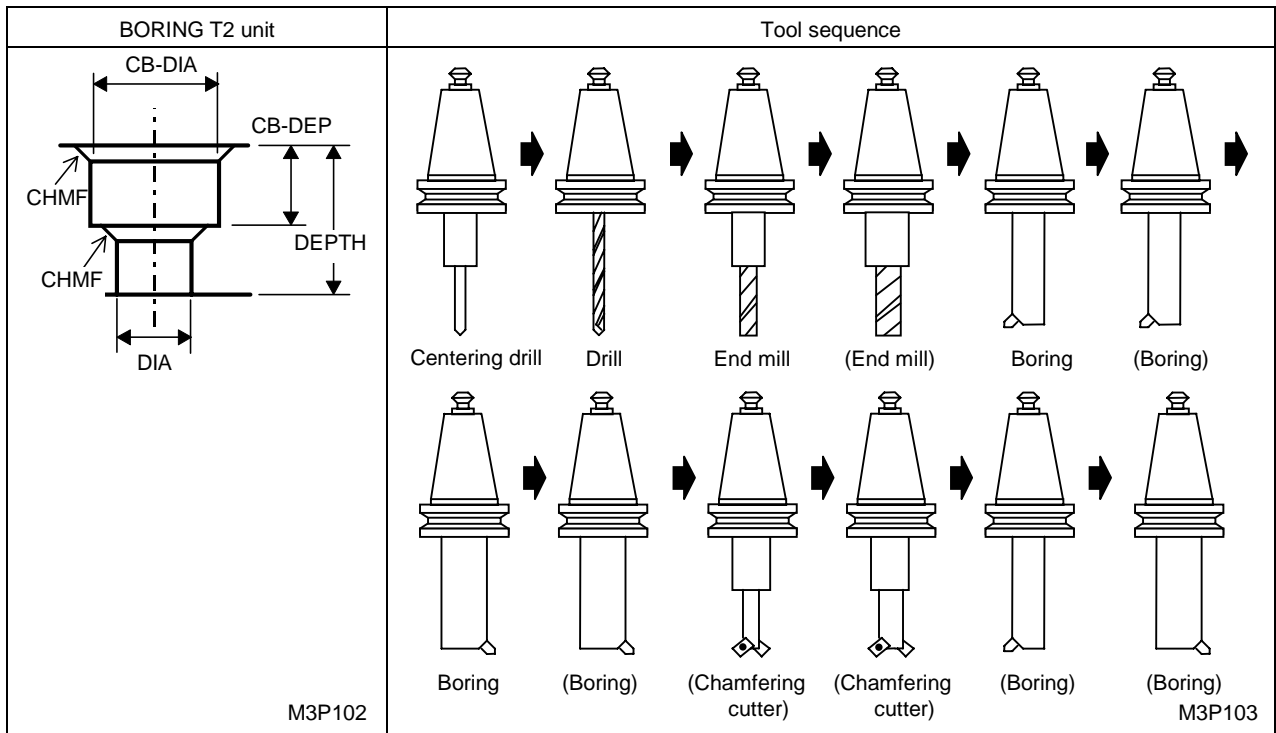
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	BTM	WAL	DIA	DEPTH	CHMF	WAL			
2	BORE T2	999.999	999.999	99.9			999.999	999.999	99.9				
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○	○	◆	◆	◆		90°	CTR-D				
2	DRILL	○	○	○	○	○	○	○	○				
3	END MILL		○	○	○	○	○	○	○				
4	END MILL		○	○	○	○	○	○	○				
5	BOR BAR	○	○	○	○	○	○	○	○				
6	BOR BAR	○	○	○	○	○	○	○	○				
7	BOR BAR	○	○	○	○	○	○	○	○				
8	BOR BAR	○	○	○	○	○	○	○	○				
9	CHAMFER		○	○	○	○	○	◆	○				
10	CHAMFER		○	○	○	○	○	◆	○				
11	BOR BAR	○	○	○	○	○	○	○	○				
12	BOR BAR	○	○	○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	Drill is always selected.
End mill	A maximum of up to two end mills are selected depending on the diameter of the hole. $0 < \text{DIA} - 6.0 < \mathbf{D8}$: One end mill is selected. $\mathbf{D8} < \text{DIA} - 6.0 \leq 999.999$: Two end mills are selected.
Boring tool	A maximum of up to three boring tools are selected 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 : One boring tool is selected. Wall roughness of hole = 3, 4 : Two boring tools are selected. Wall roughness of hole = 5, 6, 7, 8, 9 : Three boring tools are selected. ----- Wall roughness of large hole = 1, 2 : One boring tool is selected. Wall roughness of large hole = 3, 4 : Two boring tools are selected. Wall roughness of large hole = 5, 6, 7, 8, 9 : Three boring tools are selected.

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

Stepped non-through hole boring unit (BORING S2)

Select this unit for performing stepped non-through boring.

A. Data setting

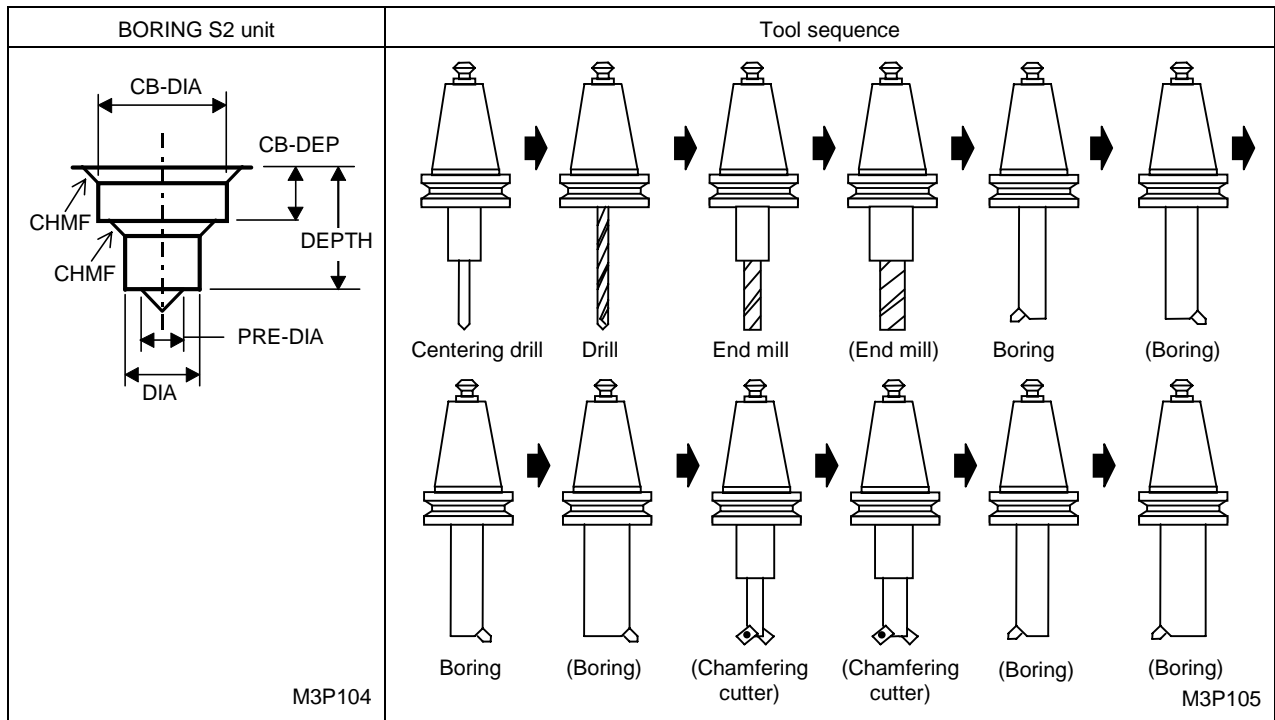
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	BTM	WAL	PRE-DIA	DIA	DEPTH	CHMF	BTM	WAL	
2	BORE S2	999.999	999.999	99.9			999.999	999.999	999.999				
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	◆				
3	END MILL			○	○	○	○	○	○				
4	END MILL			○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	BOR BAR	○		○	○	○	○	○	○				
7	BOR BAR	○		○	○	○	○	○	○				
8	BOR BAR	○		○	○	○	○	○	○				
9	CHAMFER			○	○	○	○	◆	○				
10	CHAMFER			○	○	○	○	◆	○				
11	BOR BAR	○		○	○	○	○	○	○				
12	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	Drill is always selected.
End mill	A maximum of up to two end mills are selected depending on the diameter of the hole. $0 < \text{DIA} - 6.0 < \mathbf{D8}$, $10.0 < \text{PRE-DIA}$ and $(\text{DIA} - \text{PRE-DIA}) \leq 6.0$: One end mill is selected. $\mathbf{D8} < \text{DIA} - 6.0 \leq 999.999$: Two end mills are selected.
Boring tool	A maximum of up to three boring tools are selected 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 : One boring tool is selected. Wall roughness of hole = 3, 4 : Two boring tools are selected. Wall roughness of hole = 5, 6, 7, 8, 9 : Three boring tools are selected. Wall roughness of large hole = 1, 2 : One boring tool is selected. Wall roughness of large hole = 3, 4 : Two boring tools are selected. Wall roughness of large hole = 5, 6, 7, 8, 9 : Three boring tools are selected.
Chamfering cutter	Chamfering cutter is not selected 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:

- $\text{CB-DIA} < \text{DIA}$
- $\text{DIA} < \text{PRE-DIA}$
- $\text{DEPTH} < \text{CB-DEP}$
- $\text{CB-DEP} < \text{CHMF (CB)}$
- $(\text{CB-DIA} - \text{DIA})/2 < \text{CHMF}$
- $(\text{DEPTH} - \text{CB-DEP}) < \text{CHMF (CB)}$
- $\text{DIA} \leq 6.0$
- $\text{DEPTH} < \text{CHMF}$
- $\text{B} \leq 0$

B	$\text{DIA} - 6.0$ (in case of $\text{DIA} - 6.0 < \mathbf{D8}$)
	$\mathbf{D8}$ (in case of $\mathbf{D8} \leq \text{DIA} - 6.0$)
	$\text{DEPTH} - \text{A}/3.328558 - \mathbf{D12}$ (in case of $\text{CB-DEP} = 0$)
	$\text{DEPTH} - (\text{A} - \text{PRE-DIA})/3.328558$ (in case of $\text{PRE-DIA} \neq 0$)

A	$\text{DIA} - 6.0$ (in case of $\text{DIA} - 6.0 < \mathbf{D8}$)
	$\mathbf{D8}$ (in case of $\mathbf{D8} \leq \text{DIA} - 6.0$)

7. Back boring unit (BK-CBORE)

Select this unit for performing back boring.

A. Data setting

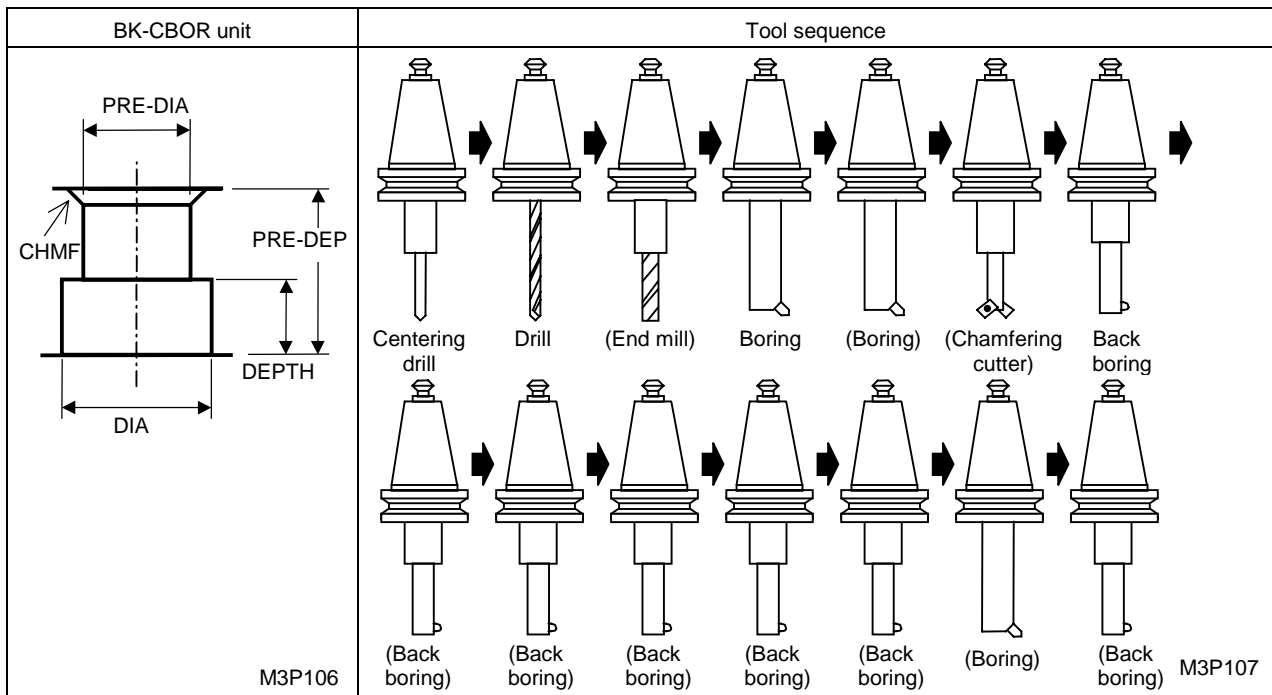
UNo.	UNIT	DIA	DEPTH	BTM	WAL	PRE-DIA	PRE-DEP	CHMF	WAL				
2	BK-CBORE	999.999	999.999			999.999	999.999	99.9					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	○	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	B-B BAR	○		○	○	○	○	○	○				
8	B-B BAR	○		○	○	○	○	○	○				
9	B-B BAR	○		○	○	○	○	○	○				
10	B-B BAR	○		○	○	○	○	○	○				
11	B-B BAR	○		○	○	○	○	○	○				
12	B-B BAR	○		○	○	○	○	○	○				
13	BOR BAR	○		○	○	○	○	○	○				
14	B-B BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	Drill is always selected.
End mill	End mill is not selected in the following case: PRE-DIA – 6.0 < D8
Boring tool	A maximum of up to three boring tools are developed depending on the wall roughness. Wall roughness of pre-hole = 1, 2 : One boring tool (Roughing) is selected. Wall roughness of pre-hole = 3, 4 : Two boring tools (Roughing, semi-finishing) are selected. Wall roughness of pre-hole = 5, 6, 7, 8, 9 : Three boring tools (Roughing, semi-finishing, finishing) are selected.
Chamfering cutter	Chamfering cutter is not selected in the following case: CHMF = 0
Back boring tool	A maximum of up to five back boring tools are selected according to the value of N (See Note below.) N = 2 : Two back boring tools are selected. N = 3 : Three back boring tools are selected. N = 4 : Four back boring tools are selected. N = 5 : Five back boring tools are selected.
Back boring tool (Semi-finishing, finishing)	A maximum of up to two back boring tools are selected depending on the wall roughness. Wall roughness of hole = 1, 2 : Not developed. Wall roughness of hole = 3, 4 : One back boring tool (Semi-finishing) is selected. Wall roughness of hole = 5, 6, 7, 8, 9 : Two back boring tools (Semi-finishing, finishing) are selected.

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.

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

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

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

8. Circular milling unit (CIRC MIL)

Select this unit for performing drilling with the end mill.

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

- TORNA.: 0 circular milling cycle
- 1 circular tornado milling cycle
- 2 precision rapid boring tornado cycle

However, the function TORNA. 2 (precision rapid boring tornado cycle) requires the shape correction option and the MAZAK precision rapid boring tornado option.

Circular milling cycle

A. Data setting

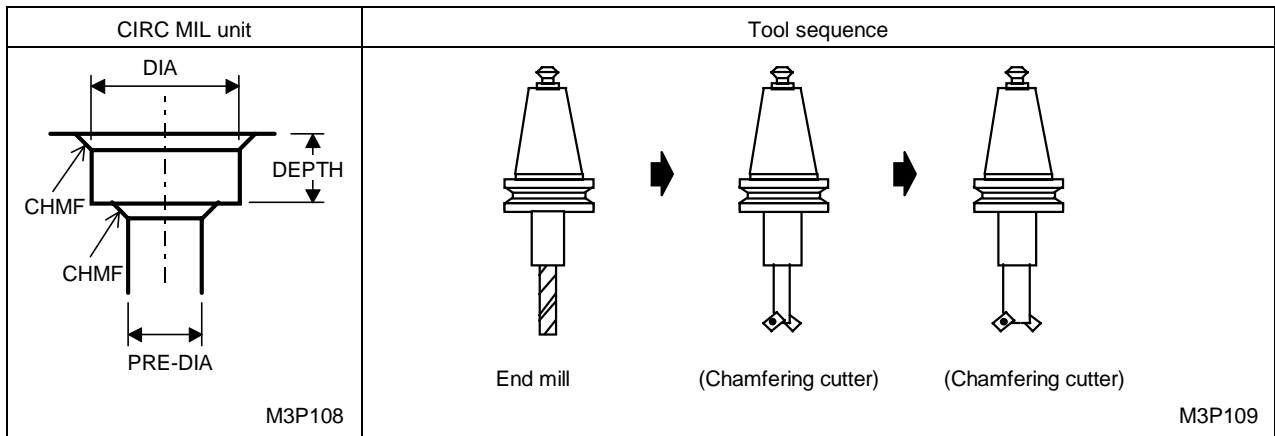
UNo.	UNIT	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2			
2	CIRC MIL	0	999.999	999.999	99.9	9	999.999	99.9	◆	◆			
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				
2	CHAMFER			○	○	○	○	◆	○				
3	CHAMFER			○	○	○	○	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
End mill	End mill is always selected.
Chamfering cutter	Chamfering cutter is not selected 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)

Circular tornado milling cycle

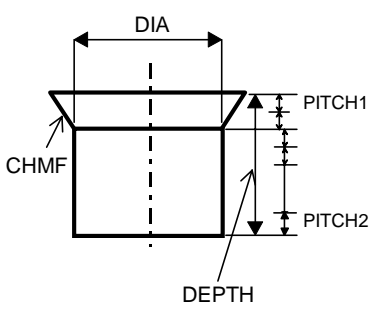
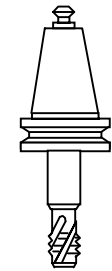
A. Data setting

UNo.	UNIT	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2			
2	CIRC MIL	1	999.999	999.999	99.9	9	◆	◆	999.999	999.999			
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				

○ : The data displayed here are automatically determined by the automatic tool development function.
 ◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.

CIRC MIL unit	Tool sequence
 <p>D735P0063</p>	 <p>End mill (Thread mill)</p> <p>D735P0064</p>

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
End mill	End mill is always selected.

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)

Precision rapid boring tornado cycle

A. Data setting

UNo.	UNIT	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2			
2	CIRC MIL	2	999.999	999.999	99.9	9	999.999	99.9	999.999	999.999			
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				

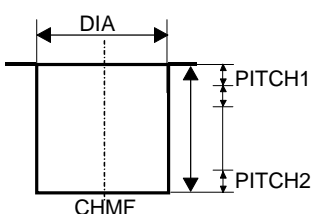
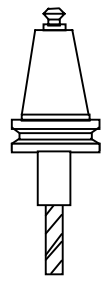
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: To set 2 for the unit data item TORNA., the shape correction option and the MAZAK precision rapid boring tornado option are required.

Note 3: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.

CIRC MIL unit	Tool sequence
	 <p>End mill</p>
D735P0071	

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
End mill	End mill is always selected.

9. Counterbore-tapping unit (CBOR-TAP)

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

A. Data setting

UNo.	UNIT	NOM-	MAJOR-φ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM	CHP		
2	CBOR-TAP		<input type="text"/>	<input type="text"/>	999.999	99.9	999.999	999.999	99.9				
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	END MILL			○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	CHAMFER			○	○	○	○	◆	○				
8	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
9	TAP	○		○	○	○	◆	○	○				

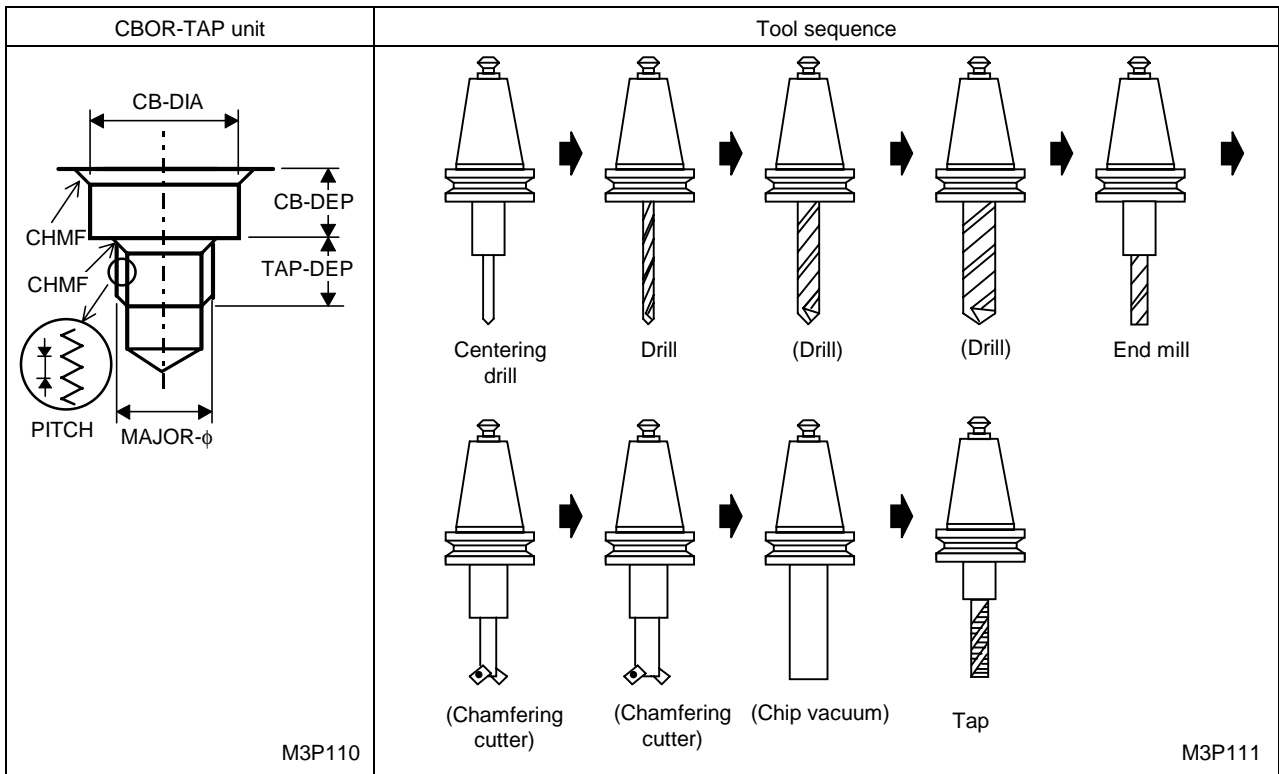
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.

Remark: Any given value for tapping can be specified as an auto-set value by editing the required text file within the hard disk. For details refer to “4-7-5 New tapping auto-setting scheme.”



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-7-6.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

<Development patterns>

Tool	Development patterns
Centering drill	Centering drill is always selected.
Drill	A maximum of up to three drills are selected depending on the diameter of the hole. $0.05 \leq \text{DIA} \leq \mathbf{D8}$: One drill is selected. $\mathbf{D8} < \text{DIA} \leq \mathbf{D9}$: Two drills are selected. $\mathbf{D9} < \text{DIA} \leq \mathbf{D10}$: Three drills are selected.
Chamfering cutter	Chamfering cutter is not selected in the following cases: CHMF (faced hole) = 0 CHMF (threaded hole) = 0
Chip vacuum	Chip vacuum is not selected if the removal of chips is not performed.

The bold codes represent the parameter addresses.

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

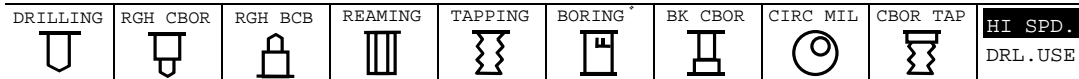
- $\text{CB-DIA} < \text{MAJOR-}\phi$
- $(\text{CB-DIA} - \text{MAJOR-}\phi)/2 < \text{CHMF}$ (threaded hole)
- $\text{PRE-DEP} < \text{CHMF}$ (faced hole)
- $\text{TAP-DEP} < \text{CHMF}$ (threaded hole)

4-7-4 Automatic tool development for cemented carbide drill

Subsection 4-7-3 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 menu key **HI SPD DRL. USE** to make the menu 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:



Automatic tool development for drilling with cemented carbide drills can be used for all the point-machining units. In the following description this function is explained using a drill unit as an example.

UNo.	UNIT	DIA	DEPTH	CHMF									
2	DRILLING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	DRILL	○		○	○	○	○	DRILL	○				
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	DIA	DEPTH	CHMF									
2	DRILLING	999.999	999.999	99.9									
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	DRILL	○			○	○	○	DRIL	○				
2	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum entry value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-7-6.

4-7-5 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/Tapping in turning mode				
	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:\nm64mdata” 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 “419 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”]

$$(\text{Chamfering}) = \{(\text{Tap outside diameter}) + (\text{Thread pitch}) \times 2 - (\text{Prehole diameter})\} / 2$$

[If parameter **D44** is set to “1” (M32 scheme)]

$$(\text{Chamfering}) = \{(\text{Tap outside diameter}) - (\text{Prehole diameter})\} / 2$$

Note 3: Even when data within the above data range is entered, particular data settings in parameter **D8** to **D10** display alarm “416 AUTO PROCESS IMPOSSIBLE” during automatic tool development. (For parameter **D8** to **D10** refer to the Parameter List.)

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 1/16		
DIAMETER_1=77230 ;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=70000 ;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)
 This item also denotes the auto-setting value for ACT- ϕ of the tap displayed in the **TOOL DATA** display.
- 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 drill 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 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 “419 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	9900000

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”]

$$(\text{Chamfering}) = \{(\text{Tap outside diameter}) + (\text{Thread pitch}) \times 2 - (\text{Prehole diameter})\} / 2$$

[If parameter **D44** is set to “1” (M32 scheme)]

$$(\text{Chamfering}) = \{(\text{Tap outside diameter}) - (\text{Prehole diameter})\} / 2$$

Note 3: Even when data within the above data range is entered, alarm “416 AUTO PROCESS IMPOSSIBLE” may be displayed during automatic tool development.

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 “PT1/16”, proceed as follows to auto-set 7.77 mm as the tap outside diameter, 27 as the number of threads, 11 mm as the thread depth, 6.25 mm as the prehole diameter, and 17 mm as the prehole depth:

(1) Open the text file “Pipescdt.txt” and move the cursor to “PT1/16.”

(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 1/16
DIAMETER_1=77700 ;Diameter(1/10000mm)
THREAD_1=270 ;Number of Thread(1/10Thread)
DEPTH_1=110000 ;Depth(1/10000mm)
PRE_DIA_1=62500 ;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 pipe 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: Do not edit the text file during processing from tapping/counterbore-tapping program creation to “Tool diameter” registration (ACT-φ) in the **TOOL DATA** display mode. Otherwise, the auto-set MAJOR-φ value of the tapping unit/counterbore-tapping unit in the program and the auto-set ACT-φ value of that tap in the **TOOL DATA** display mode will mismatch.

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.

4-7-6 Tool sequence data

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

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

List of tool sequence data

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

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

Note 1: CHIP VAC (chip vacuum) is an option.

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

Note 3: Different data items must be set for CTR DR, if CHAMF is selected for DEPTH.

1. Tool designation: TOOL

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

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

Note: The chip vacuum does not exist on the menu. (It can only be selected in automatic tool development mode.)

2. Nominal diameter of tool: NOM-φ

Used to specify the nominal diameter of the tool.

Setting range (0 to 999.9 mm)

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

Note 2: The alarm “434 NO ASSIGNED TOOL IN TOOL FILE” is given if the tool entered has not been previously recorded in the **TOOL FILE** display.

3. Tool identification

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

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

4. Priority number: No.

Used to specify the machining priority number (prior machining priority number/subsequent machining priority number).

There are the following three types of setting:

- Prior machining priority number:
To be set by means of numeric keys. The number is displayed in yellow.
Setting range (1 to 99)
- Subsequent machining priority number:
To be set by means of numeric keys after having reversed the display of **DELAY PRIORITY** by selecting its menu key.
The characters are displayed in reversed status.
Setting range (1 to 99)
- Without entry (Ordinary machining)
The priority number is not entered.
For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

5. Diameter of machining hole: HOLE- ϕ

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

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

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

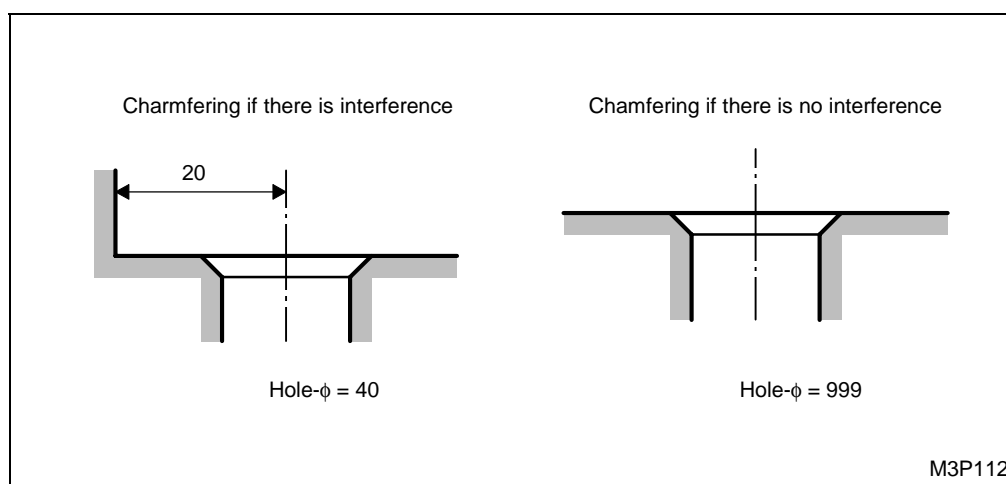


Fig. 4-2 Specification of diameter of machining hole

6. Depth of machining hole: HOLE-DEP

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

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

Note 1: For the chamfering cutter or the chamfering cycle of the centering drill, this item is specified as illustrated below.

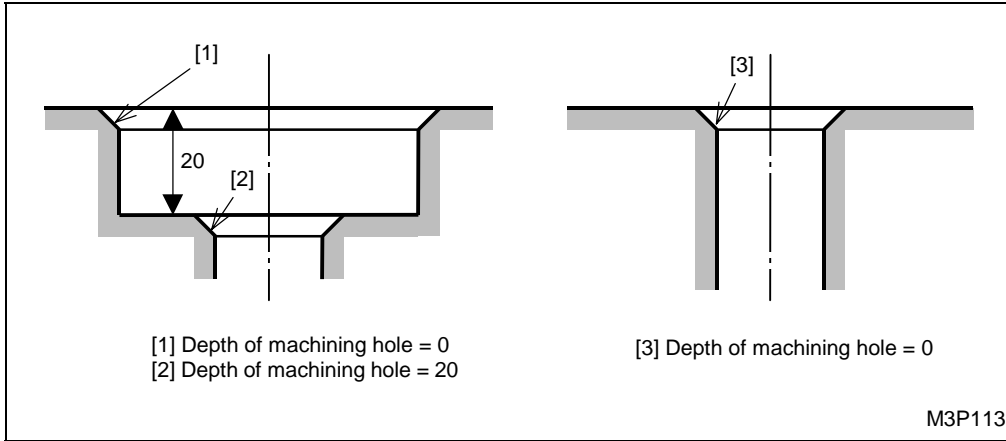
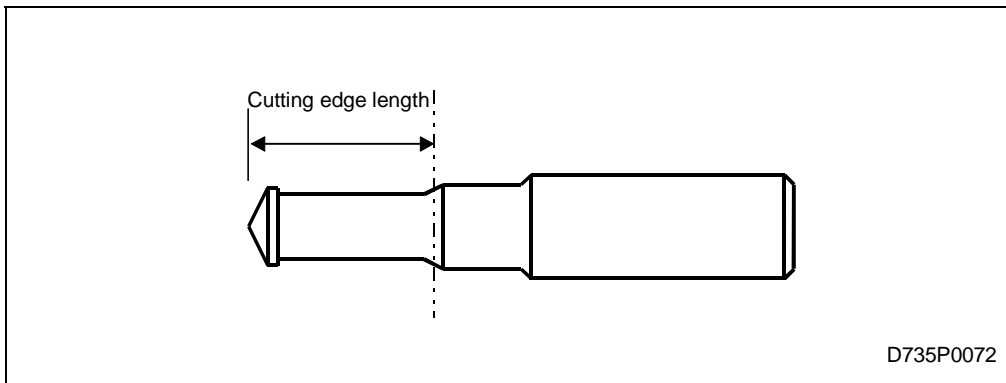


Fig. 4-3 Specification of depth of machining hole

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.



7. Diameter of pre-hole: PRE-DIA

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

The data for this item 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 4-7-7 “Tool path”, “8. Boring tool.”

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

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

TAPPING CYCLE	PECKING CYCLE 2	PLANET CYCLE							
------------------	--------------------	-----------------	--	--	--	--	--	--	--

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

8. Depth of the pre-hole: PRE-DEP

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

The data for this item 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 item.

Note 2: Enter the depth of the faced hole in the case of boring for this item. 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 item.

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 circular tornado cycle and precision rapid boring 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 4-7-7 “Tool path”, “4. End mill.”

Note 5: Data setting is not required for TAPPING CYCLE. Set “Cutting depth per peck” for PECKING CYCLE 2. The pecking cycle, however, will not occur if the item is left blank or zero is set. The value of the **D50** parameter “Pre-hole machining feed” is set for PLANET CYCLE automatically.

9. Cutting surface roughness: RGH

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

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

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

90°	118°	60°						
-----	------	-----	--	--	--	--	--	--

Note 2: For the drill, the drilling cycle can be selected from the menu. In automatic tool development mode, DRILLING CYCLE and PECKING CYCLE 1 or PECKING CYCLE 2 are determined automatically from the machining depth and the drilling diameter depending upon a parameter setting. These cycles can also be selected from the following menu.

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 4-7-7 “Tool path”, “2. Drill.”

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

10. Cutting depth: DEPTH

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

- Cutting depth on Z-axis per pass in the case of drill.
First Z-axial cutting depth in the case of drilling (DECREME PECKING CYCLE).
Data setting is not required in the case of drilling (AUTOPECK CYCLE).
- Amount of chamfering in the case of chamfering cutter.
- Cutting depth in the radial direction per pass in the case of end mill.
Data setting is not required in the case of end milling (precision rapid boring tornade cycle).
- 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 automatic tool 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.
- In the case of the centering drill, the machining cycle can be selected from the menu. In the tool data automatic development mode CTR-DRIL CYCLE (centering drilling cycle) is selected. For details refer to Subsection 4-7-7 “Tool path”, “1. Centering drill.”

CTR-DRIL CYCLE	CHAMFER CYCLE							
----------------	---------------	--	--	--	--	--	--	--

11. Circumferential speed: C-SP

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

The tool material types in the menu are the same as those which have been set on the **CUTTING CONDITION (W.-MAT./T.-MAT.)** display.

To register new tool material types, refer to Section 8-1, "CUTTING CONDITION (W.-MAT./T.-MAT.) Display," of the relevant Operating Manual.

HASS AUTO	CARBIDE AUTO								
--------------	-----------------	--	--	--	--	--	--	--	--

Data can also be set using the numeric keys.

The tool data window can be displayed if the cursor is set to "C-SP" or "FR."

12. Feedrate: FR

Used to specify the feedrate of the tool.

Same as the circumferential speed, the entry of data is done by means of menu keys or numeric keys.

Note: The value of the **D51** parameter is set automatically for the planetary tapping cycle.

13. M-code: M

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

The entry of data is done by the selection of the desired menu key or by using the numeric keys.

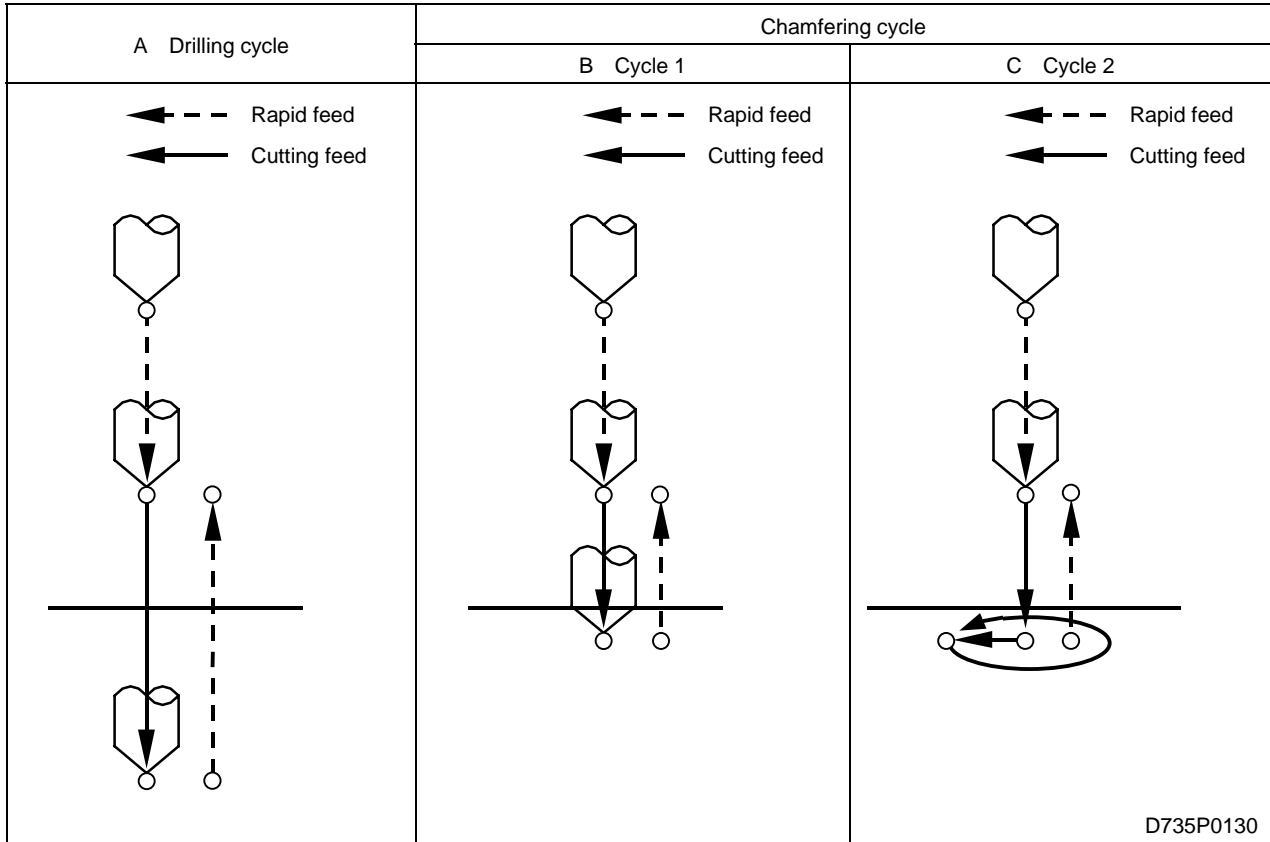
01 OPT. STOP	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST COOLANT	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST	51 THR COOLANT	>>>
00 PROG STOP	19 SPNDL ORIENT	35 T-BRK DETECT	38 SPNDL GEAR L/M	39 SPNDL GEAR H	33 OUT MSR UNIT	34 IN MSR UNIT			>>>
16 OPEN ATC CVR	15 CLOSE ATC CVR	11 TOOL UNCLAMP	10 TOOL CLAMP	53 CHIP VAC ON	09 CHIP VAC OFF				>>>

Note: The M-codes listed in the menu vary from machine to machine.

4-7-7 Tool path

1. Centering drill

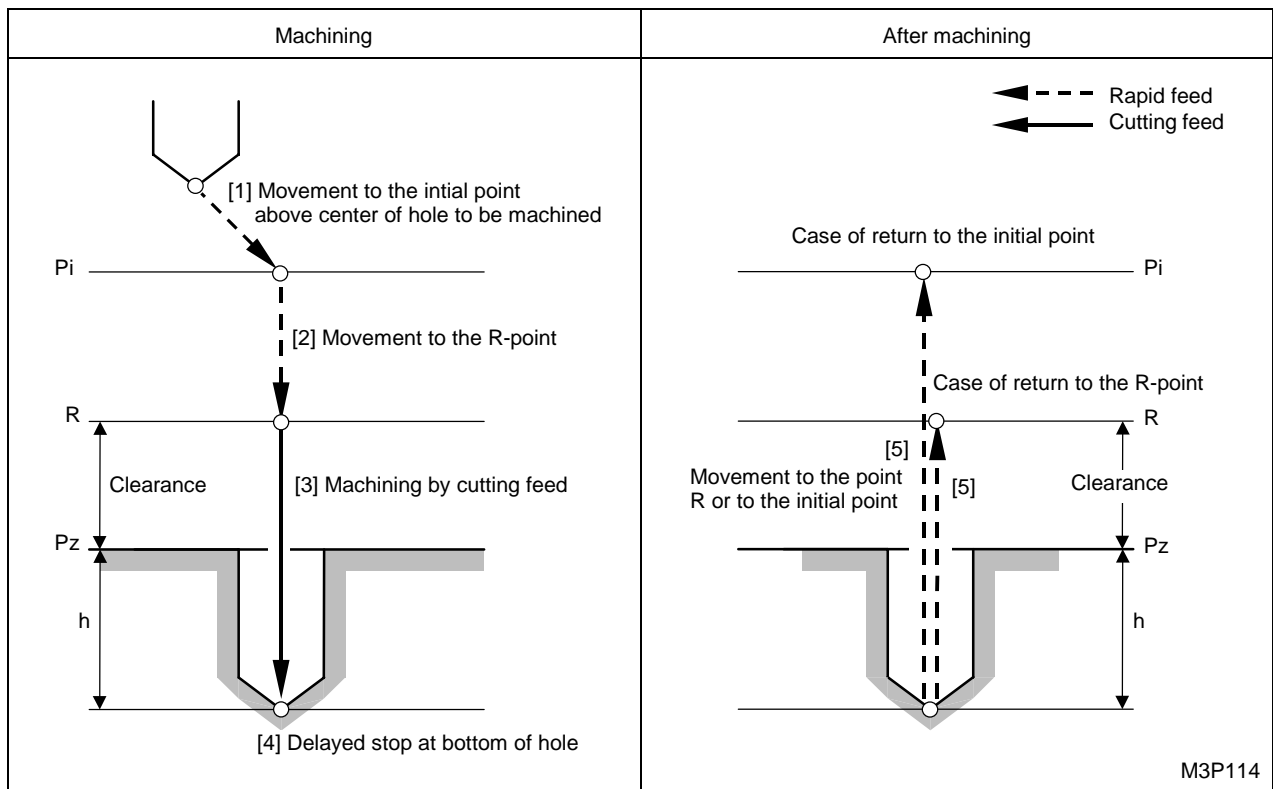
The cycle of machining with a centering drill is available in the following three types.



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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of machining surface to be entered in the shape sequence

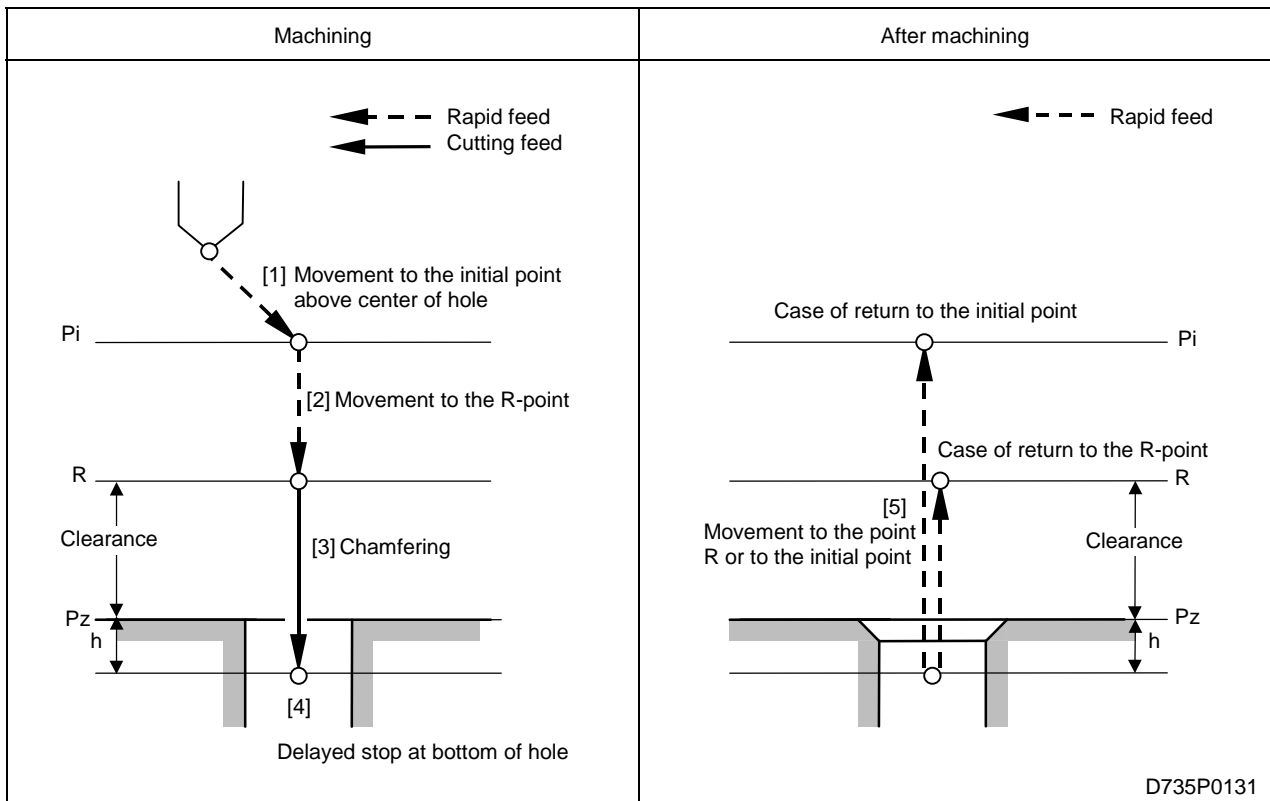
R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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

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

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

B. Cycle 1 of chamfering cycle

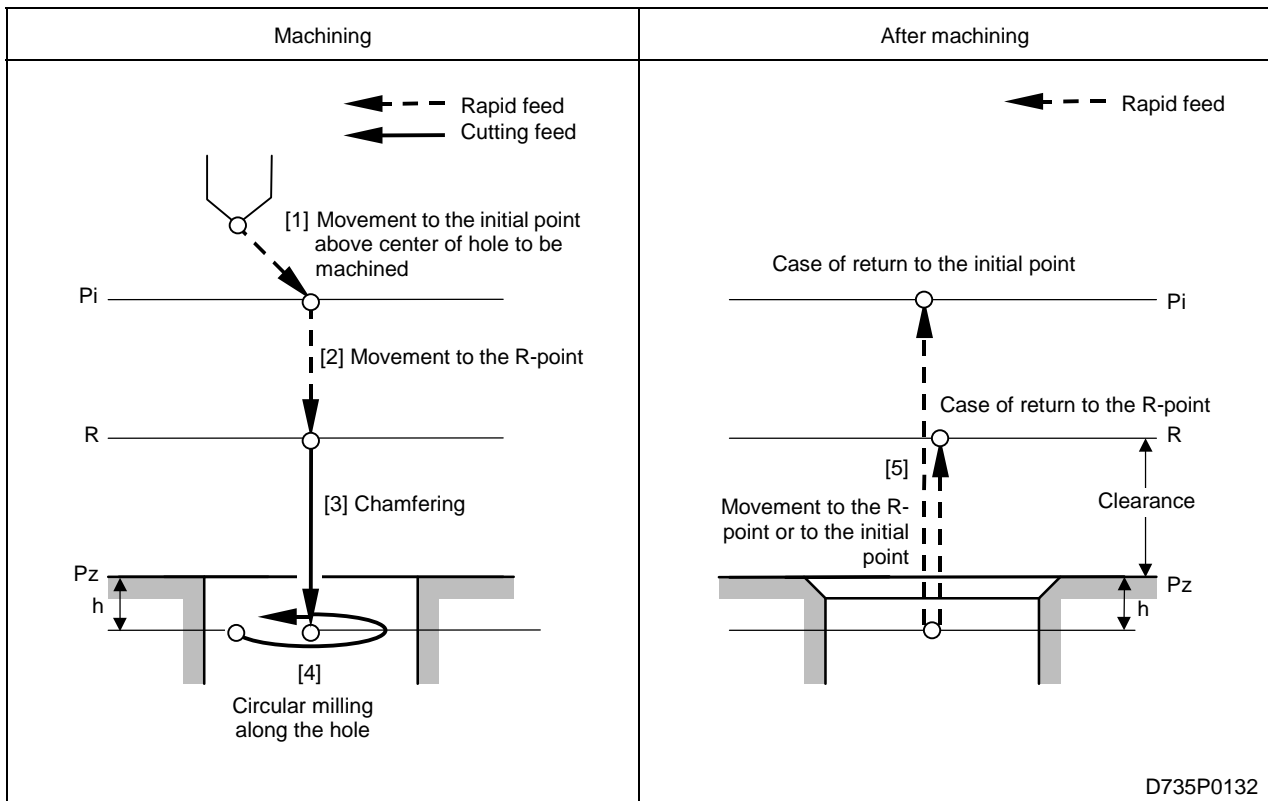


The bold codes represent the parameter addresses.

- Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- Pz: Z coordinate of the machining surface to be entered in the shape sequence
- R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- 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 feed on Z-axis 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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.

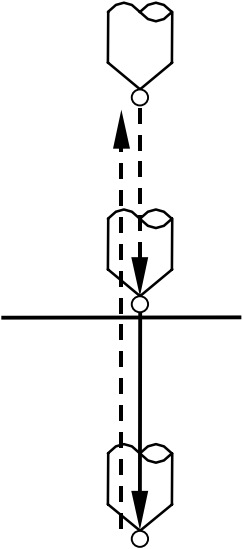
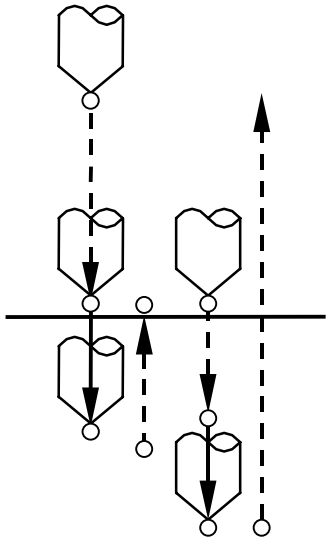
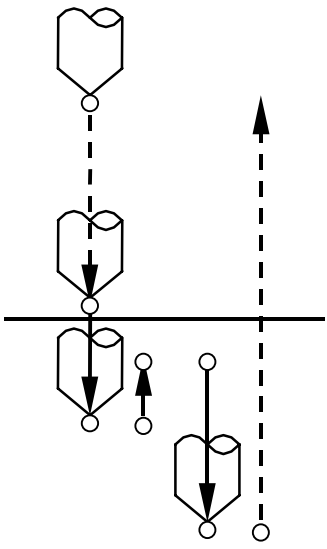
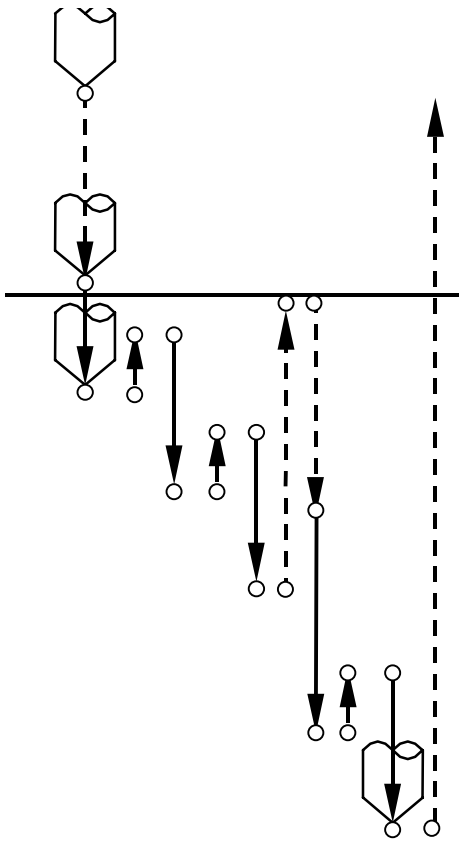
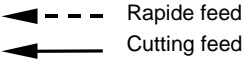
- Case where the bit 7 of parameter **D91** is 1.

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

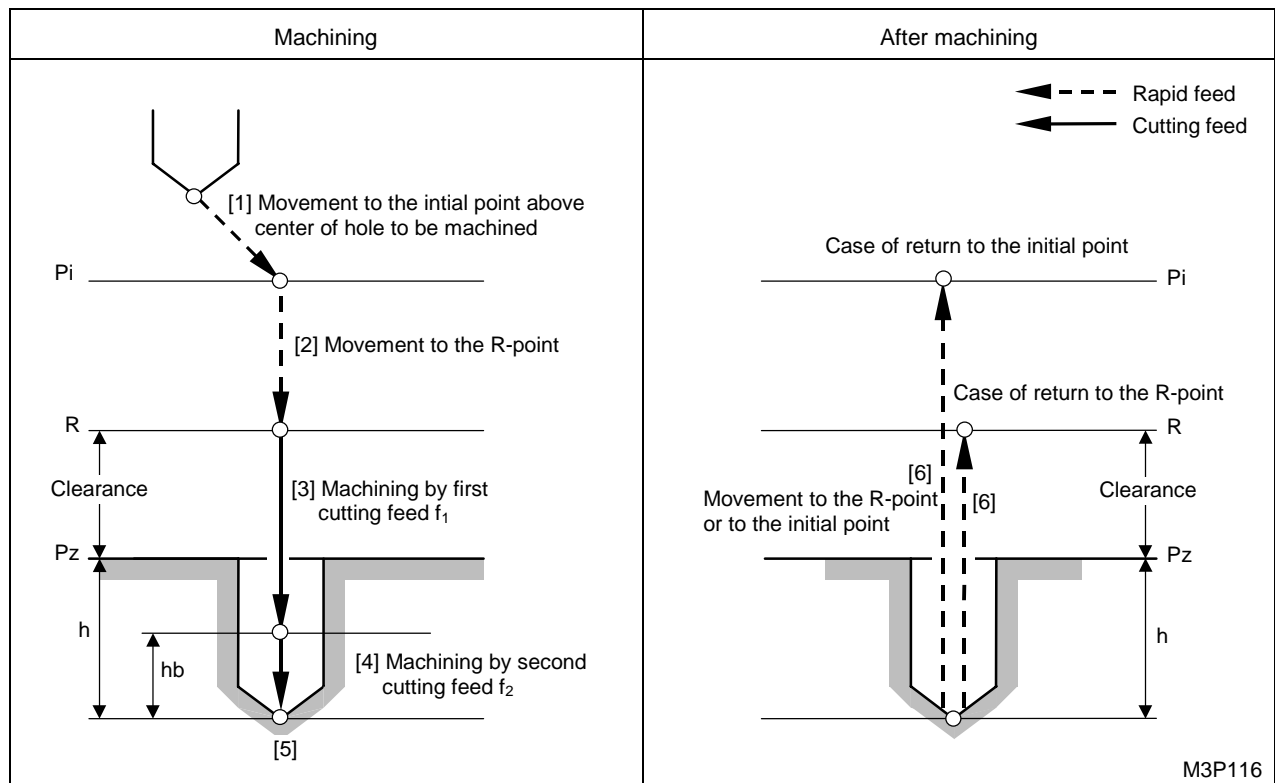
The cycle of machining with drill is available in the following eight types.

<p>A. DRILLING CYCLE</p> 	<p>B. Deep-hole drilling cycle (PECKING CYCLE 2)</p> 	<p>C. High speed deep-hole drilling cycle (PECKING CYCLE 1)</p> 
<p>D. Very deep-hole drilling cycle (PECKING CYCLE 3)</p> 	<p>E. Auto-pecking cycle of the cutting load detection type (AUTOPECK CYCLE)</p> <p>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.</p> <p>F. Deep-hole drilling cycle for gradual depth reduction (DECREME. PECKING CYCLE 2)</p> <p>In this machining cycle, the cutting depth is decremented with respect to a normal deep-hole drilling cycle each time the workpiece is cut.</p> <p>G. High speed deep-hole drilling cycle for gradual depth reduction (DECREME. PECKING CYCLE 1)</p> <p>In this machining cycle, the cutting depth is decremented with respect to a high deep-hole drilling cycle each time the workpiece is cut.</p> <p>H. Very deep-hole drilling cycle for gradual depth reduction (DECREME. PECKING CYCLE 3)</p> <p>In this machining cycle, the cutting depth is decremented with respect to a very deep-hole drilling cycle each time the workpiece is cut.</p> <div style="text-align: right;">  <p>← - - - Rapide feed ← Cutting feed</p> </div> <p style="text-align: right;">D734P0012</p>	

Remark 1: See Items A to F 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**.

A. Drilling cycle



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

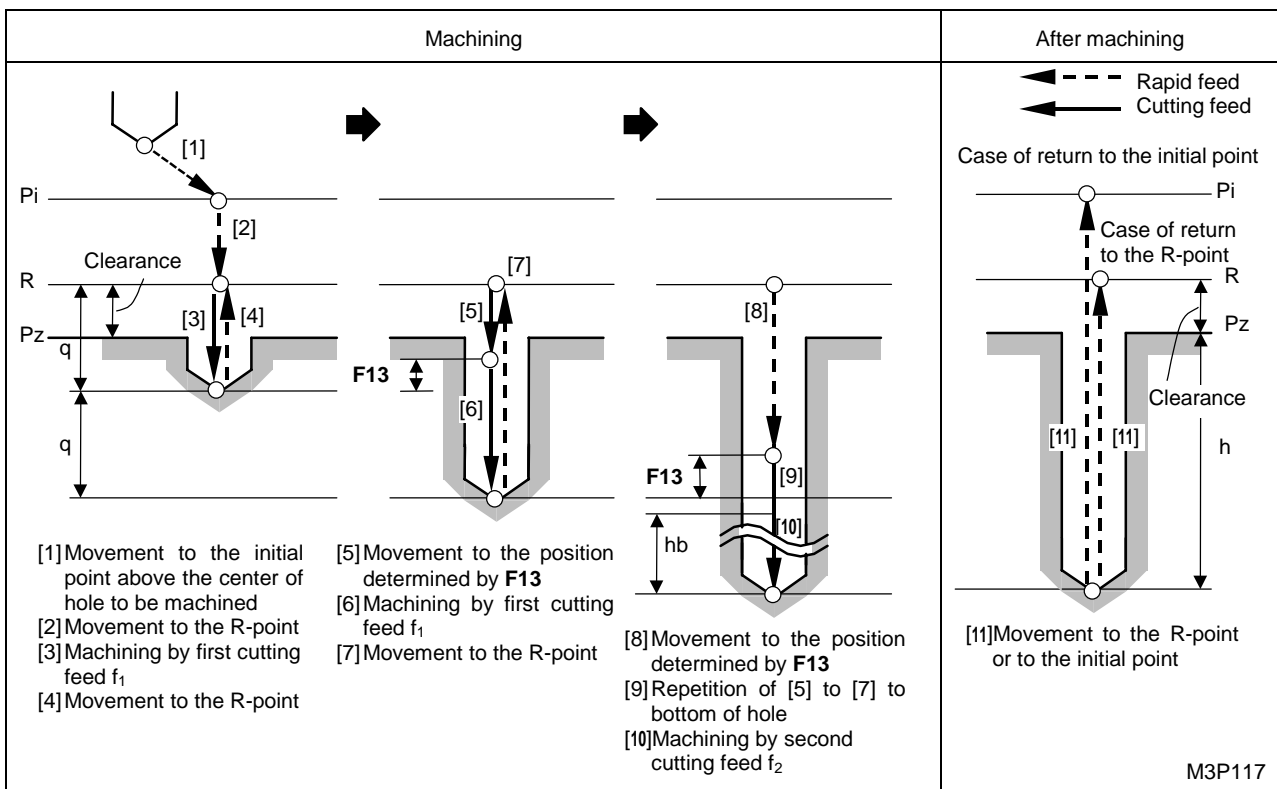
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

f₁: Feedrate (FR) to be set for the tool sequence

f₂: Feedrate to be modified by the data PRE-DEP (feedrate updating rate)

$f_2 = f_1 \times \text{Feedrate updating rate}$

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



The bold codes represent the parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

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

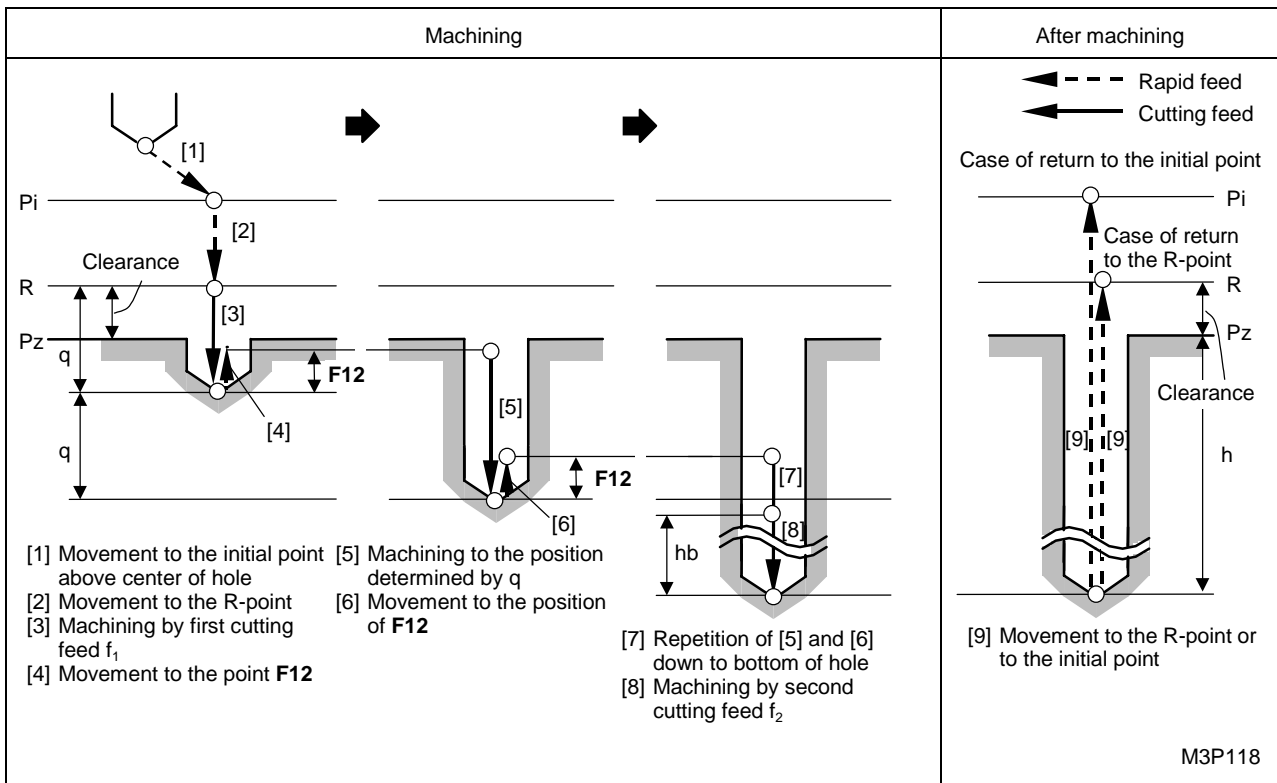
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

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

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

$f_2 = f_1 \times \text{Feedrate updating rate}$

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



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

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

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

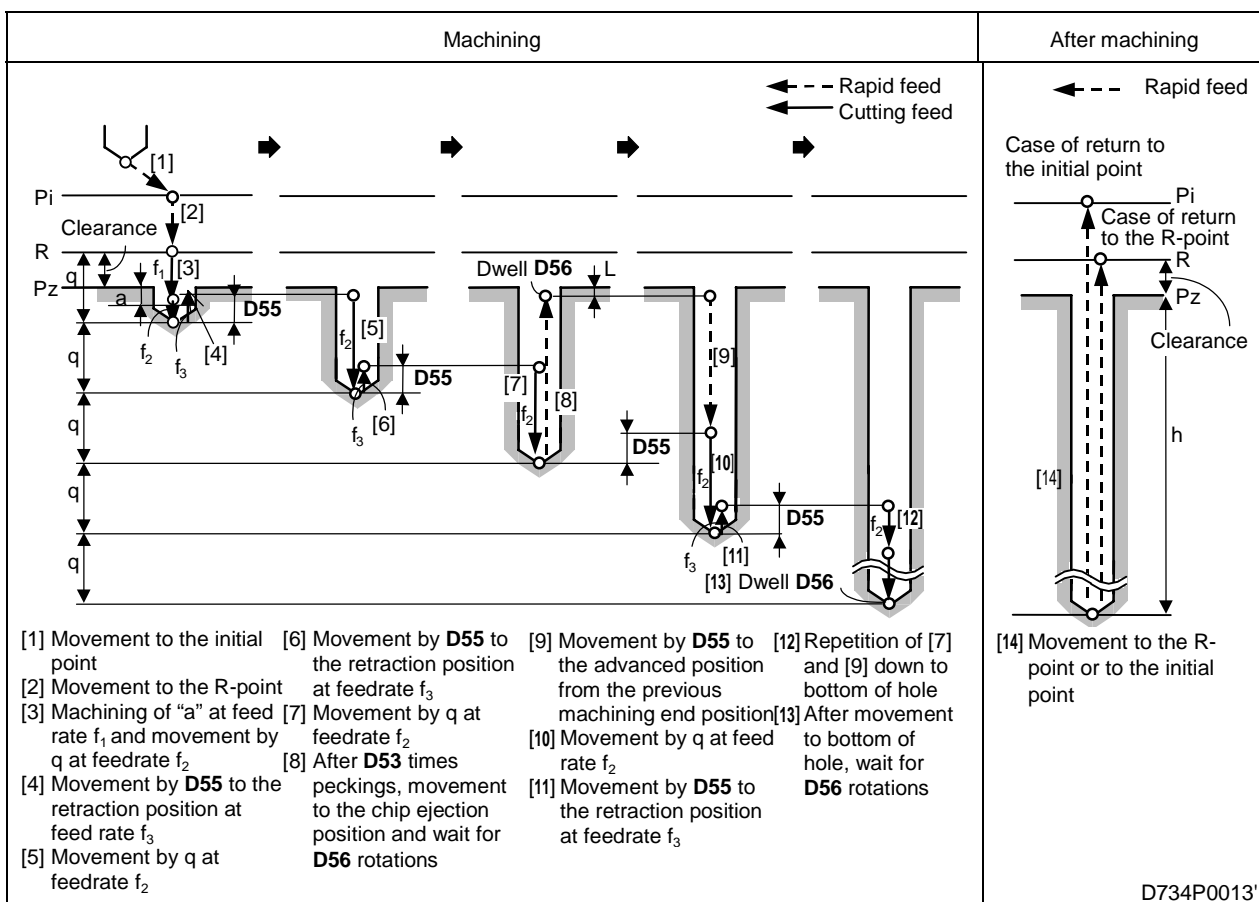
f₁: Feedrate (FR) to be set for the tool sequence

f₂: Feedrate to be modified by the data PRE-DEP (feedrate updating rate)

$f_2 = f_1 \times \text{Feedrate updating rate}$

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)



D734P0013'

The bold codes represent the parameter addresses.

During the "n"th cutting operation, if $(q \times n) < \mathbf{D55}$, 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].

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

a: Cutting area (= data LENG COMP. (tool correction) on the **TOOL DATA** display)

q: Cutting 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.

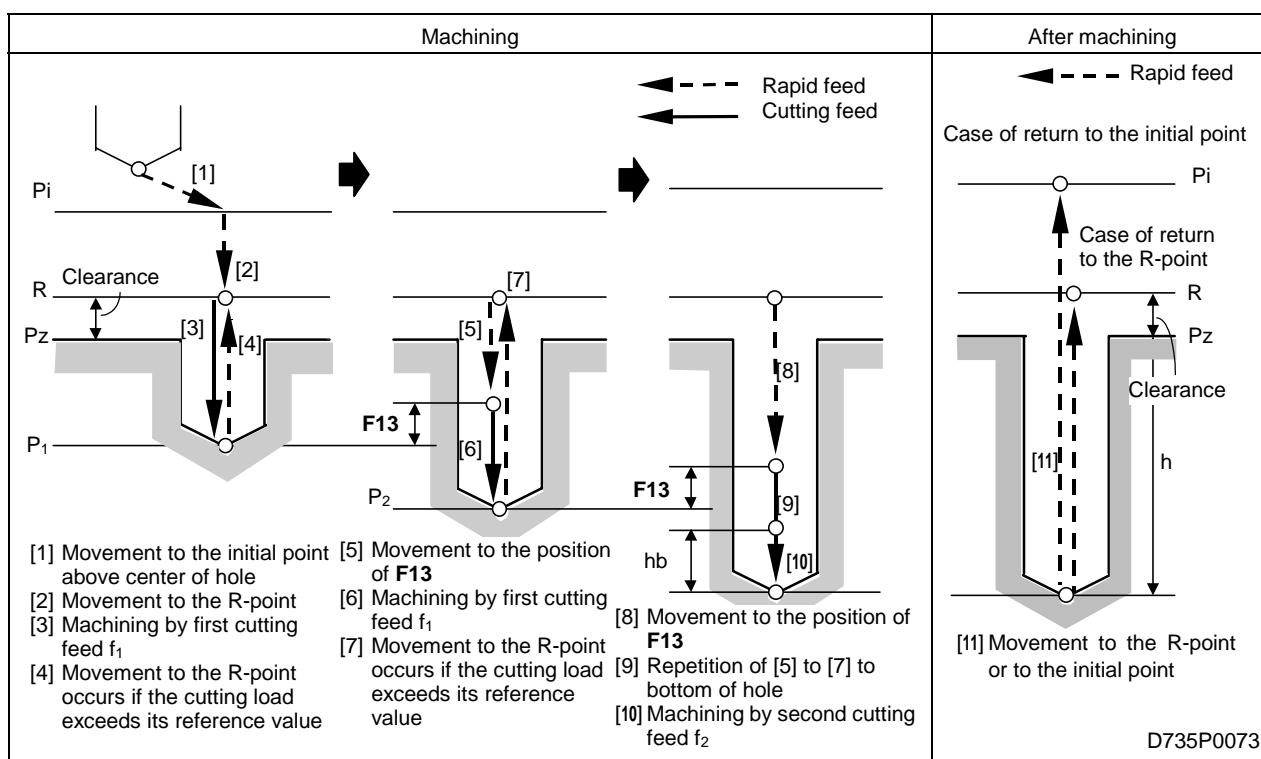
L: Chip ejection distance calculated from the data ACT- ϕ (tool diameter: D) and data LENG COMP. (tool correction) on the **TOOL DATA** display

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

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

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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

P₁, P₂: The positions where autonomous pecking will occur if the cutting load exceeds its reference value

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

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

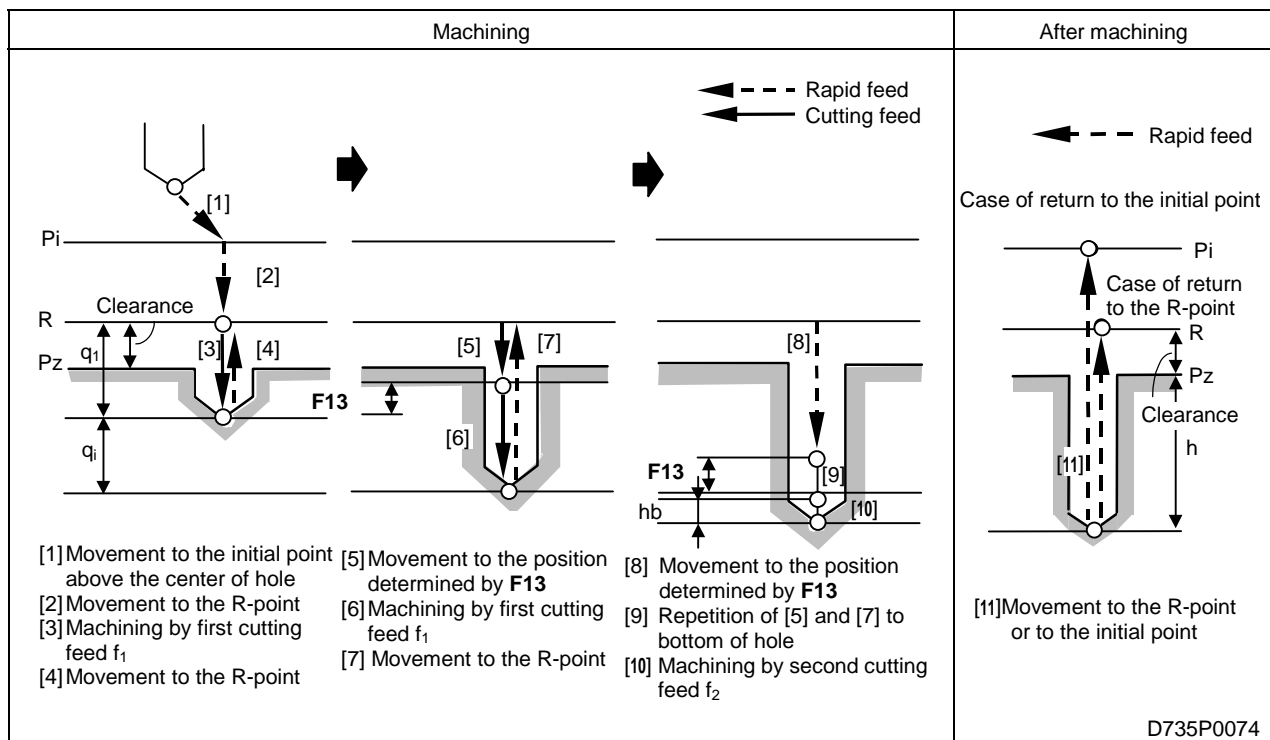
f₁: Feedrate (FR) to be set for the tool sequence

f₂: Feedrate to be modified by the data PRE-DEP (feedrate updating rate)

$f_2 = f_1 \times \text{Feedrate updating rate}$

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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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.

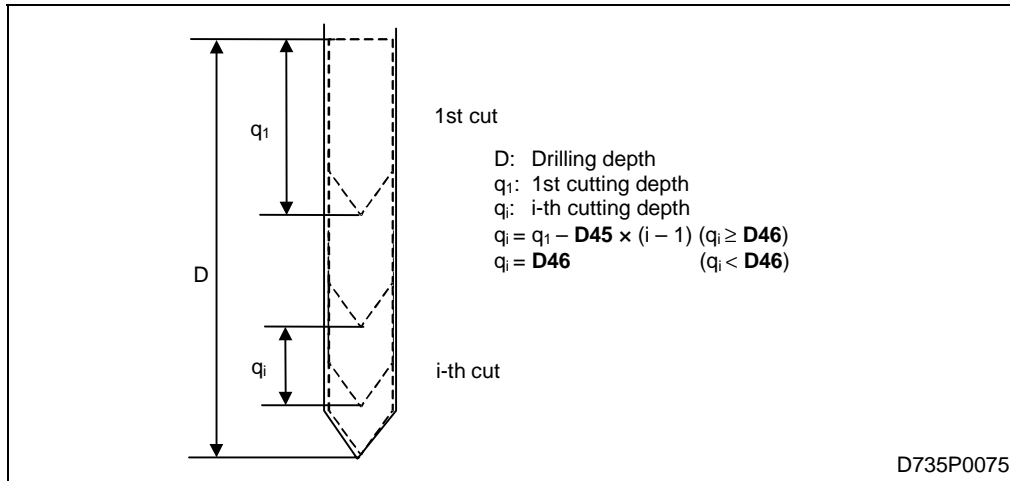
h: Hole depth to be calculated by the data HOLE-DEP entered in the tool sequence and also the data LENG COMP. (tool correction) on the **TOOL DATA** display

$h = \text{Depth of machining hole} + \text{Tool correction}$

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.



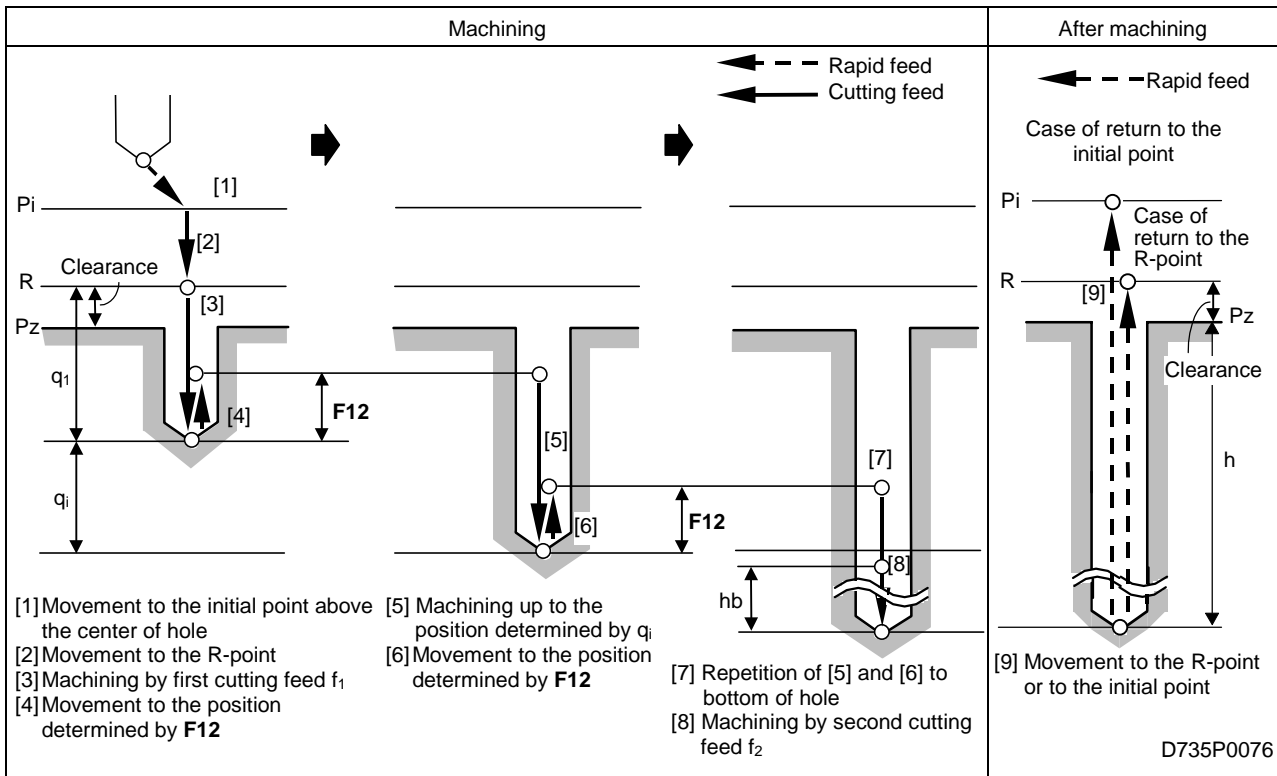
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

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)

$$f_2 = f_1 \times \text{Feedrate updating rate}$$

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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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.

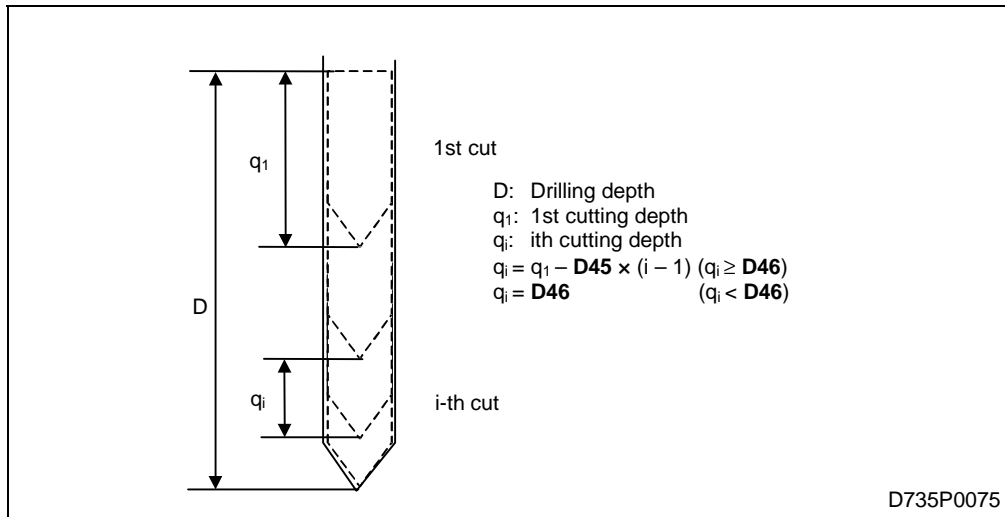
h: Hole depth to be calculated by the data HOLE-DEP entered in the tool sequence and also the data LENG COMP. (tool correction) on the **TOOL DATA** display

$h = \text{Depth of machining hole} + \text{Tool correction}$

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

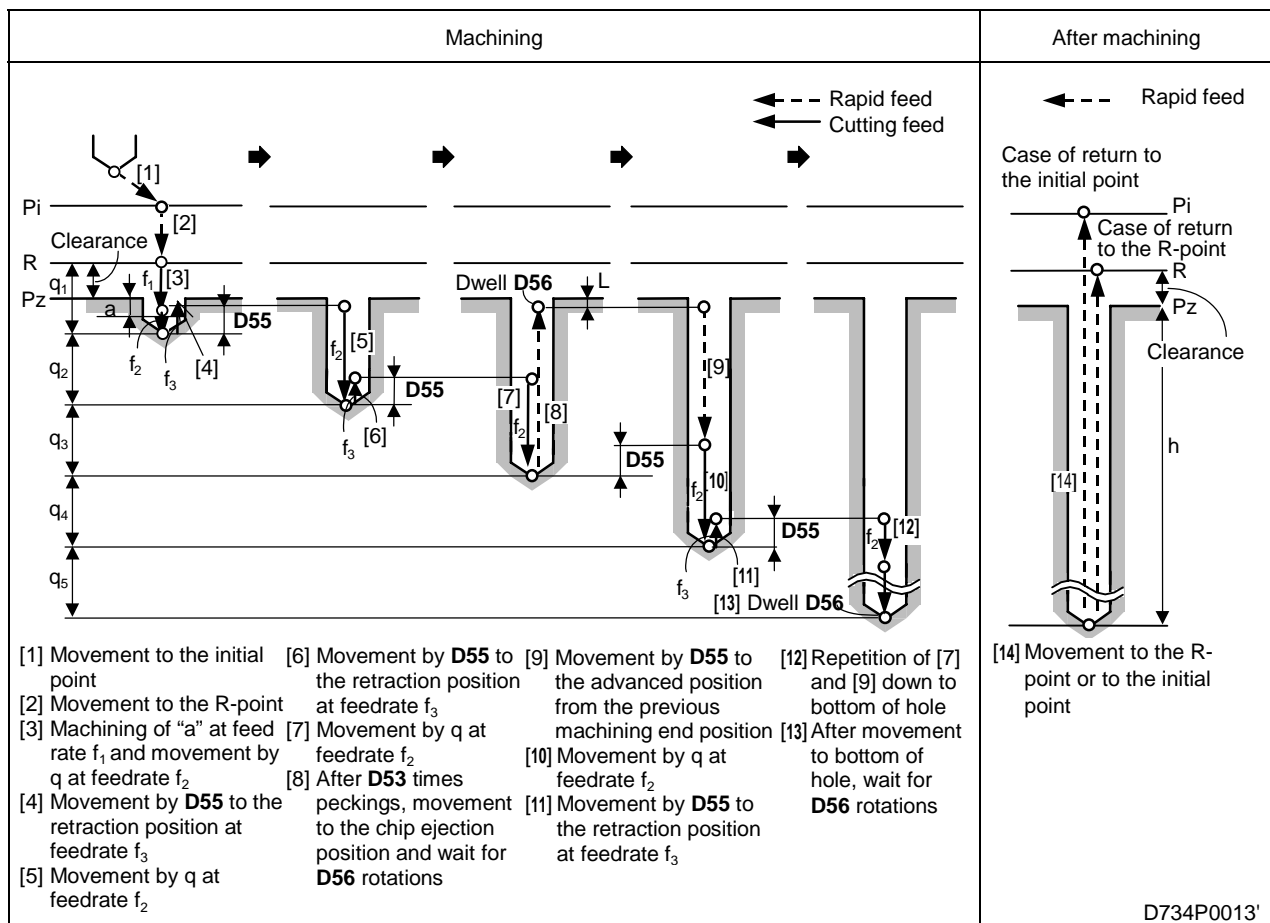
f_1 : Feedrate (FR) to be set for the tool sequence

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

$$f_2 = f_1 \times \text{Feedrate updating rate}$$

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.

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



The bold codes represent the parameter addresses.

During the "n"th cutting operation, if $(q \times n) < \mathbf{D55}$, 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].

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

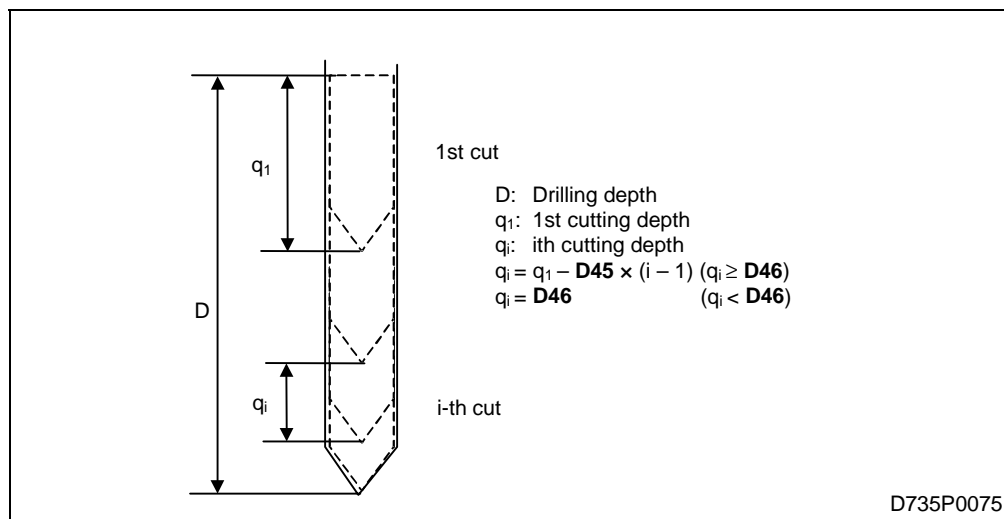
a: Cutting area (= data LENG COMP. (tool correction) on the **TOOL DATA** display)

q_1 : Cutting depth to be entered in the tool sequence data

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.

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

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

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

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.

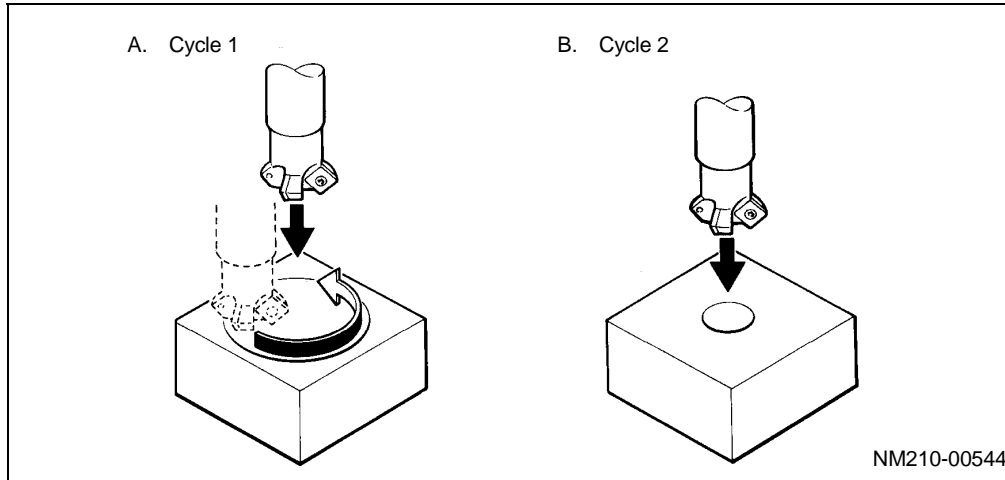
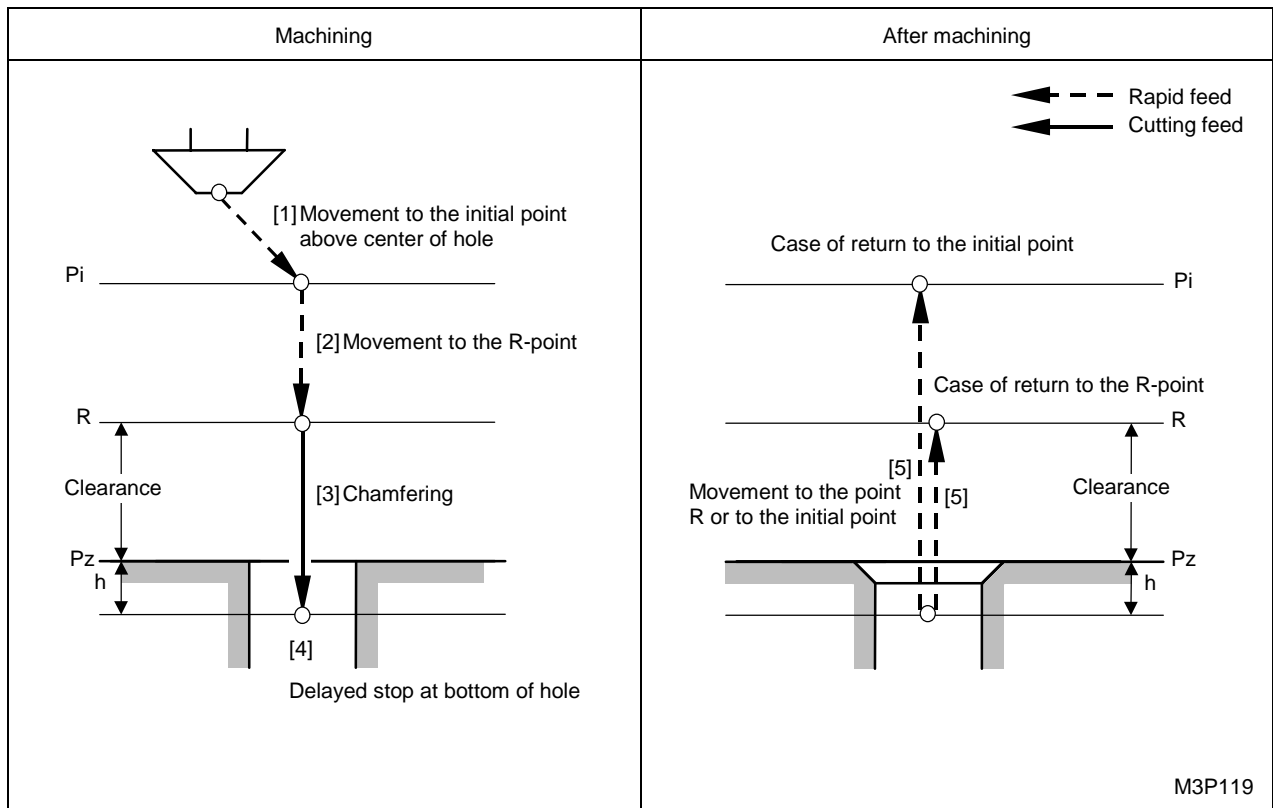


Fig. 4-4 Cycle 1 and cycle 2

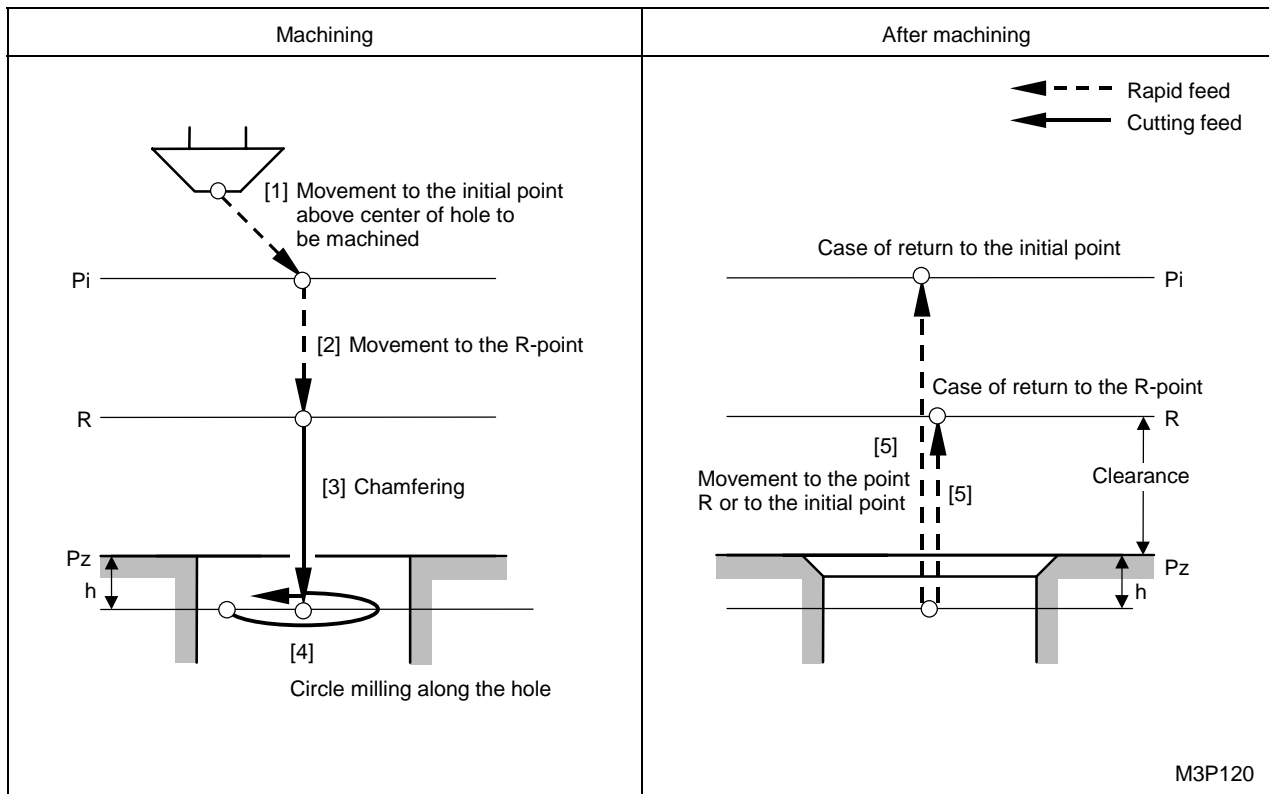
A. Cycle 1



The bold codes represent the parameter addresses.

- Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- Pz: Z coordinate of the machining surface to be entered in the shape sequence
- R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- h: Optimum distance to be automatically calculated by the data HOLE-φ and HOLE-DEP in the tool sequence and also the data ANG on the **TOOL FILE** display

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

B. Cycle 2

The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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 2: The time of delayed stop of the feed on Z-axis at bottom of hole is set by the parameter **D16**.

Note 3: 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**.

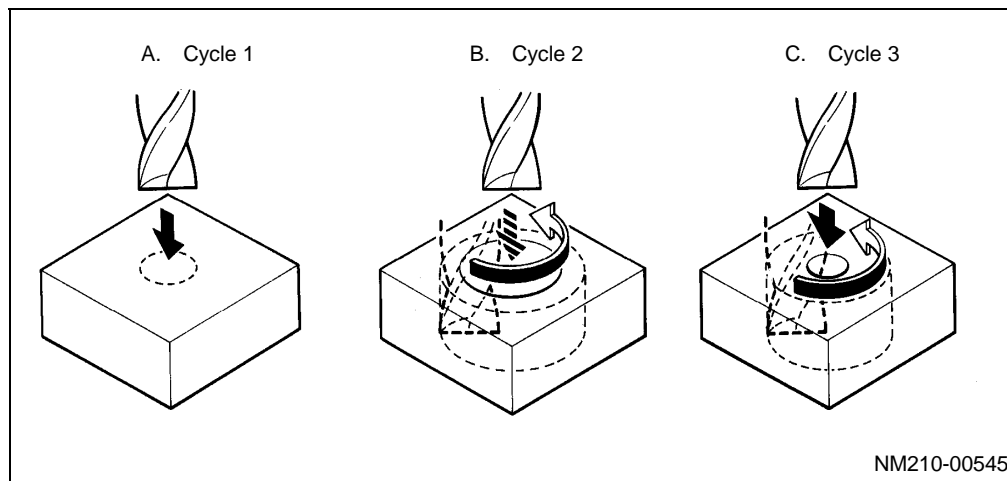
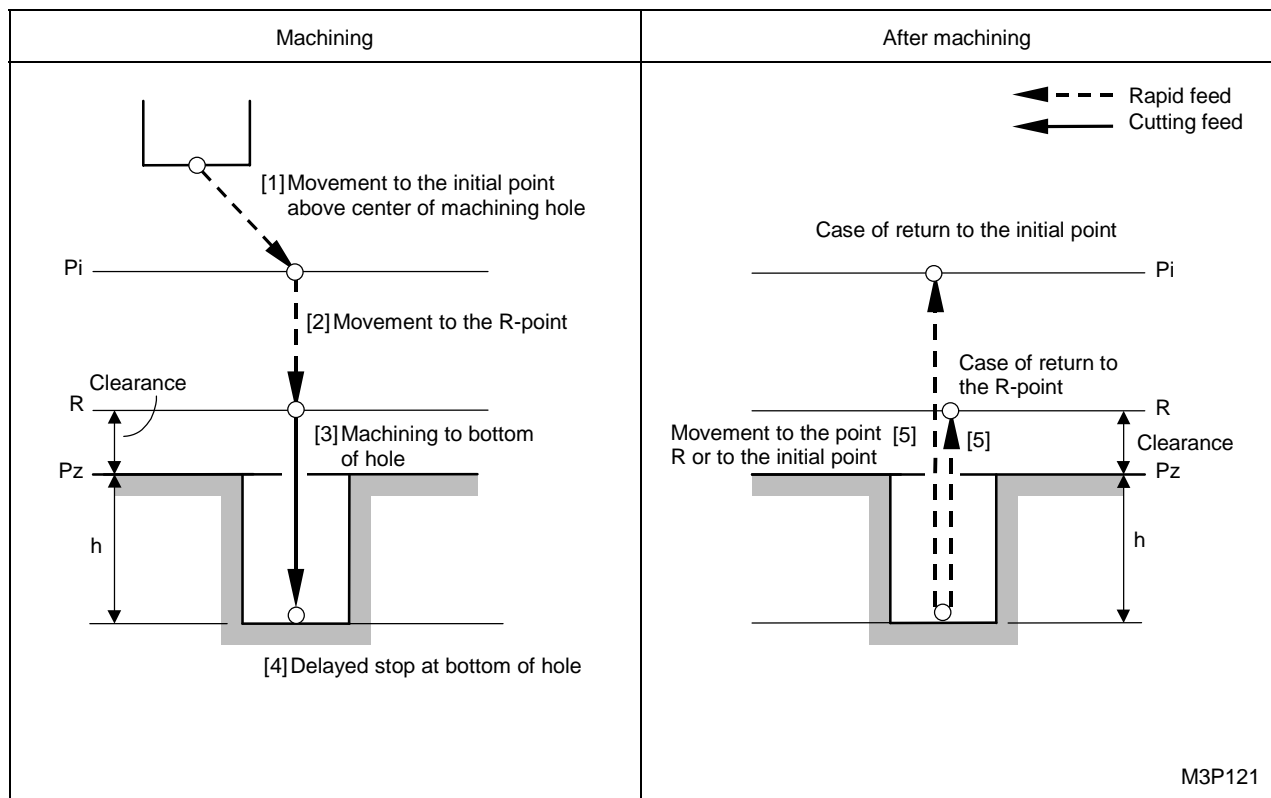


Fig. 4-5 Circular milling cycles 1, 2 and 3

A. Cycle 1



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

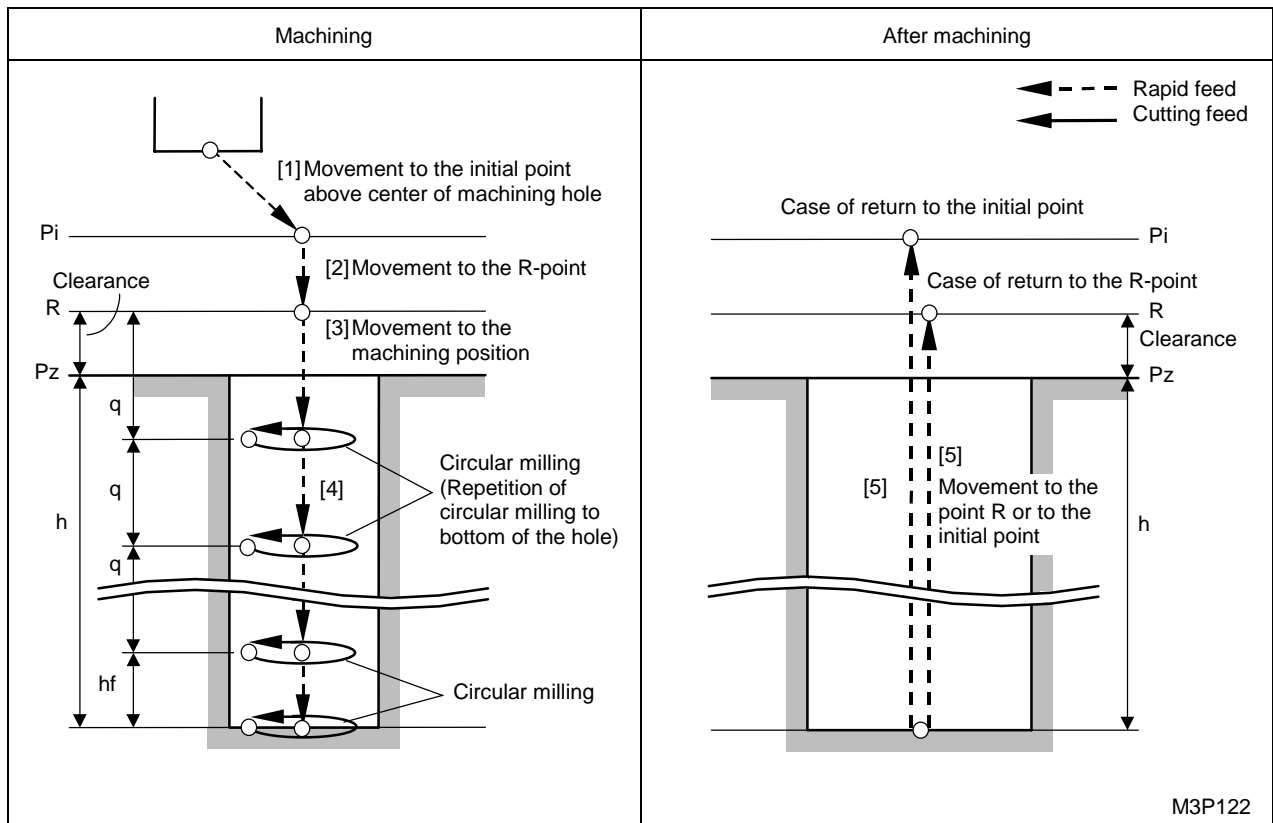
Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

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

B. Cycle 2



M3P122

The bold codes represent the parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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

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

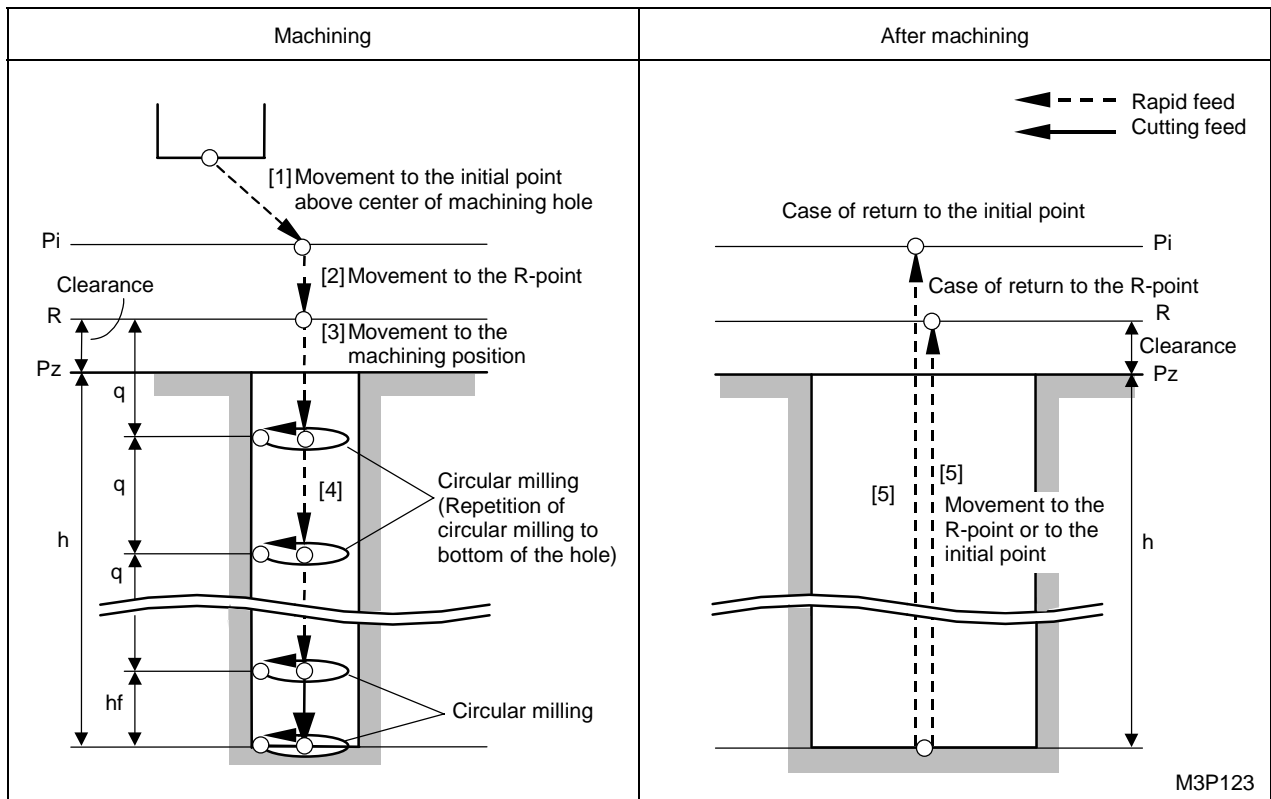
q: Cutting depth on Z per pass to be determined by:

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

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

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

C. Cycle 3



Pi: Initial point (See “Positions of the initial point and R-point” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and R-point” in Section 4-6.)

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

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

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

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

(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.

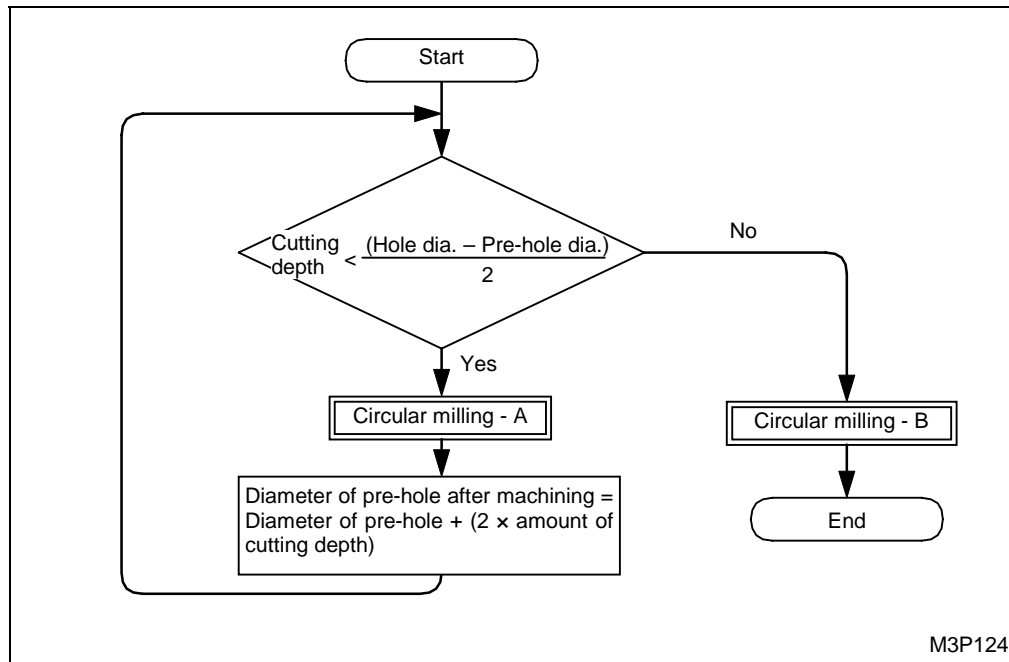


Fig. 4-6 Circular milling

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

1. Circular milling-A

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

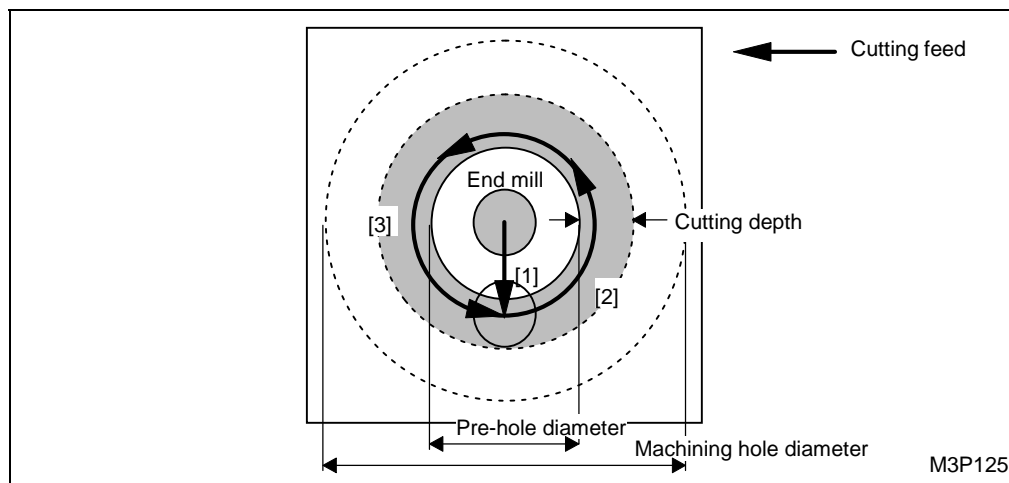


Fig. 4-7 Circular milling-A

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

2. Circular milling-B

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

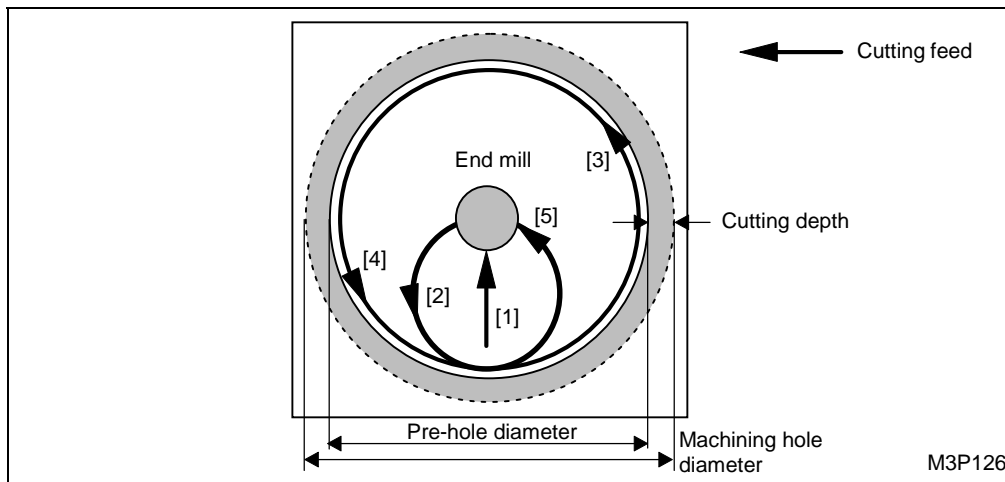


Fig. 4-8 Circular milling-B

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

Note: However, when bit 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.

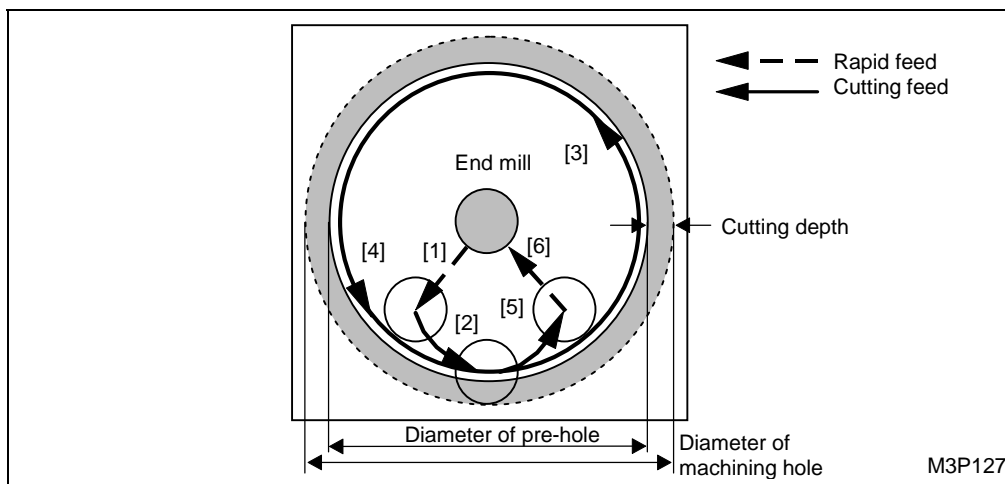
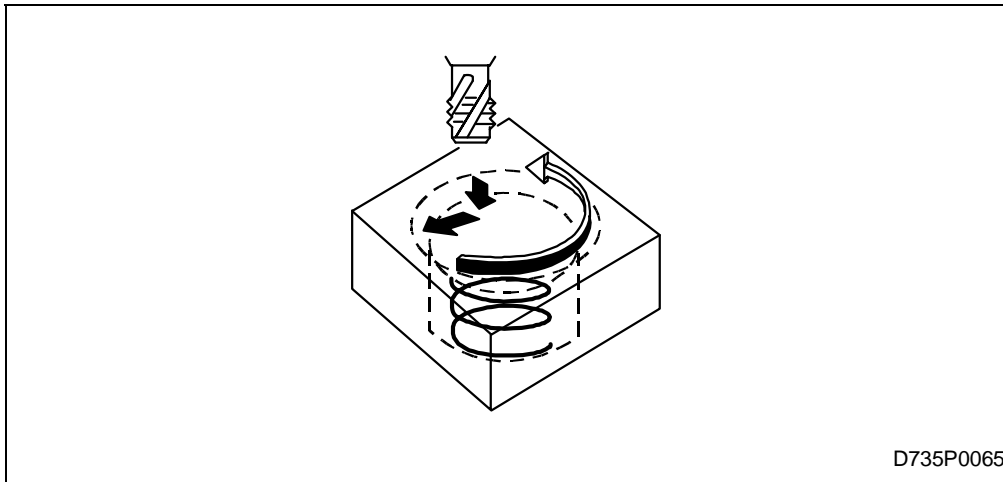


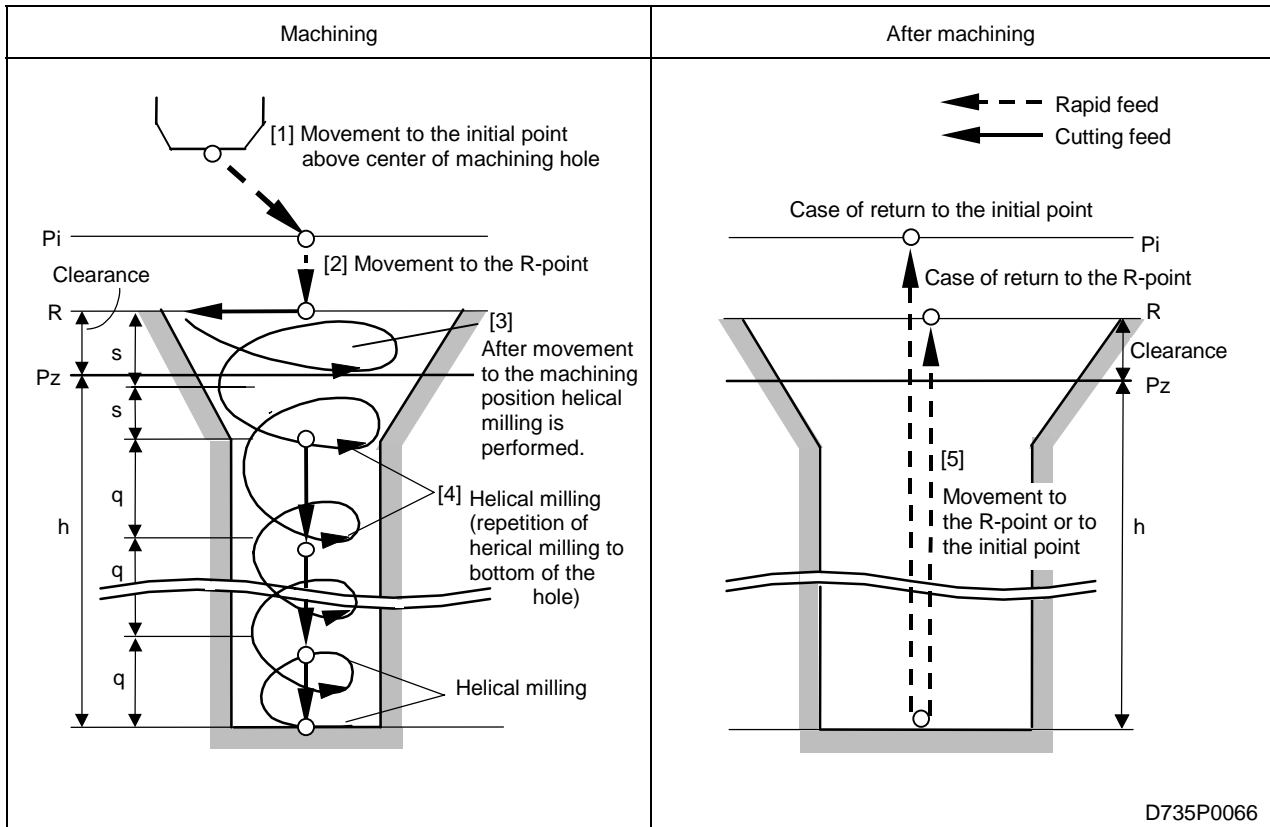
Fig. 4-9 Circular milling-B (case of shortening in chamfering)

- 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>



D735P0065



D735P0066

The bold codes represent the parameter addresses.

- Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)
- Pz: Z coordinate of the machining surface to be entered in the shape sequence
- R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)
- 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. 4-10.

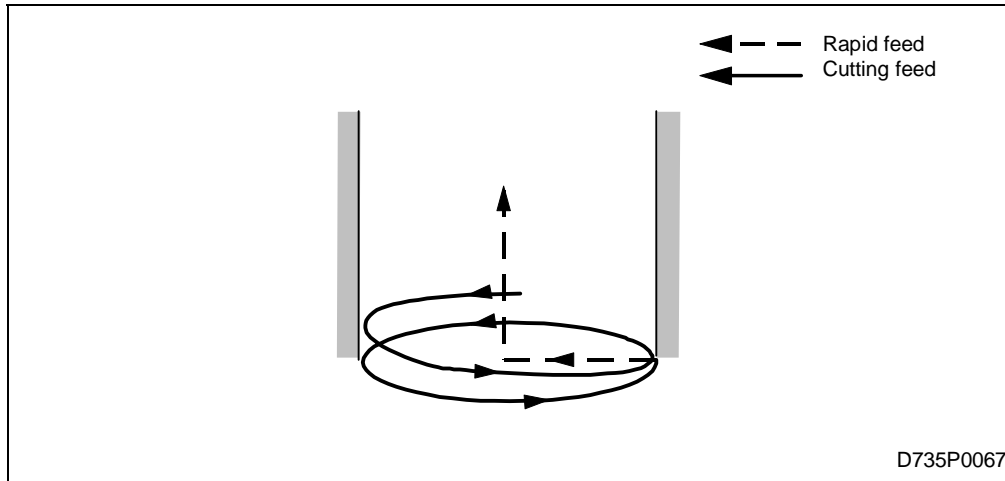


Fig. 4-10 Circular helical processing (with bottom finishing)

After helical interpolation down to the bottom of the hole, one entire circumference of arc interpolation occurs. Next, the tool moves to the center of the hole and then moves in the rapid feed rate to its initial point or to 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. 4-11.

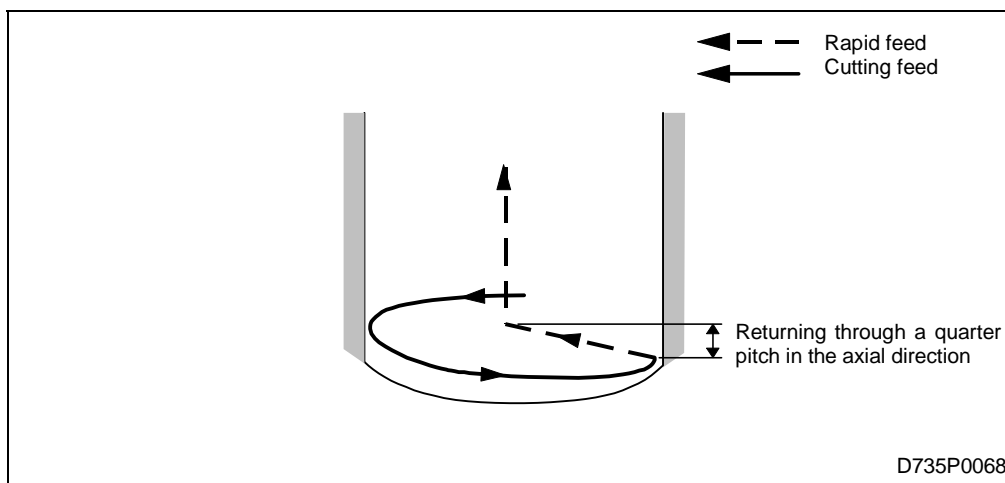
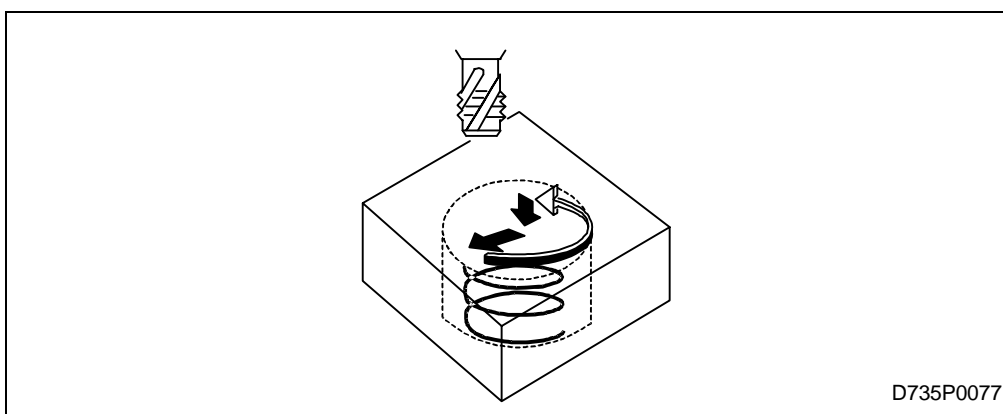


Fig. 4-11 Circular helical processing (without bottom finishing)

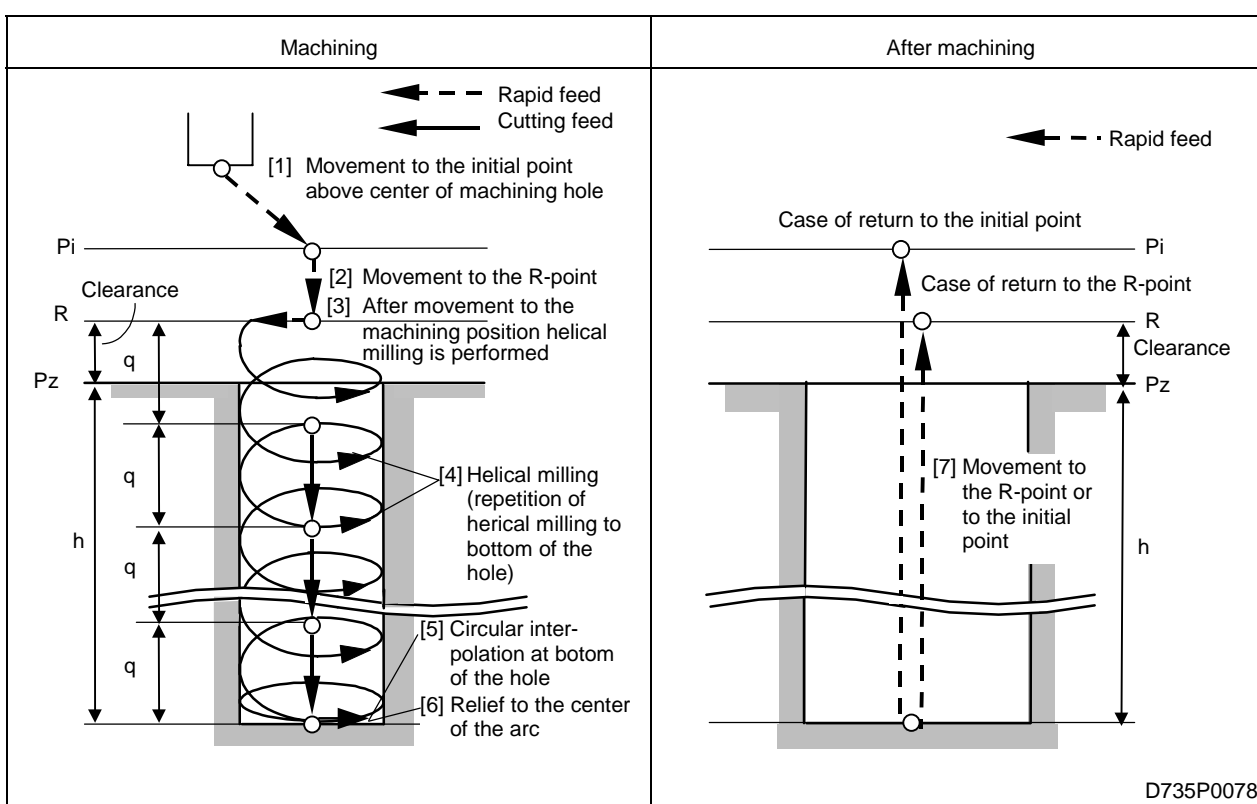
After helical interpolation down to the bottom of the hole, the tool moves to the center of the hole by returning through a quarter pitch in the axial direction and then moves in rapid feed rate to its initial point or to R-point in the axial direction.

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.



The bold codes represent the parameter addresses.

- Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)
 - Pz: Z coordinate of the machining surface to be entered in the shape sequence
 - R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)
 - 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.

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

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

Circular milling

The movement of circular milling is as shown below.

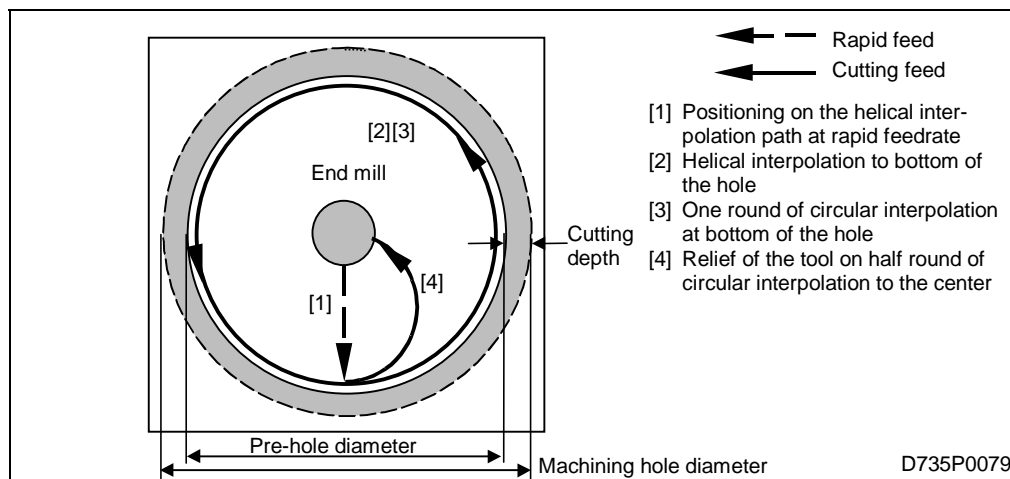


Fig. 4-12 Circular milling

- 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.

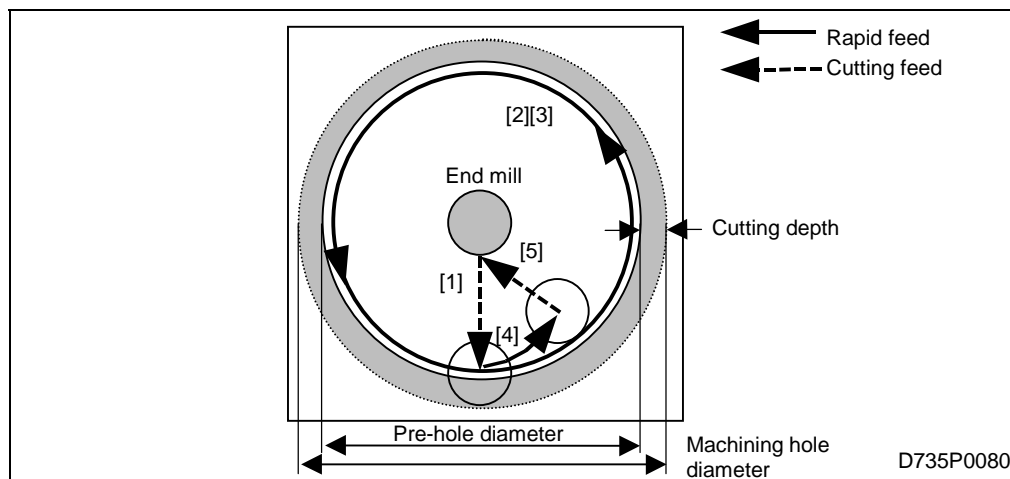
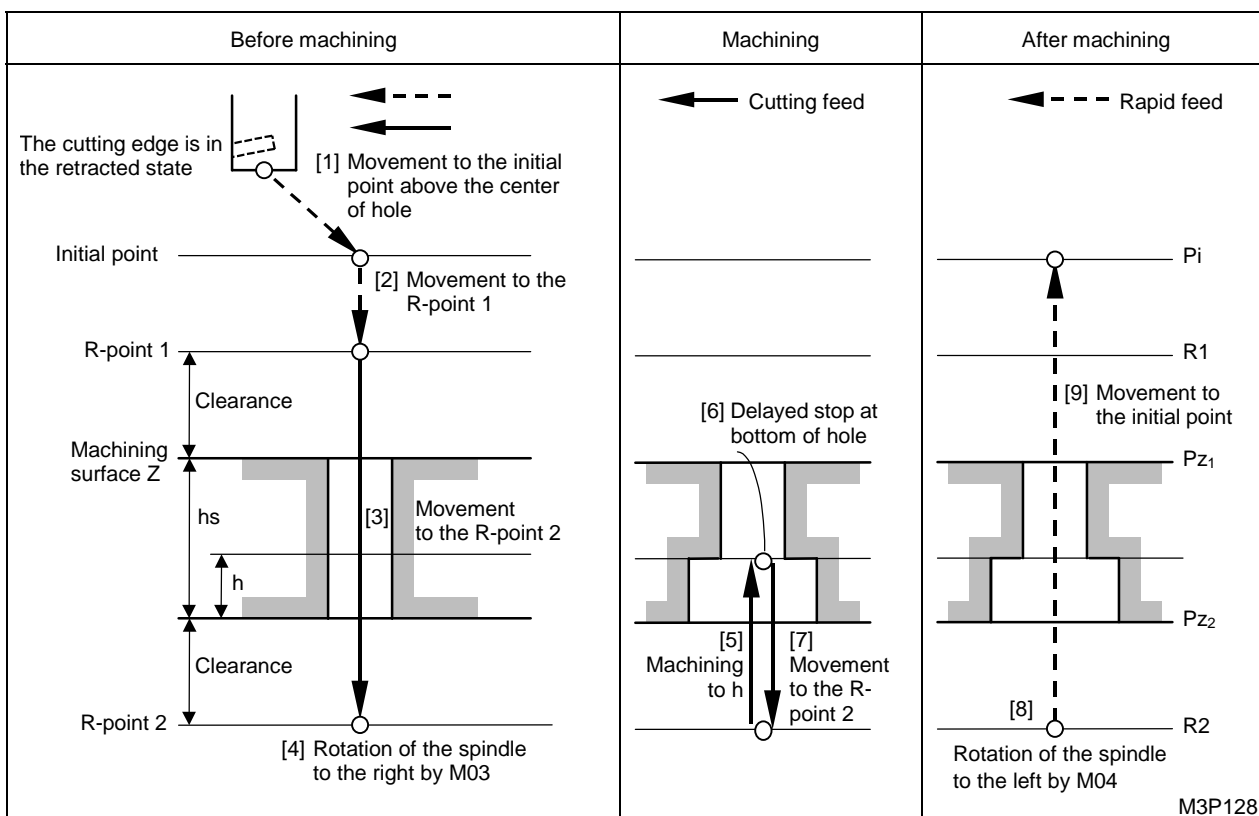


Fig. 4-13 Circular milling (case of shortening in chamfering)

- 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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz₁: Z coordinate of the machining surface to be entered in the shape sequence

Pz₂: Position at a distance of hs from Pz₁

R1, R2: Safety clearance above the points Pz₁, Pz₂ respectively (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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

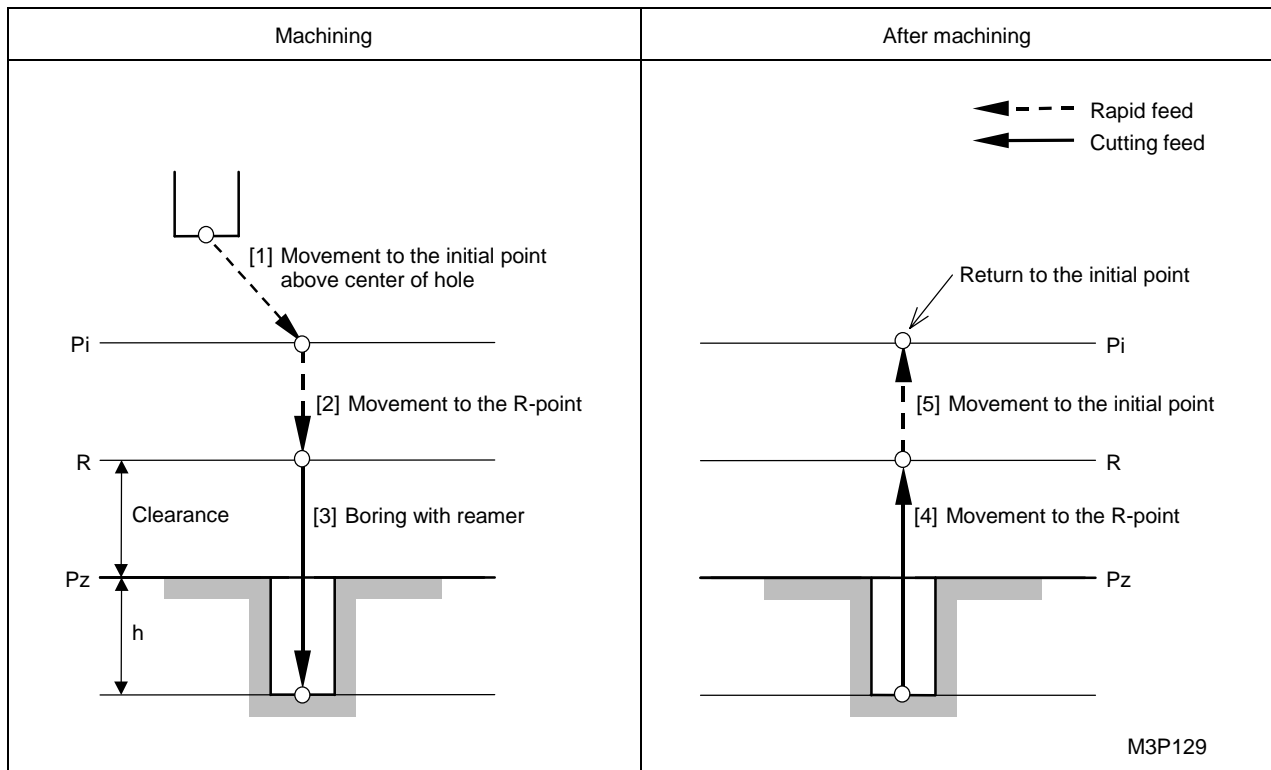
hs: Distance equal to the sum of the depth of pre-hole entered in the tool sequence and the tool data LENG COMP. (tool correction) on the **TOOL DATA** display

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

Note 2: Feed speed on the tool path [3] and [6] 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



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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.

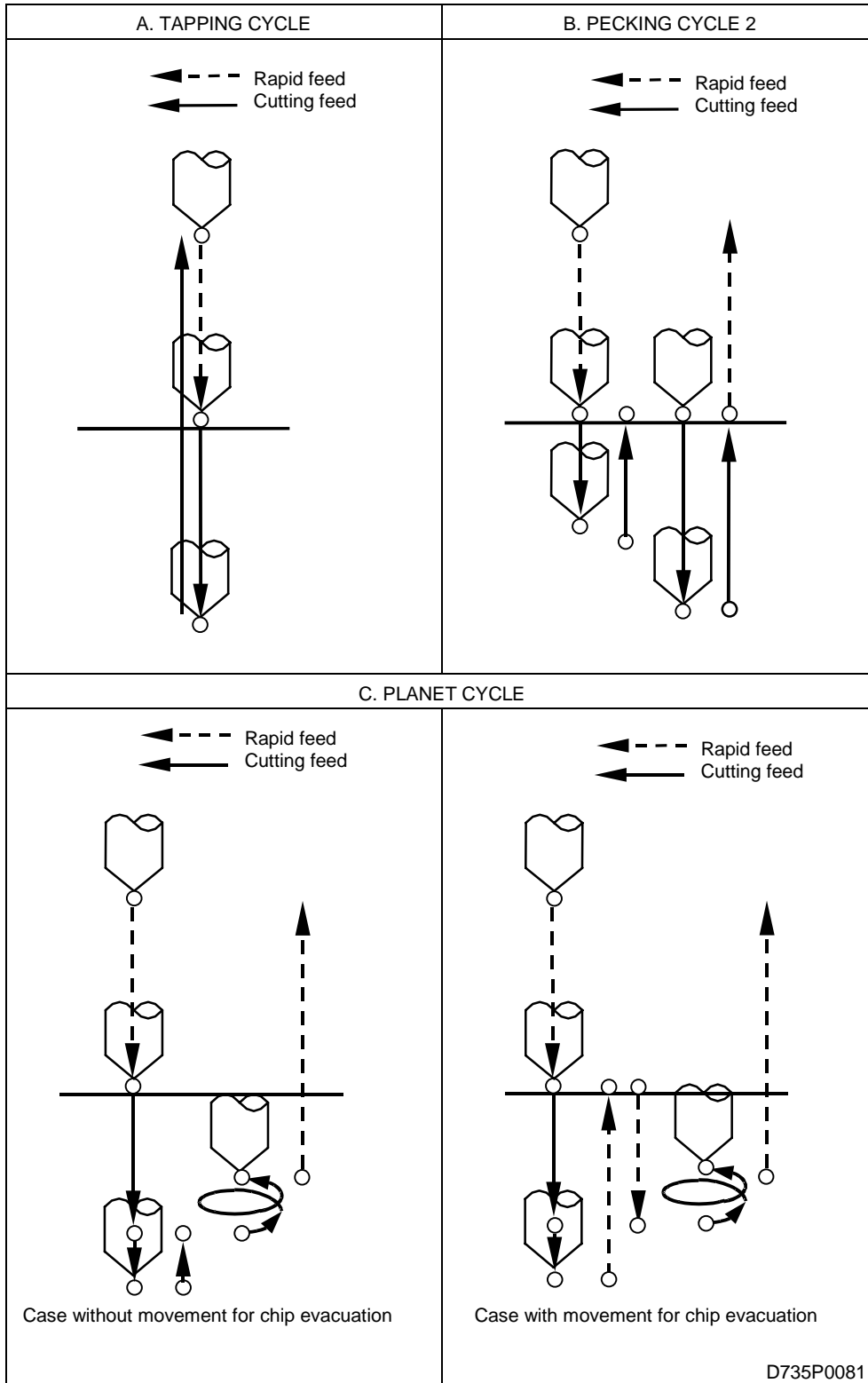
When menu key **CUT G01** is pressed Parameter **D18**

When menu key **RAPID G00** is pressed Rapid feed

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

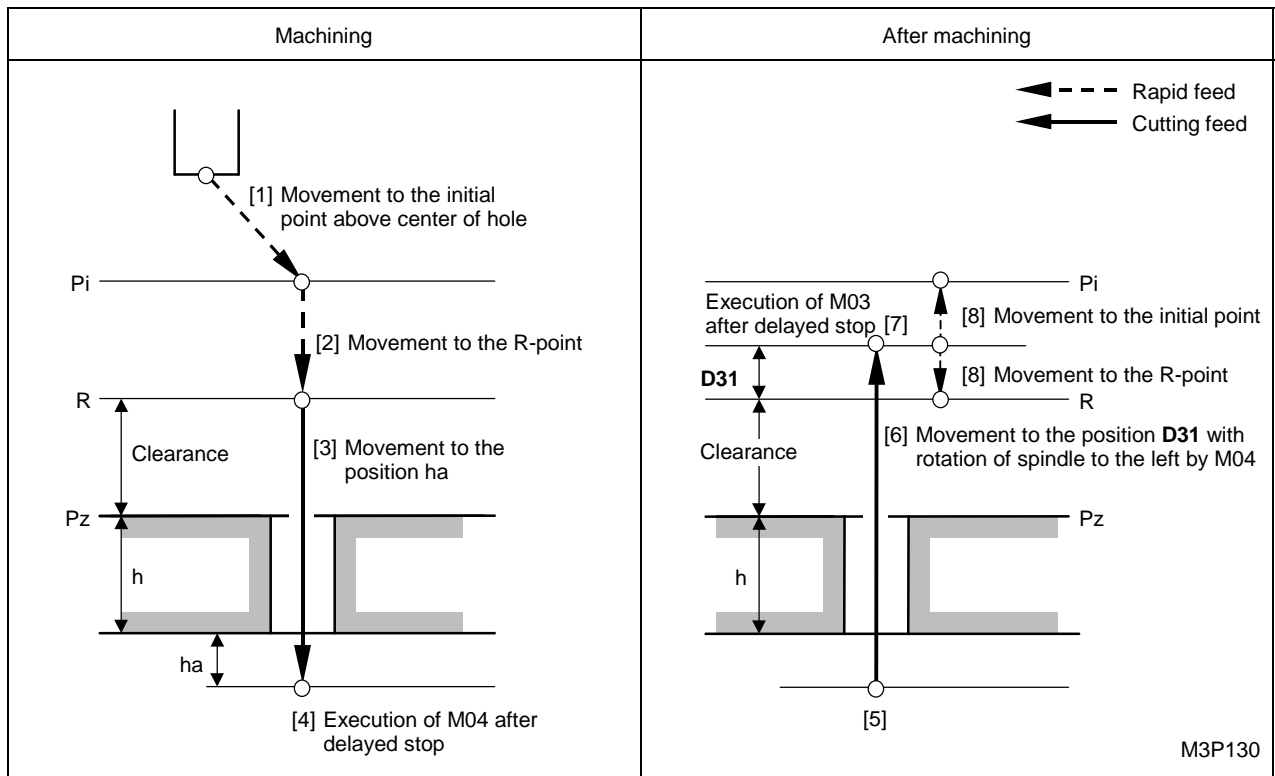
7. Tap

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



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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

$h = \text{Depth of machining hole} + \text{Tool correction}$

ha: Distance to be determined by $(A - \mathbf{D32}) \times Pt$

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]

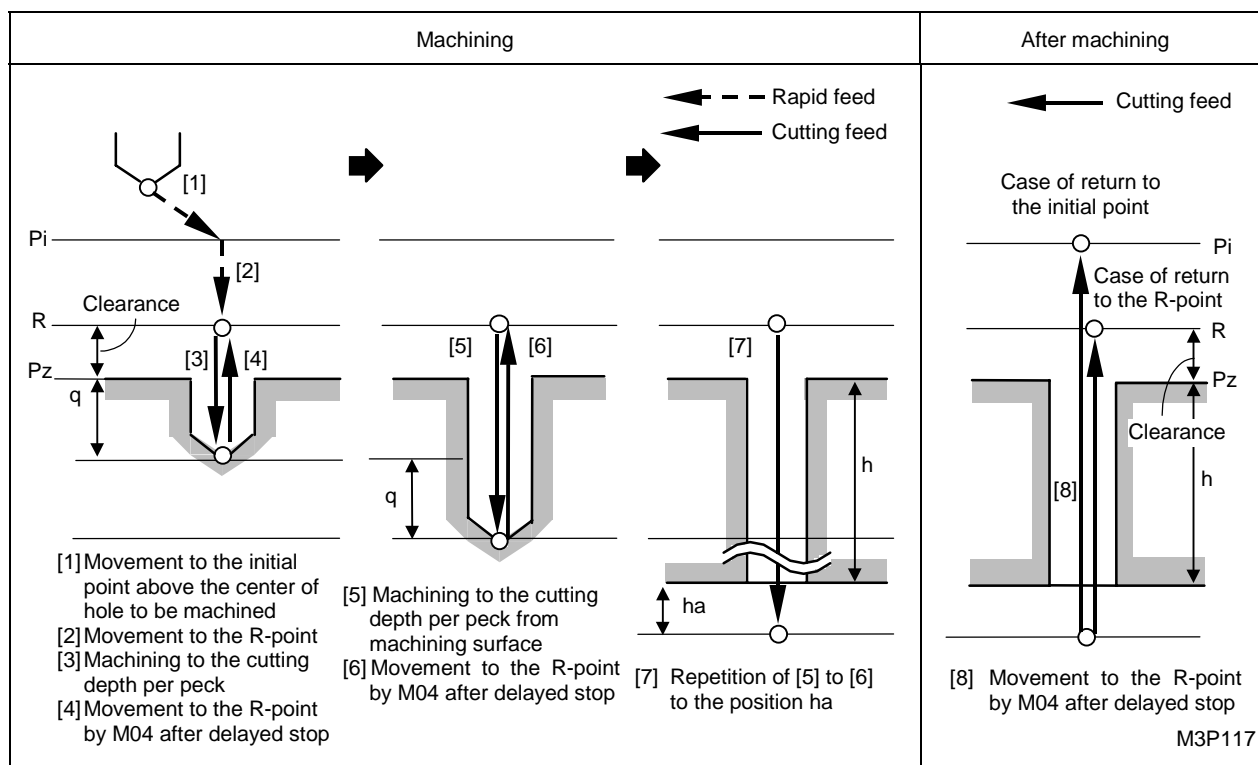
Bit 1 Delayed stop after execution of M04 at bottom of hole [5]

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)



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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 $(A - D32) \times Pt$

A **D30** when using metric and unified screws, **D43** when using pipe screws

Pt Pitch entered in the machining unit

q: Cutting 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**.

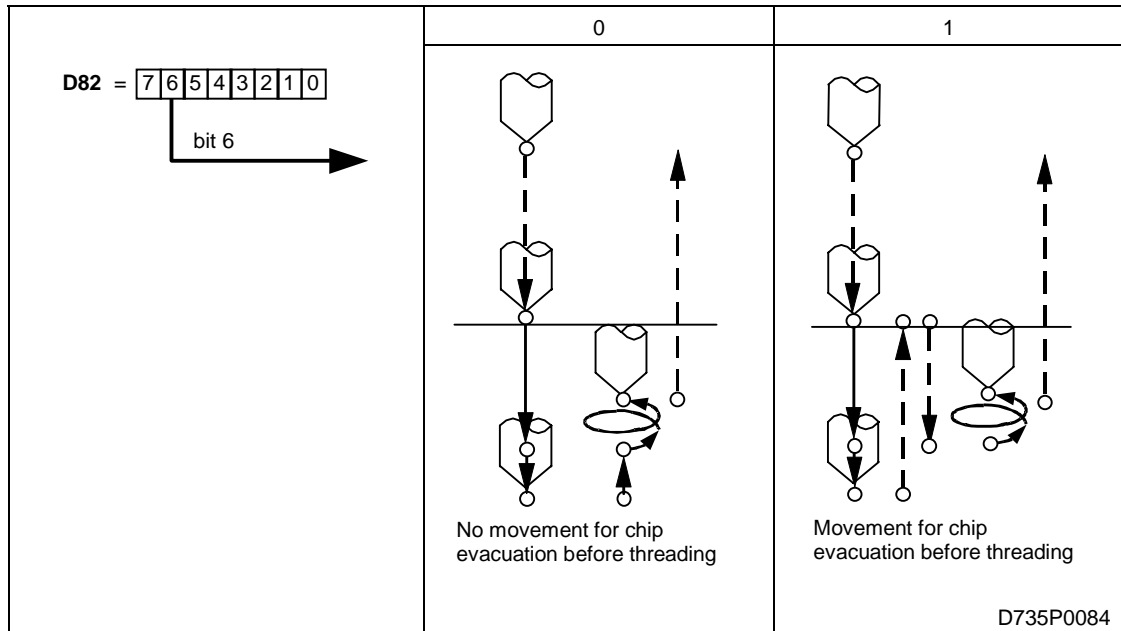
Note 2: When M04 is entered in the tool sequence, inversed tapping will be executed.

C. Planetary tapping (PLANET CYCLE)

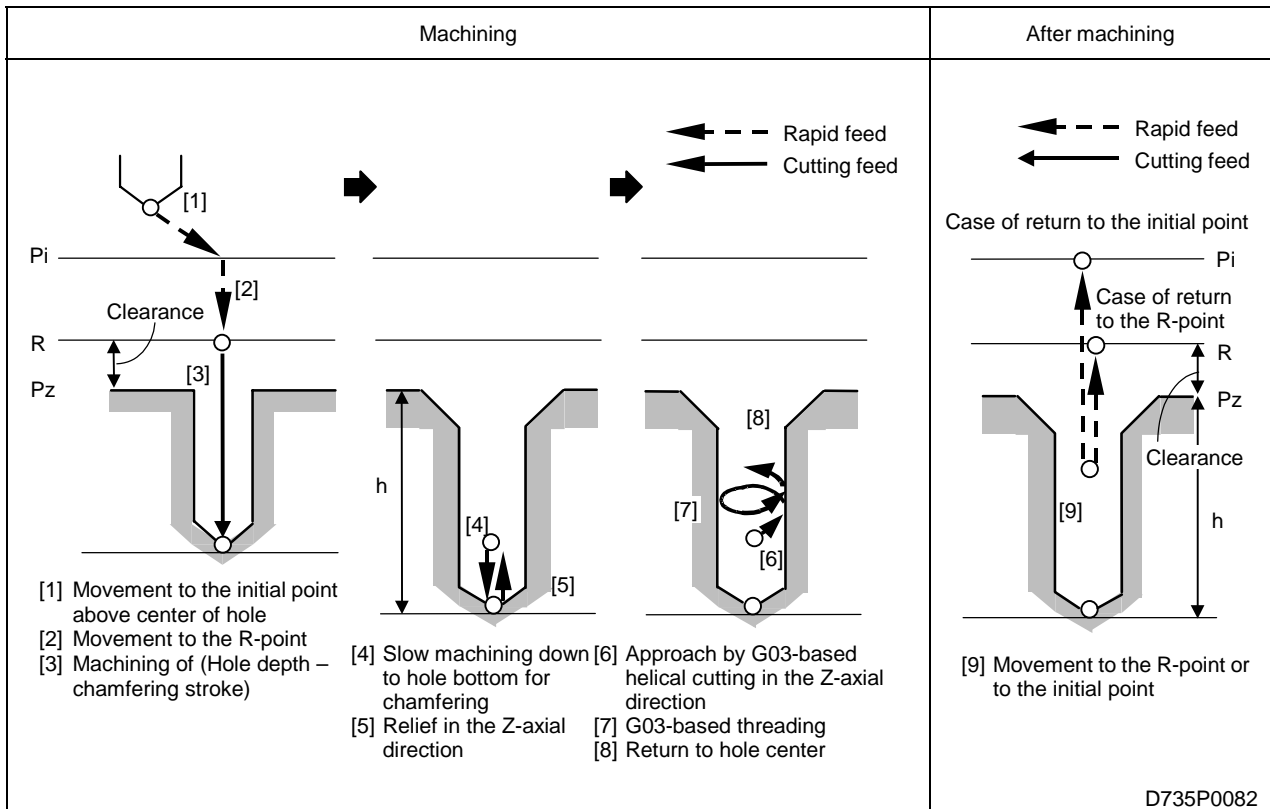
The planetary tapping cycle allows three types of machining (pre-hole machining, chamfering, and female threading) with one tool.

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

- Set either 0 or 1 in the bit 6.



1. Case without movement for chip evacuation



The bold codes represent the parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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:

$$\text{Chamfering feed} = \text{Pre-hole machining feed} \times \text{Chamfering feed override (parameter D48)}/100$$

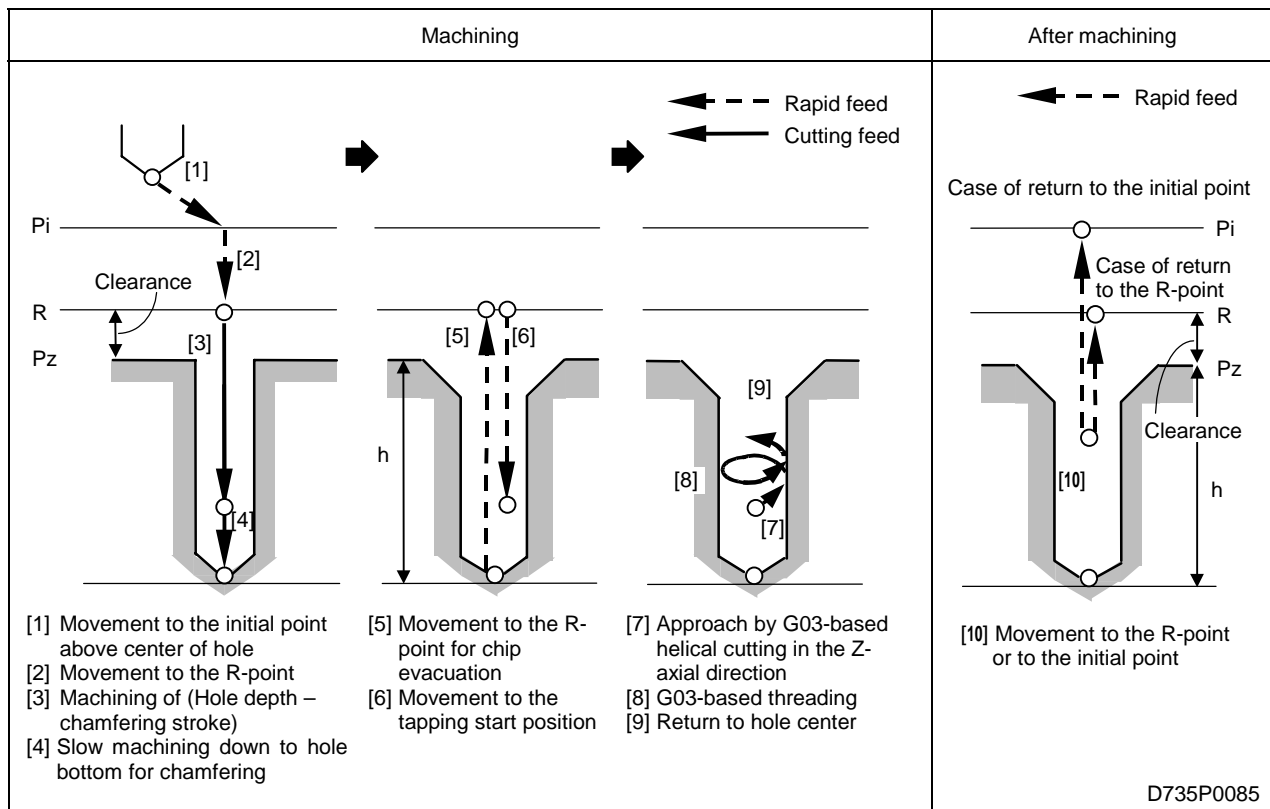
Note 3: The amount of return from hole bottom on tool path [5] is calculated as follows:

$$\text{Amount of return} = \text{Tapping pitch} \times \text{Number of thread (parameter D49)}/10$$

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.

2. Case with movement for chip evacuation



The bold codes represent the parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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.

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.

	Run-off on Z-axis Yes/No.	Delayed stop Yes/No.	Cycle		
			1	2	3
Roughness 0	No	No			
Roughness 1	No	Yes			
Roughness 2 - 9	Yes	Yes			

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.

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.

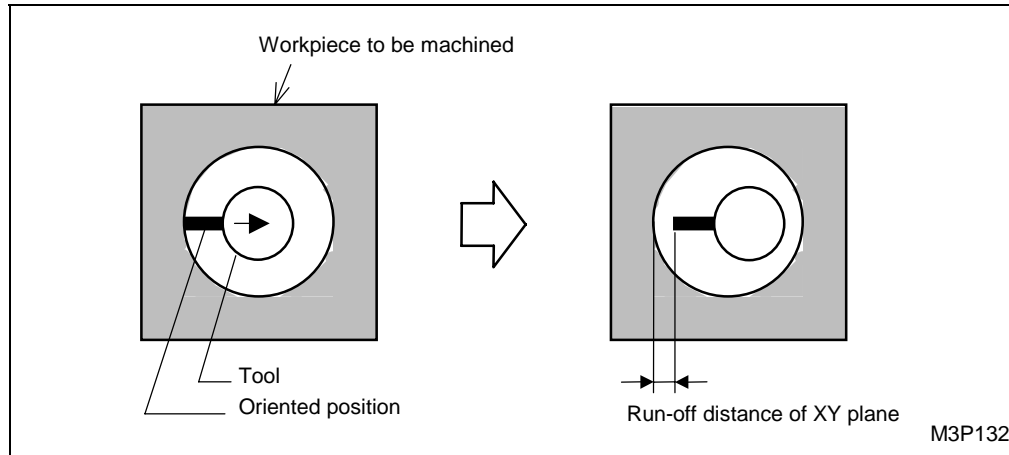


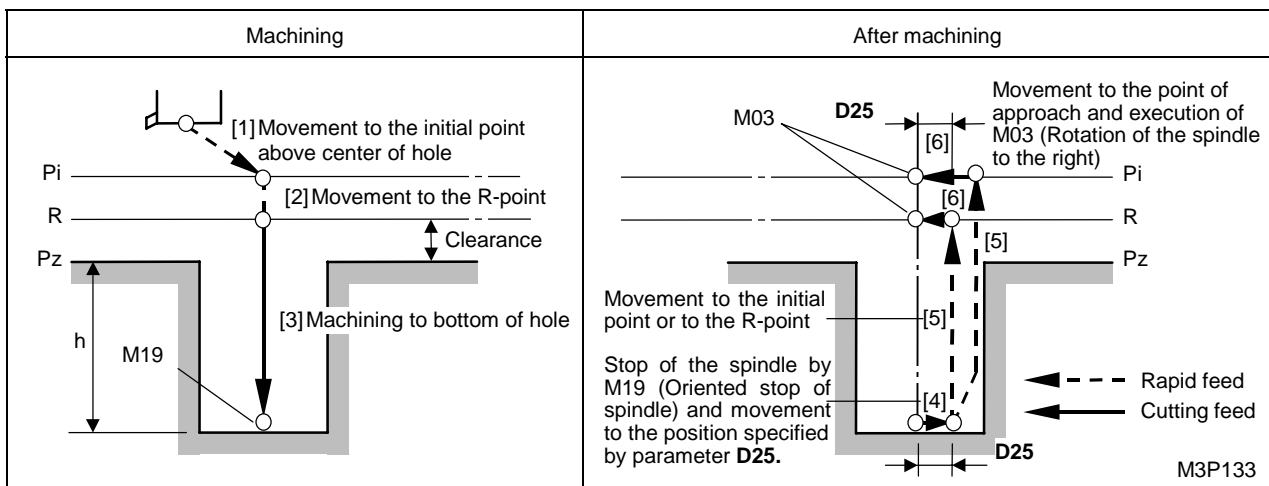
Fig. 4-14 Run-off distance on the X-Y plane

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.

A. Cycle 1 with roughness 0



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of machining surface to be entered in the shape sequence

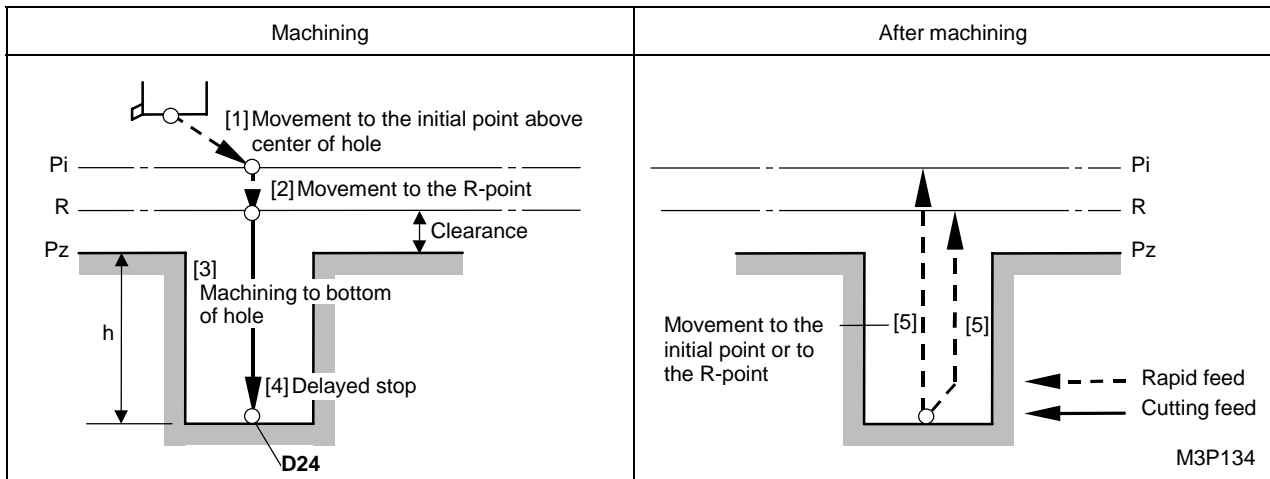
R: Safety clearance on Z-axis Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

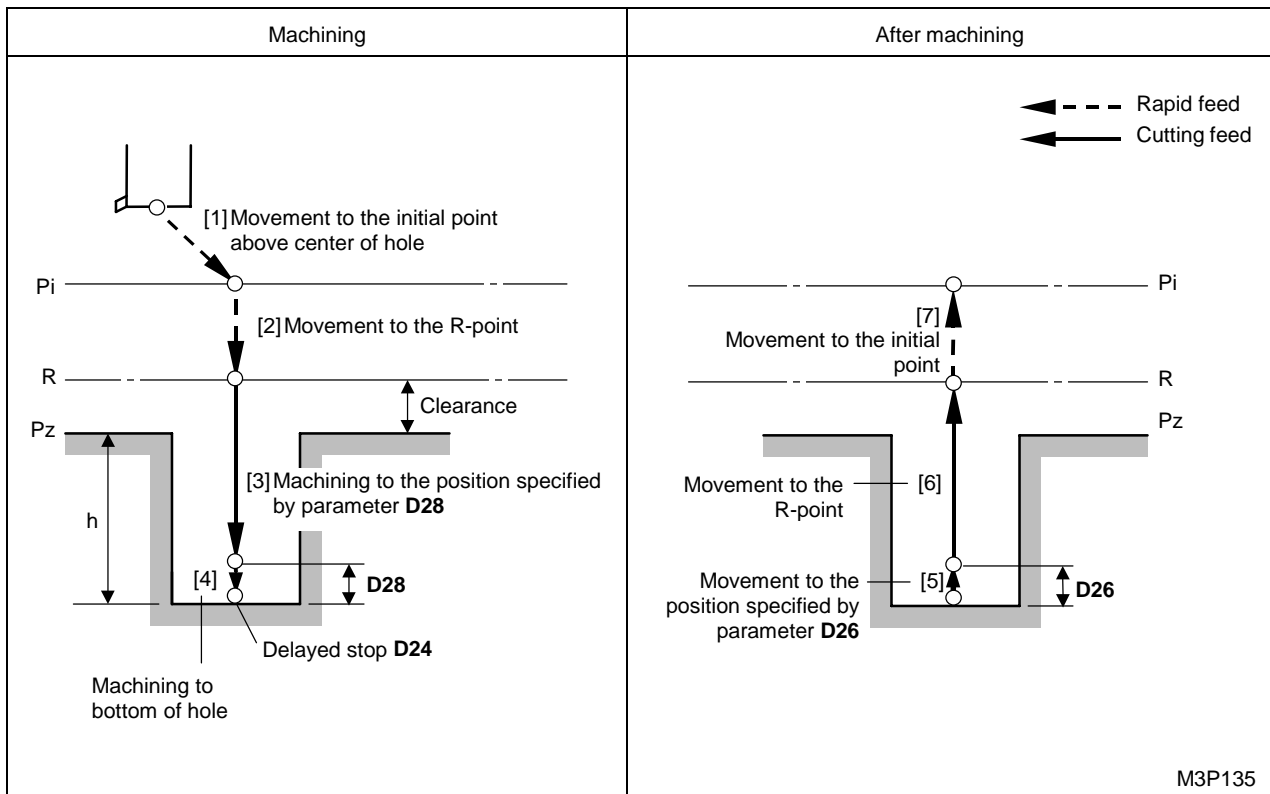
Pz: Z coordinate of machining surface to be entered in the shape sequence

R: Safety clearance on Z-axis (See “Positions of the initial point and reference point (R)” in Section 4-6.)

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 feed on Z-axis 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of machining surface to be entered in the shape sequence

R: Safety clearance on Z-axis (See "Positions of the initial point and reference point (R)" in Section 4-6.)

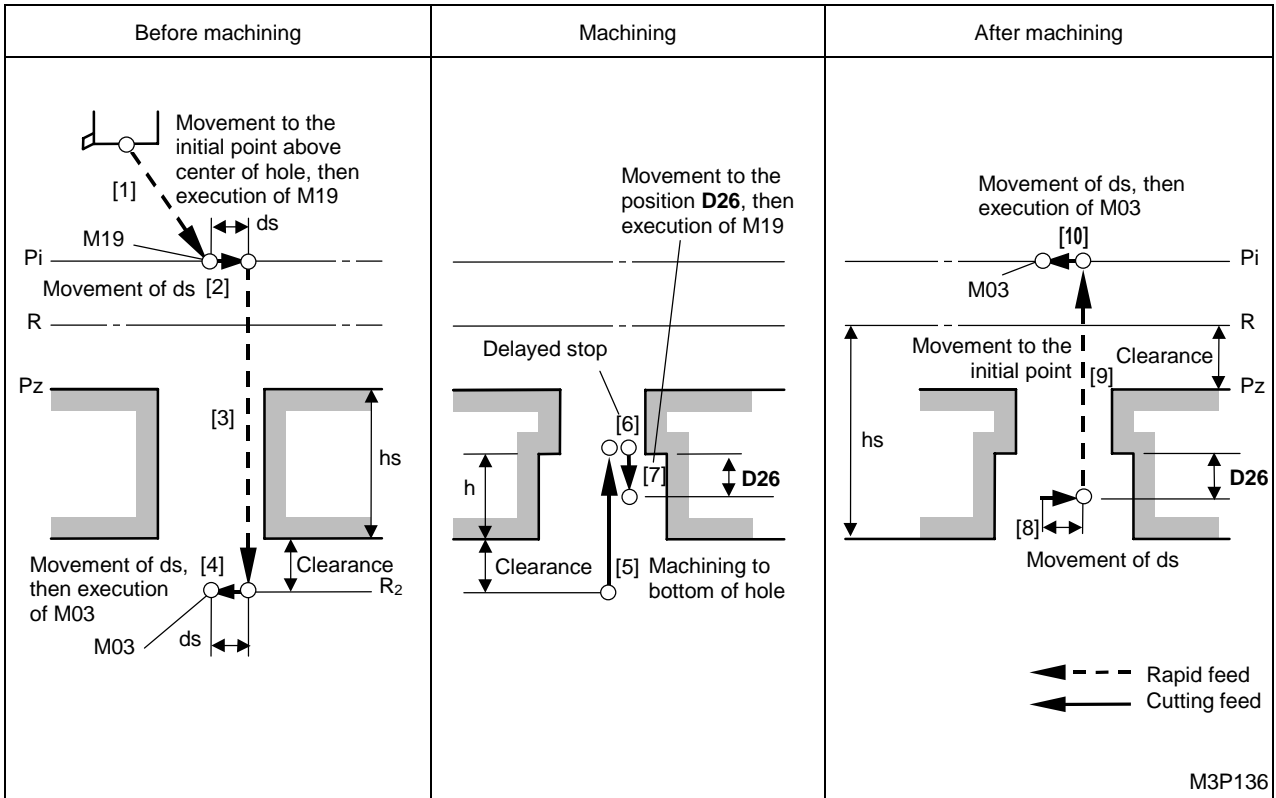
h: Distance equal to the sum of the depth of hole (HOLE-DEP) entered in the tool sequence and the data LENG COMP. (tool correction) in the **TOOL DATA** display

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 feed on Z-axis at bottom of hole is set by the parameter **D24**.

9. Back boring tool



The bold codes represent the parameter addresses.

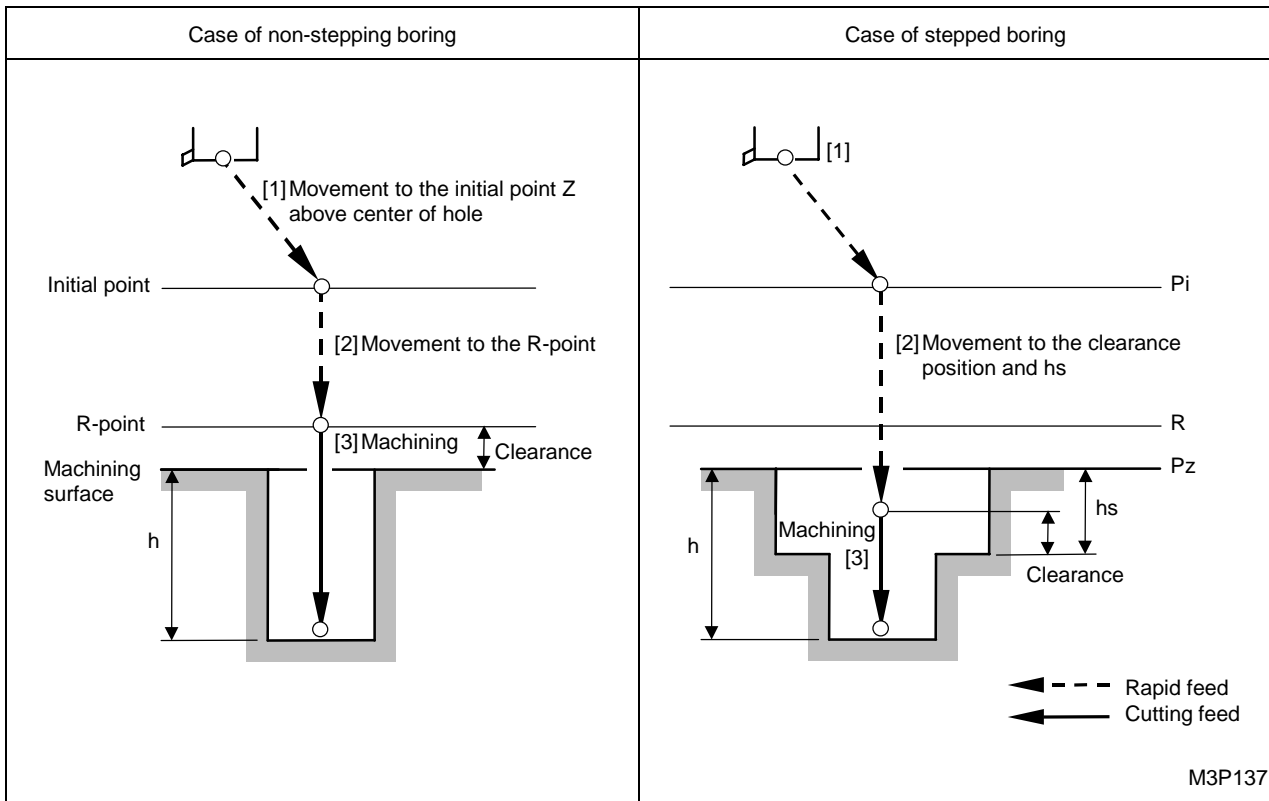
- Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- Pz: Z coordinate of machining surface to be entered in the shape sequence
- R, R₂: Safety clearance on Z-axis (See “Positions of the initial point and reference point (R)” in Section 4-6.)
- h: Distance 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 $\frac{d_1 + d_2}{2} + \mathbf{D33}$
 - d₁ Diameter of hole (HOLE-φ) entered in the tool sequence
 - d₂ 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 feed on Z-axis is set by the parameter **D40**.

General precautions concerning the path of the boring tool

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



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

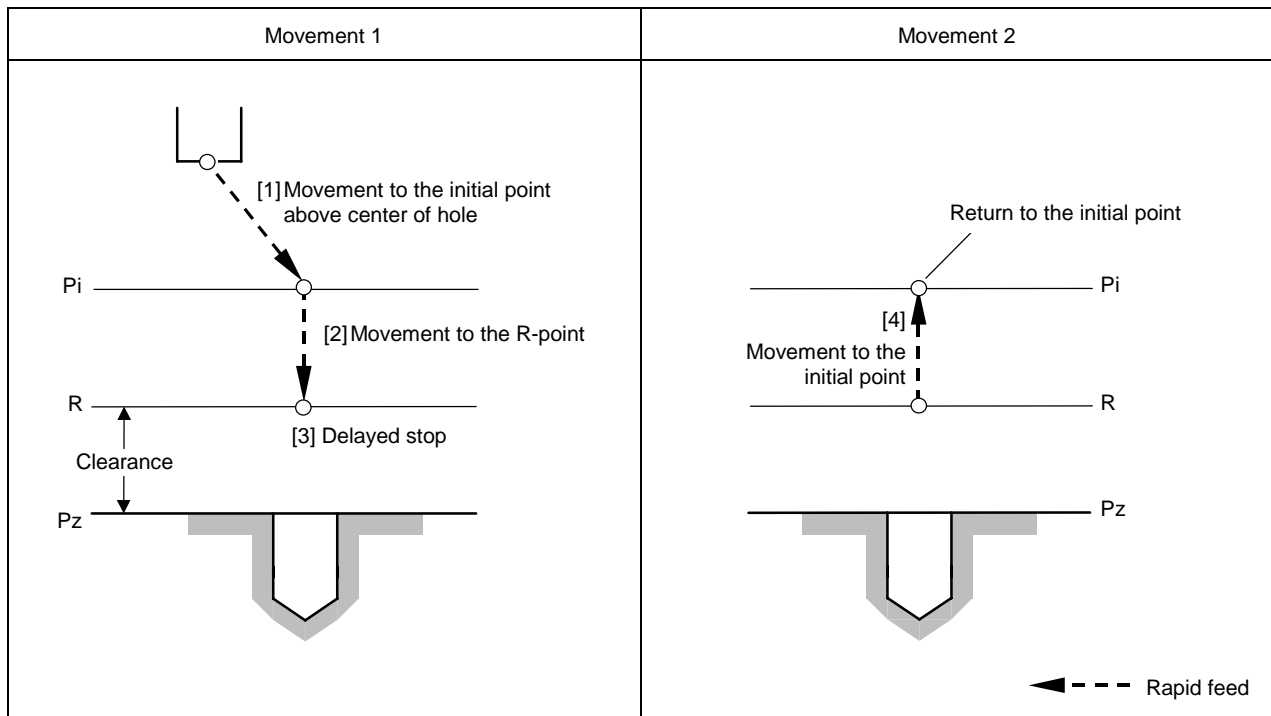
R: Safety clearance on Z-axis (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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 h_s (depth of pre-hole).

10. Chip vacuuming tool (option)



The bold codes represent the parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pz: Z coordinate of the machining surface to be entered in the shape sequence

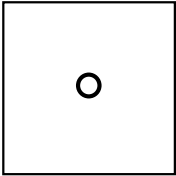
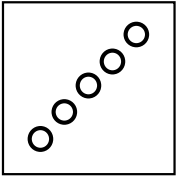
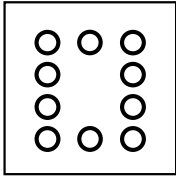
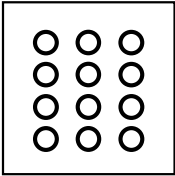
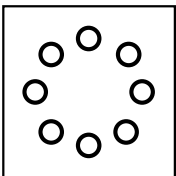
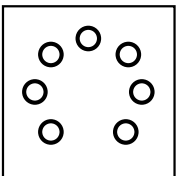
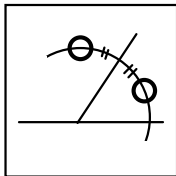
R: Safety clearance above the point Pz (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Note: The delayed stop time of the feed on Z-axis is set by the parameter **D29**.

4-7-8 Shape sequence

Once the entering of the data of the machining unit and of the tool sequence is completed, the shape sequence is entered.

1. Types of point machining shape

A. POINT	B. LINE	C. SQUARE	D. GRID
			
E. CIRCLE	F. ARC	G. CHORD	
			M3P139

2. Common data (Z and R)

The shape sequence of the point machining has the following common data: Z and R.

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1		(a)												(b)

(a) Z (Z coordinate of the machining surface)

Coordinate Z of the machining surface is understood to mean the distance on the Z-axis from the workpiece zero-point to the machining surface. The relation between the machining surface and the item Z is as shown below.

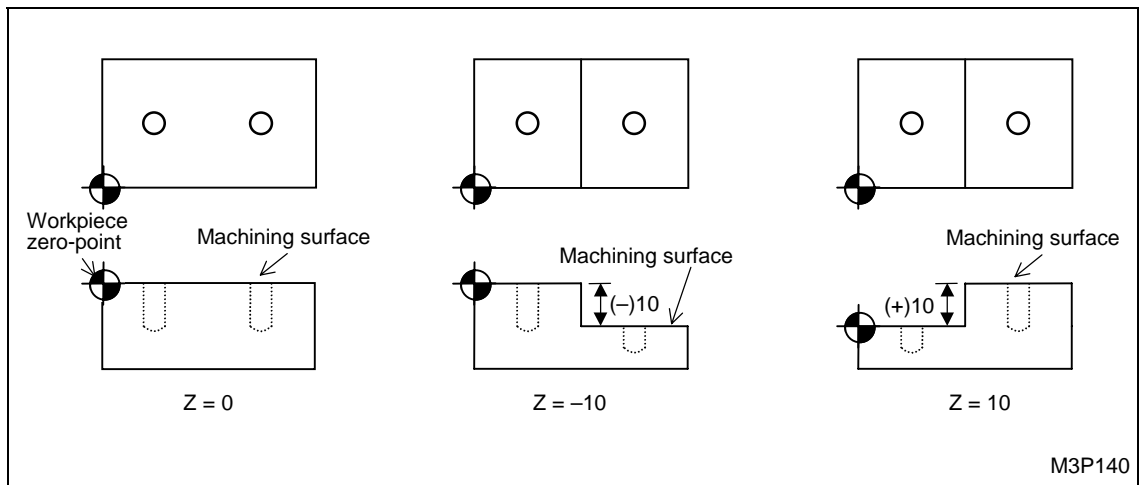
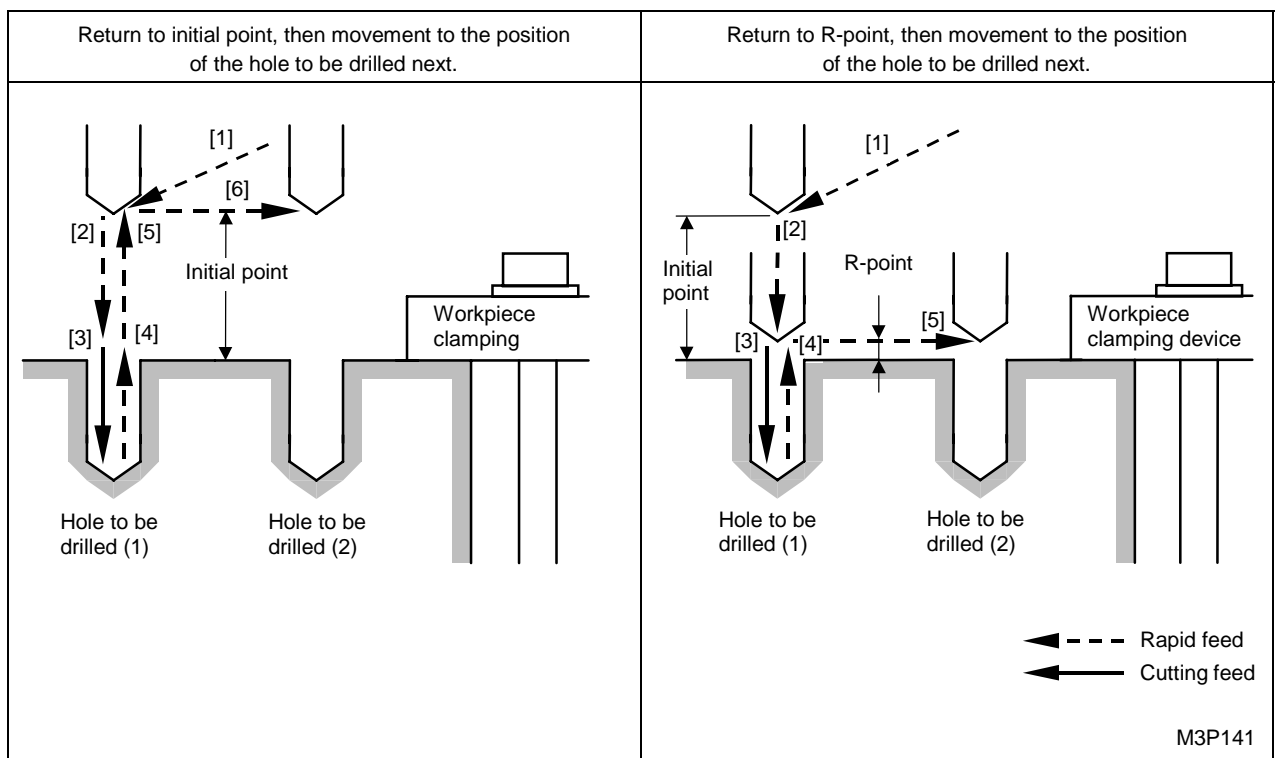


Fig. 4-15 Coordinate Z of the machining surface

(b) R (return level)

After machining, two types of return are possible.



3. Entry of shape sequence data

The following explains the data entry method of the point machining shape sequence for each type of shape, taking the drawing below as an example.

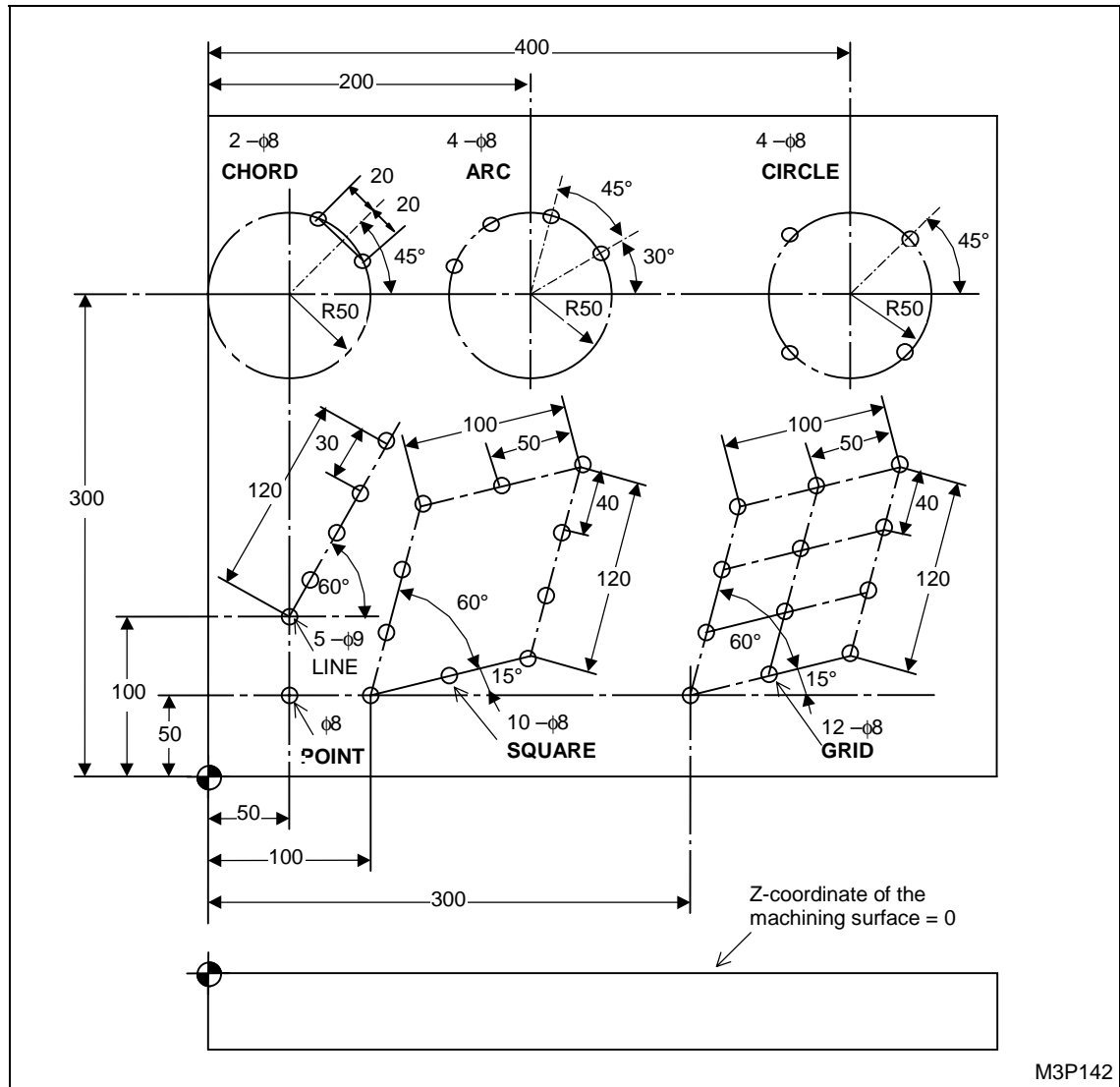
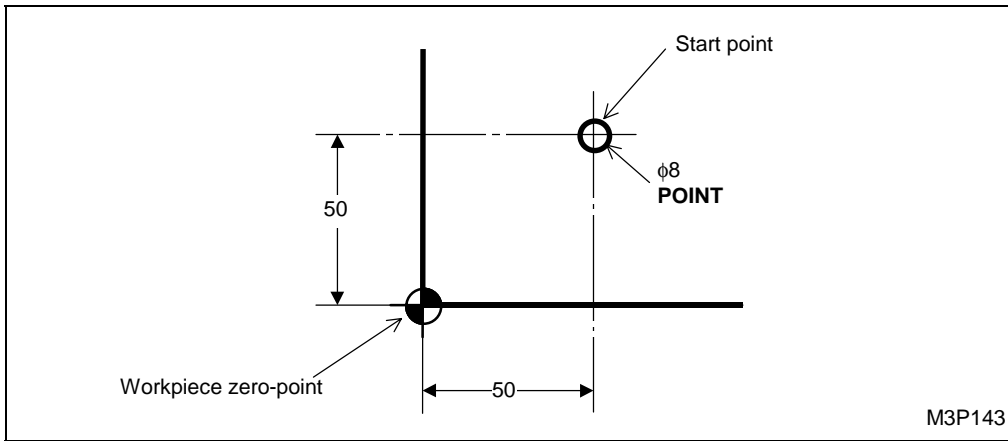


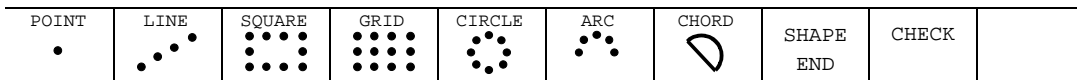
Fig. 4-16 Drawing for entry of shape sequence data

A. POINT



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.



Press the menu key **POINT**.

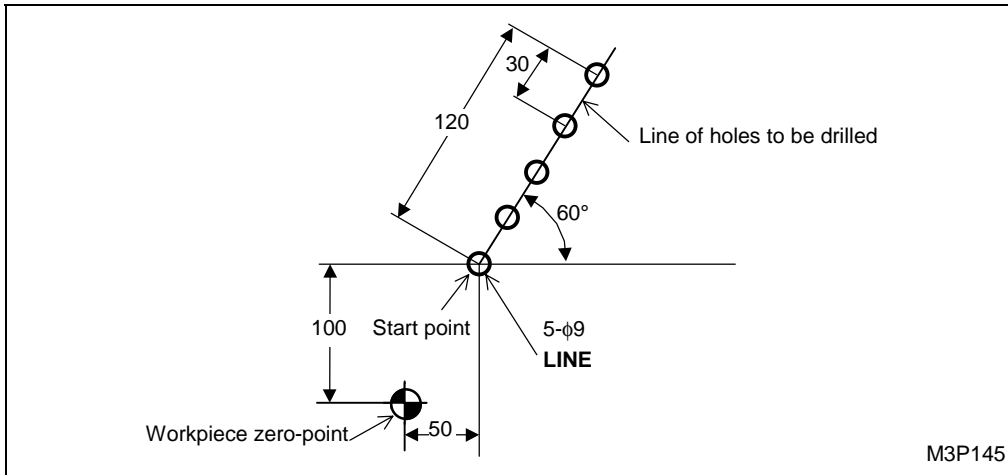
(b) Data setting

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0.	50.	50.	◆	◆	◆	◆	◆	◆	◆	0	0	0

◆: Data are not necessary to be set here.

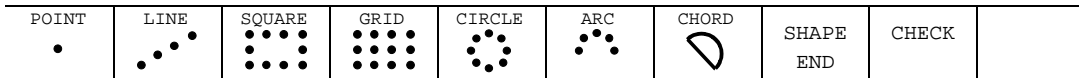
Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the first hole to be drilled. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the first hole to be drilled. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
P	<p>Specify the tool path.</p> <p>Simultaneous movement on X- and Y-axes <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>Movement on Y-axis, then on X-axis <input type="text" value="1"/> <input type="button" value="↔"/></p> <p>Movement on X-axis, then on Y-axis <input type="text" value="2"/> <input type="button" value="↔"/></p> <p>M3P144</p>
Q	<p>Specify if the machining at the start point is executed or not.</p> <p>Actual execution of machining <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>Only positioning without machining <input type="text" value="1"/> <input type="button" value="↔"/></p>
R	<p>Specify the position to which the tool returns after machining.</p> <p>Initial point <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>R-point <input type="text" value="1"/> <input type="button" value="↔"/></p>

B. LINE



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.



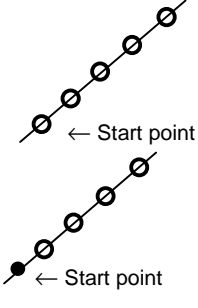
Press the menu key **LINE**.

(b) Data setting

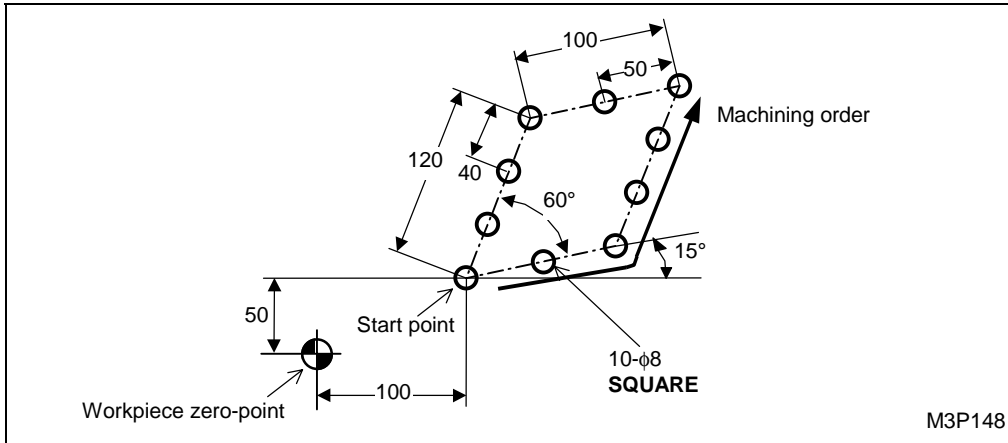
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
2	LIN	0.	50.	100.	60.	◆	30.	◆	0	5	◆	◆	0	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the first hole to be drilled. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the first hole to be drilled. <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
AN1	<p>Specify the angle θ_1 formed by the line of holes to be drilled and the X-axis.</p> <p>There are two types of θ_1</p> <p>CCW direction: $\theta_1 = 60^\circ$ (to the left) (+) <input type="text" value="6"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>CW direction: $\theta_1' = -300^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>M3P146</p>
T1	<p>Specify the spacing between holes to be drilled or the total distance between the first hole to be drilled and the last hole.</p> <p>ℓ_1: Spacing between holes <input type="text" value="3"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>ℓ_2: Total distance between the first hole to be drilled and last hole <input type="text" value="1"/> <input type="text" value="2"/> <input type="text" value="0"/> <input type="button" value="↔"/></p>

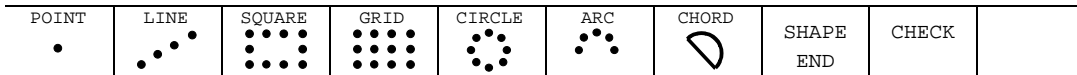
Cursor position	Description
F	Specify whether the data entered in T1, concern the spacing or the total distance. Spacing <input type="text" value="0"/> <input type="button" value="↔"/> Total distance <input type="text" value="1"/> <input type="button" value="↔"/>
M	Specify the number of holes to be drilled <input type="text" value="5"/> <input type="button" value="↔"/>
O	Specify whether the machining at the start point is executed or not.  <p style="text-align: right;">Actual execution of the machining <input type="text" value="0"/> <input type="button" value="↔"/></p> <p style="text-align: right;">Only positioning without machining <input type="text" value="1"/> <input type="button" value="↔"/></p> <p style="text-align: center;">M3P147</p>
R	Specify the position to which the tool returns after machining. Initial point <input type="text" value="0"/> <input type="button" value="↔"/> R-point..... <input type="text" value="1"/> <input type="button" value="↔"/>

C. SQUARE



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

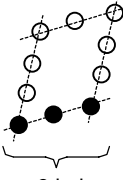
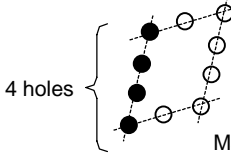
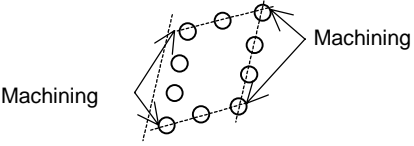
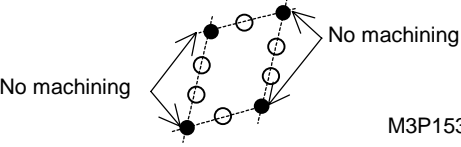


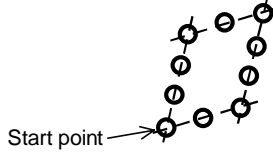
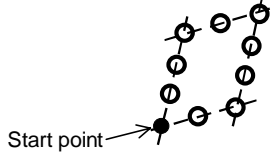
Press the menu key **SQUARE**.

(b) Data setting

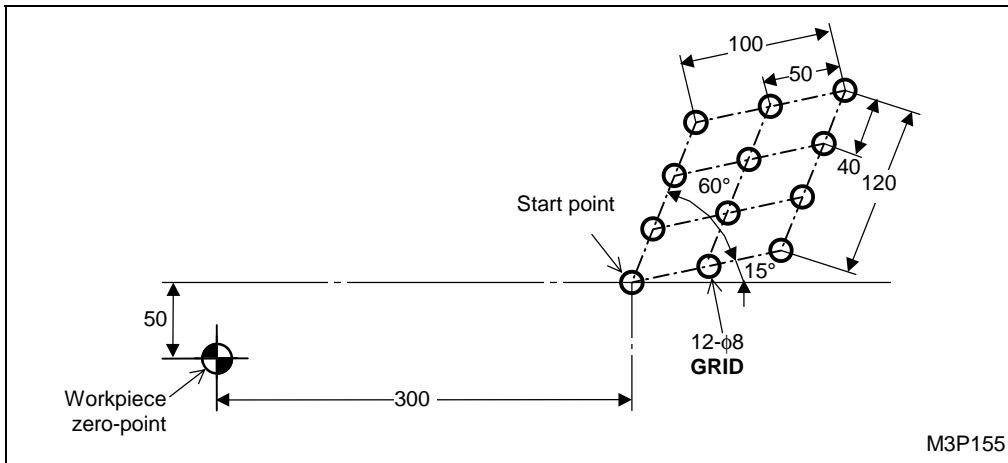
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
3	SQR	0.	100.	50.	15.	60.	50.	40.	0	3	4	0	0	0

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the first hole to be drilled. <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the first hole to be drilled. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
AN1	<p>Specify the angle θ'_1 formed by the line of holes to be drilled firstly and the X-axis.</p> <p>There are two types of θ_1</p> <p>CCW direction: $\theta_1 = 15^\circ$ (to the left) (+) <input type="text" value="1"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>CW direction: $\theta'_1 = -345^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="4"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>M3P149</p>
AN2	<p>Specify the angle θ'_2 formed by two lines of holes to be drilled.</p> <p>There are two types of θ_2</p> <p>CCW direction: $\theta_2 = 60^\circ$ (to the left) (+) <input type="text" value="6"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>CW direction: $\theta'_2 = -300^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>M3P150</p>

Cursor position	Description
T1	Specify the spacing between holes or the total length of the lines of holes to be drilled firstly l_1 : Spacing between holes of the line of holes to be drilled firstly..... <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/> l_2 : Total length of the line of holes to be drilled firstly..... <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/> (Designate l_1 or l_2 .)
T2	Specify the spacing between holes or the total length of the line of holes to be drilled lastly. t_1 : Spacing between holes of the line of holes to be drilled lastly..... <input type="text" value="4"/> <input type="text" value="0"/> <input type="button" value="↔"/> t_2 : Total length of the line of holes to be drilled lastly..... <input type="text" value="1"/> <input type="text" value="2"/> <input type="text" value="0"/> <input type="button" value="↔"/> (Designate the spacing if it is specified in T1 or the total length if it is specified therein.)
F	Specify whether the data entered in T1 and T2 concern the spacing or the total length. Spacing <input type="text" value="0"/> <input type="button" value="↔"/> Total length..... <input type="text" value="1"/> <input type="button" value="↔"/>
M	Specify the number of holes on the line of holes to be drilled firstly..... <input type="text" value="3"/> <input type="button" value="↔"/>  <p style="text-align: center;">3 holes M3P151</p>
N	Specify the number of holes on the line of holes to be drilled lastly..... <input type="text" value="4"/> <input type="button" value="↔"/>  <p style="text-align: center;">4 holes M3P152</p>
P	Specify if the machining at the four corners is executed or not. Machining at the four corners..... <input type="text" value="0"/> <input type="button" value="↔"/>  <p style="text-align: center;">Machining</p> No machining at the four corners <input type="text" value="1"/> <input type="button" value="↔"/>  <p style="text-align: center;">No machining M3P153</p>

Cursor position	Description
Q	<p>Specify if the machining at the start point is executed or not.</p> <p>Actual execution of the machining..... <input type="text" value="0"/> <input type="button" value="↔"/></p>  <p>Start point →</p> <p>Only positioning without machining..... <input type="text" value="1"/> <input type="button" value="↔"/></p>  <p>Start point →</p> <p style="text-align: right;">M3P154</p>
R	<p>Specify the position to which the tool returns after the machining.</p> <p>Initial point <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>R-point..... <input type="text" value="1"/> <input type="button" value="↔"/></p>

D. GRID



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

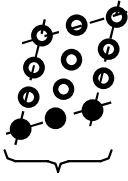
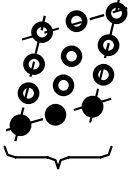
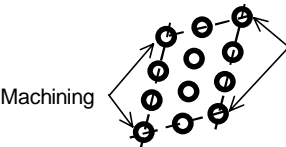
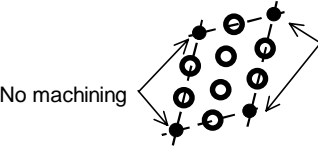
POINT •	LINE •••	SQUARE ••••	GRID ••••	CIRCLE ••••	ARC •••	CHORD ◌	SHAPE END	CHECK
------------	-------------	----------------	--------------	----------------	------------	------------	--------------	-------

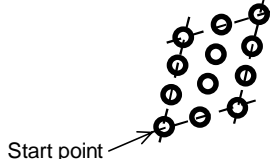
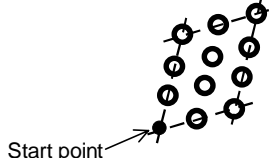
Press the menu key **GRID**.

(b) Data setting

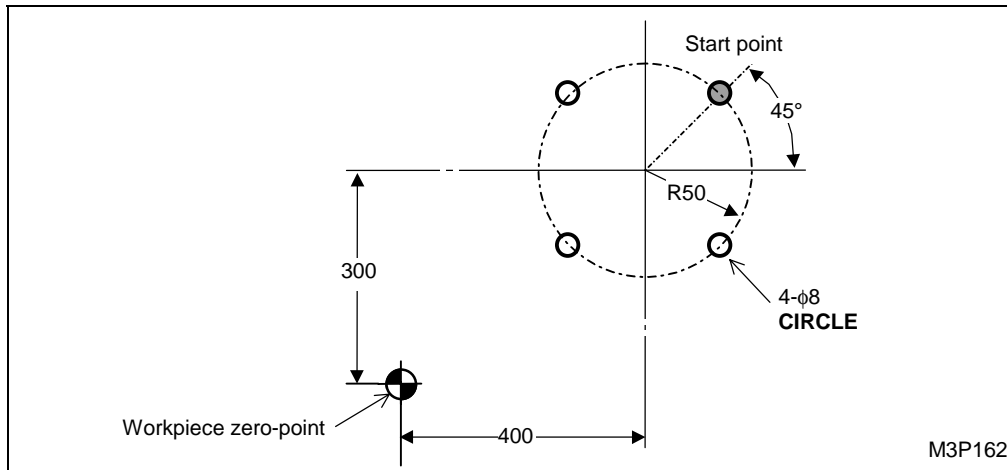
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
4	GRD	0.	100.	50.	15.	60.	50.	40.	0	3	4	0	0	0

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the first hole to be drilled. <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the first hole to be drilled. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
AN1	<p>Specify the angle θ_1 formed by the line of holes to be drilled firstly and the X-axis.</p> <p>There are two types of θ_1 CCW direction: $\theta_1 = 15^\circ$ (to the left) (+) <input type="text" value="1"/> <input type="text" value="5"/> <input type="button" value="↔"/> CW direction: $\theta_1 = -345^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="4"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>M3P156</p>
AN2	<p>Specify the angle θ_2 formed by two lines of holes to be drilled.</p> <p>There are two types of θ_2 CCW direction: $\theta_2 = 60^\circ$ (to the left) (+) <input type="text" value="6"/> <input type="text" value="0"/> <input type="button" value="↔"/> CW direction: $\theta_2 = -300^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>M3P157</p>

Cursor position	Description
T1	Specify the spacing between holes or the total length of the lines of holes to be drilled firstly. l_1 : Spacing between holes of the line of holes to be drilled firstly..... <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/> l_2 : Total length of the line of holes to be drilled firstly..... <input type="text" value="1"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/> (Designate l_1 or l_2 .)
T2	Specify the spacing between holes or the total length of the line of holes to be drilled lastly. t_1 : Spacing between holes of the line of holes to be drilled lastly..... <input type="text" value="4"/> <input type="text" value="0"/> <input type="button" value="↔"/> t_2 : Total length of the line of holes to be drilled lastly..... <input type="text" value="1"/> <input type="text" value="2"/> <input type="text" value="0"/> <input type="button" value="↔"/> (Designate the spacing if it is specified in T1 or the total length if it is specified therein).
F	Specify whether the data entered in T1 and T2 concern the spacing or the total length. Spacing <input type="text" value="0"/> <input type="button" value="↔"/> Total length..... <input type="text" value="1"/> <input type="button" value="↔"/>
M	Specify the number of holes on the line of holes to be drilled firstly..... <input type="text" value="3"/> <input type="button" value="↔"/>  3 holes M3P158
N	Specify the number of holes on the line of holes to be drilled lastly. <input type="text" value="4"/> <input type="button" value="↔"/>  3 holes M3P158
P	Specify if the machining at the four corners is executed or not. Machining at the four corners..... <input type="text" value="0"/> <input type="button" value="↔"/>  Machining Machining No machining at the four corners <input type="text" value="1"/> <input type="button" value="↔"/>  No machining No machining M3P160

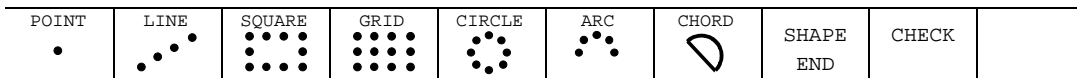
Cursor position	Description
Q	<p>Specify if the machining at the start point is executed or not.</p> <p>Actual execution of the machining..... <input type="text" value="0"/> <input type="button" value="↕"/></p>  <p>Only positioning without machining..... <input type="text" value="1"/> <input type="button" value="↕"/></p>  <p style="text-align: right;">M3P161</p>
R	<p>Specify the position to which the tool returns after the machining.</p> <p>Initial point <input type="text" value="0"/> <input type="button" value="↕"/></p> <p>R-point..... <input type="text" value="1"/> <input type="button" value="↕"/></p>

E. CIRCLE



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

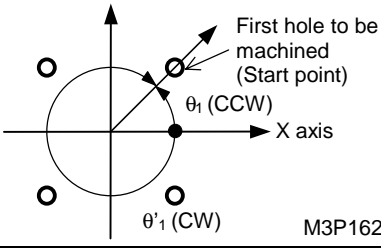


Press the menu key **CIRCLE**.

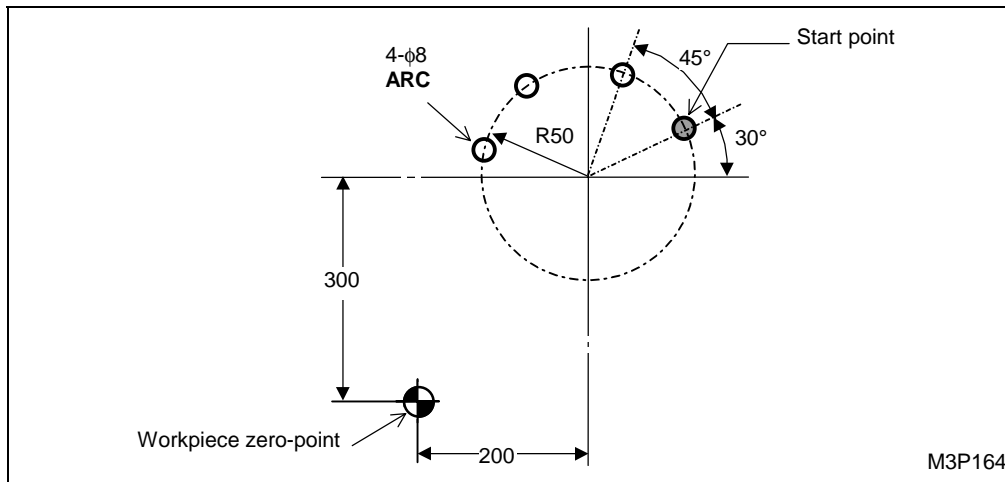
(b) Data setting

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
5	CIR	0.	400.	300.	45.	◆	50.	◆	◆	4	◆	◆	◆	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the center of the circle. <input type="text" value="4"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the center of the circle. <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
AN1	Specify the angle θ_1 formed by the start point and the X-axis.  <p>There are two types of θ_1</p> <p>CCW direction: $\theta_1 = 45^\circ$ (to the left) (+) <input type="text" value="4"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>CW direction: $\theta'_1 = -315^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="1"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>M3P162</p>
T1	Specify the radius of the circle. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
M	Specify the number of holes to be machined..... <input type="text" value="4"/> <input type="button" value="↔"/>
R	Specify the position to which the tool returns after the machining. Initial point <input type="text" value="0"/> <input type="button" value="↔"/> R-point..... <input type="text" value="1"/> <input type="button" value="↔"/>

F. ARC



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

POINT •	LINE •••	SQUARE ••••	GRID ••••	CIRCLE ••••	ARC •••	CHORD ◌	SHAPE END	CHECK
------------	-------------	----------------	--------------	----------------	------------	------------	--------------	-------

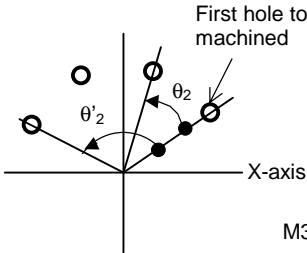
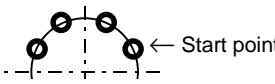
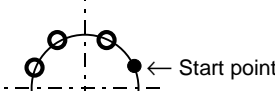
Press the menu key **ARC**.

(b) Data setting

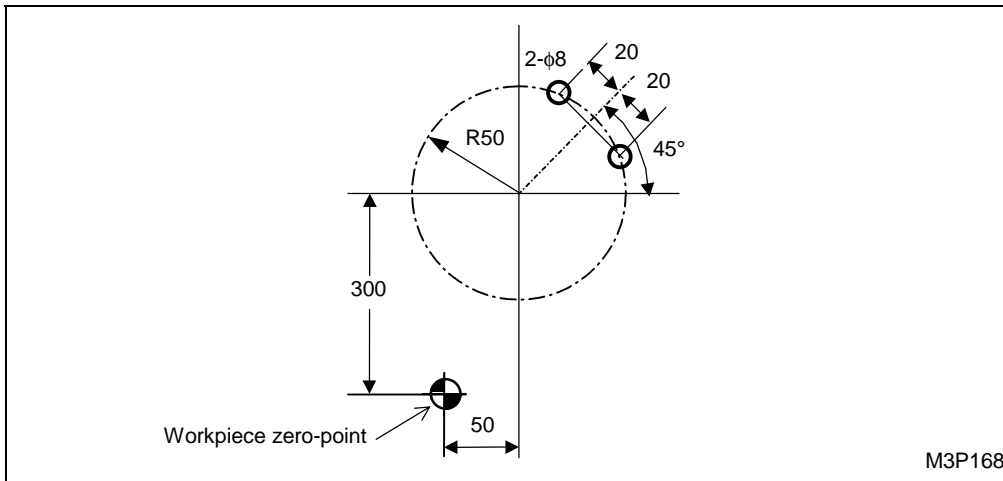
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
6	ARC	0.	200.	300.	30.	45.	50.	◆	0	4	◆	◆	0	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↕"/>
X	Specify the X coordinate of the center of the arc. <input type="text" value="2"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↕"/>
Y	Specify the Y coordinate of the center of the arc. <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↕"/>
AN1	<p>Specify the angle θ_1 formed by the start point and the X-axis.</p> <p>First hole to be machined</p> <p>θ_1 (CCW)</p> <p>θ'_1 (CW)</p> <p>M3P165</p> <p>There are two types of θ_1</p> <p>CCW direction: $\theta_1 = 30^\circ$ (to the left) (+) <input type="text" value="3"/> <input type="text" value="0"/> <input type="button" value="↕"/></p> <p>CW direction: $\theta'_1 = -330^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="3"/> <input type="text" value="0"/> <input type="button" value="↕"/></p>

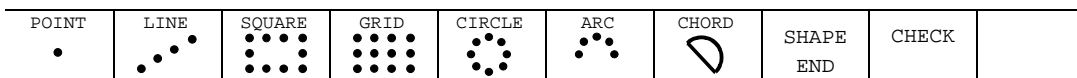
Cursor position	Description
AN2	<p>Specify the angle of pitch between two adjacent holes or the angle between the first hole and the last hole.</p>  <p>θ_2: Angle of pitch between two adjacent holes $\theta_2 = 45^\circ$ <input type="text" value="4"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>θ'_2: Total angle between the first hole and the last hole M3P166 $\theta'_2 = 135^\circ$ <input type="text" value="1"/> <input type="text" value="3"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>(Designate θ_2 or θ'_2)</p>
T1	Specify the radius of the arc <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>
F	<p>Specify whether the data entered in AN2 concern the angle of pitch or the total angle.</p> <p>Angle of pitch..... <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>Total angle..... <input type="text" value="1"/> <input type="button" value="↔"/></p>
M	Specify the number of holes to be machined..... <input type="text" value="4"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Q	<p>Specify the machining at the starting point is executed or not:</p>  <p>Actual execution of the machining..... <input type="text" value="0"/> <input type="button" value="↔"/></p>  <p>Only positioning without machining <input type="text" value="1"/> <input type="button" value="↔"/></p> <p>M3P167</p>
R	Specify the position to which tool returns after machining. <p>Initial point <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>R-point..... <input type="text" value="1"/> <input type="button" value="↔"/></p>

G. CHORD



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.






Press the menu key **CHORD**.

(b) Data setting

FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
7	CRD	0.	50.	300.	45.	◆	50.	40.	◆	◆	◆	0	◆	0

◆: Data are not necessary to be set here.

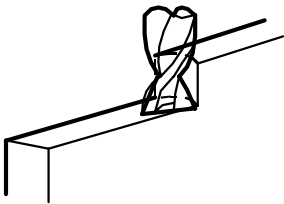
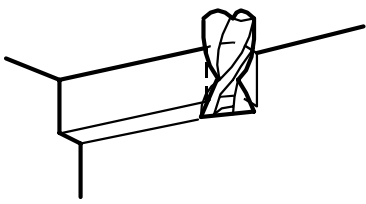
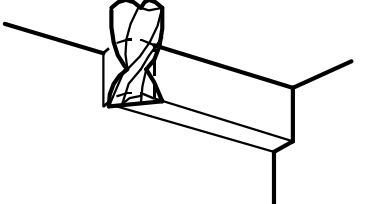
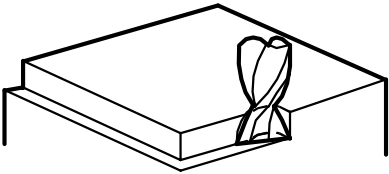
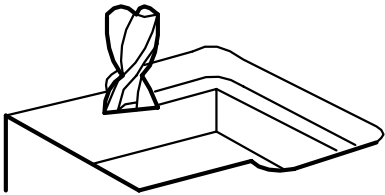
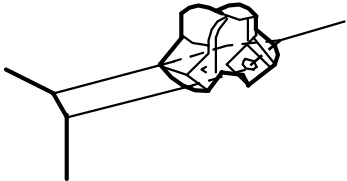
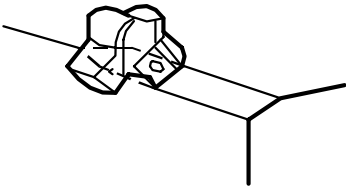
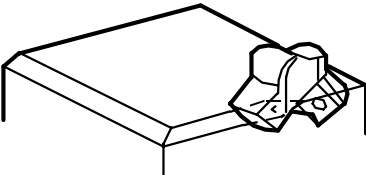
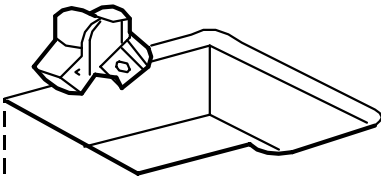
Cursor position	Description
Z	Specify the Z coordinate of the machining surface. <input type="text" value="0"/> <input type="button" value="↔"/>
X	Specify the X coordinate of the center of the circle..... <input type="text" value="5"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
Y	Specify the Y coordinate of the center of the circle. <input type="text" value="3"/> <input type="text" value="0"/> <input type="text" value="0"/> <input type="button" value="↔"/>
AN1	<p>Specify the angle θ_1 formed by the bisectrix of the chord and the X-axis.</p> <p>There are two types of θ_1 CCW direction: $\theta_1 = 45^\circ$ (to the left) (+) <input type="text" value="4"/> <input type="text" value="5"/> <input type="button" value="↔"/> CW direction: $\theta'_1 = -315^\circ$ (to the right) (-) <input type="text" value="-"/> <input type="text" value="3"/> <input type="text" value="1"/> <input type="text" value="5"/> <input type="button" value="↔"/></p> <p>M3P169</p>
T1	Specify the radius of the circle. <input type="text" value="5"/> <input type="text" value="0"/> <input type="button" value="↔"/>

Cursor position	Description
T2	<p>Specify the total length for the machining of the holes on both sides of the bisectrix or 1/2 of the total length for the machining on a single side of the bisectrix.</p> <p>Total length..... <input type="text" value="4"/> <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>1/2 of total length <input type="text" value="2"/> <input type="text" value="0"/> <input type="button" value="↔"/></p>
R	<p>Specify the position of the hole to be machined.</p> <p>Right  Left <input type="text" value="0"/> <input type="button" value="↔"/> Machining on both sides of the chord</p> <p>Right  <input type="text" value="1"/> <input type="button" value="↔"/> Machining of the right side of the chord</p> <p> Left <input type="text" value="2"/> <input type="button" value="↔"/> Machining of the left side of the chord</p> <p>M3P170</p>
R	<p>Specify the position to which to tool returns after machining.</p> <p>Initial point <input type="text" value="0"/> <input type="button" value="↔"/></p> <p>R-point..... <input type="text" value="1"/> <input type="button" value="↔"/></p>

4-8 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.

4-8-1 Types of line machining units

1. Central linear machining	2. Right-hand linear machining	3. Left-hand linear machining
		
4. Outside linear machining	5. Inside linear machining	6. Right-hand chamfering
		
7. Left-hand chamfering	8. Outside chamfering	9. Inside chamfering
		

M3P171










Fig. 4-17 Types of line machining units

4-8-2 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	LINE	FACE	TURNING	MANUAL	WPC	OFFSET	END	SHAPE	>>>
MACH-ING	MACH-ING	MACH-ING		PROGRAM				CHECK	

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

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

- (3) Press the appropriate menu key of the desired machining unit.
Enter the data in each machining unit by moving the cursor in each item.

4-8-3 Unit data, automatic tool development and tool path

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	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z							
1	LINE CTR	999.999	999.999	999.999	9	999.999							
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

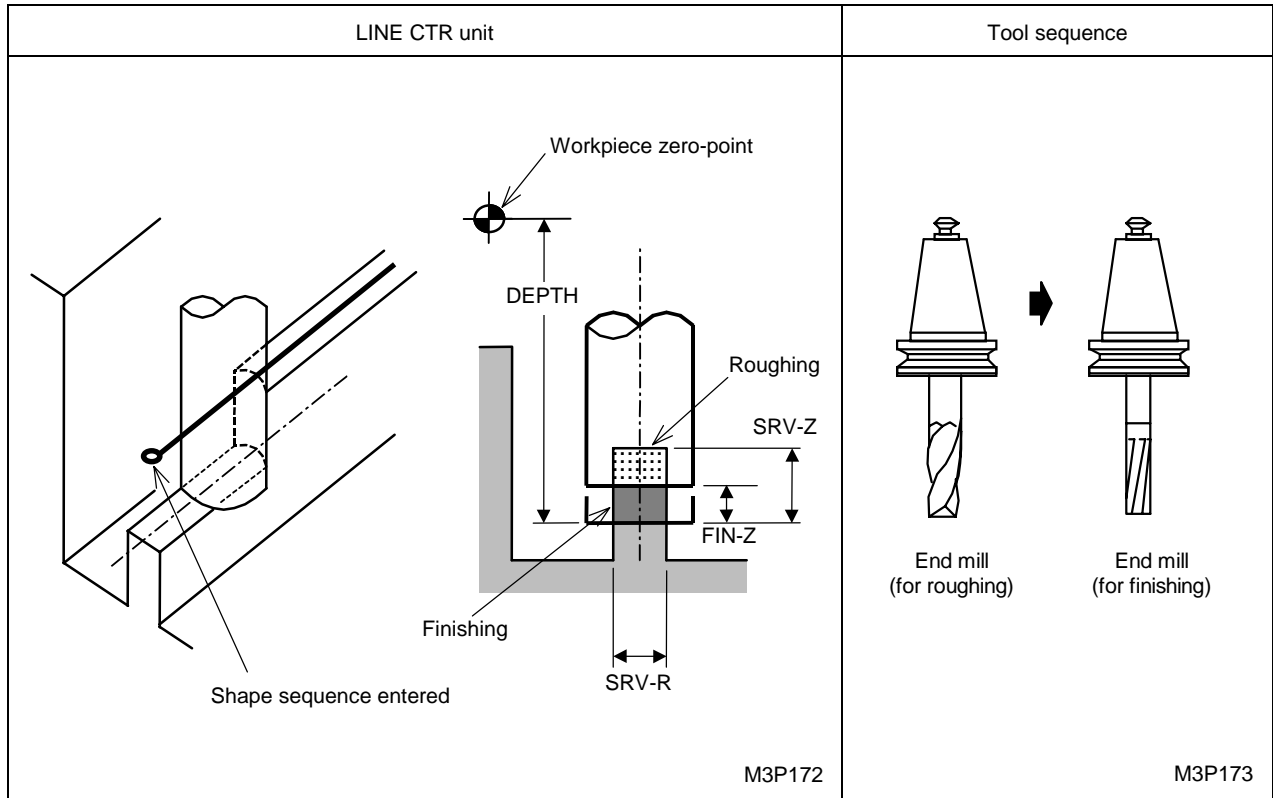
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to either face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z and on FIN-Z.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-8-4.

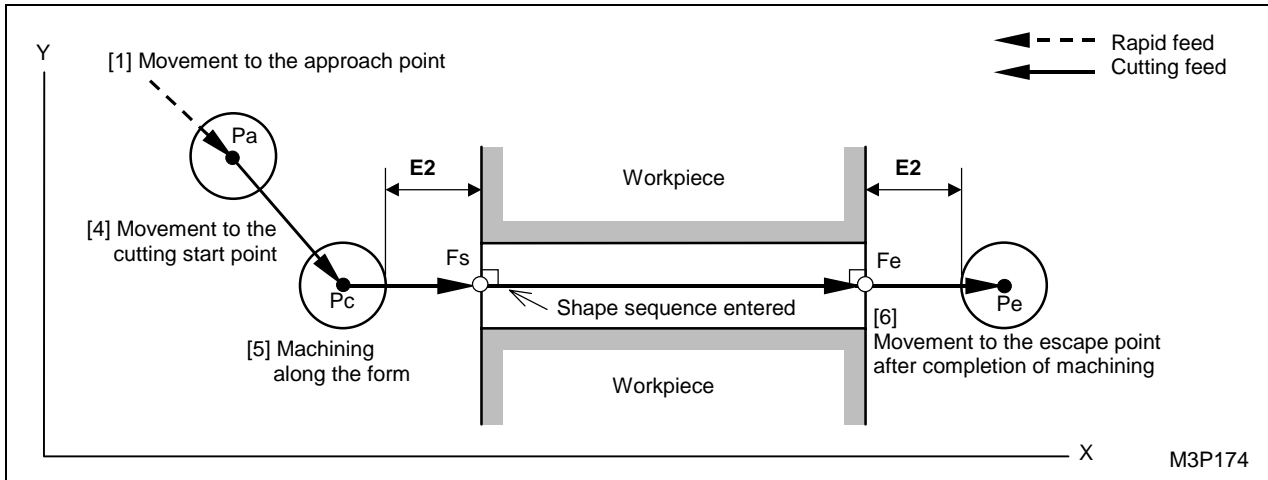


RGH: A roughness code should be selected out of the menu.

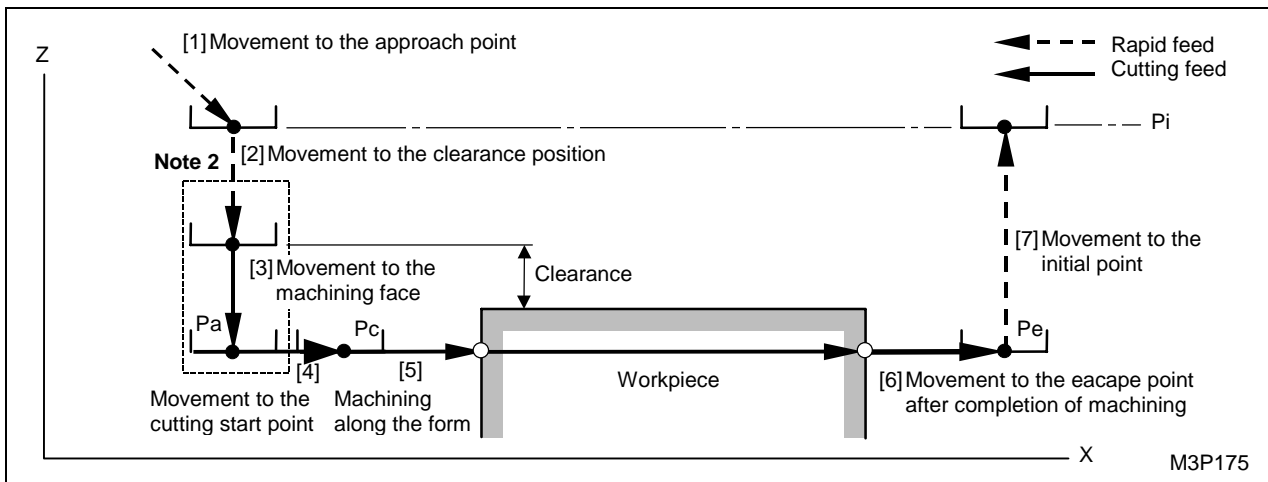
FIN-Z: A Z-axial finishing allowance is automatically entered once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

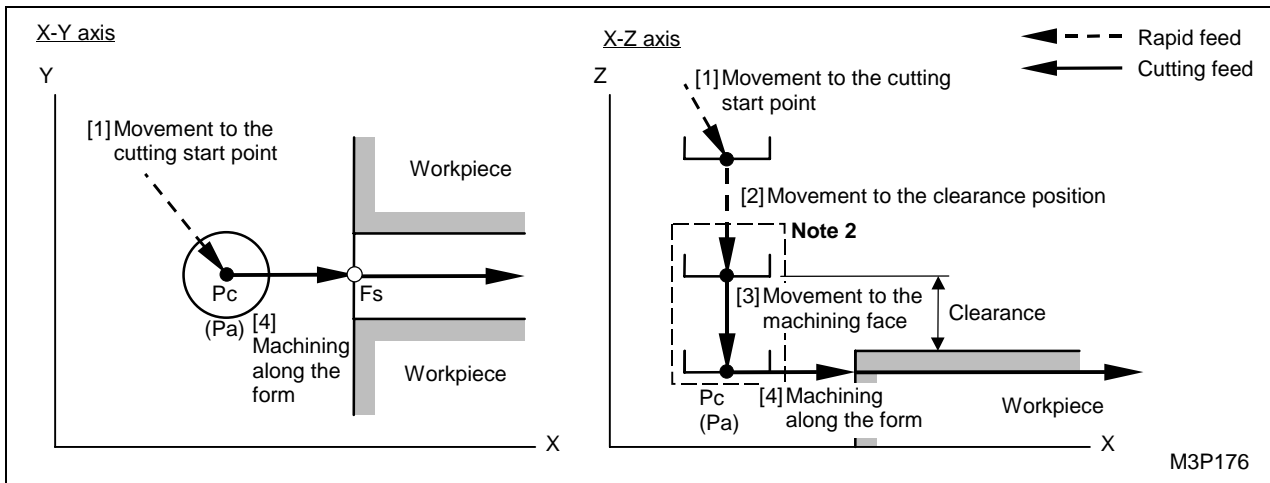
Fs: Start point of form to be entered in the shape sequence

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

Pe: Escape point to be determined automatically

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

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



Note 2: See Subsection 4-8-6, "Precautions in line machining."

Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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

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

A. Data setting

UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE RGT	999.999	999.999	999.999	9	999.999	999.999						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

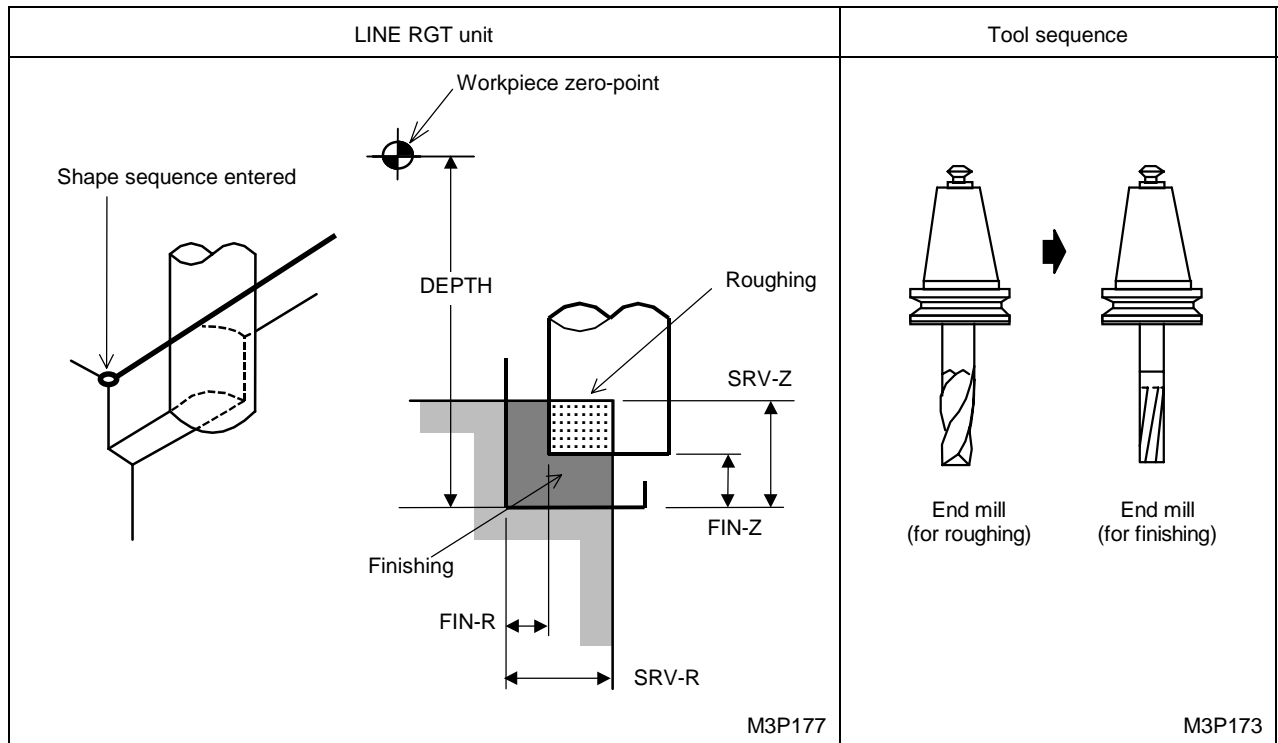
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-8-4.



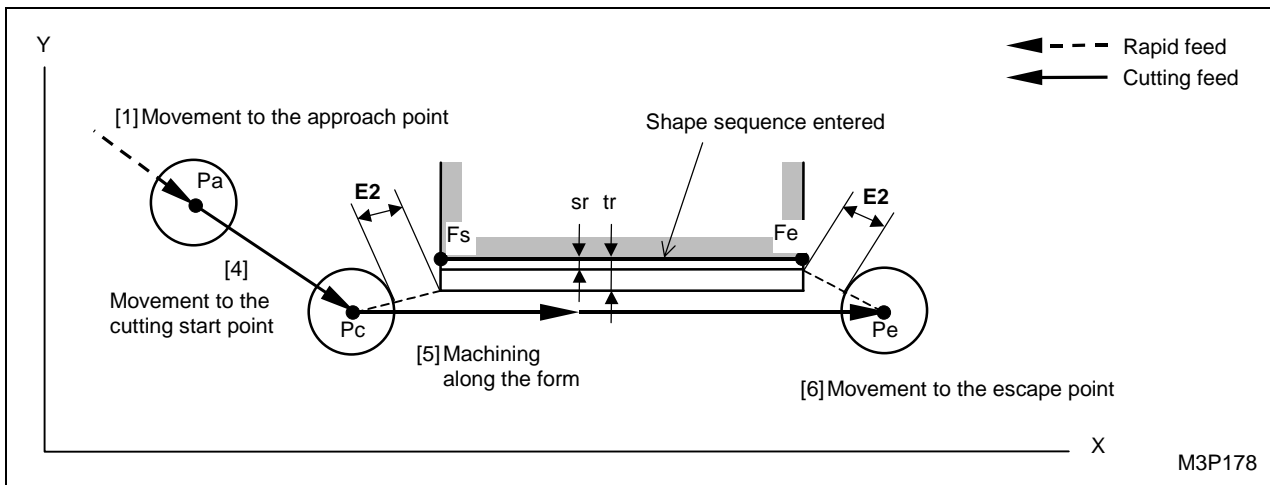
RGH: A roughness code should be selected out of the menu.

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

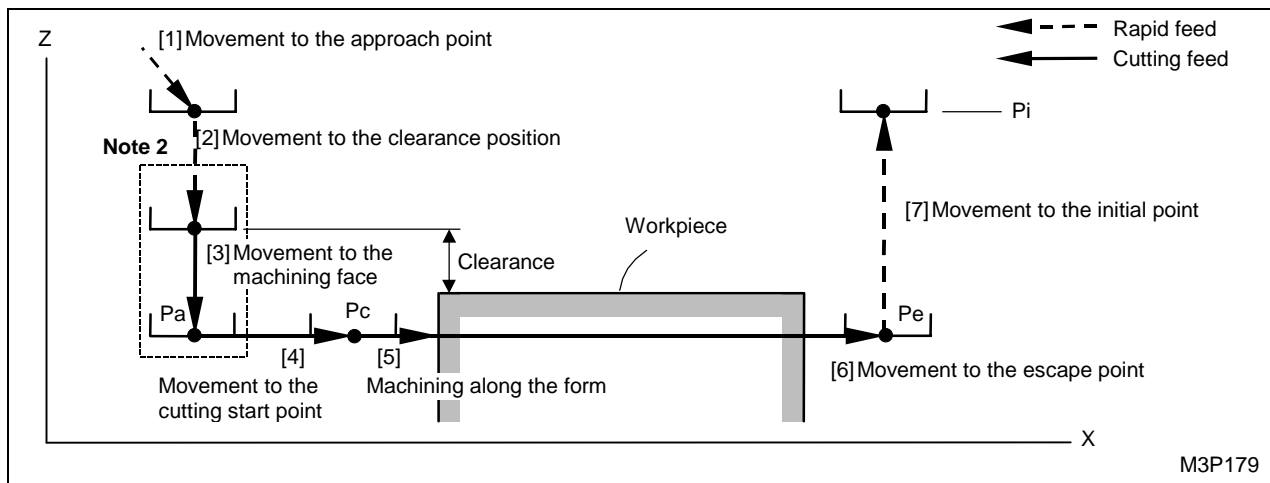
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y 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

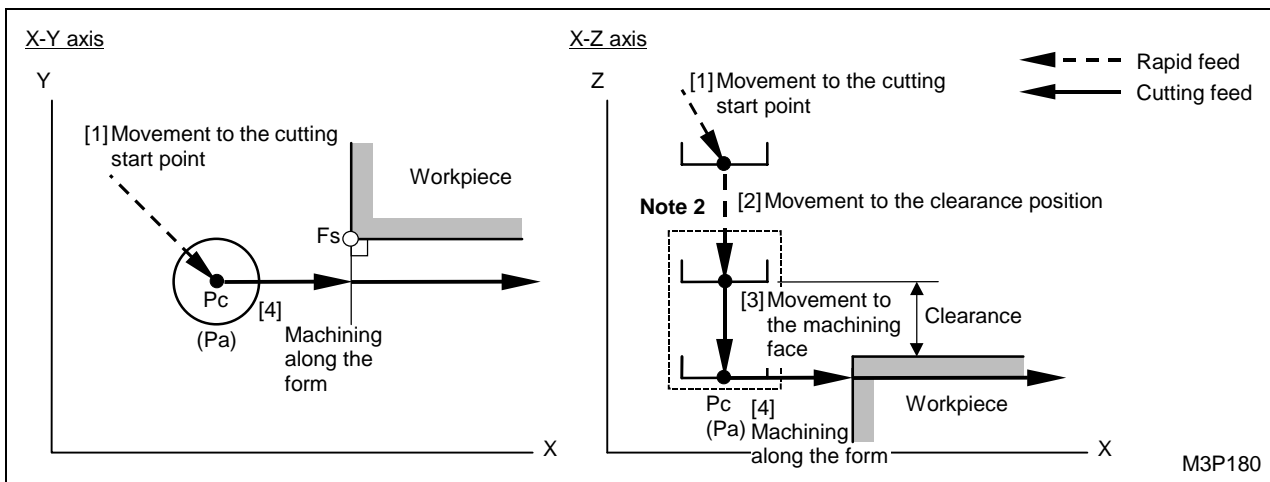
Pe: Escape point to be automatically established

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

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

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



Note 2: See Subsection 4-8-6 "Precautions in line machining."

Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis 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	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE LFT	999.999	999.999	999.999	9	999.999	999.999						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

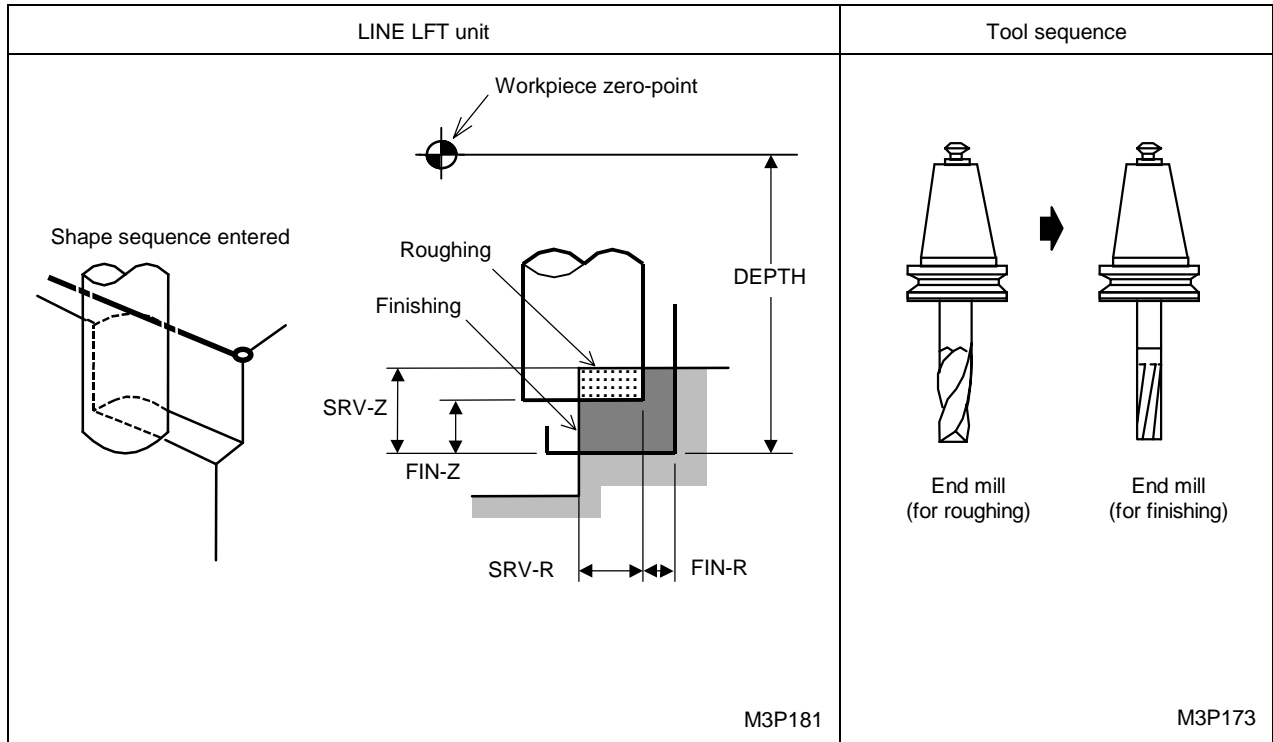
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-8-4.



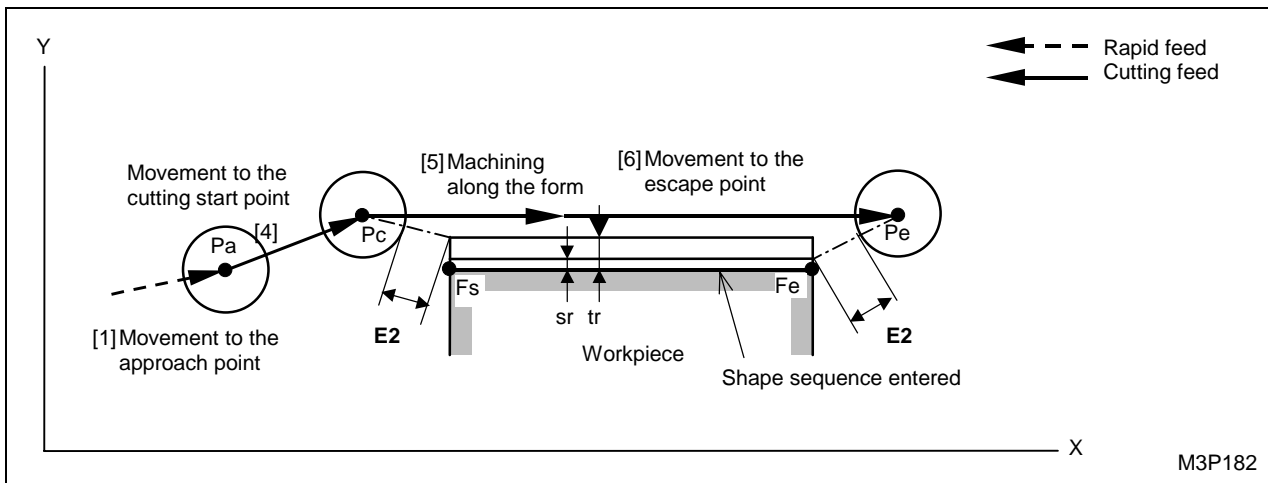
RGH: A roughness code should be selected out of the menu.

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

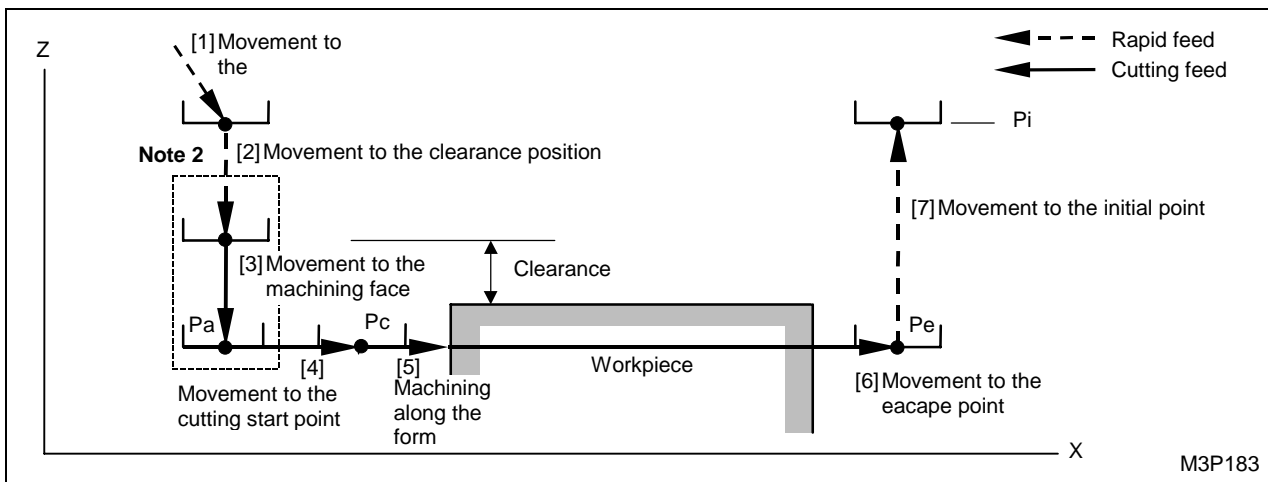
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y 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

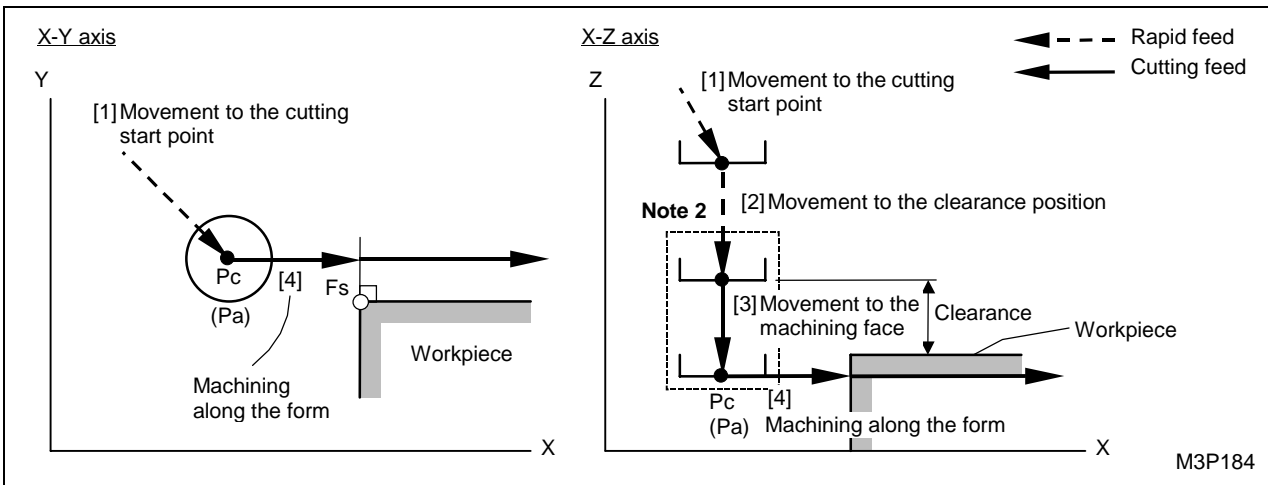
Pe: Escape point to be automatically established

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

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



Note 2: See Subsection 4-8-6, “Precautions in line machining.”

Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis 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	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE OUT	999.999	999.999	999.999	9	999.999	999.999						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				

Remark 1: Data in unit represent a maximum input value.

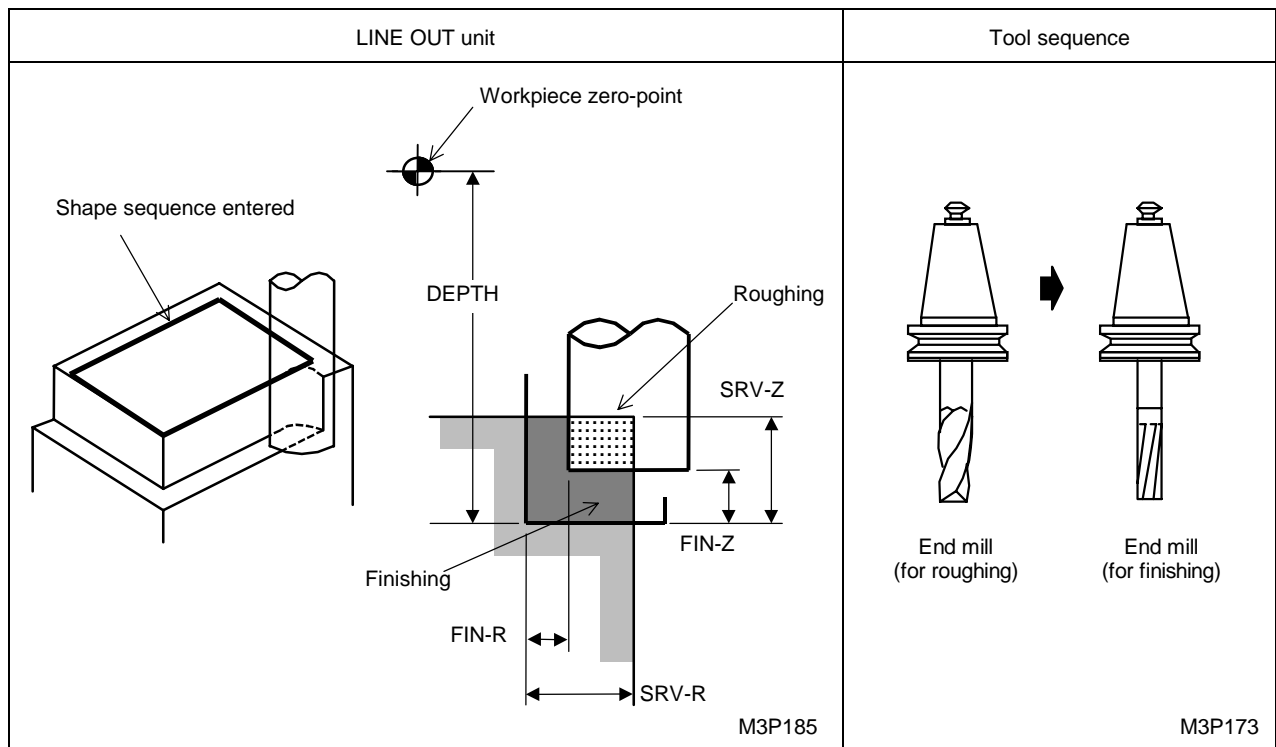
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-8-4.



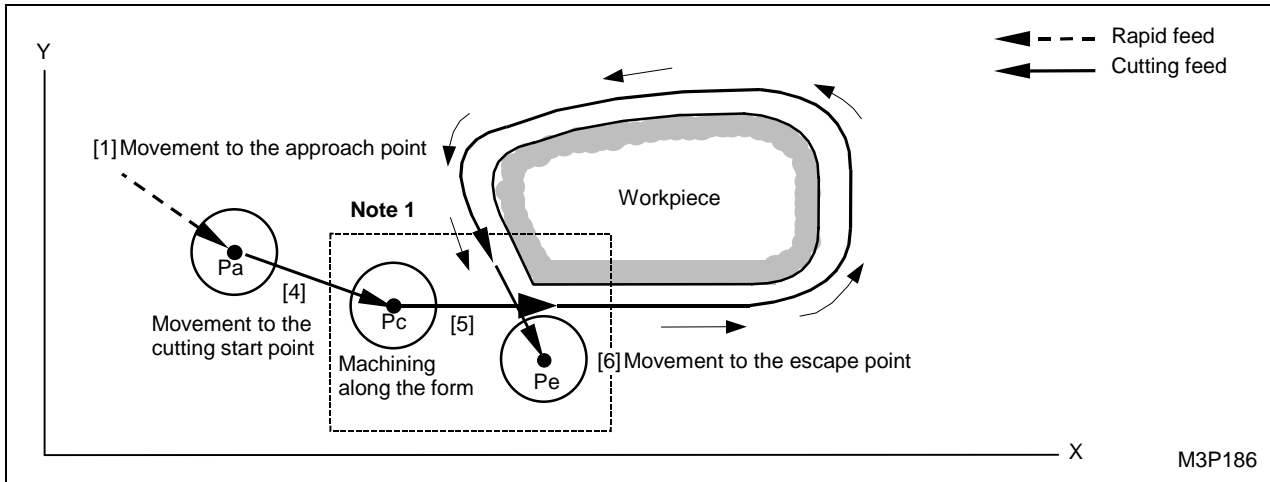
RGH: A roughness code should be selected out of the menu.

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

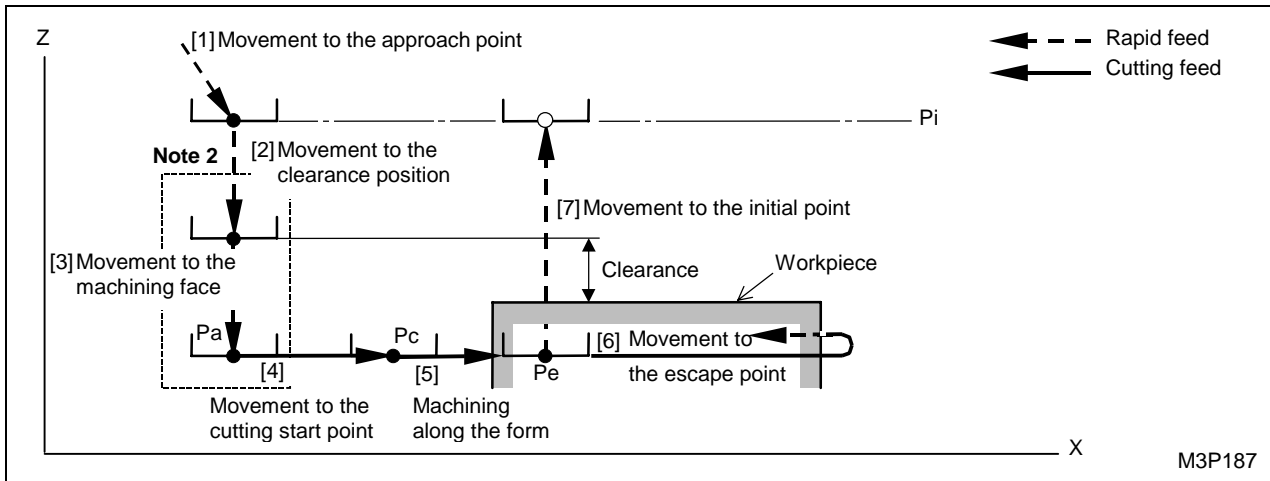
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

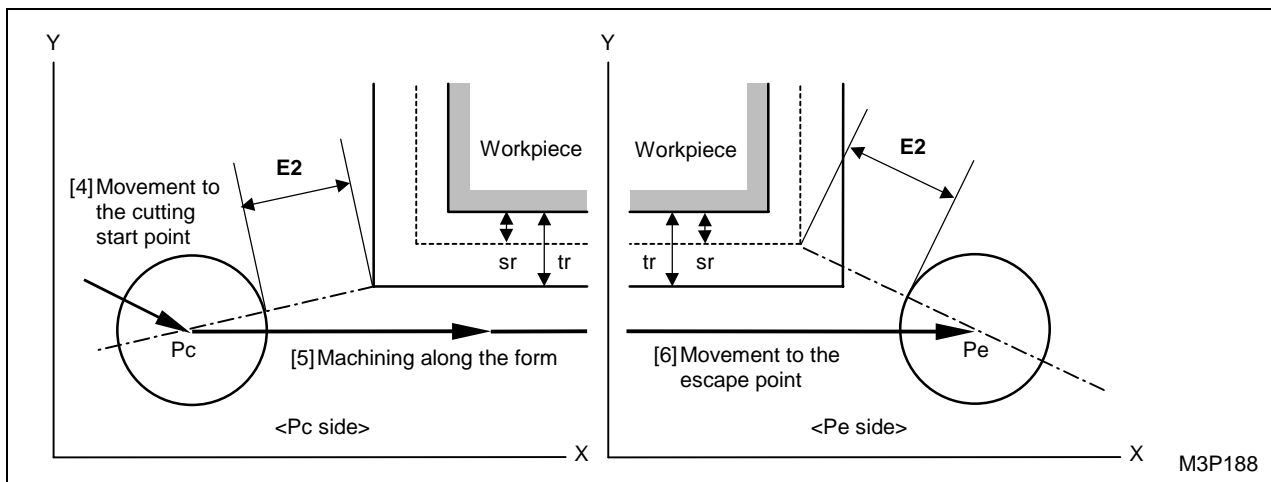
Pe: Escape point to be automatically established

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

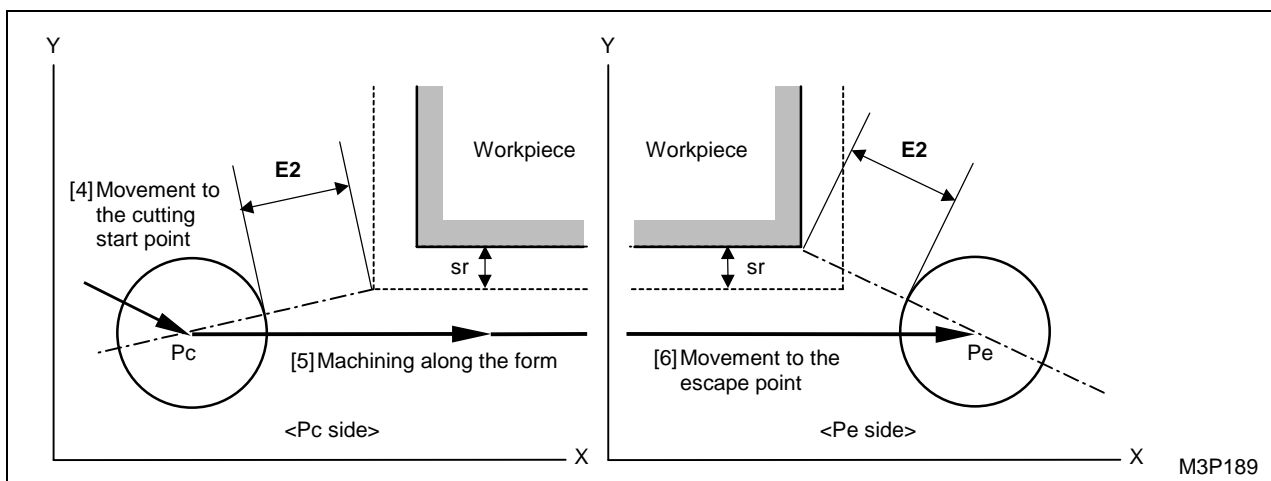
Note 1: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form

- In case of roughing



- In case of finishing



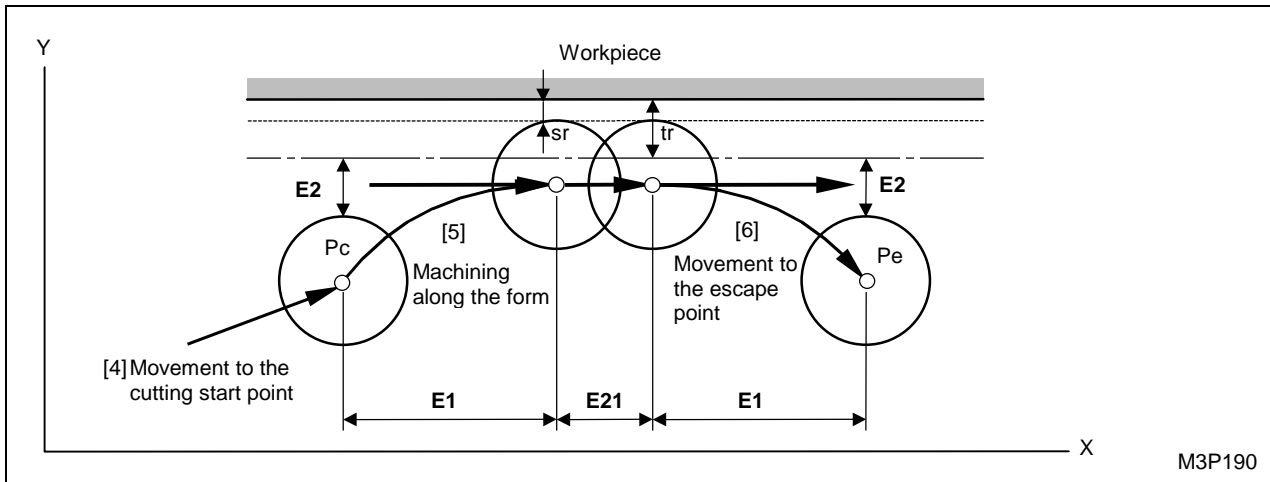
The bold codes represent parameter addresses.

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

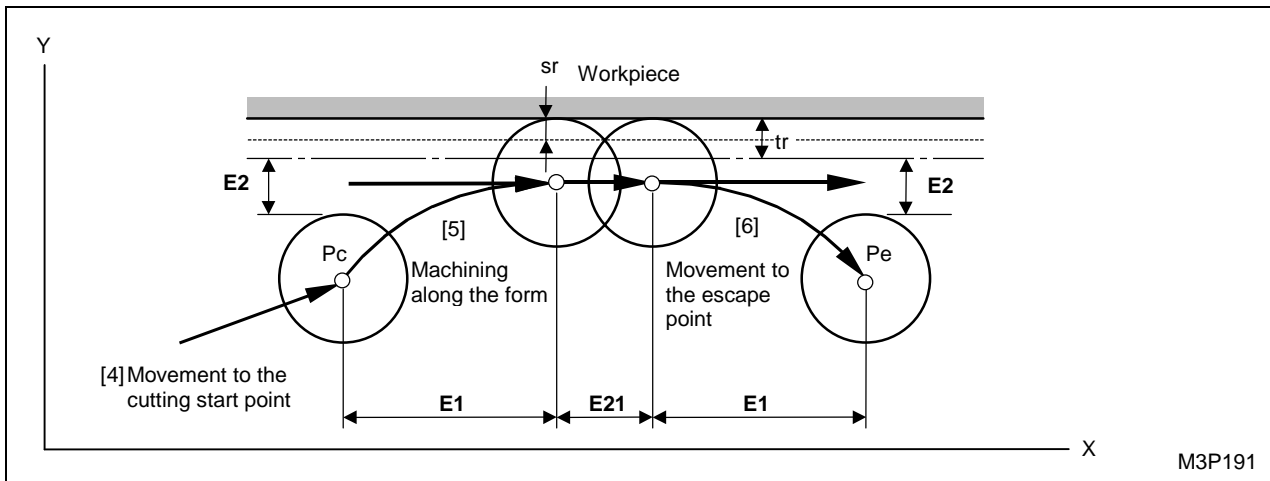
sr: Radial finishing allowance determined by the data FIN-R in the machining unit

When 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 to be determined by the data FIN-R in the machining unit

Note 2: See Subsection 4-8-6, "Precautions in line machining."

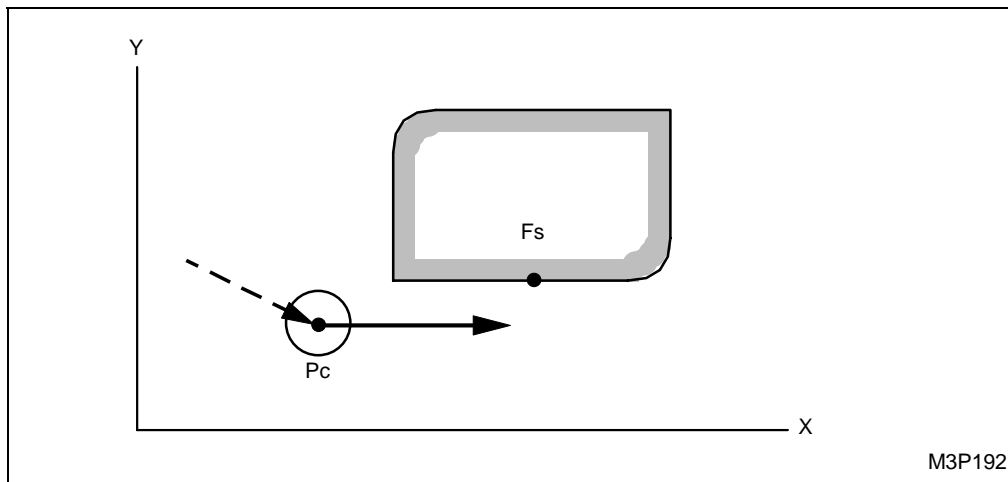
Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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).

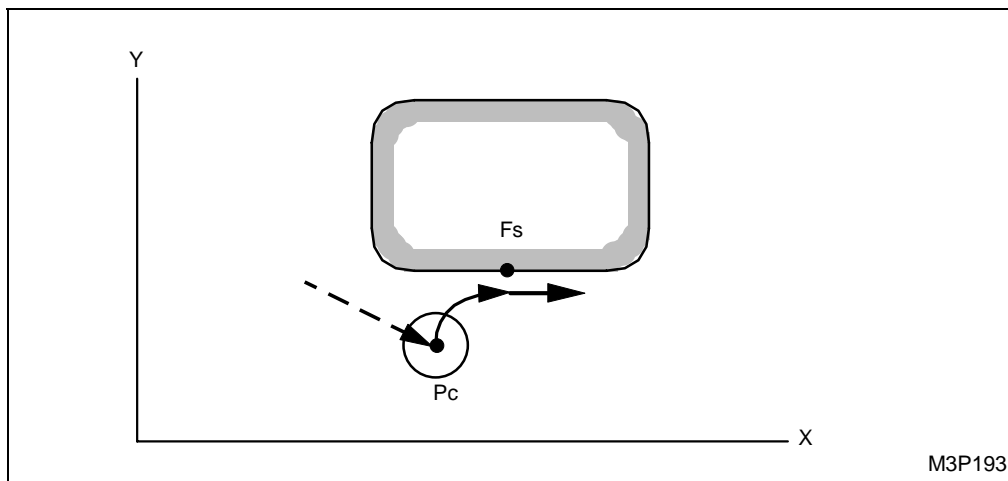
When ? is displayed in the items APRCH-X, -Y

- Form having a convex point:



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

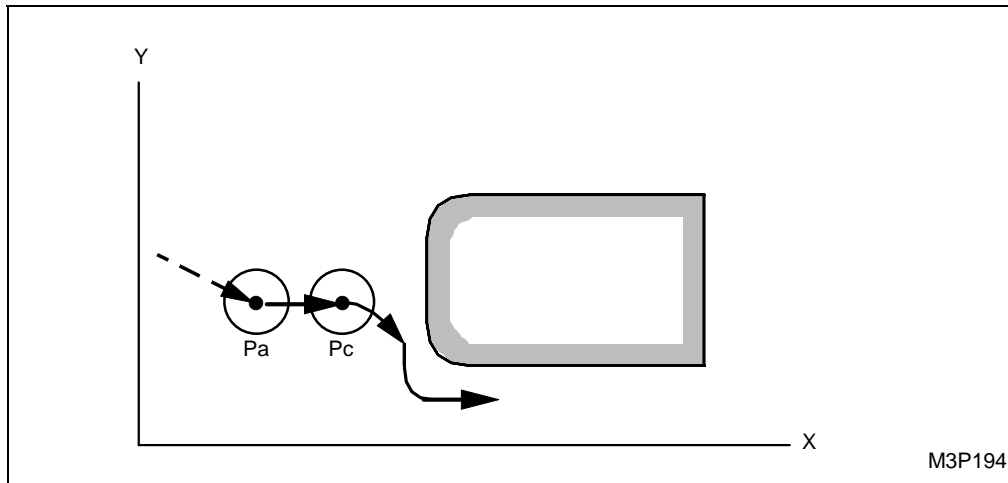
- Form having no convex point:



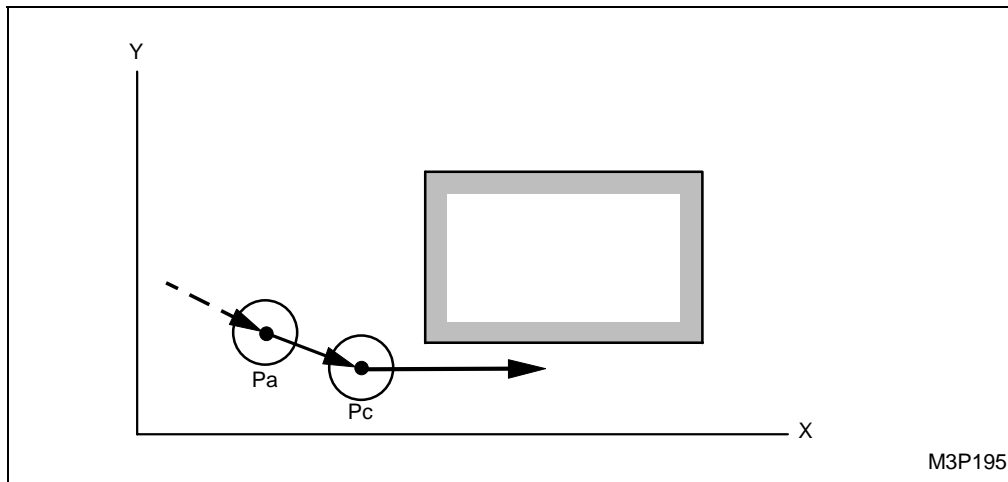
Cutting is started from the start point (F_s) entered in the shape sequence.

When the data is entered in the items APRCH-X, -Y

- 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 menu key **AUTO SET**, 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	DEPTH	SRV-Z	SRV-R	RGH			FIN-Z	FIN-R				
1	LINE IN	999.999	999.999	999.999	9			999.999	999.999				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				

Remark 1: Data in unit represent a maximum input value.

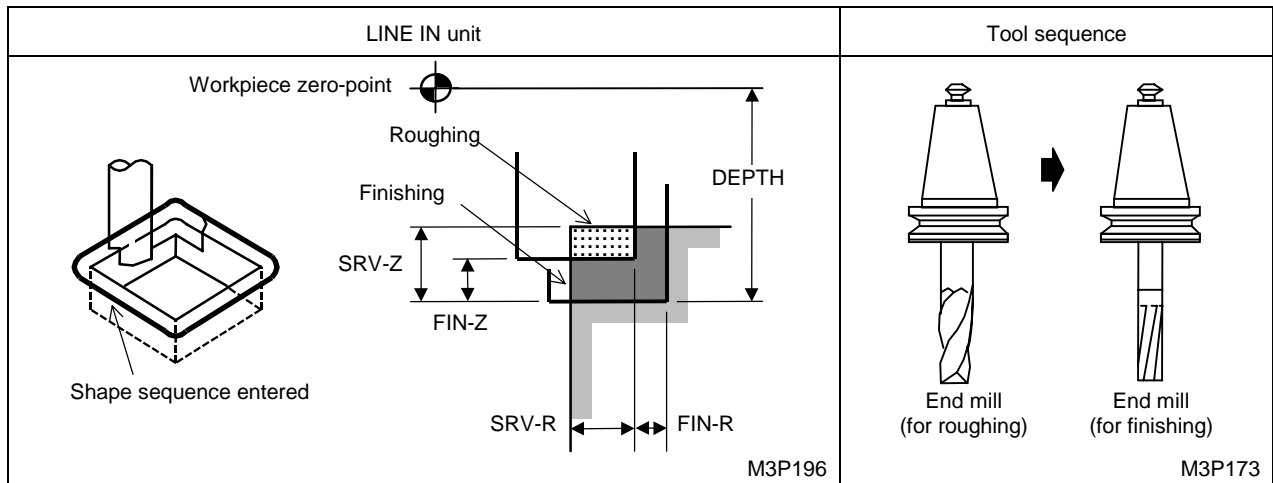
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.



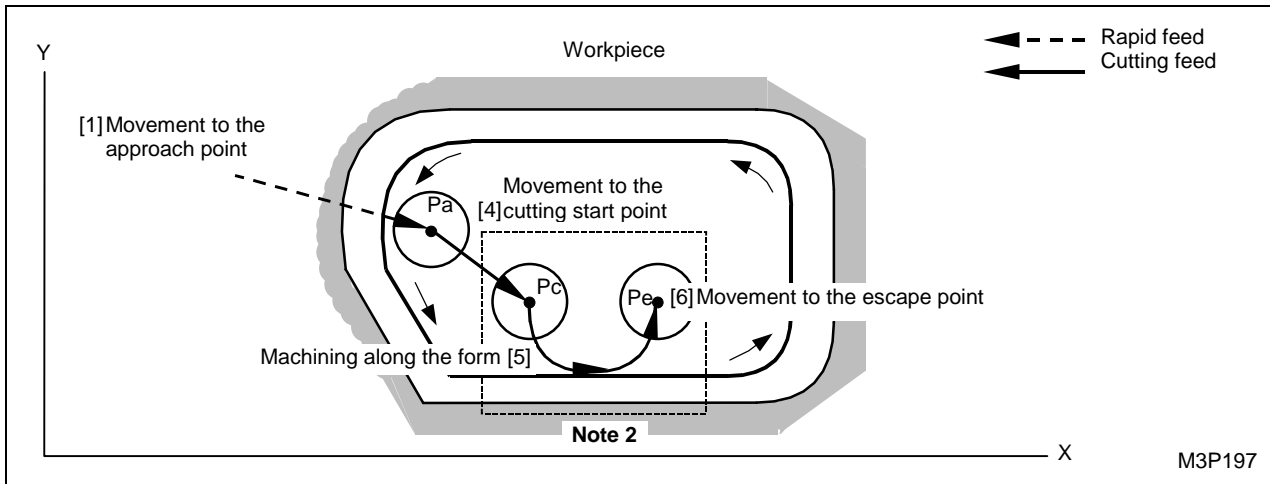
RGH: A roughness code should be selected out of the menu.

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

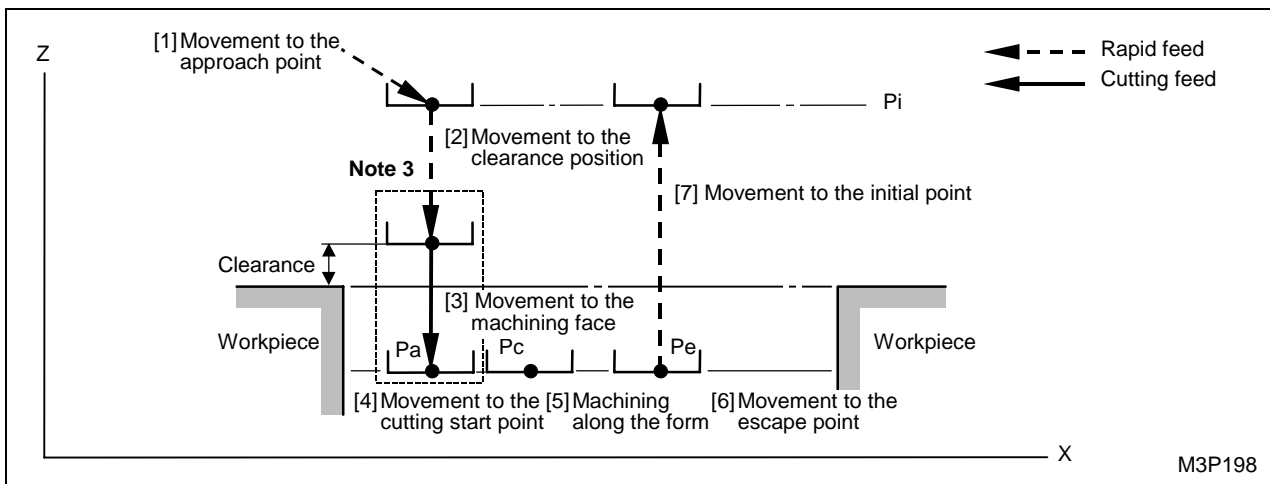
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

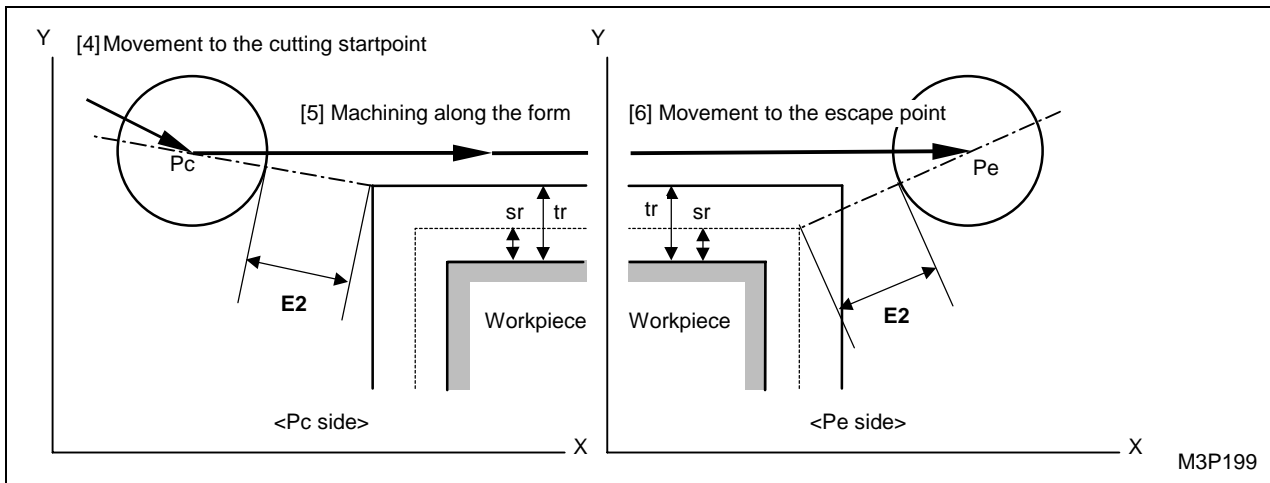
Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

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

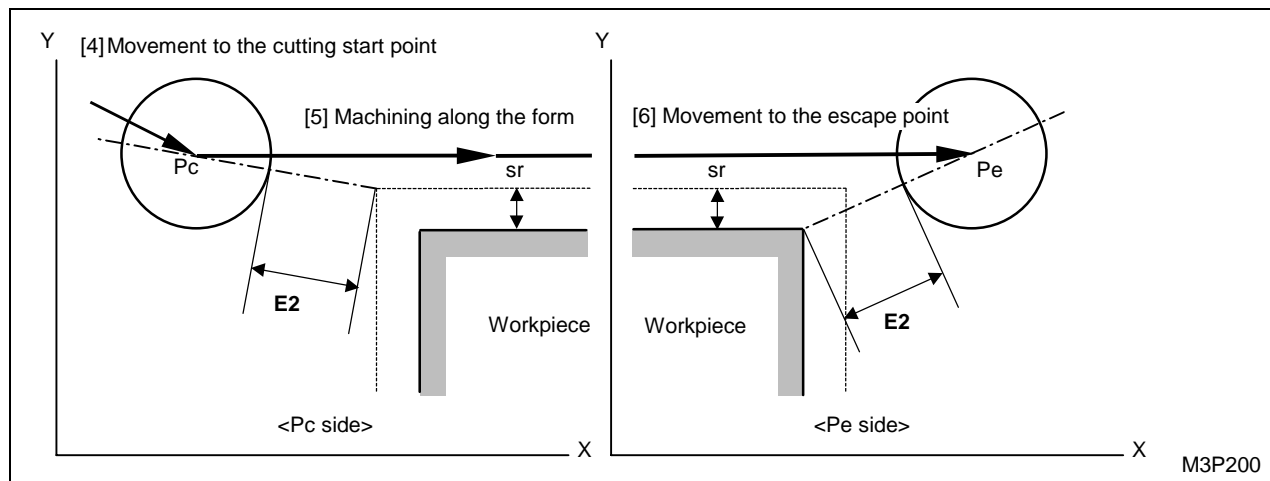
Note 2: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form

- In case of roughing



- In case of finishing



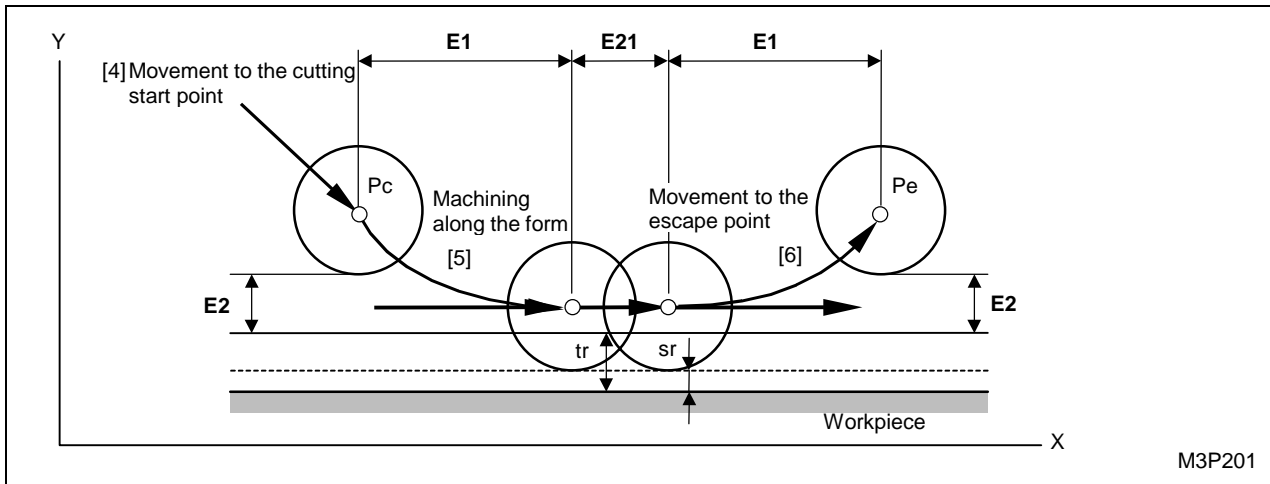
The bold codes represent parameter addresses.

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

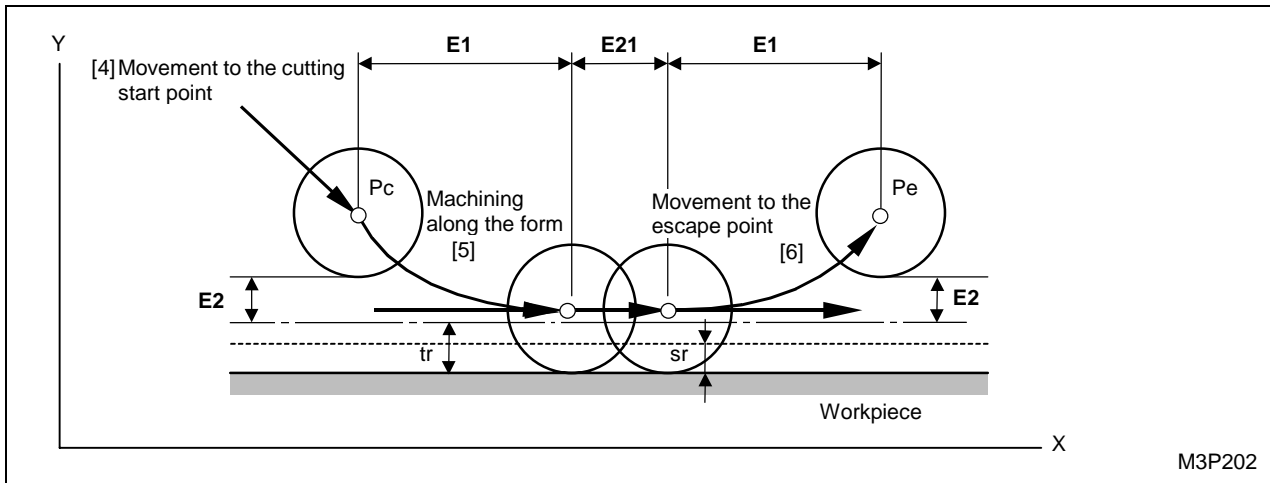
sr: Radial finishing allowance determined by the data **FIN-R** in the machining unit

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

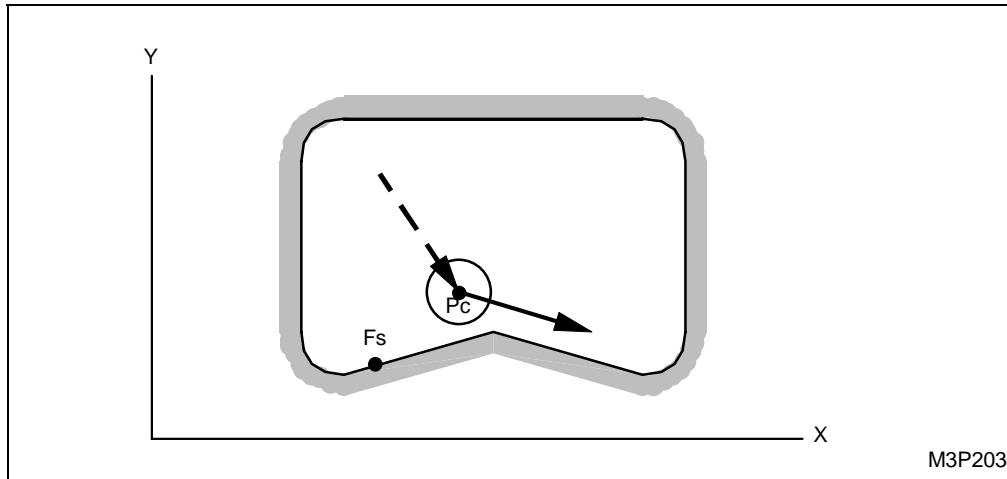
Note 3: See Subsection 4-8-6, "Precautions in line machining."

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).

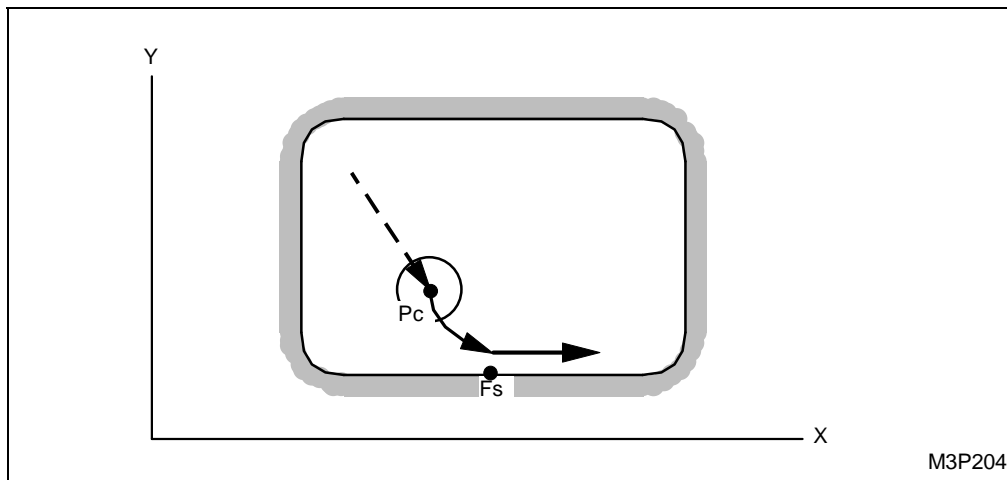
When ? is displayed in the items APRCH-X, -Y

- Form having a convex point:



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

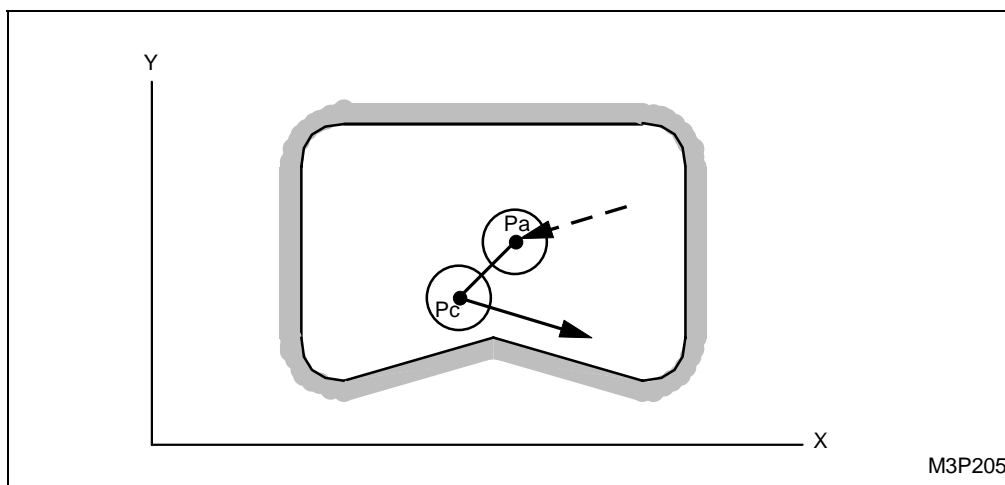
- Form having no convex point:



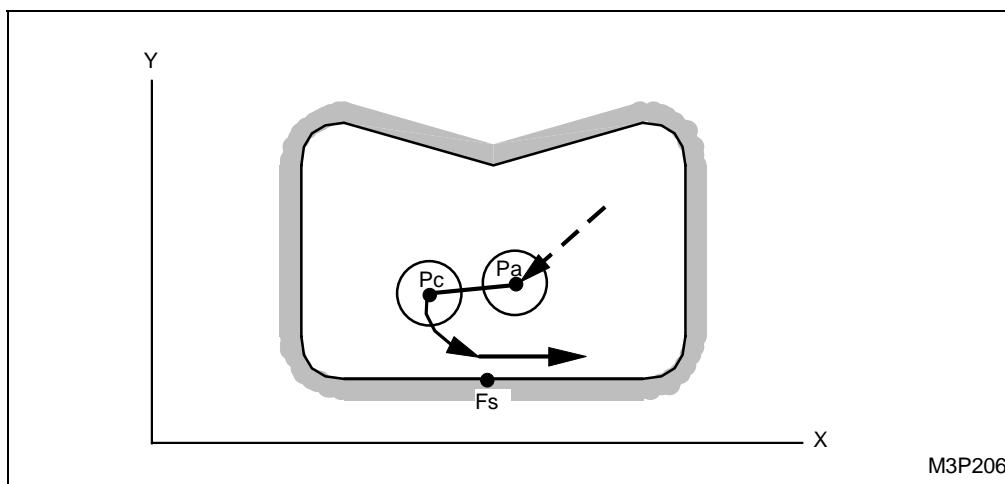
Cutting is started from the start point (F_s) entered in the shape sequence.

When the data is entered in the items APRCH-X, -Y

- 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 menu key **AUTO SET**, 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	DEPTH	INTER-Z	INTER-R	CHMF								
1	CHMF RGT	999.999	99.999	99.999	99.99								
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER					◆		◆	◆				

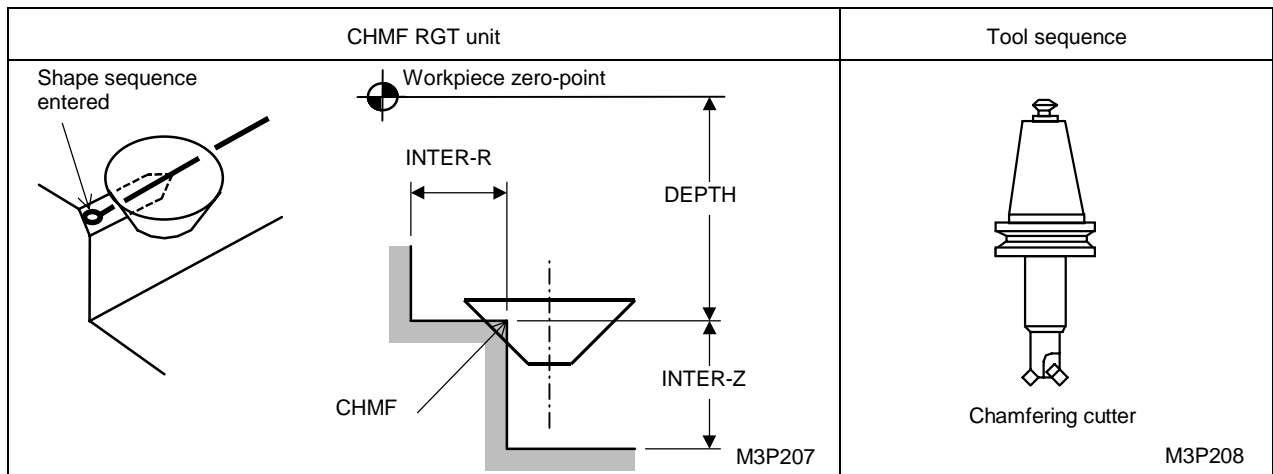
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

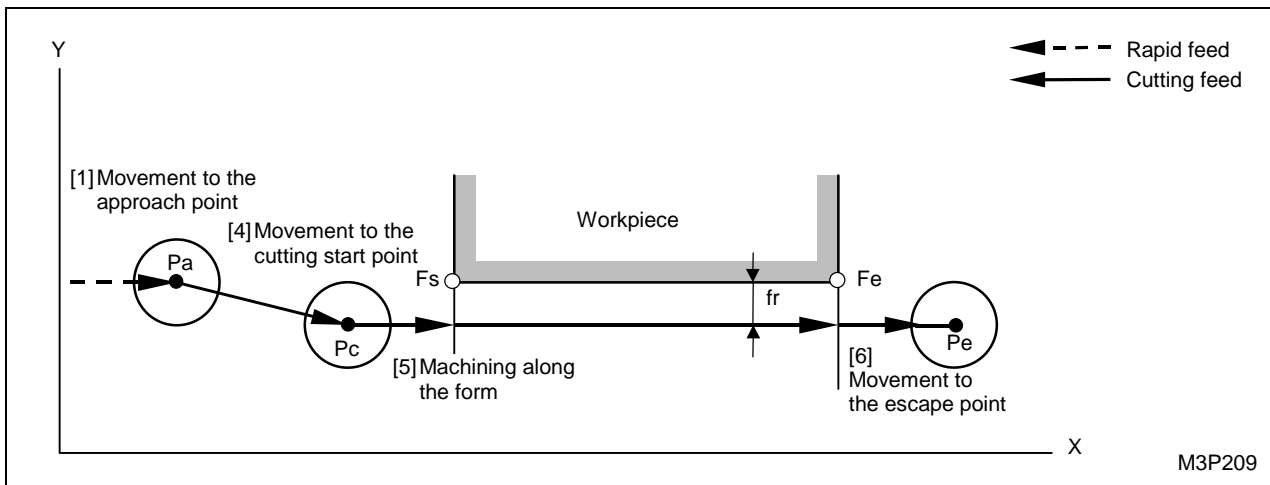
Remark 4: For the tool sequence data setting, refer to Subsection 4-8-4.

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

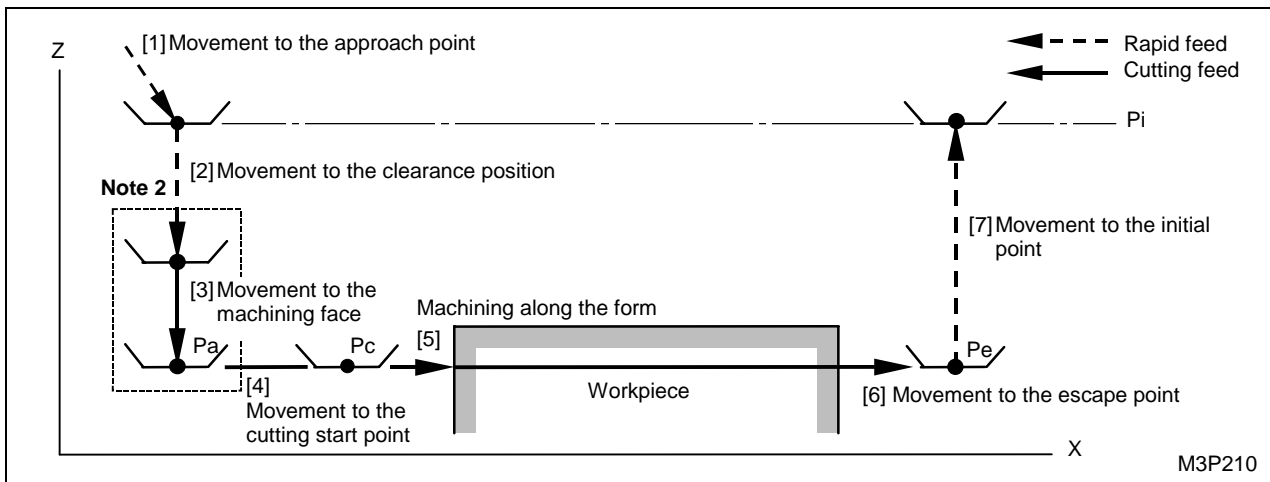


B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

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

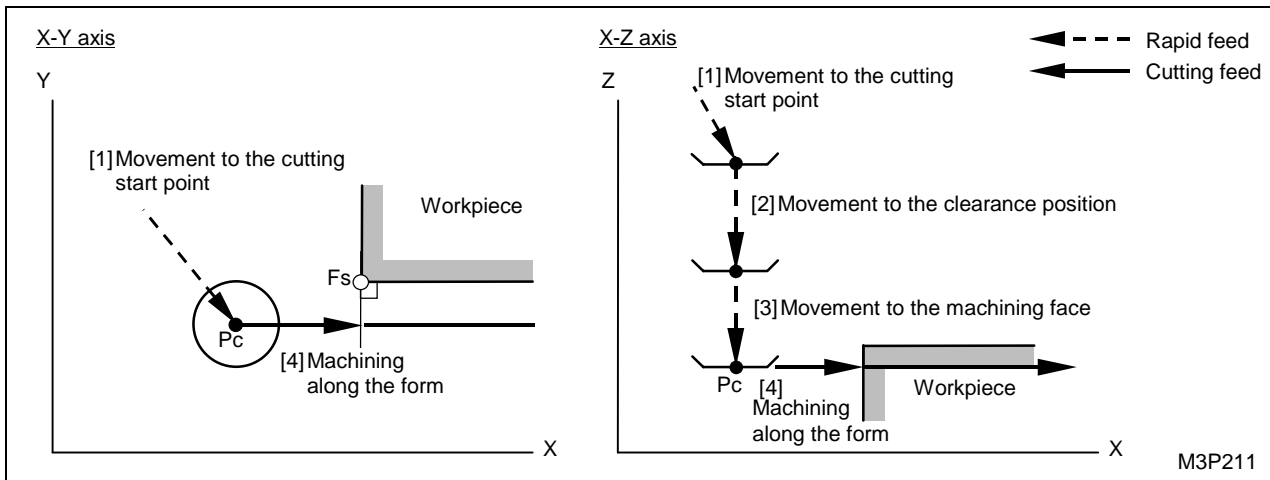
Fe: End point of form to be entered in the shape sequence

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

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

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



The bold codes represent parameter addresses.

Note 2: See Subsection 4-8-6, "Precautions in line machining."

Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis 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	DEPTH	INTER-Z	INTER-R	CHMF								
1	CHMF LFT	999.999	99.999	99.999	99.9								
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER					◆		◆	◆				

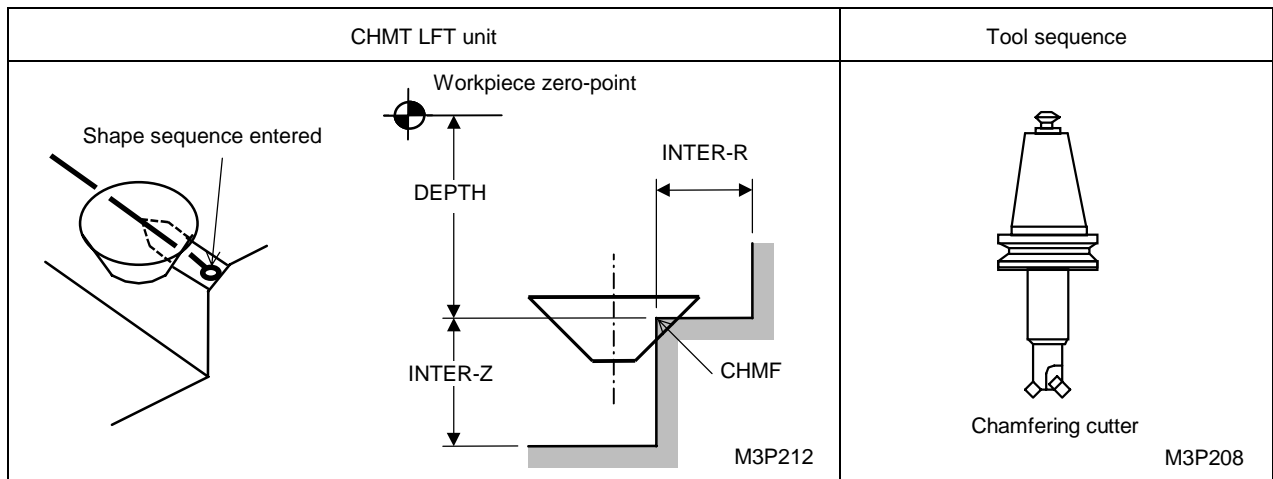
Remark 1: Data in unit represent a maximum input value.

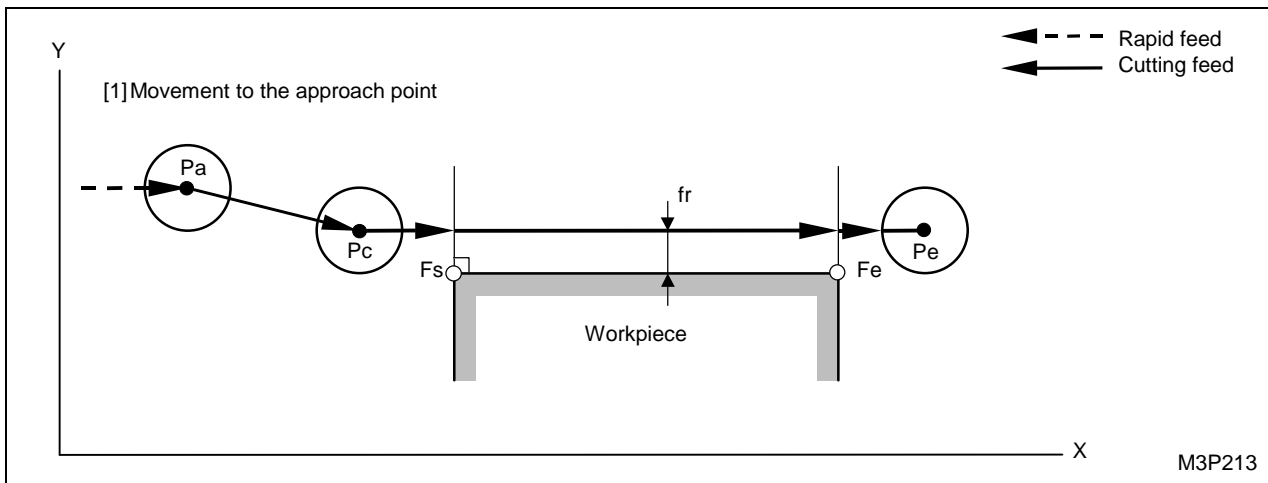
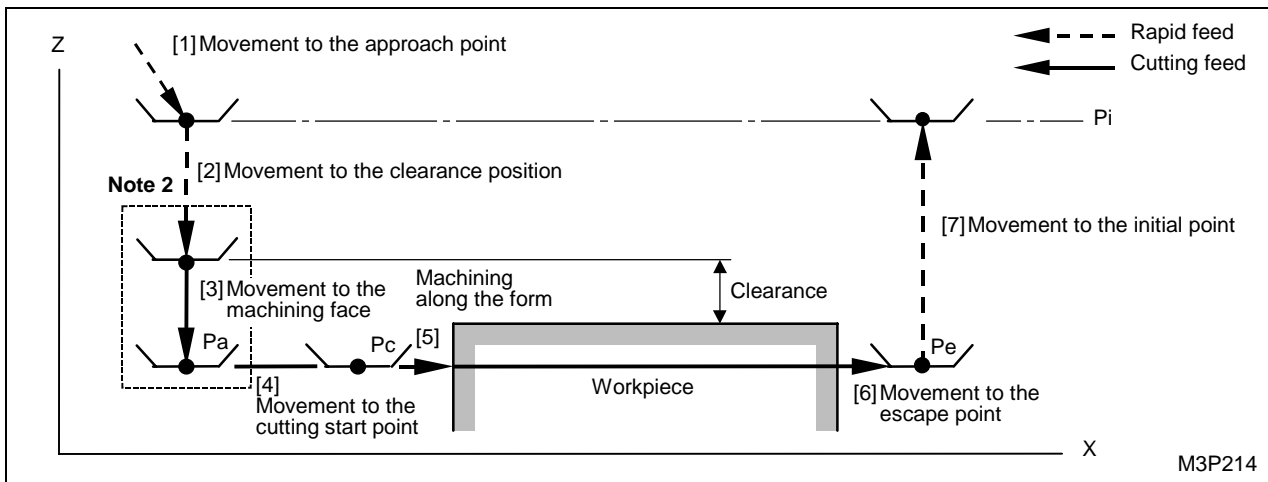
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For the tool sequence data setting, refer to Subsection 4-8-4.

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



B. Tool pathX-Y axisX-Z axis

The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

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

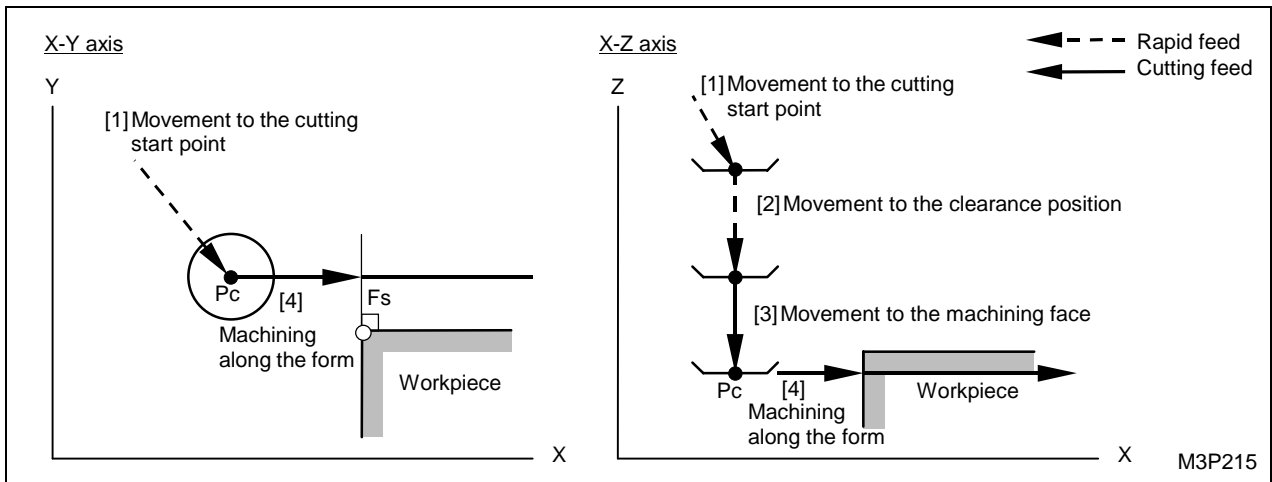
Fe: End point of form to be entered in the shape sequence

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

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



Note 2: See Subsection 4-8-6, "Precautions in line machining."

Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis 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	DEPTH	INTER-Z	INTER-R	CHMF								
1	CHMF OUT	999.999	99.999	99.999	99.99								
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER							◆	◆				

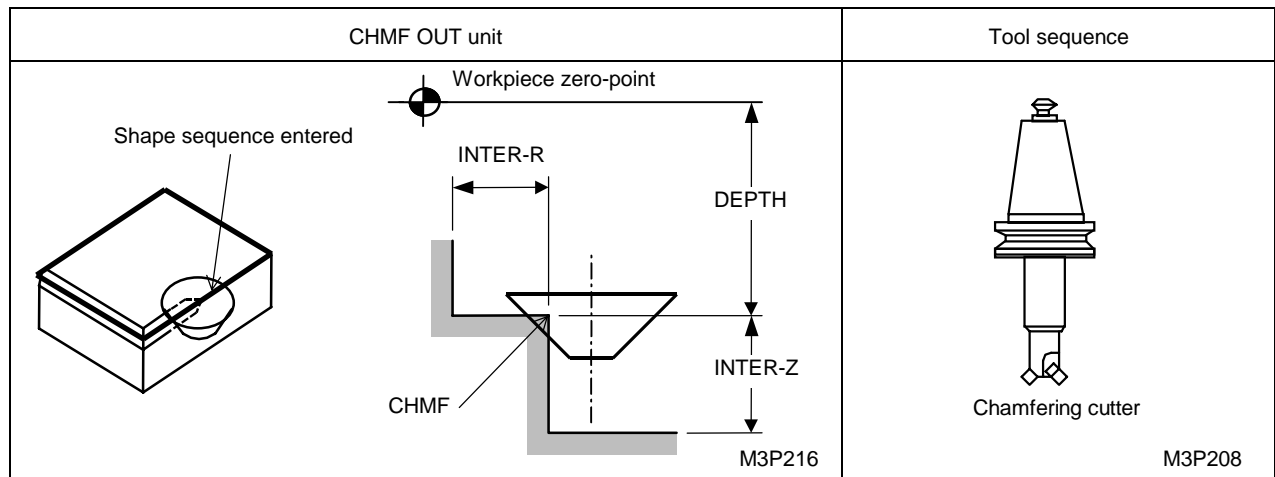
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be here.

Remark 3: In this unit, chamfering tools are automatically developed. Instead of the chamfering cutter, a centering drill can be used.

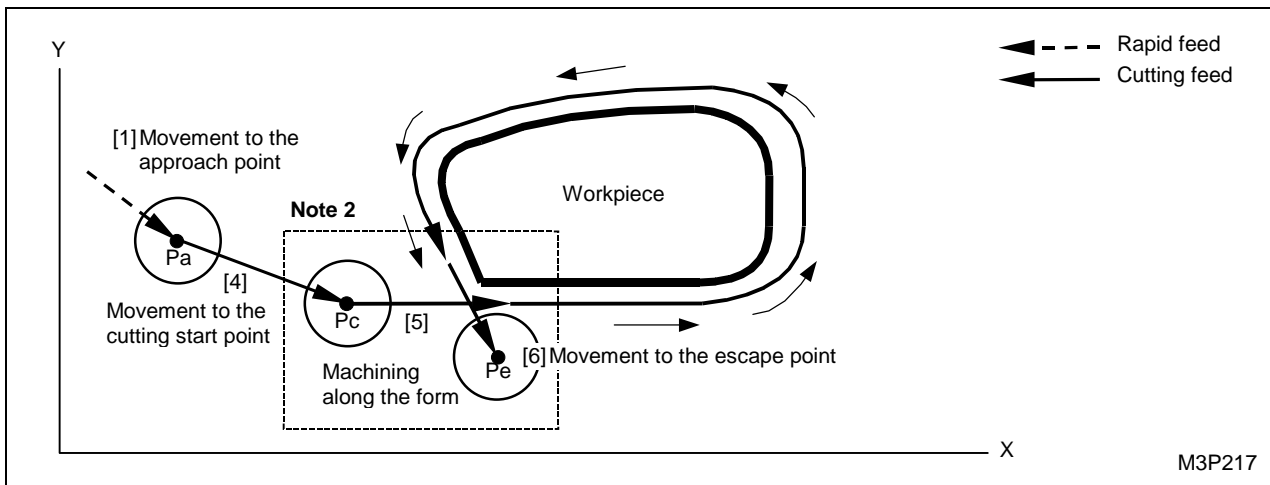
Remark 4: For tool sequence data setting, refer to Subsection 4-8-4.

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

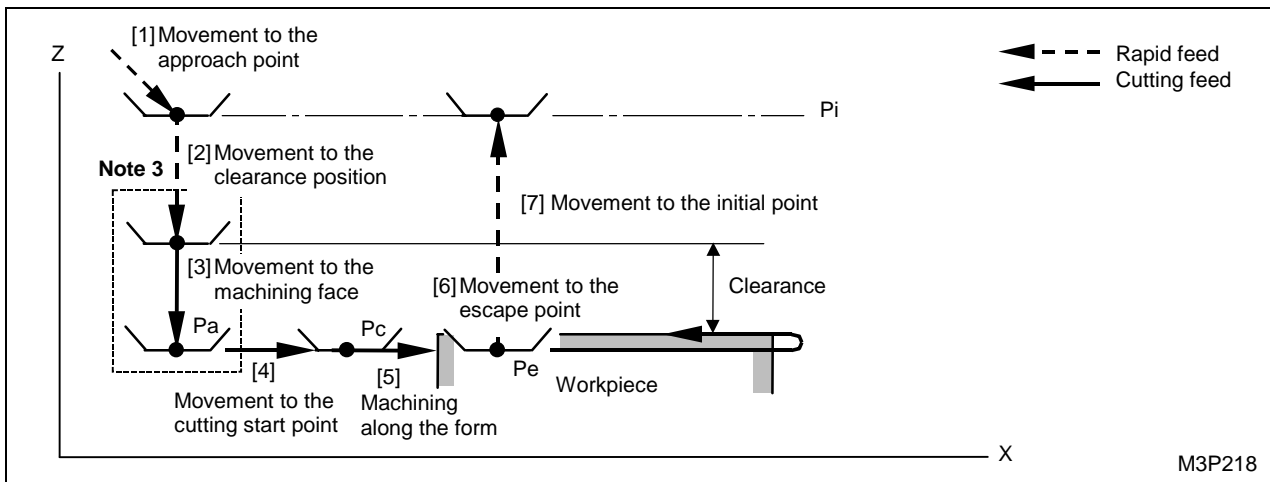


B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

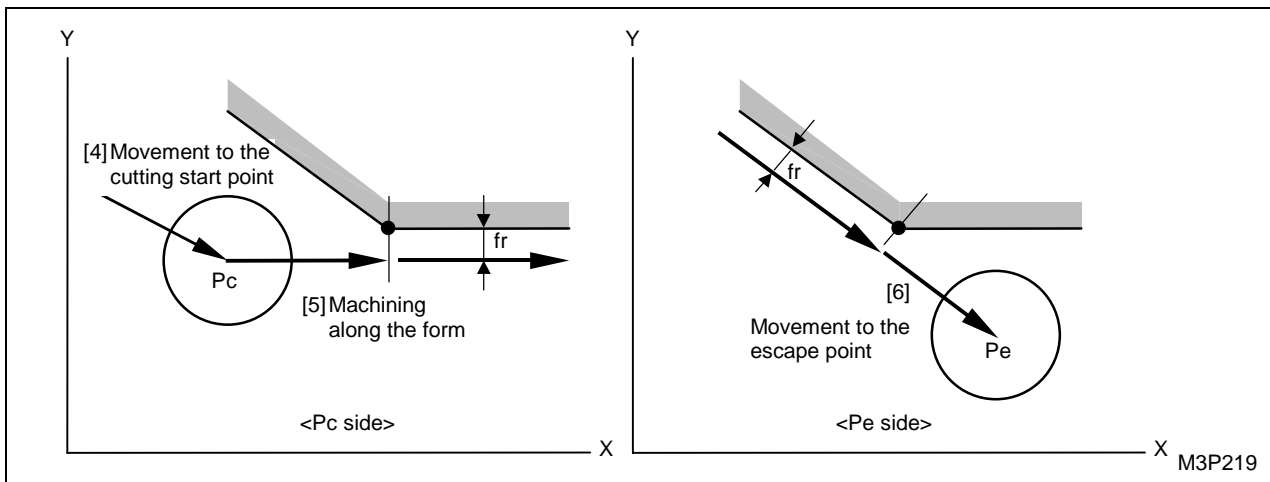
Pe: Escape point to be automatically established

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

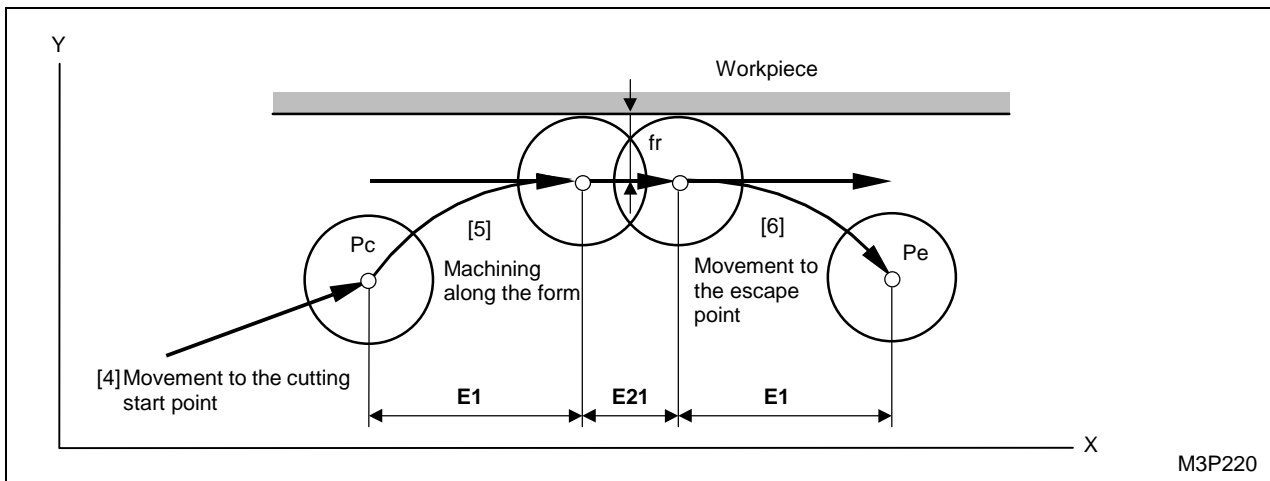
Note 1: The feedrate on tool path [3] is dependent upon the ZFD (Z-axis feed) in the tool sequence.

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

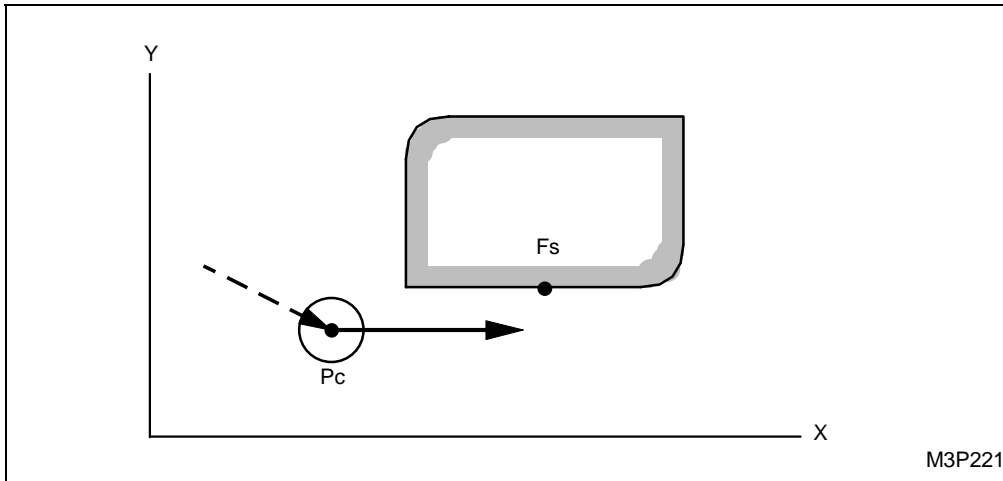
Note 3: See Subsection 4-8-6, "Precautions in line machining."

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).

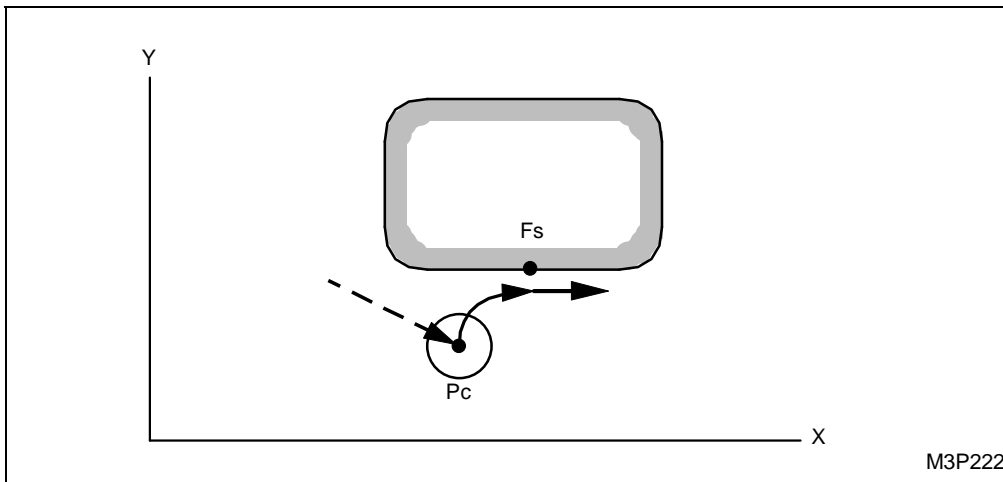
When ? is displayed in the items APRCH-X, -Y

- Form having a convex point:



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

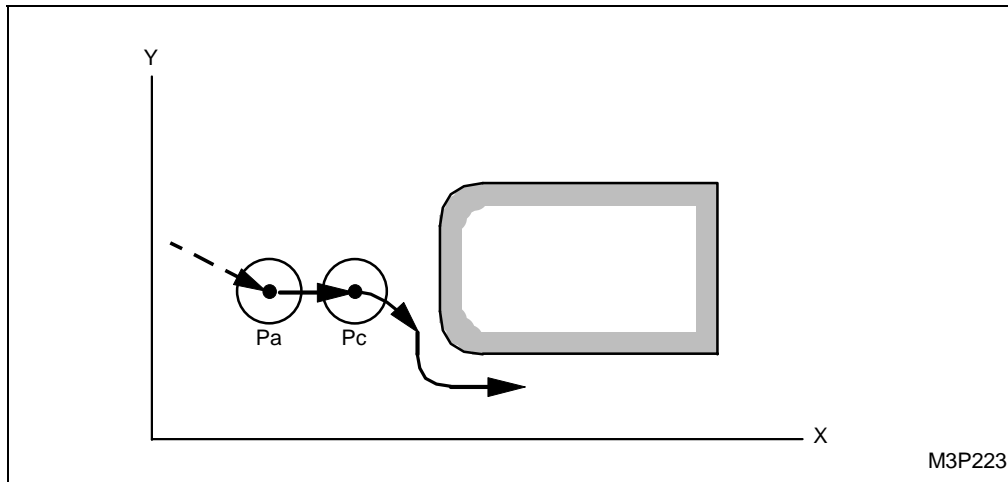
- Form having no convex point:



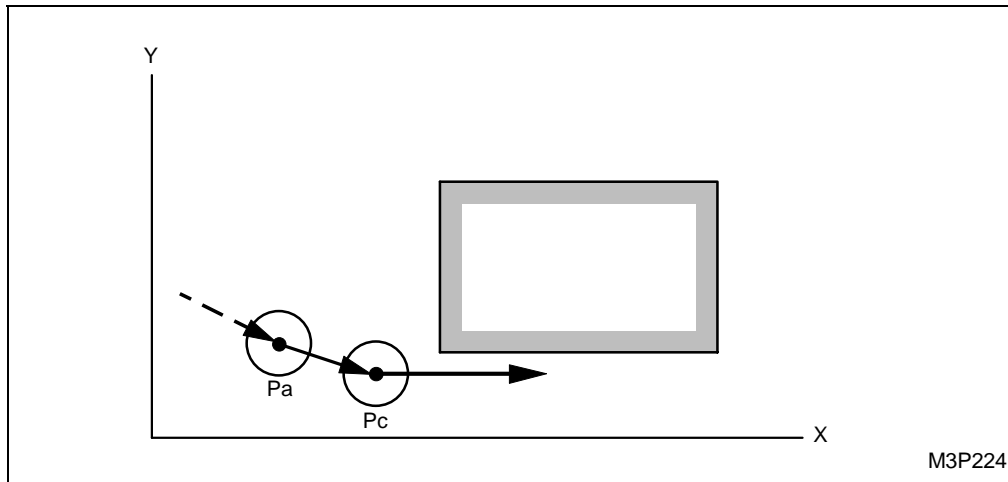
Cutting is started from the start point (Fs) entered in the shape sequence.

When the data is entered in the items APRCH-X, -Y

- 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 menu key **AUTO SET**, 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	DEPTH	INTER-Z	INTER-R	CHMF								
1	CHMF IN	999.999	99.999	99.999	99.9								
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER							◆	◆				

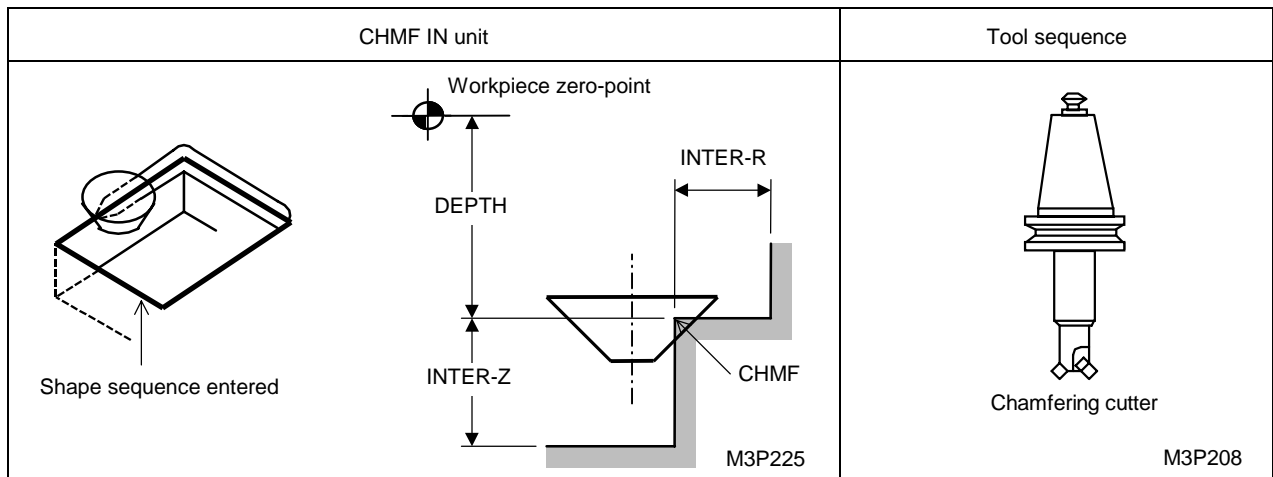
Remark 1: Data in unit represent a maximum input value.

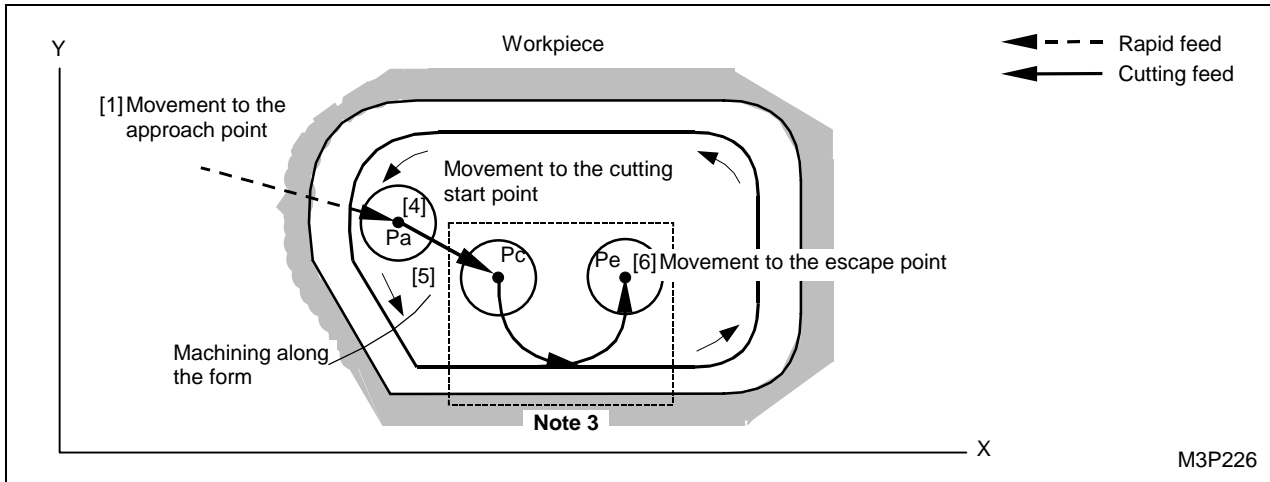
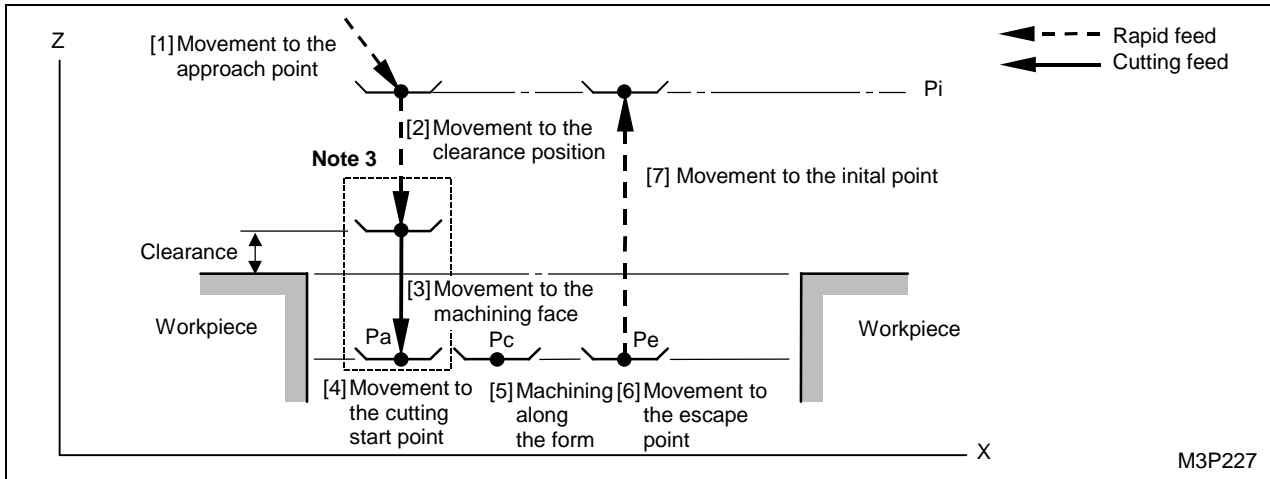
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For sequence data setting, refer to Subsection 4-8-4.

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



B. Tool pathX-Y axisX-Z axis

The bold codes represent parameter addresses.

Pi: Initial point Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

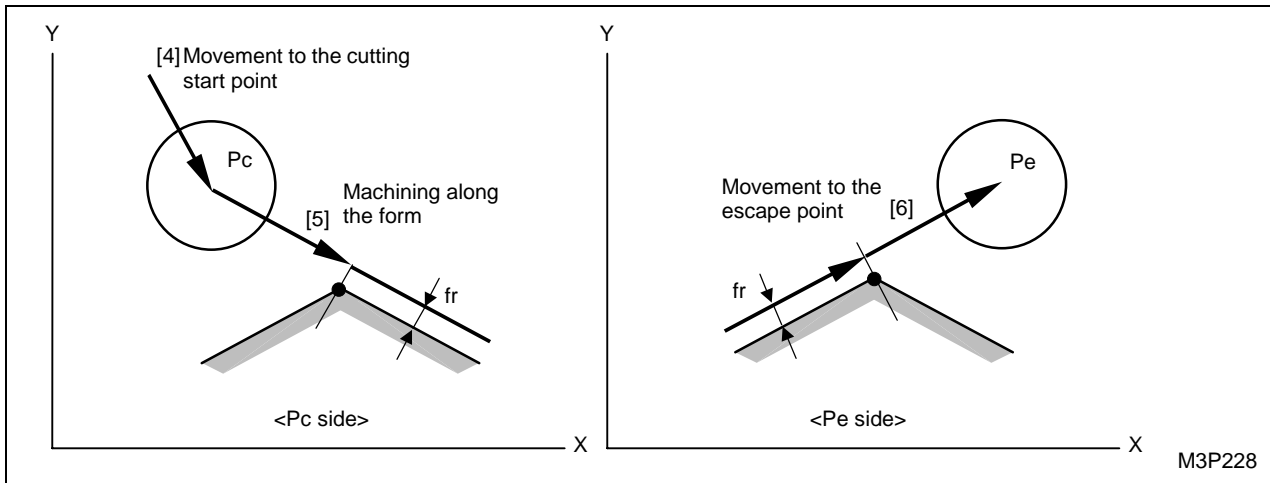
Pe: Escape point to be automatically established

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

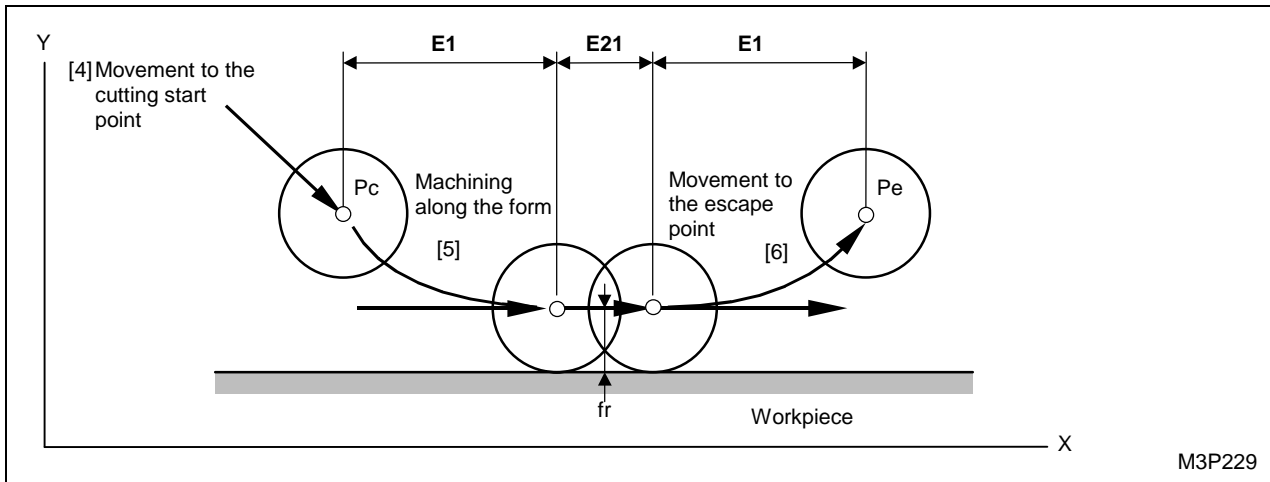
Note 1: The feedrate on tool path [3] is dependent upon the ZFD (Z-axis feed) in the tool sequence.

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

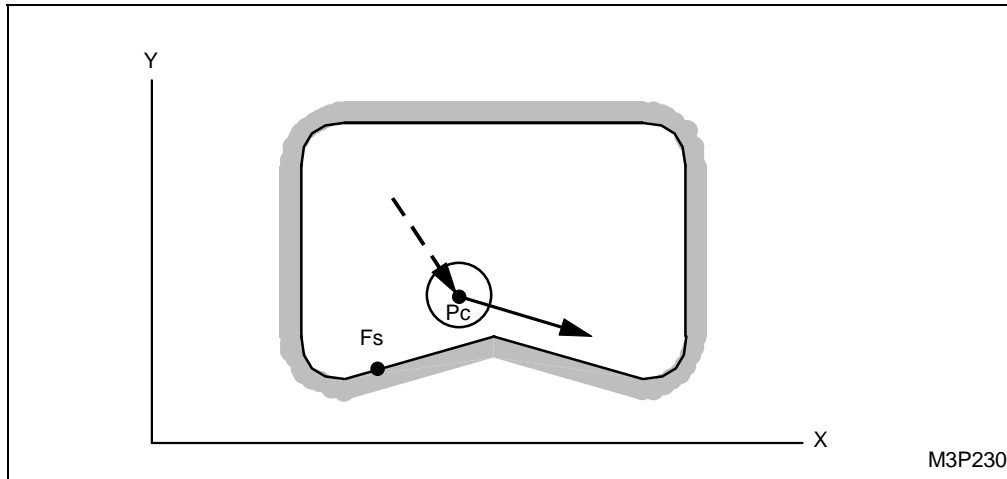
Note 3: See Subsection 4-8-6, "Precautions in line machining."

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).

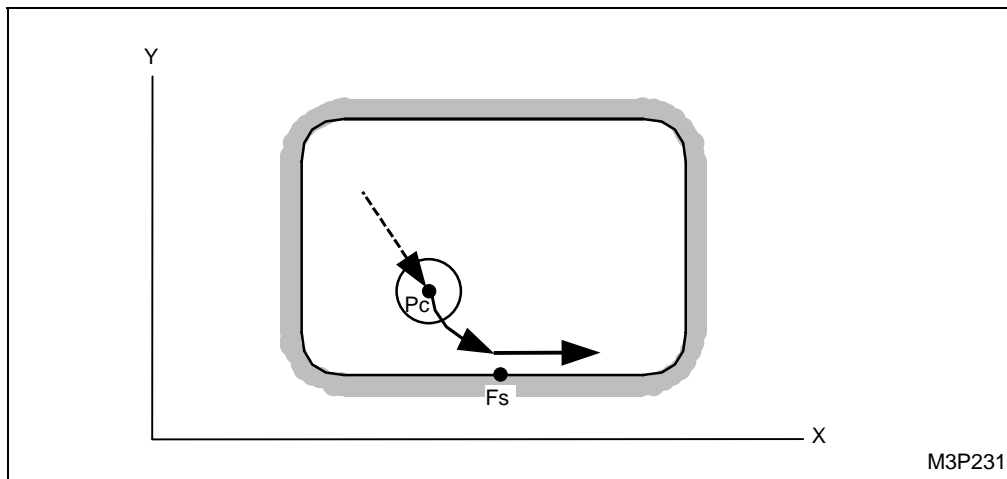
When ? is displayed in the items APRCH-X, -Y

- Form having a convex point:



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

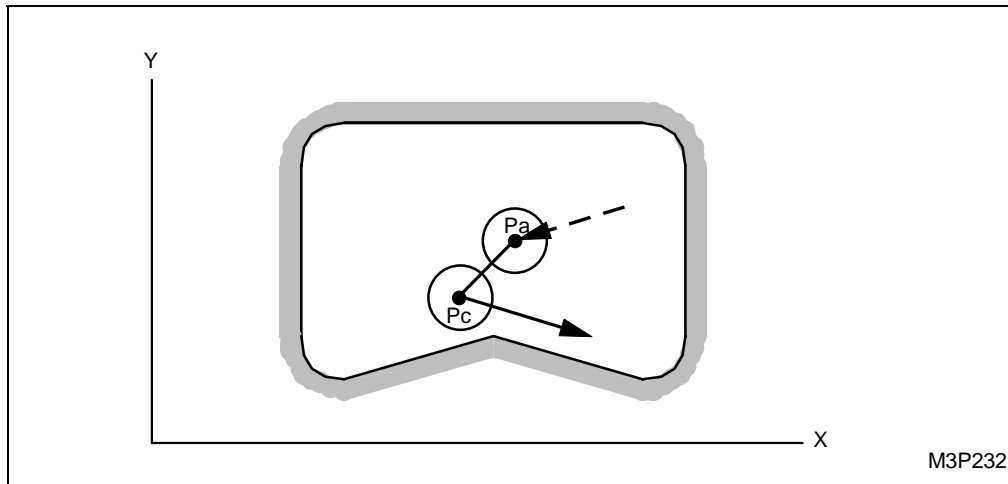
- Form having no convex point:



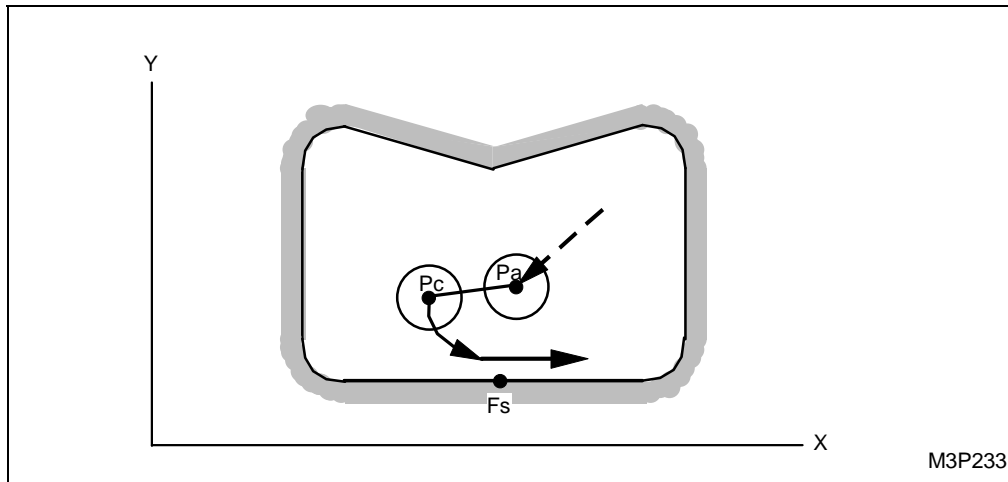
Cutting is started from the start point (F_s) entered in the shape sequence.

When the data is entered in the items APRCH-X, -Y

- 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 menu key **AUTO SET**, 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

4-8-4 Line machining tool sequence data

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.

SN _o .	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				
	↑	↑	↑	↑	↑	↑	↑	↑		↑	↑	↑	↑
	1	2	3	4	5	6	7	8		9	9	10	10

◆: Not necessary to be set here.

1. Tool designation: TOOL

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

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	CENTER DRILL						
---------	----------	-------------------	-----------------	-----------------	--	--	--	--	--	--

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.

2. Nominal diameter of tool: NOM-φ

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

3. Tool identification code

A code should be selected out of the menu to identify those tools which are of identical type (having 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 menu key **HEAVY TOOL** 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. Priority number: No.

Enter the priority number for prior machining and subsequent machining. There are three following entry methods.

A. Prior machining priority number

To be set by means of numeric keys. The number is displayed in yellow.

Setting range (1 to 99).

B. Subsequent machining priority number

To be set by means of numeric keys after having reversed the display of **DELAY PRIORITY** by selecting its menu key.

The characters are displayed in reversed status.

Setting range (1 to 99).

C. Without entry (ordinary machining)

The priority number is not entered.

For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

5. Coordinates X and Y of the approach point: APRCH-X, APRCH-Y

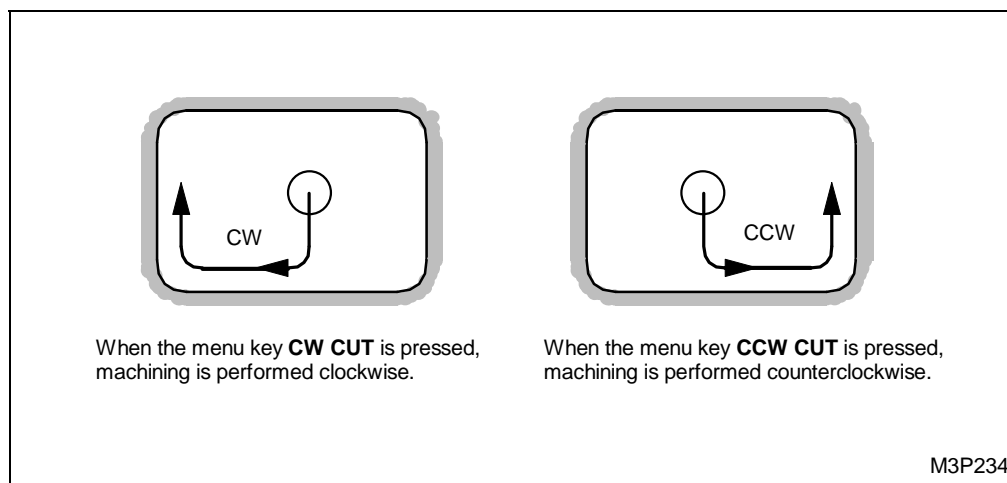
Enter an X, Y coordinates of the position at which a tool is to cut in axially.

With the menu key **AUTO SET** selected, ? is displayed. After the tool path check is performed, ? will automatically change over to the coordinates of a cutting start point. (Refer to tool path by each unit.)

6. Machining method: TYPE

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

CW CUT 	CCW CUT 								
---	--	--	--	--	--	--	--	--	--



7. Z-axis feedrate: ZFD

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

CUT	RAPID							
G01	G00							

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

M3P235

8. Cutting stroke-Z: DEP-Z

In roughing, a maximum axial cutting stroke in one cycle is entered. With the menu key **AUTO SET** selected, a smaller value is entered, either the data SRV-Z 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-Z, SRV-Z and FIN-Z, both in the machining unit. (For calculation formula, see Subsection 4-8-6, "Precautions in line machining.")

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

A spindle speed and a cutting feedrate are entered.

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

10. M-codes: M

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

4-8-5 Line machining shape sequence data

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 4-9-7.

4-8-6 Precautions in line machining

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

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

- A. Cutting start position along the Z-axis
- B. Type of routing through approach points
- C. Type of escape along the Z-axis after machining

[Basic tool path]

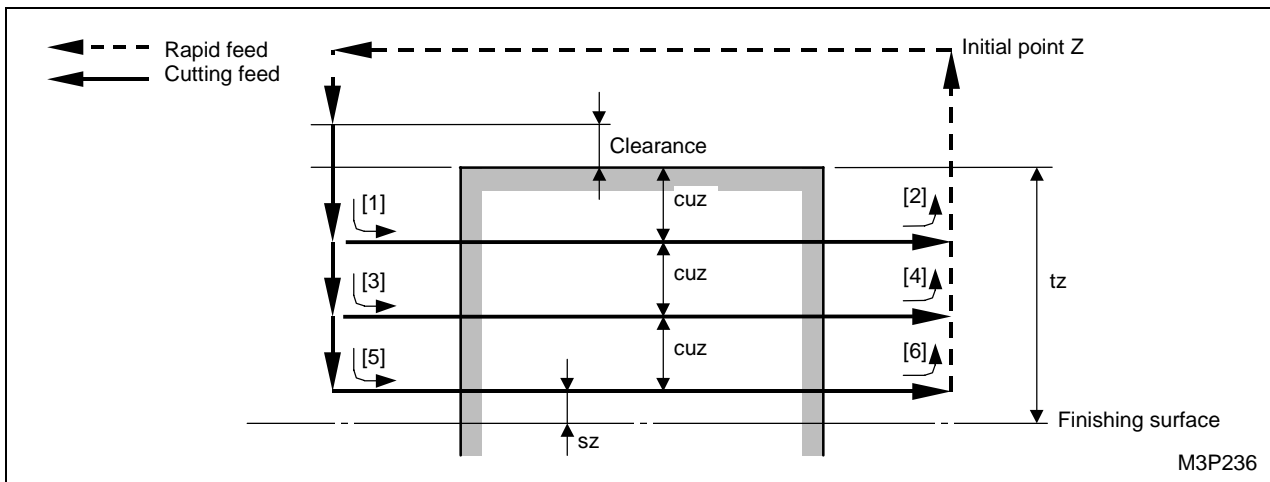


Fig. 4-18 Basic tool path

cuz: Cutting depth in the Z-axis per pass

Calculation of cuz:

$$\text{cuz} = \frac{\text{tz} - \text{sz}}{\text{n}}$$

$$\text{n} = \frac{\text{tz} - \text{sz}}{\text{cz}}$$

tz: Z-axial cutting allowance SRV-Z to be entered in the machining unit

sz: Z-axial finishing allowance FIN-Z to be entered in the machining unit

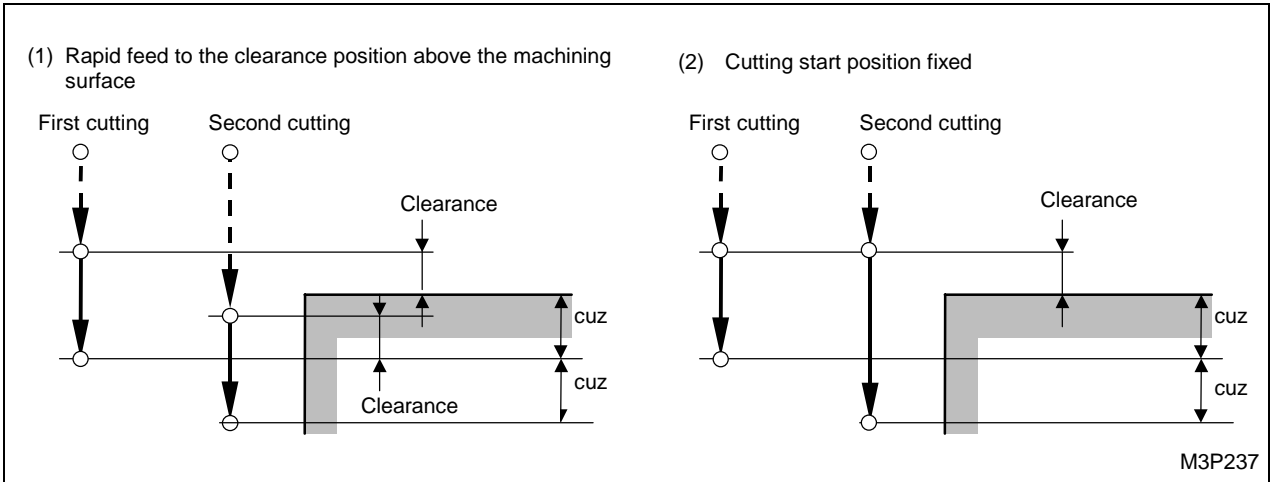
cz: Z-axial cutting depth DEP-Z to be entered in the tool sequence

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

A. Cutting start position along the Z-axis

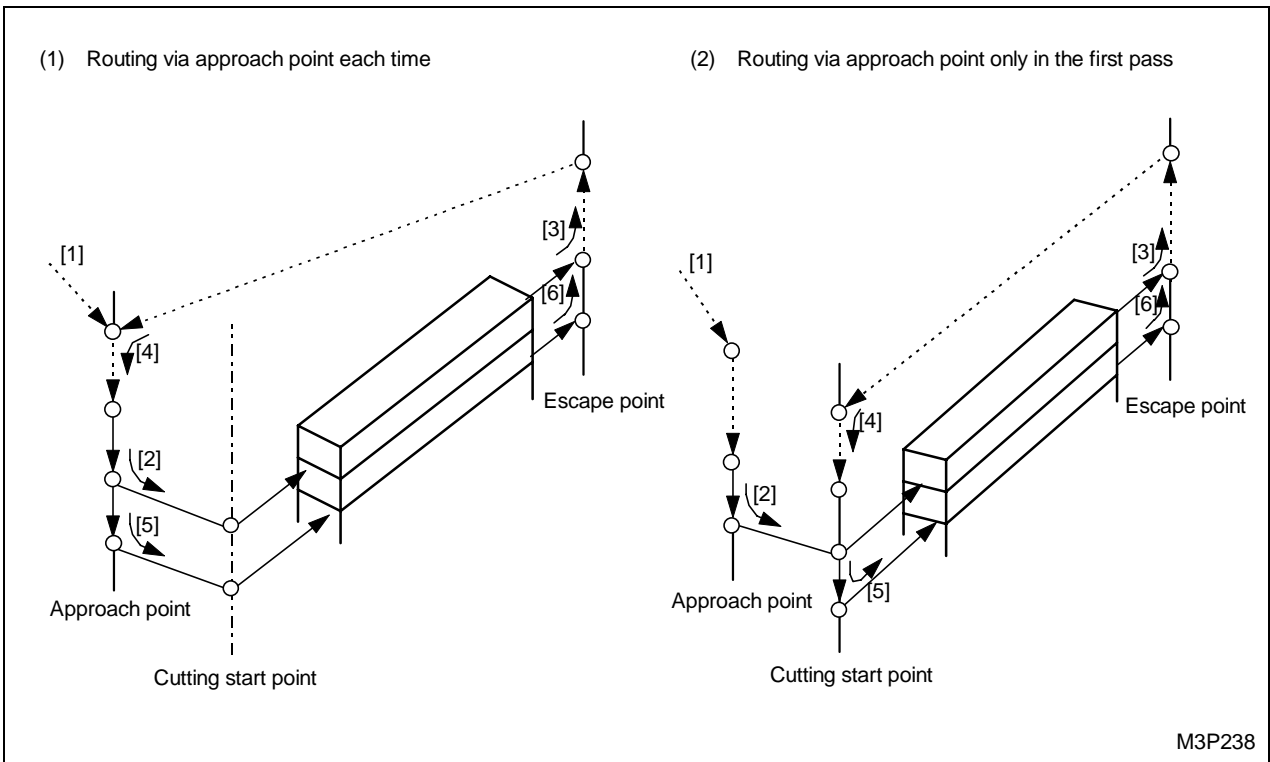
Select one of the following two types:



The bold codes represent parameter addresses.

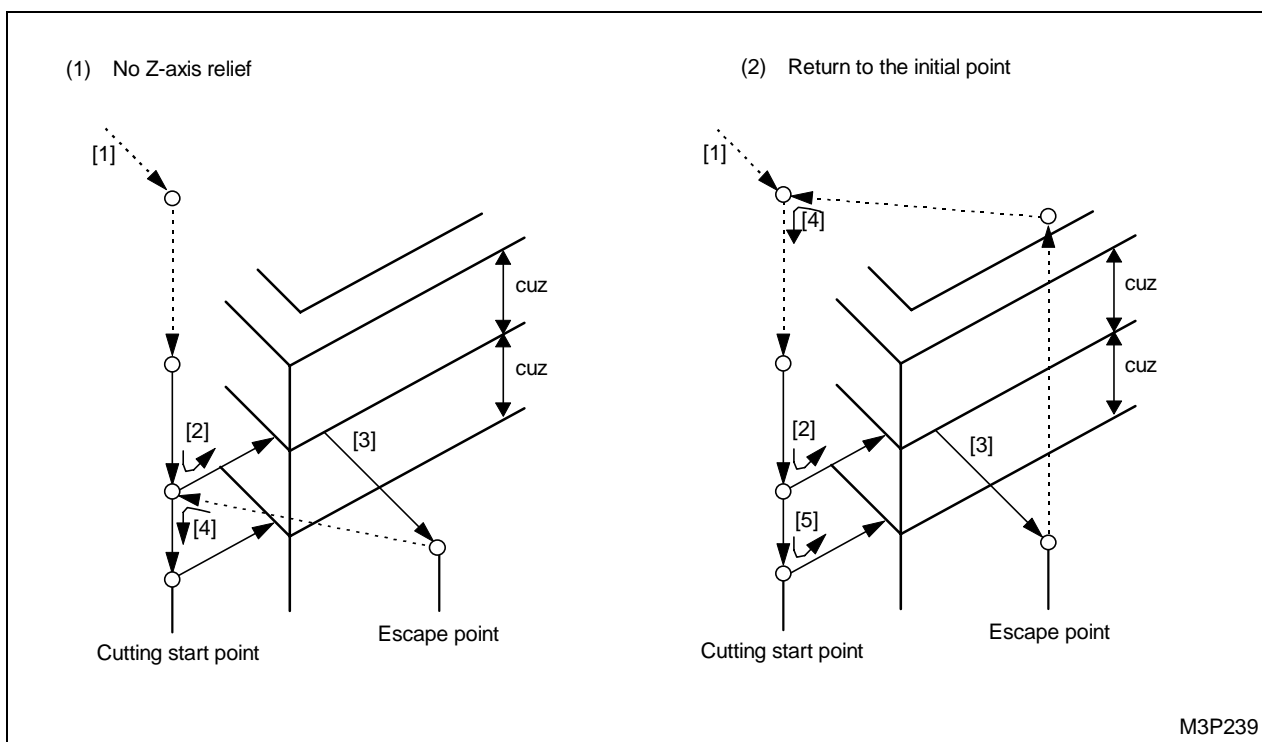
B. Type of routing via approach point

Select one of the following two types:



C. Type of escape along the Z-axis after machining

Select one of the following two types:



M3P239

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)

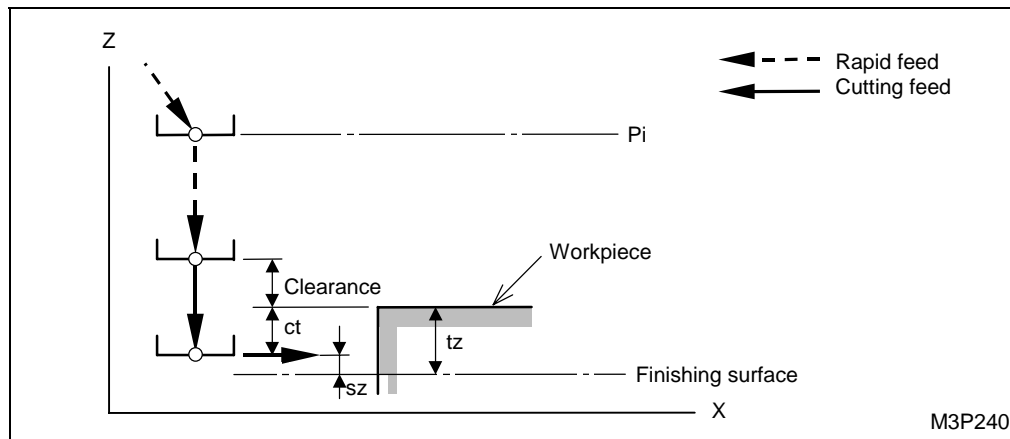
1: No escape along the Z-axis -- (1)

Note 1: Both A and B can be used for all line-machining units, whereas C can only be used for inside linear and outside linear machining units.

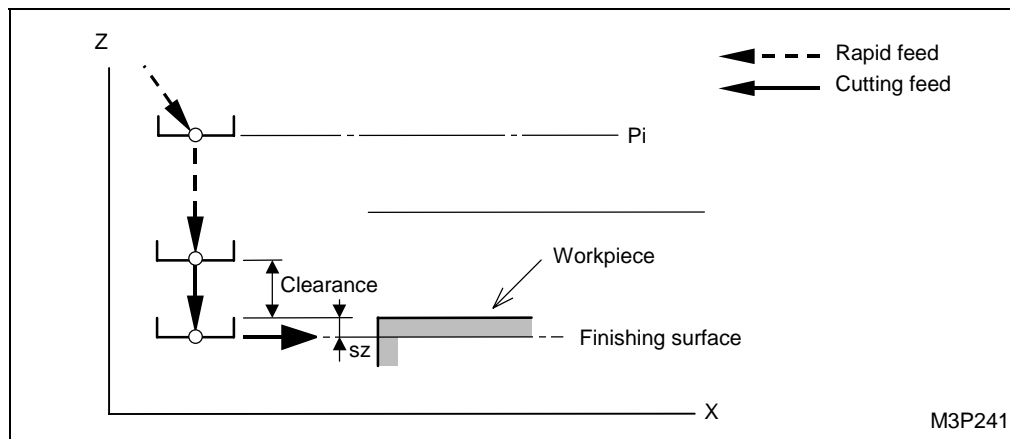
Note 2: The tool path shown at basic tool path above is selected automatically for machining units that are not subject to the selection of the parameter **E95**.

2. Detail tool path of a Z-axis cut-in

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

ct: Z-axial cutting depth DET-Z to be entered in the tool sequence

tz: Z-axial cutting allowance SRV-Z to be entered in a machining unit

sz: Z-axial finishing allowance FIN-Z to be entered in a machining unit

Note 1: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 2: 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 3: 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 sequence.
- The machining unit is either outside linear or inside linear machining.

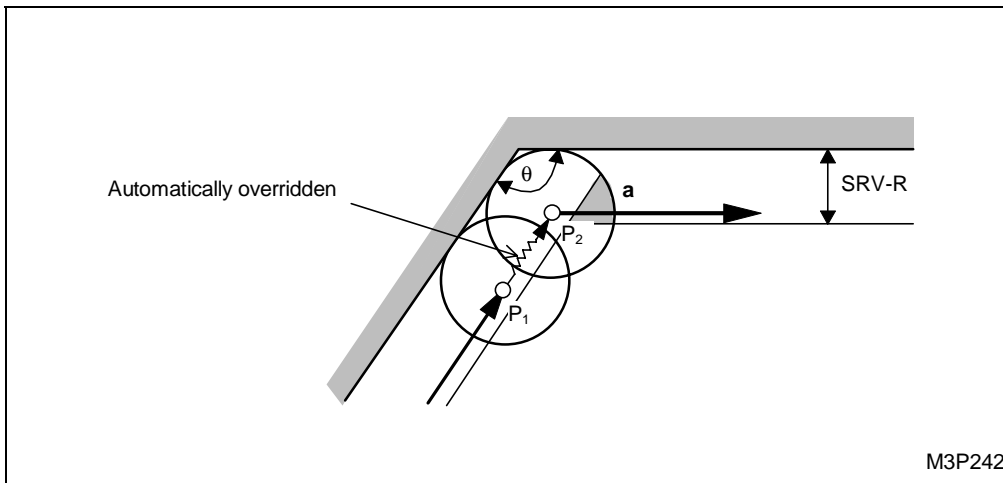
3. Other precaution on tool path

If shape data, tool data or parameter are modified after the automatic determination of X and Y 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.

4-8-7 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.

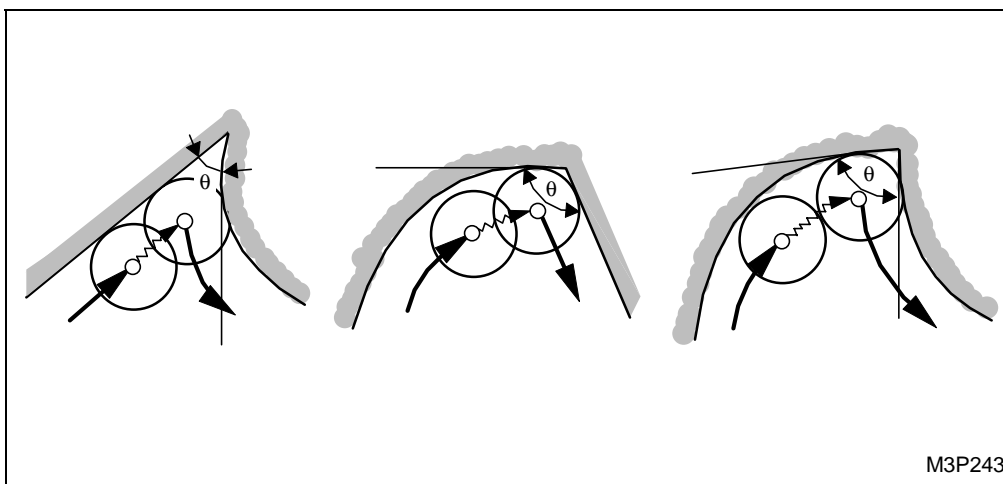
1. Operating conditions



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 overridden.

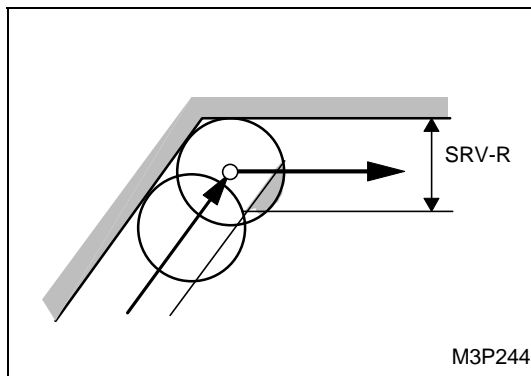
This override, however, will be valid only when all of the following requirements (A, B, C) are satisfied:

- A. Inside corner angle θ is equal to or less than the value entered in the parameter **E25** (with $\theta \leq \mathbf{E25}$).



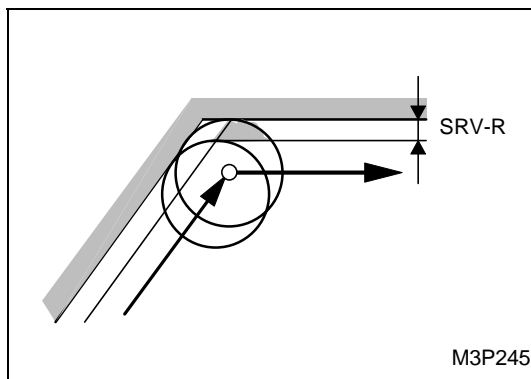
- B. A radial cutting allowance is equal to or less than the value entered in the parameter **E23**
($SRV-R \leq \text{tool diameter} \times E23/100$)

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



- C. A radial cutting allowance is equal to or less than the value entered in the parameter **E24**
($SRV-R \leq \text{tool diameter} \times E24/100$)

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



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.

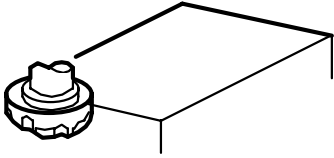
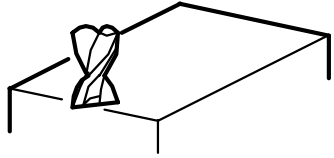
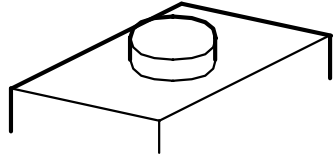
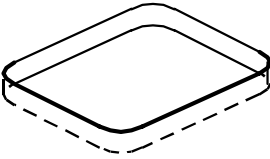
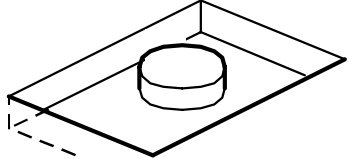
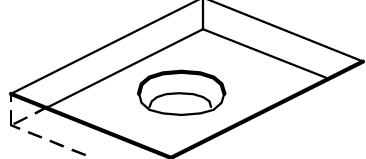
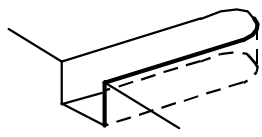
3. Override rate

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.

4-9 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.

4-9-1 Types of face machining units

1. Face milling	2. End milling-top	3. End milling-step
		
4. Pocket milling	5. Pocket milling-mountain	6. Pocket milling-valley
		
7. End milling-slot		

M3P246








Fig. 4-19 Types of face machining unit

4-9-2 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	WPC	OFFSET	END	SHAPE CHECK	>>>
-------------------	------------------	------------------	---------	-------------------	-----	--------	-----	----------------	-----

(2) Pressing on the menu key **FACE MACH-ING** displays the following machining unit menu.

FCE MILL 	TOP EMILL 	STEP 	POCKET 	PCKT MT 	PCKT VLY 	SLOT 			
---	--	---	---	--	---	---	--	--	--

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

4-9-3 Unit data, automatic tool development and tool path

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	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
1	FCE MILL	999.999	999.999	9	◆	999.999	◆						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	FCE MILL						◆						
F2	FCE MILL						◆	◆					

Remark 1: Data in unit represent a maximum input value.

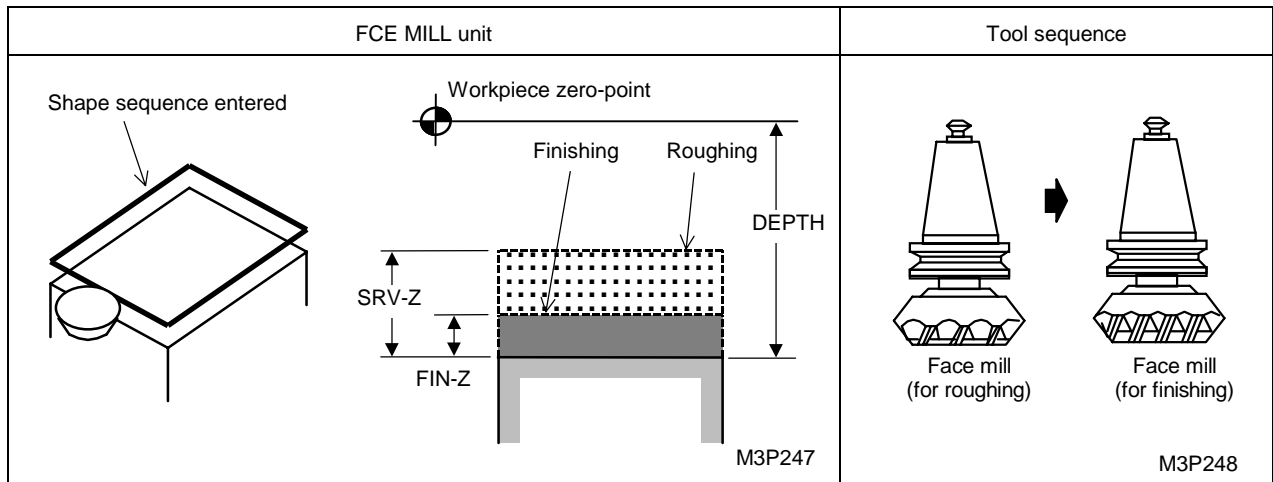
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, face mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z and on FIN-Z.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.

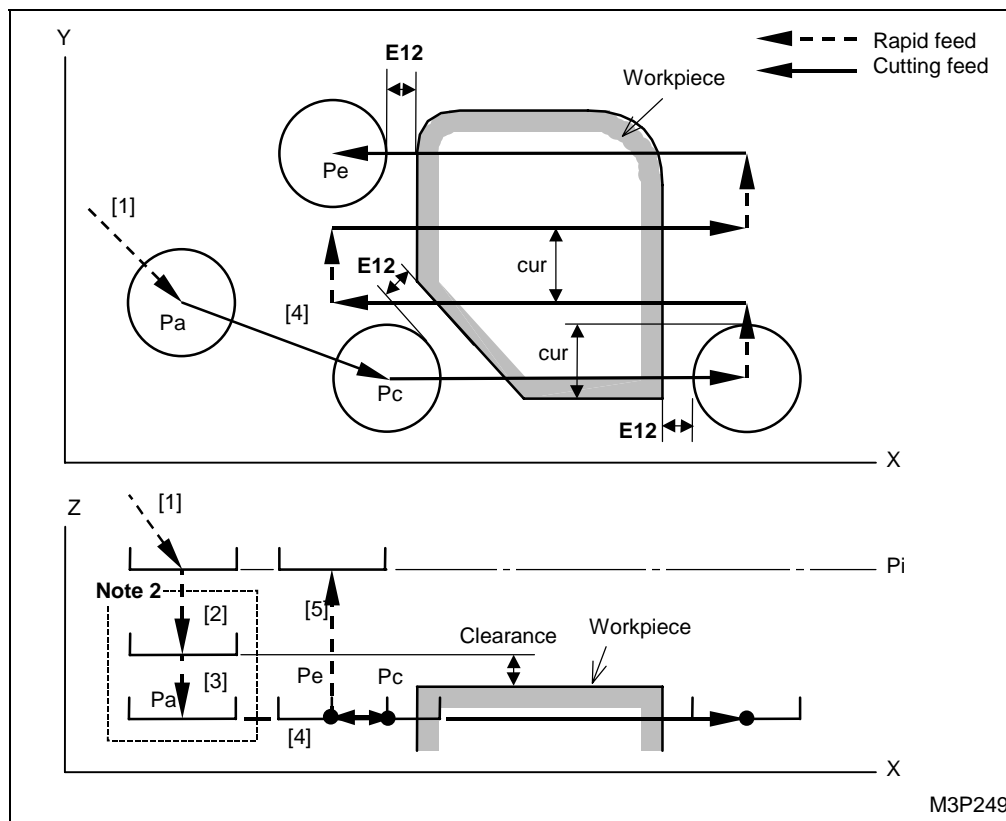


BTM: A bottom roughness code is selected out of the menu.

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

B. Tool path

When the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

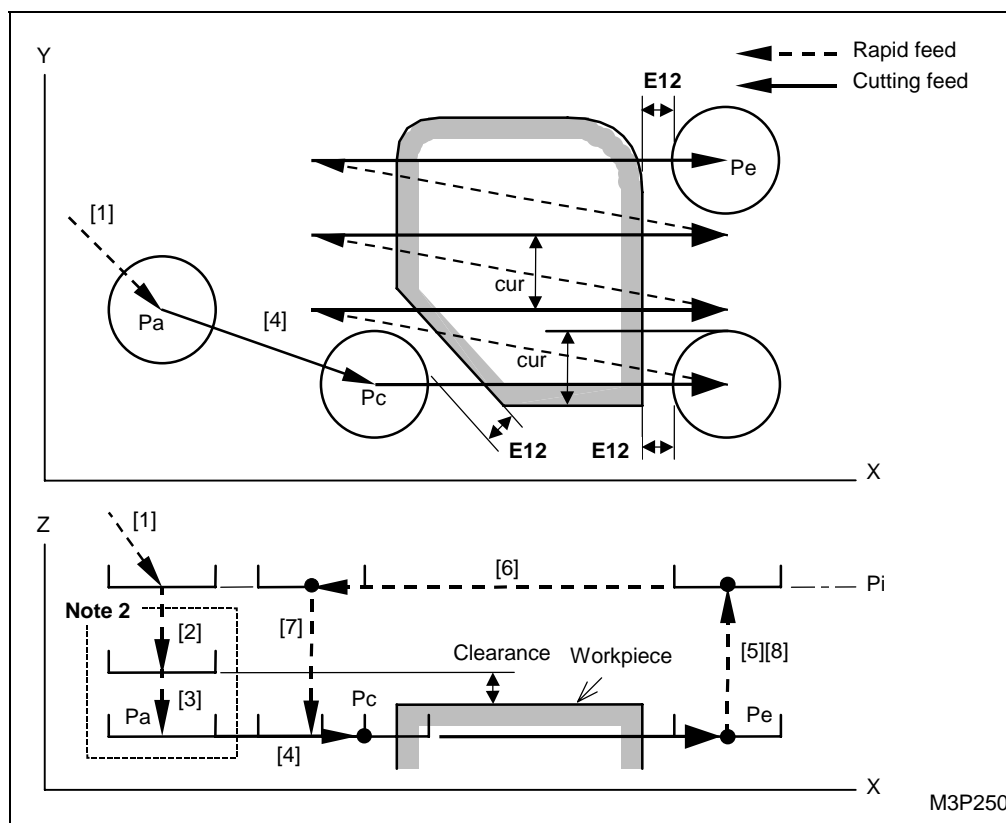
cur: Radial cutting depth to be determined by the data WID-R in the tool sequence

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

<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 the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

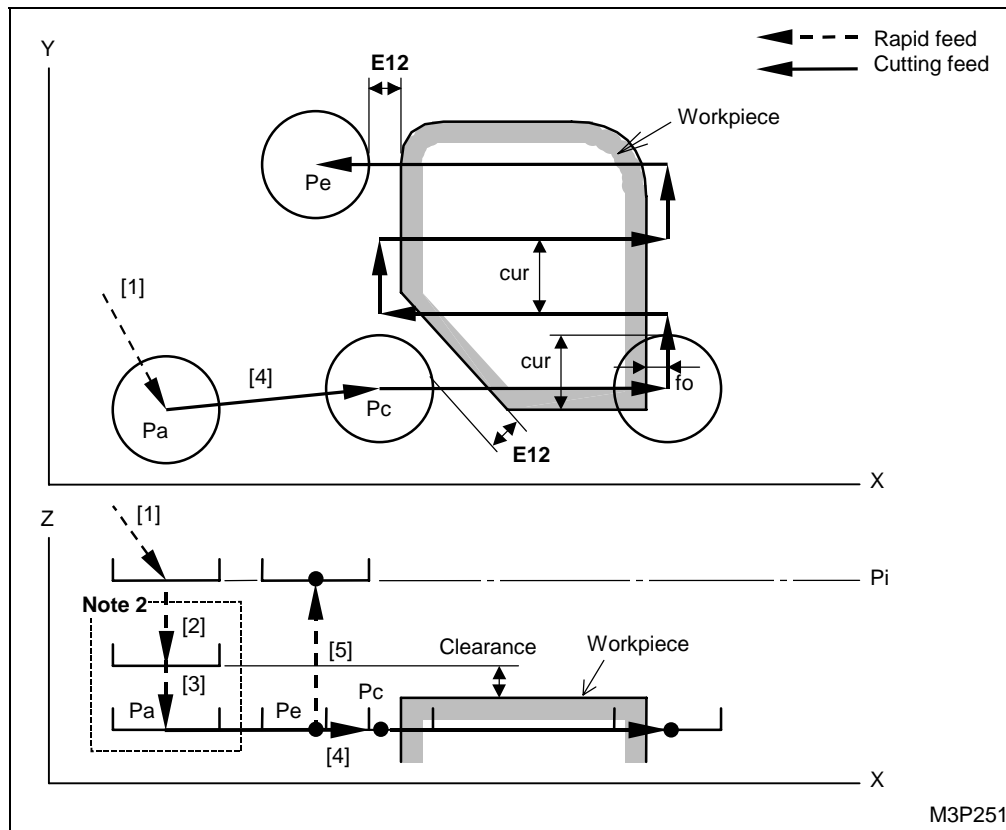
cur: Radial cutting depth to be determined by the data WID-R in the tool sequence

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

<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 the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

fo: Form-offsetting clearance

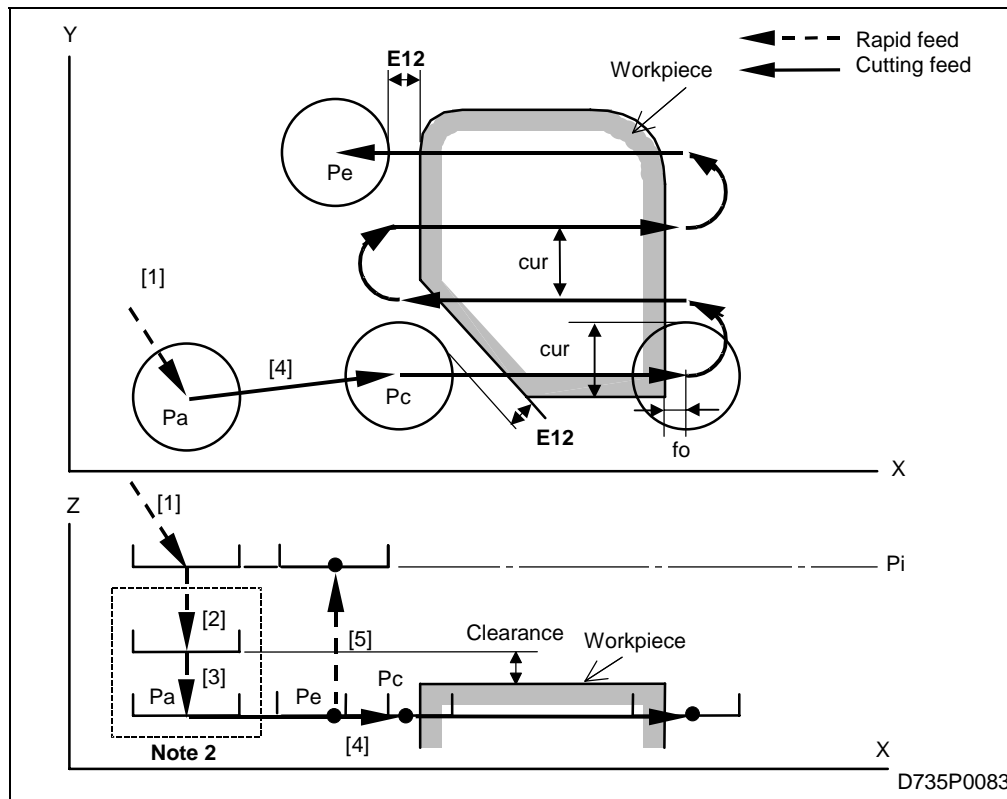
$$fo = \text{tool diameter} \times \frac{\mathbf{E15}}{10}$$

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

<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 the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

fo: Form-offsetting clearance

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

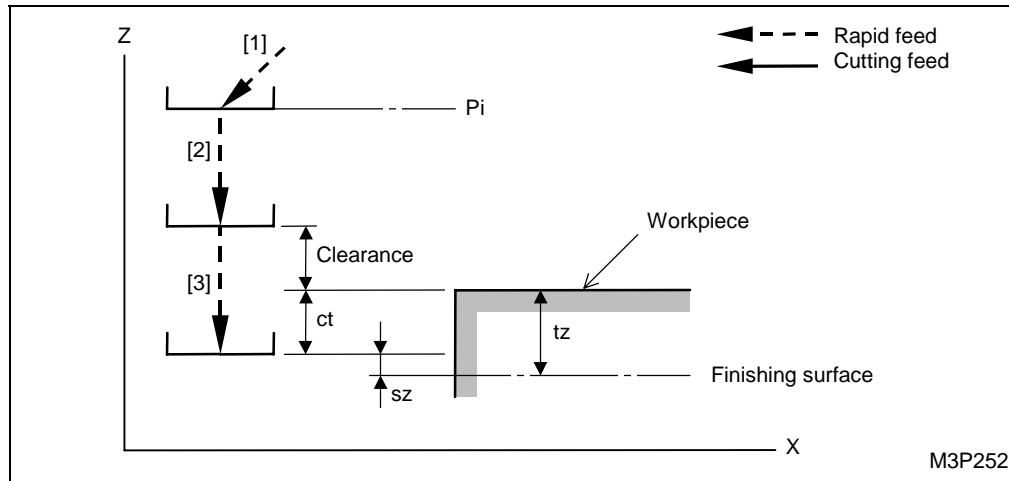
<Route on which tool is to move>

- [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.

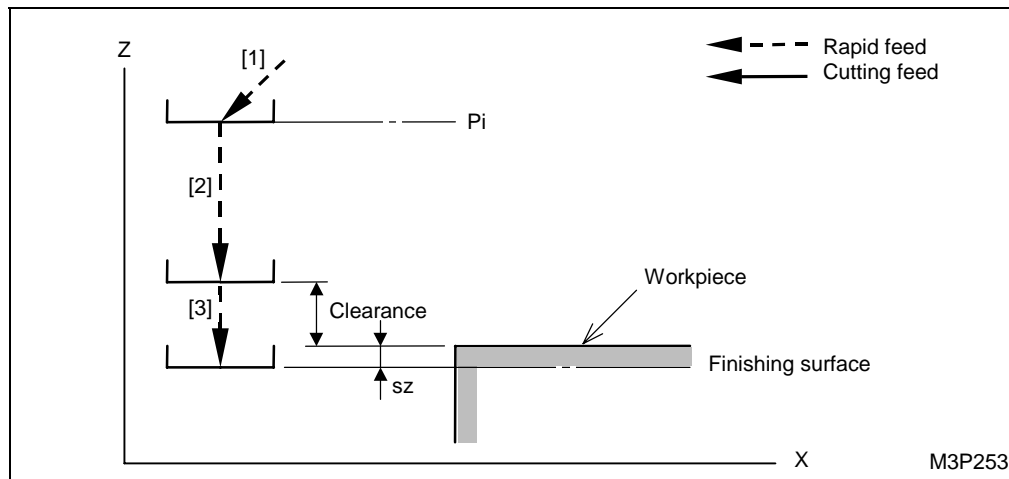
Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate of cutting start point will be entered in these items.

Note 2: Detail of the Z-axial tool path. (See Subsection 4-9-5, "Precautions in face machining.")

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

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

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

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	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
1	TOP EMIL	999.999	999.999	9	◆	999.999	◆						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL							◆					

Remark 1: Data in unit represent a maximum input value.

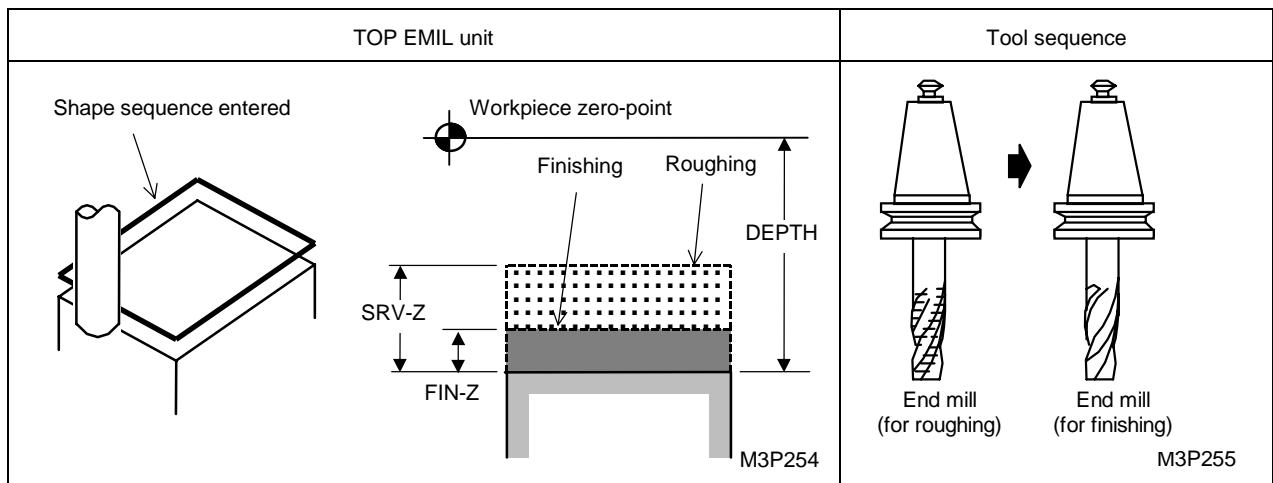
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z and on FIN-Z.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.

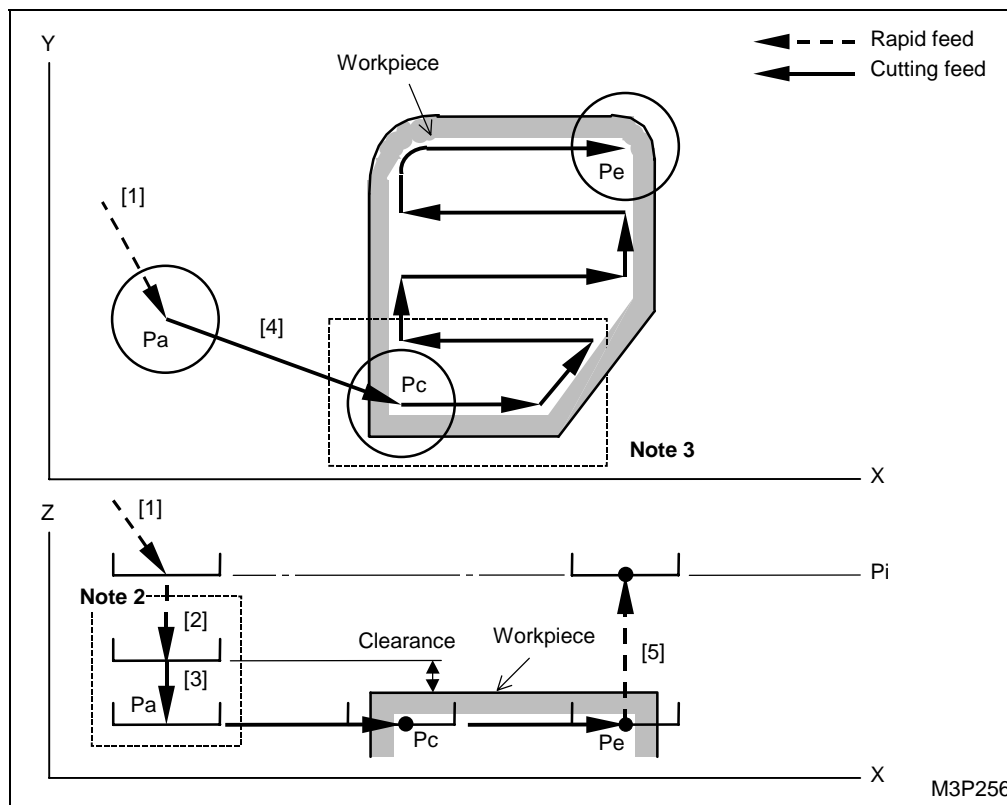


BTM: A bottom roughness code is selected out of the menu.

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

B. Tool path

When the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

<Route on which tool is to move>

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

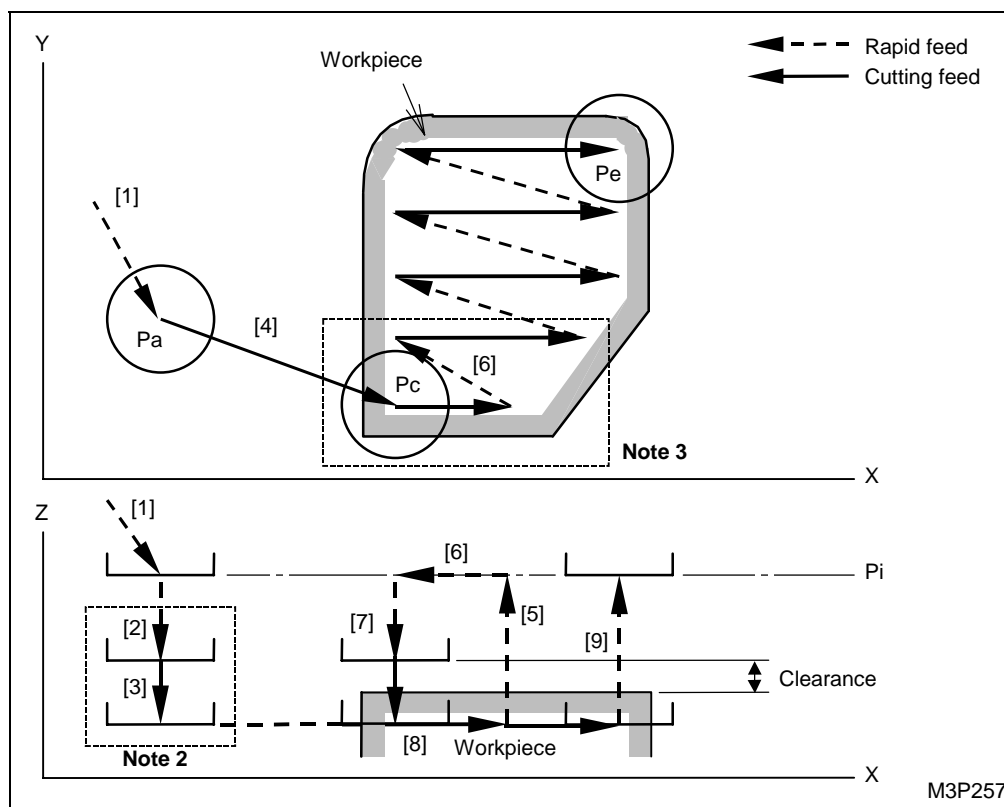
[2] The tool moves at a rapid feedrate to the clearance position.

[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data ZFD 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 the menu key **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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

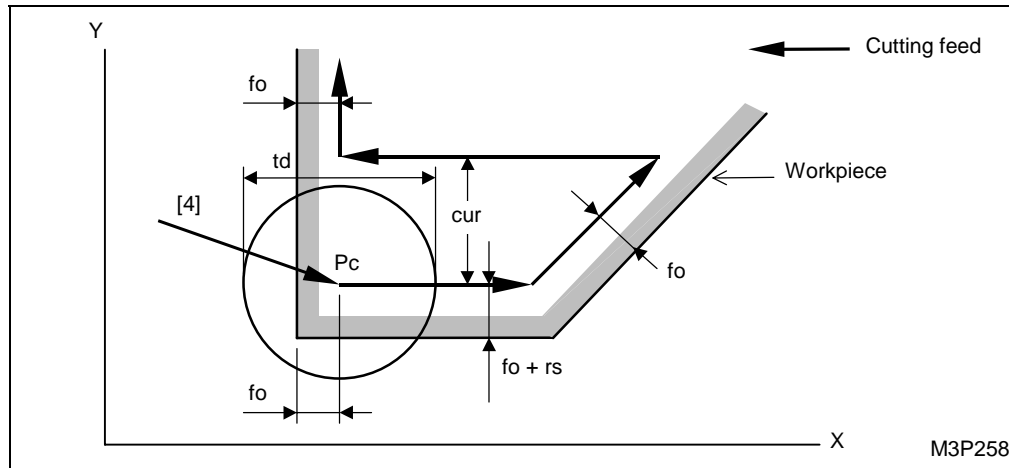
<Route on which tool is to move>

- [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 ZFD 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-X, -Y by pressing the menu key **AUTO SET**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinates of cutting start point will be entered in these items.

Note 2: See Subsection 4-9-5 "Precautions in face machining."

Note 3: Detail description of tool path



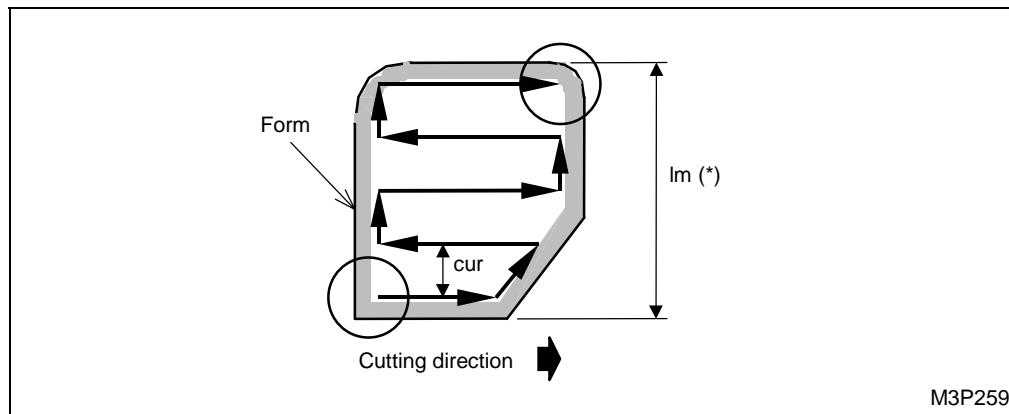
td: Diameter of a tool

fo: Form offset clearance dependent upon both td and parameter **E13**

$$fo = td \times \frac{E13}{10}$$

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	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
1	STEP	999.999	999.999	9	9	999.999	999.999						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL												

Remark 1: Data in unit represent a maximum input value.

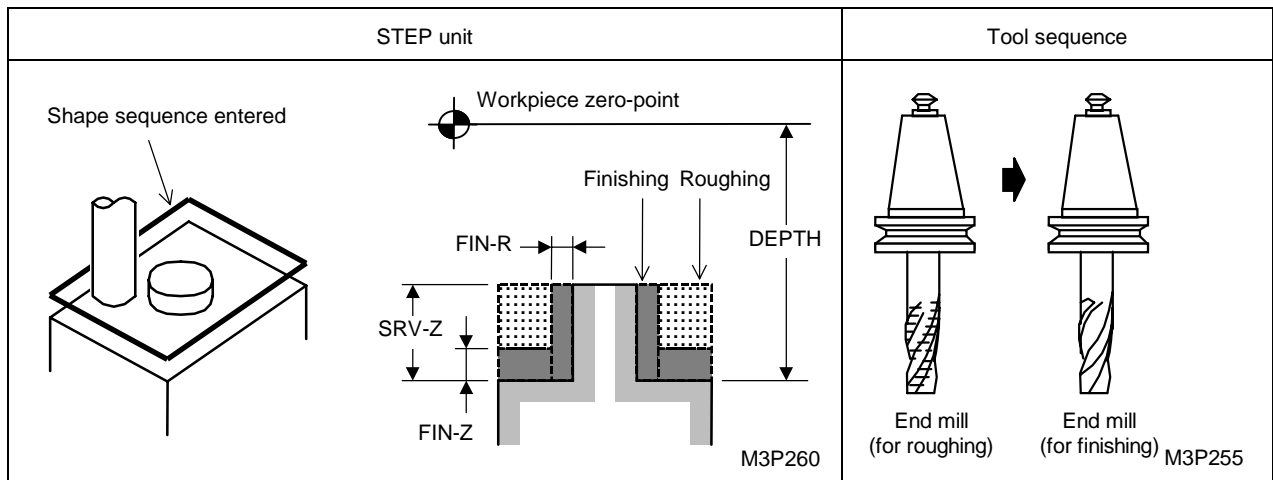
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.



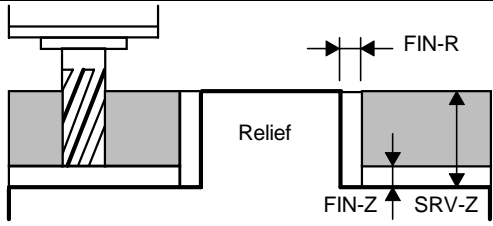
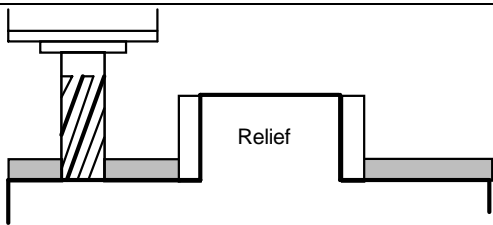
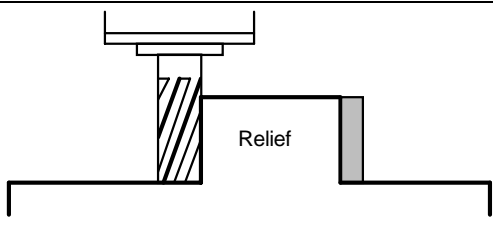
BTM: A bottom roughness code is selected out of the menu.

WAL: A wall roughness code is selected out of the menu.

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

B. Machining sequence

End milling-step is performed in the following order.

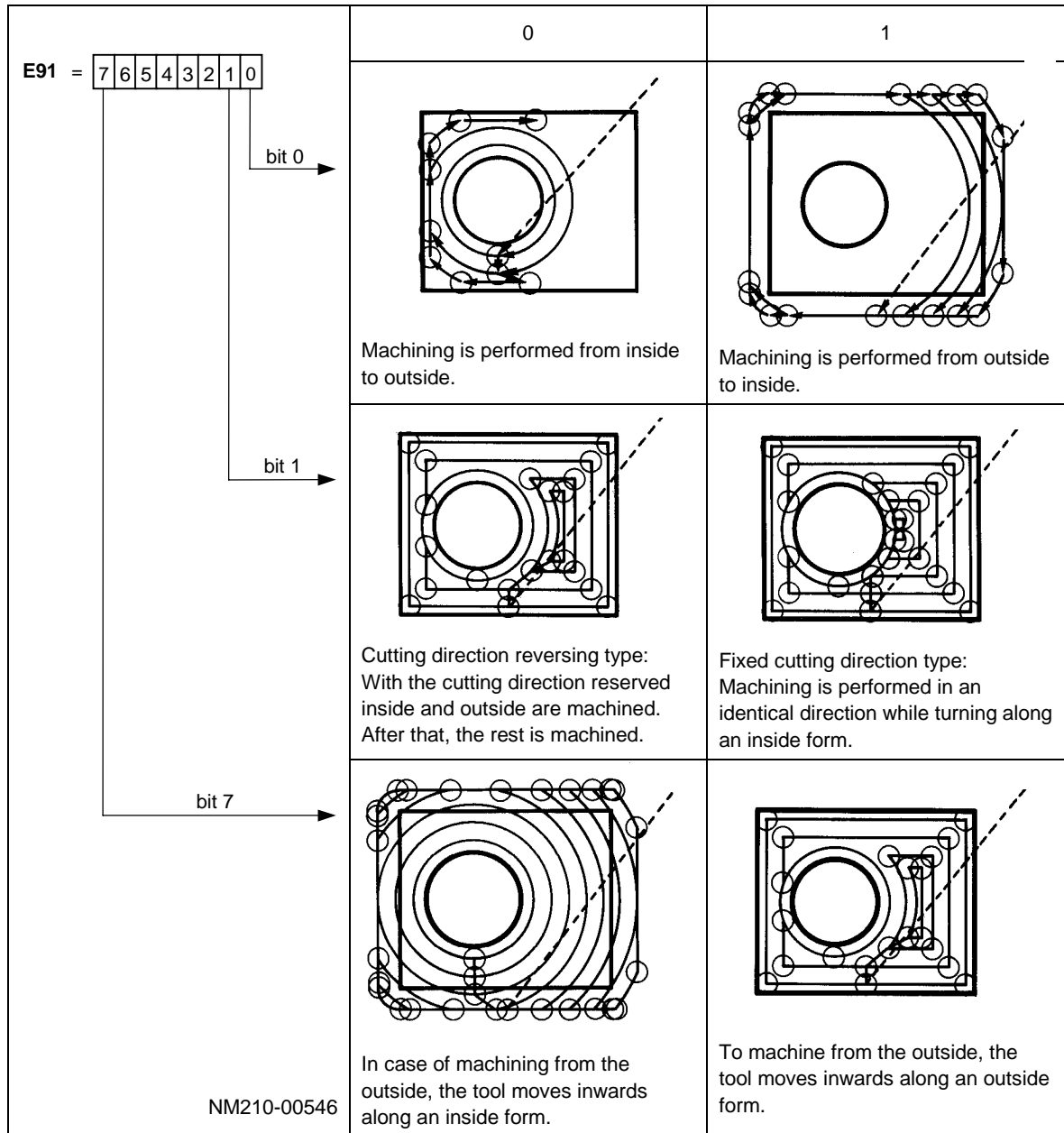
Roughing		Machining is performed with the end mill developed in the tool sequence R1. With $SRV-Z = FIN-Z$, this machining is not performed.
Finishing	Bottom 	Machining is performed with the end mill developed in the tool sequence F1. With $FIN-Z = 0$, this machining is not performed.
	Wall 	Machining is performed with the end mill developed in the tool sequence F1. With $FIN-R = 0$, this machining is not performed.

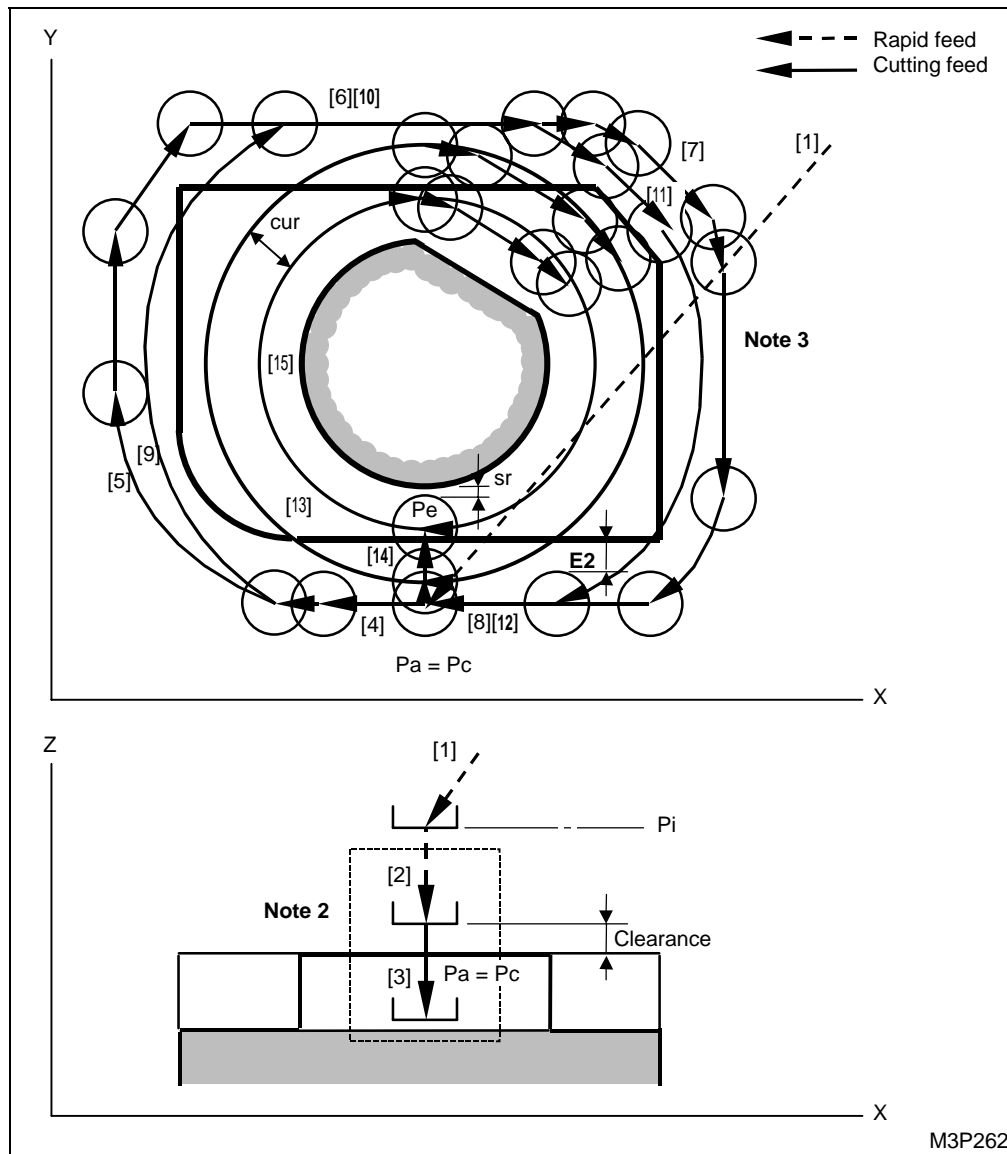
M3P261

C. 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 Parameter List.



D. Tool pathRoughing or bottom finishing

The bold codes represent parameter addresses.

Pa, Pc: Approach point to be determined by the data APRCH-X, -Y 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

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

Note 2: See Subsection 4-9-5, "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 ZFD 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.

E. Finishing

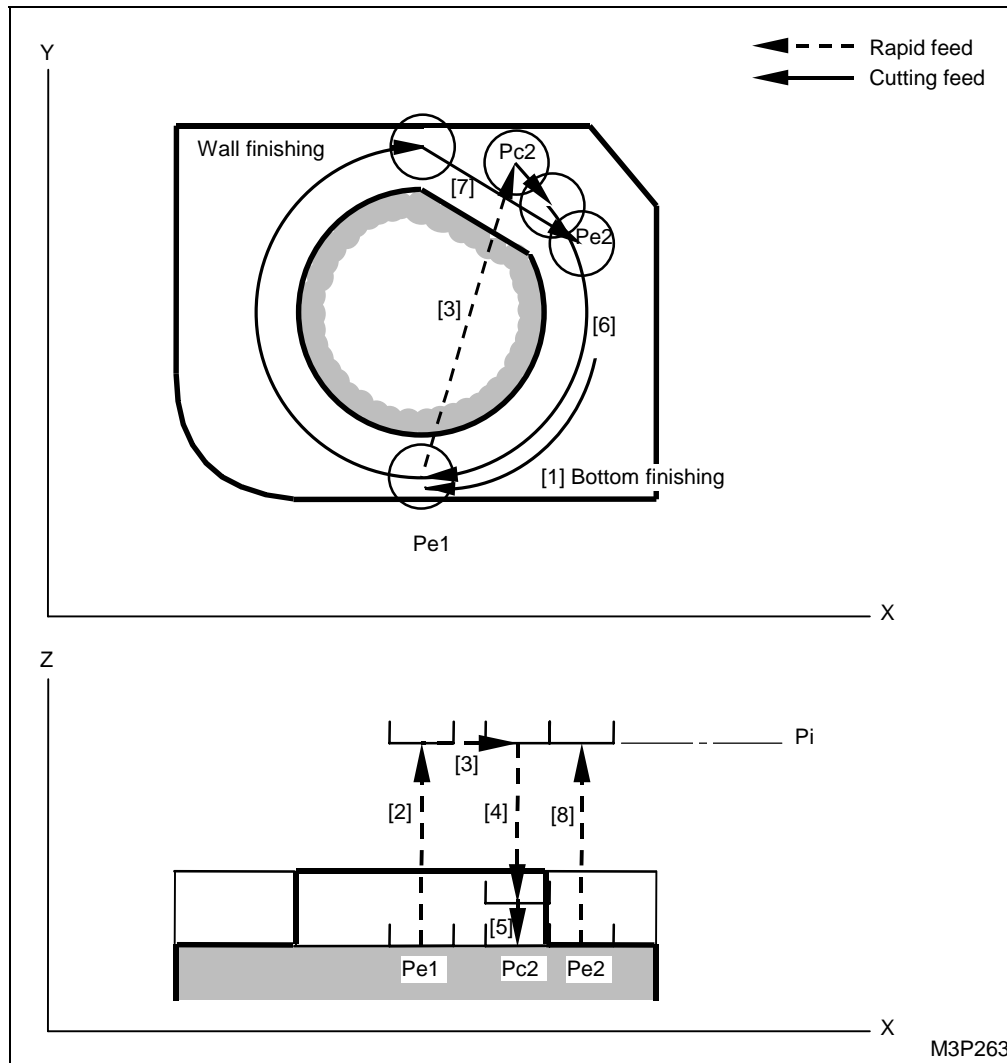
Finishing is performed, based on the entered data FIN-Z and FIN-R.

Bottom finishing is performed, with $0 < \text{FIN-Z}$.

Wall finishing is performed, with $0 < \text{FIN-R}$.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-X, -Y 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Note 2: See Subsection 4-9-5, "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**.

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	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R	INTER-R	CHMF				
1	POCKET	999.999	999.999	9	9	999.999	999.999	999.999	99.9				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL							◆					
3	CHAMFER							◆	◆				

Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

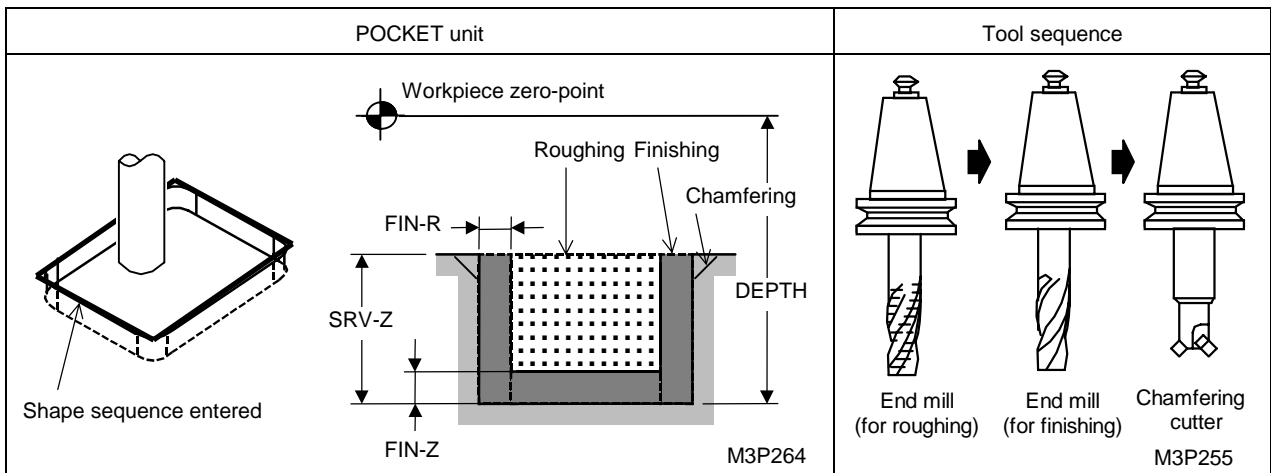
Remark 3: In this unit, end mills and a chamfering cutter are automatically developed. Instead of the chamfering cutter, a centering drill can be used for chamfering.

Remark 4: In the tool sequence, a maximum of up to three tools are automatically developed, based on SRV-Z, FIN-Z, FIN-R and CHMF.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.

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



BTM: A bottom roughness code is selected out of the menu.

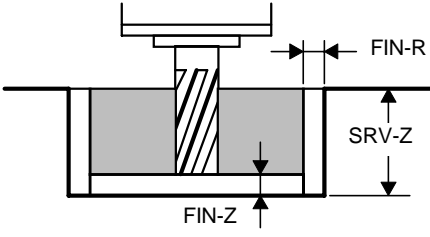
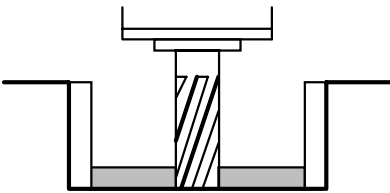
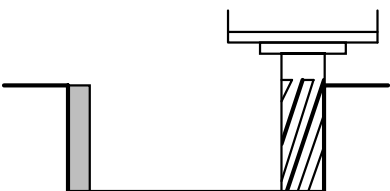
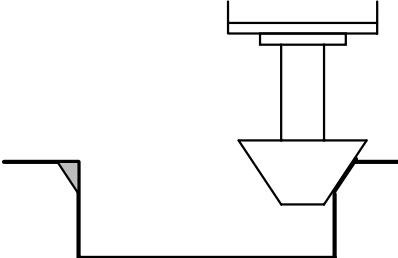
WAL: A wall roughness code is selected out of the menu.

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

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

B. Machining sequence

Pocket milling is performed in the following order.

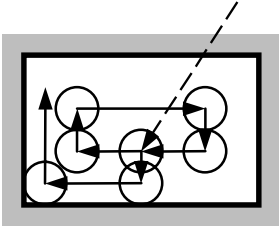
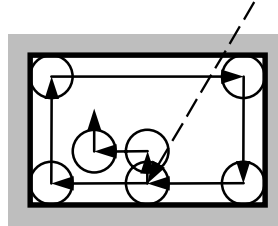
<p>Roughing</p>		<p>Machining is performed with the end mill developed in the tool sequence R1. With SRV-Z = FIN-Z, this machining is not performed.</p>	
<p>Finishing</p>	<p>Bottom</p>		<p>Machining is performed with the end mill developed in the tool sequence F1. With FIN-Z = 0, this machining is not performed.</p>
	<p>Wall</p>		<p>Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.</p>
<p>Chamfering</p>		<p>Machining is performed with the chamfering cutter or centering drill in the tool sequence. With CHMF = 0, this machining is not performed.</p>	

M3P265

C. Machining pattern

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

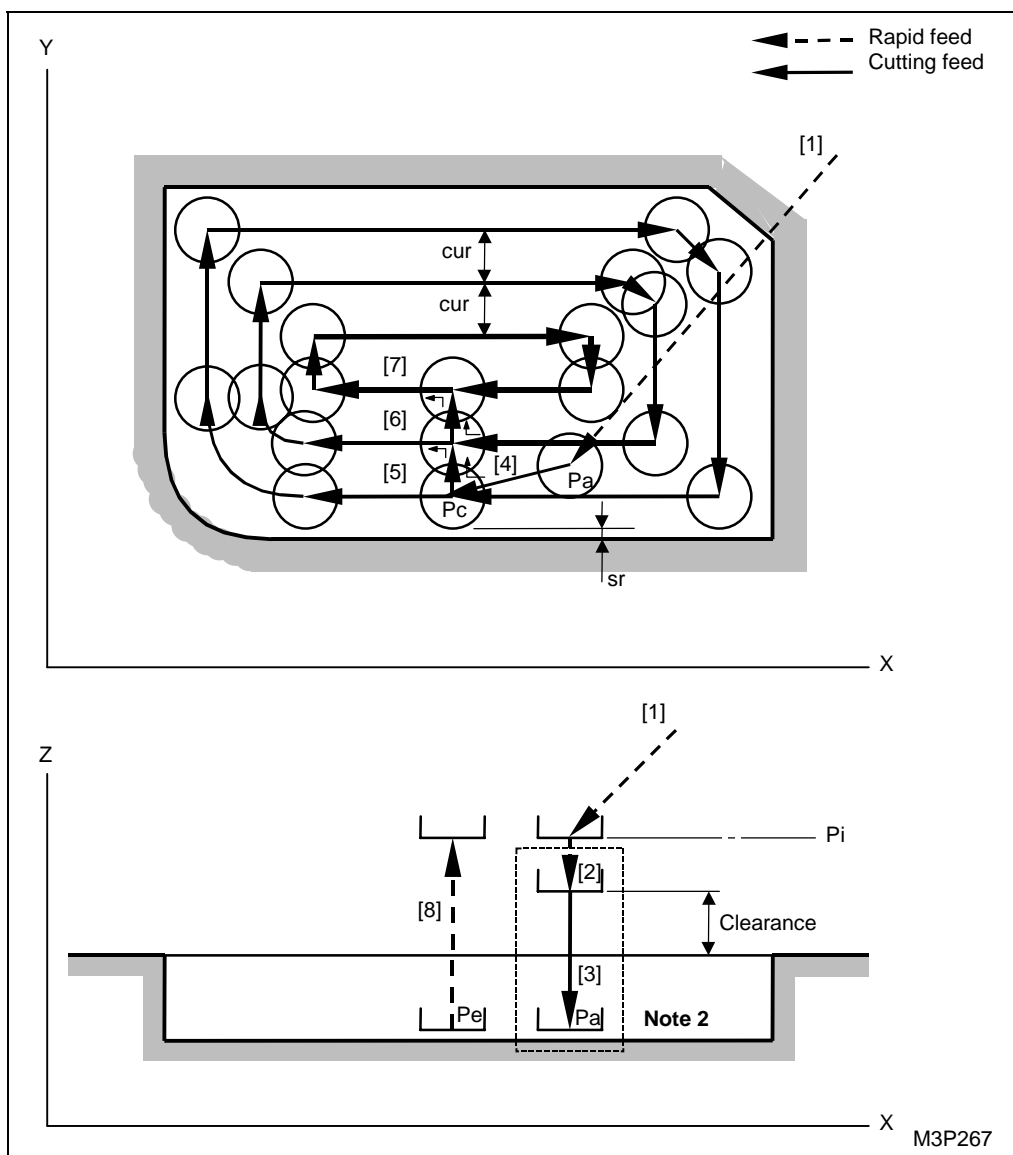
- Either 0 or 1 is entered in the related bit accordingly

<p>E92 = <table border="1" style="display: inline-table; vertical-align: middle;"> <tr> <td>7</td><td>6</td><td>5</td><td>4</td><td>3</td><td>2</td><td>1</td><td>0</td> </tr> </table></p> <p style="margin-left: 150px;">bit 0</p>	7	6	5	4	3	2	1	0	<p style="text-align: center;">0</p>  <p>Machining is performed from inside to outside.</p>	<p style="text-align: center;">1</p>  <p>Machining is performed from outside to inside.</p>
7	6	5	4	3	2	1	0			

M3P266

D. Tool path

Machining from outside (roughing or bottom finishing)



The bold codes represent parameter addresses.

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

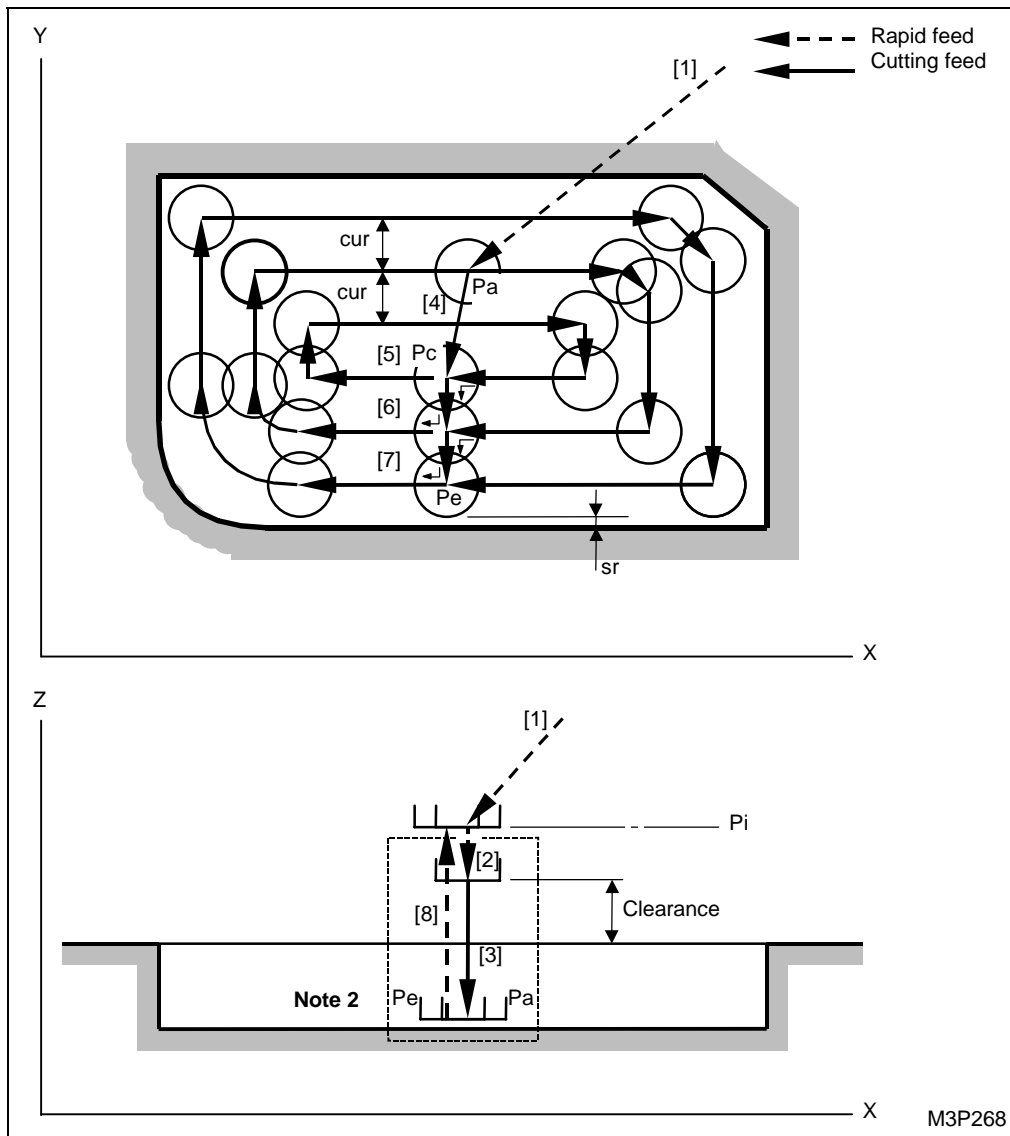
Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, the tool is positioned directly at the cutting start point and operation [2] and [3] are performed. In this case, the coordinates of cutting start point will be entered in these items.

Note 2: See Subsection 4-9-5, "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 ZFD 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)



The bold codes represent parameter addresses.

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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 4-9-5, "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 ZFD 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.

Wall finishing

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

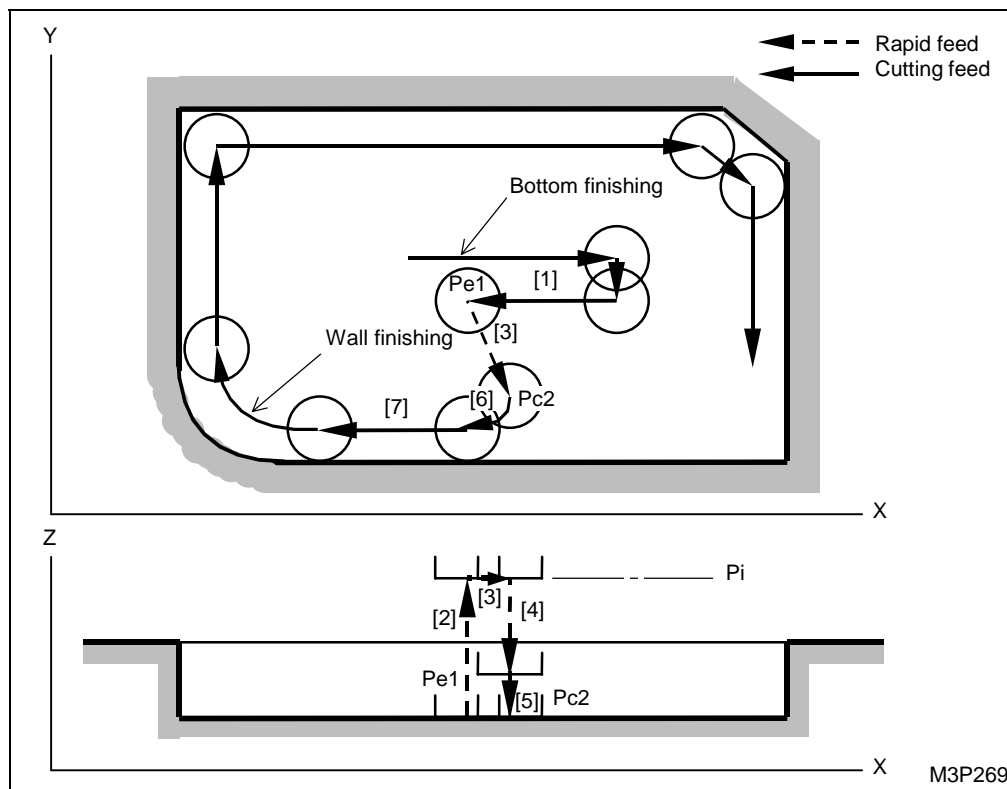
E. Finishing

Finishing is performed, based on the entered values of FIN-Z and FIN-R.

- Bottom finishing is performed, with $0 < \text{FIN-Z}$.
- Wall finishing is performed, with $0 < \text{FIN-R}$.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-X, -Y 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Note: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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	DEPTH	SRV-Z		BTM	WAL	FIN-Z	FIN-R					
1	PCKT MT	999.999	999.999		9	9	999.999	999.999					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL												

Remark 1: Data in unit represent a maximum input value.

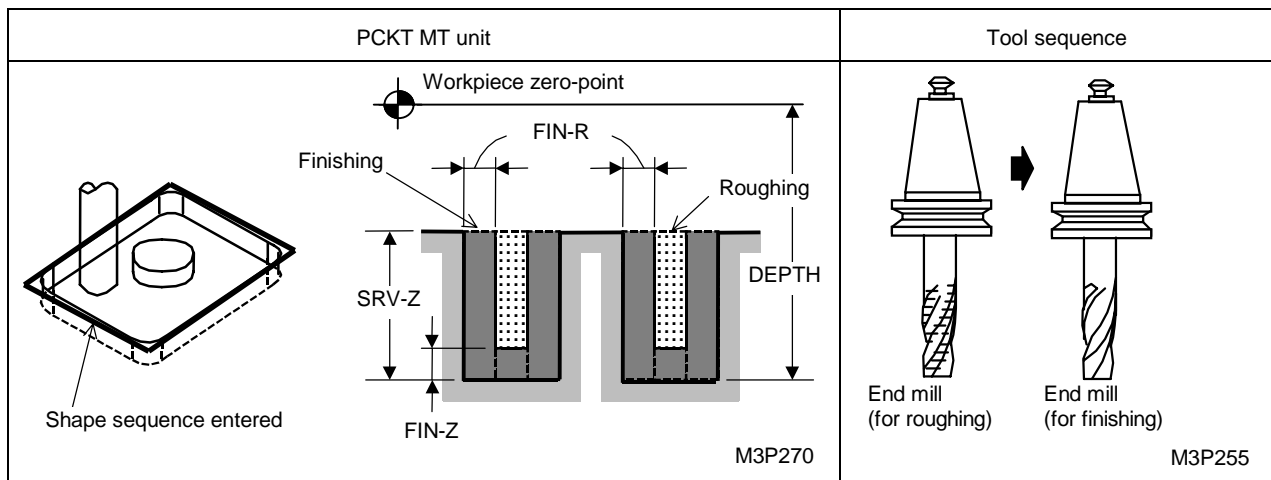
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.



BTM: A bottom roughness code is selected out of the menu.

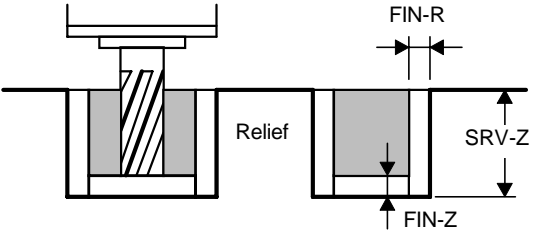
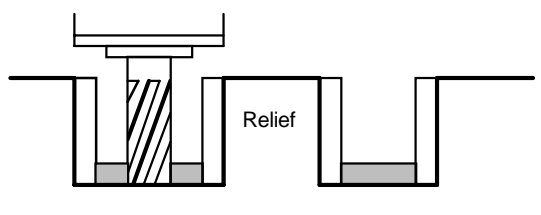
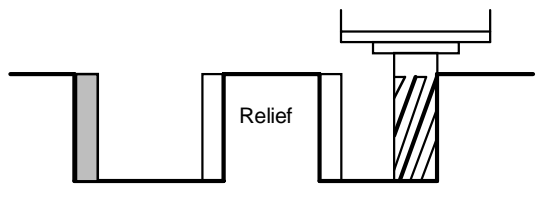
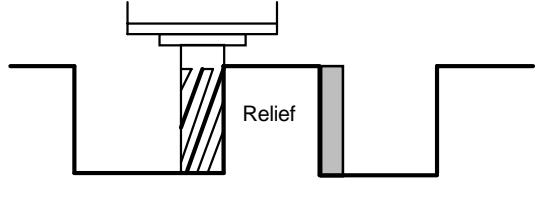
WAL: A wall roughness code is selected out of the menu.

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

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

B. Machining sequence

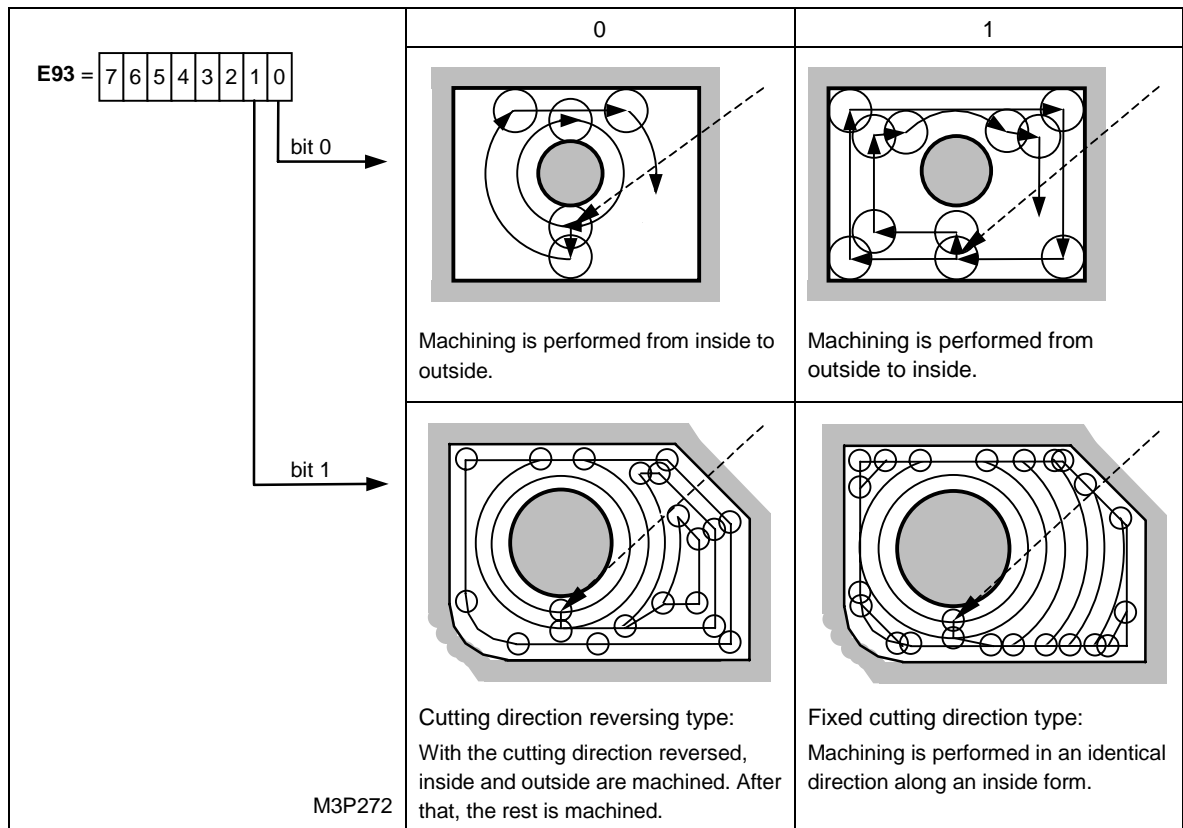
Pocket milling is performed in the following order.

Roughing		<p>Machining is performed with the end mill developed in the tool sequence R1. With $SRV-Z = FIN-Z$, this machining is not performed.</p>	
Finishing	Bottom		<p>Machining is performed with the end mill developed in the tool sequence F1. With $FIN-Z = 0$, this machining is not performed.</p>
	Outer wall		<p>Machining is performed with the end mill developed in the tool sequence F1. With $FIN-R = 0$, this machining is not performed.</p>
	Inner wall		<p>M3P271</p>

C. Machining pattern

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

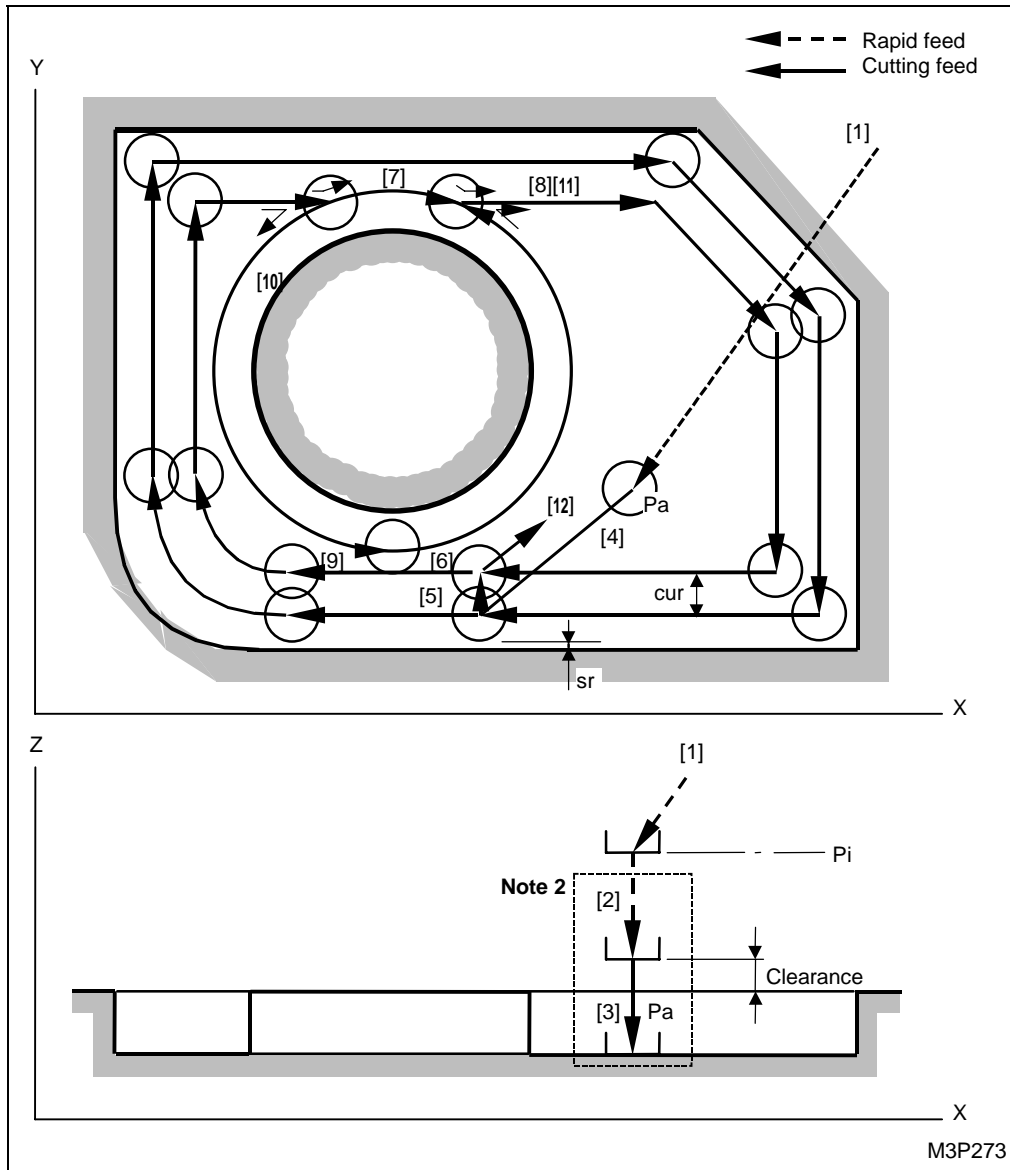
- Enter 0 or 1 in the related bit accordingly.



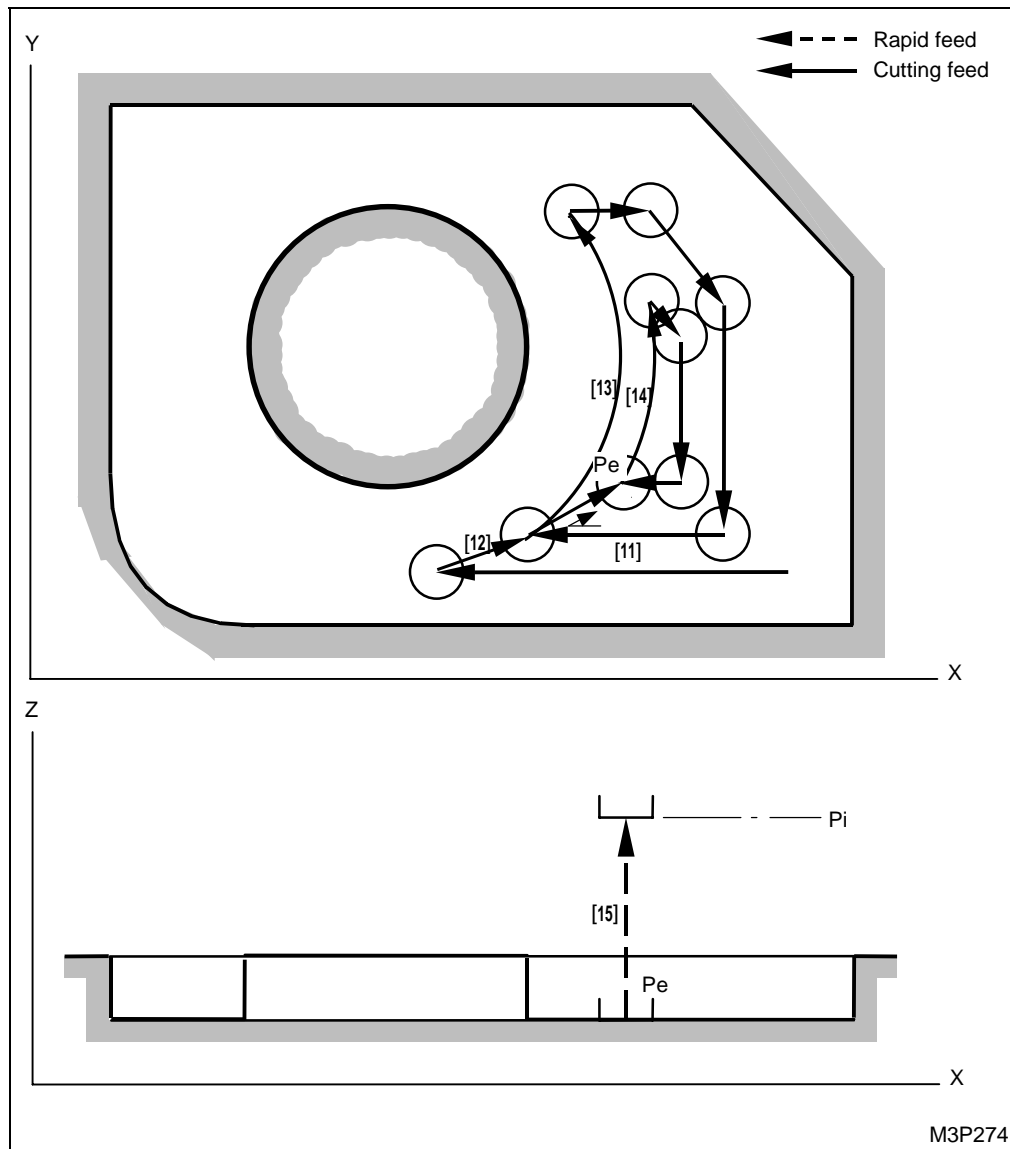
D. Tool path

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-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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 4-9-5, "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 ZFD 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.

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.

E. Finishing

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

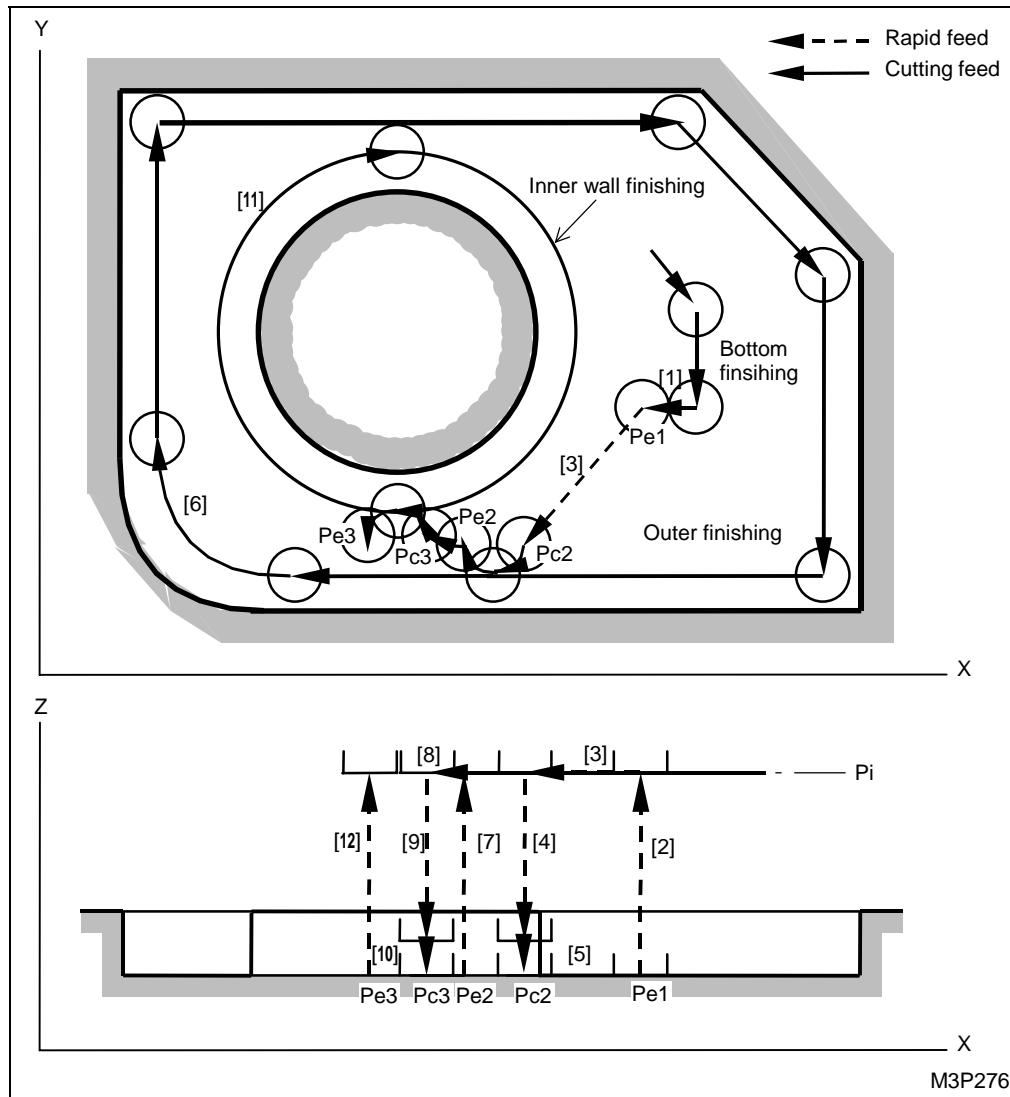
- Bottom finishing is performed, with $0 < \text{FIN-Z}$.
- Outer and inner walls are finished, with $0 < \text{FIN-R}$.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-X, -Y 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Note: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
1	PCKT VLY	999.999	999.999	9	9	999.999	999.999						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL												

Remark 1: Data in unit represent a maximum input value.

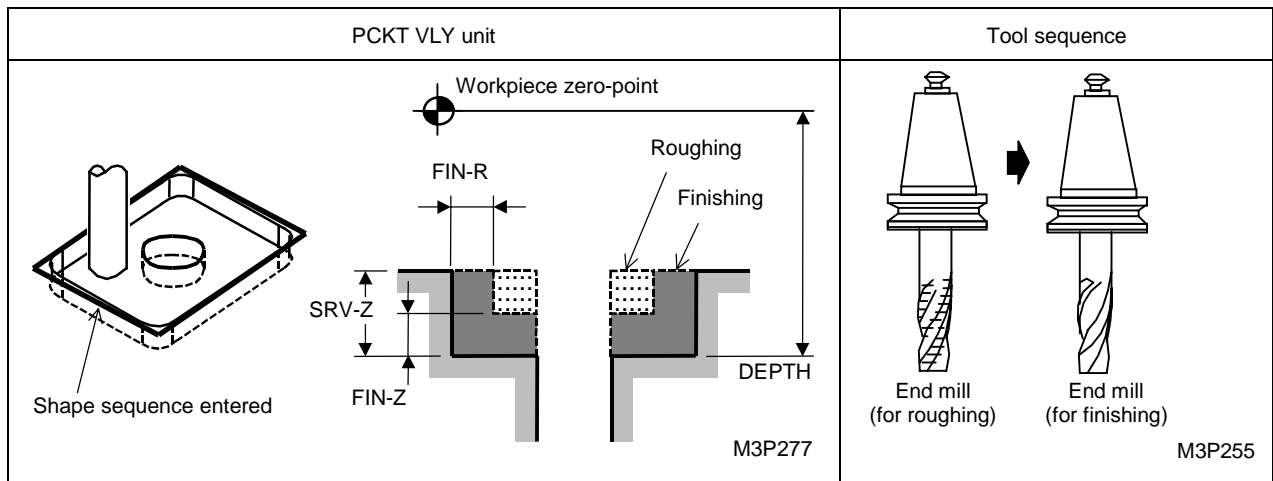
Remark 2: ◆: Data are not necessary to be here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.



BTM: A bottom roughness code is selected out of the menu.

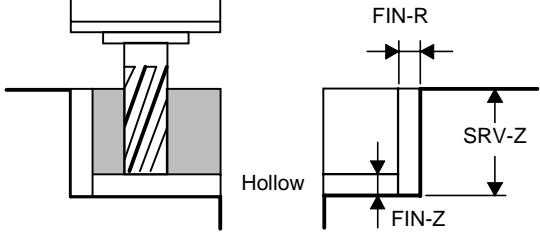
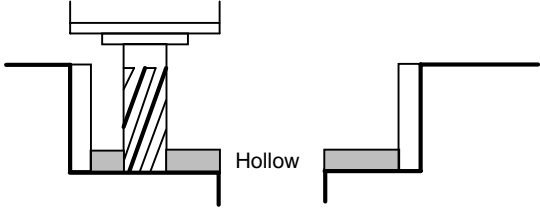
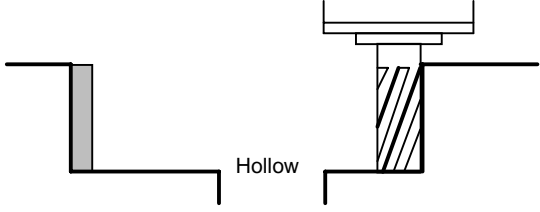
WAL: A wall roughness code is selected out of the menu.

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

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

B. Machining sequence

Pocket milling-valley is performed in the following order.

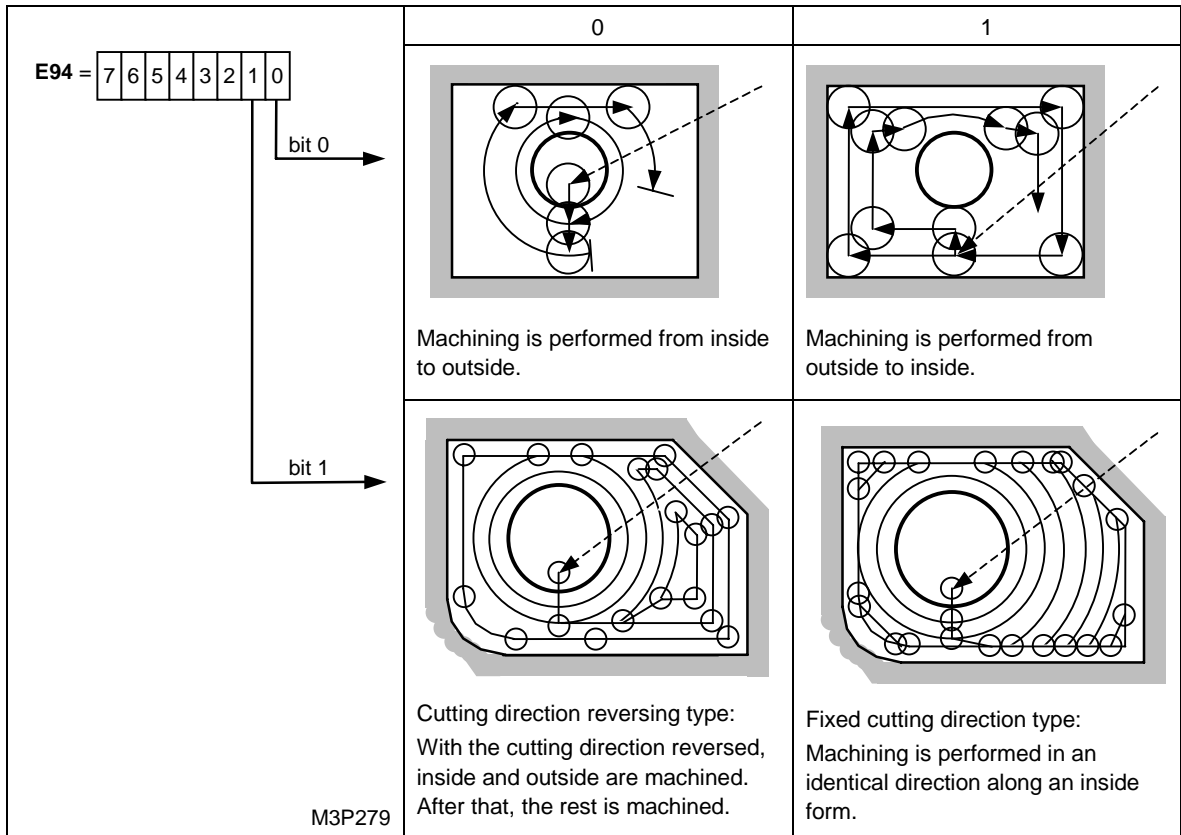
<p>Roughing</p>		<p>Machining is performed with the end mill developed in the tool sequence R1. With $SRV-Z = FIN-Z$, this machining is not performed.</p>	
<p>Finishing</p>	<p>Bottom</p>		<p>Machining is performed with the end mill developed in the tool sequence F1. With $FIN-Z = 0$, this machining is not performed.</p>
<p>Finishing</p>	<p>Wall</p>		<p>Machining is performed with the end mill developed in the tool sequence F1. With $FIN-R = 0$, this machining is not performed.</p>

M3P278

C. Machining pattern

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

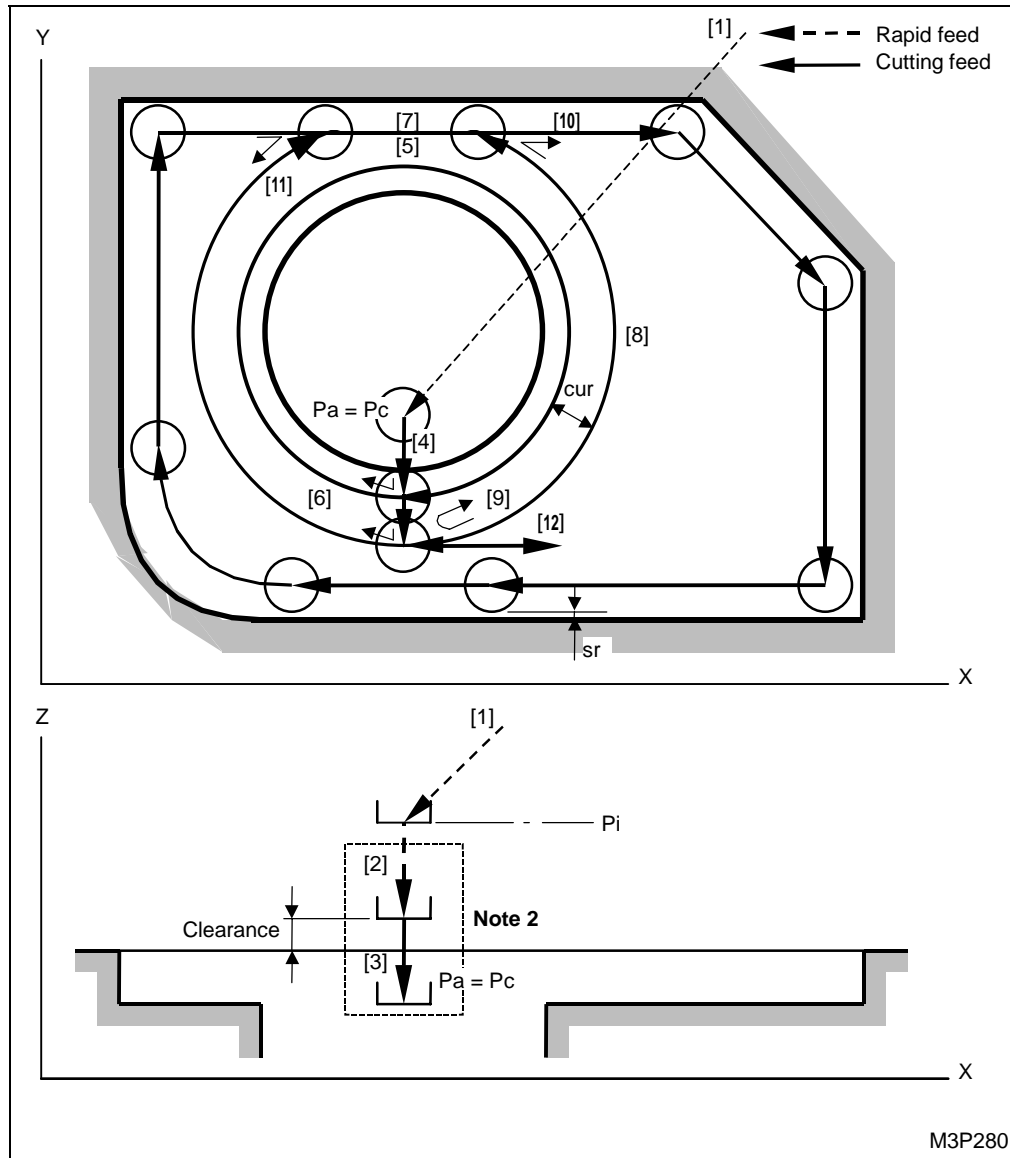
- Enter 0 or 1 in the related bit accordingly.



D. Tool path

Machining from inside (roughing or bottom finishing)

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



The bold codes represent parameter addresses.

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

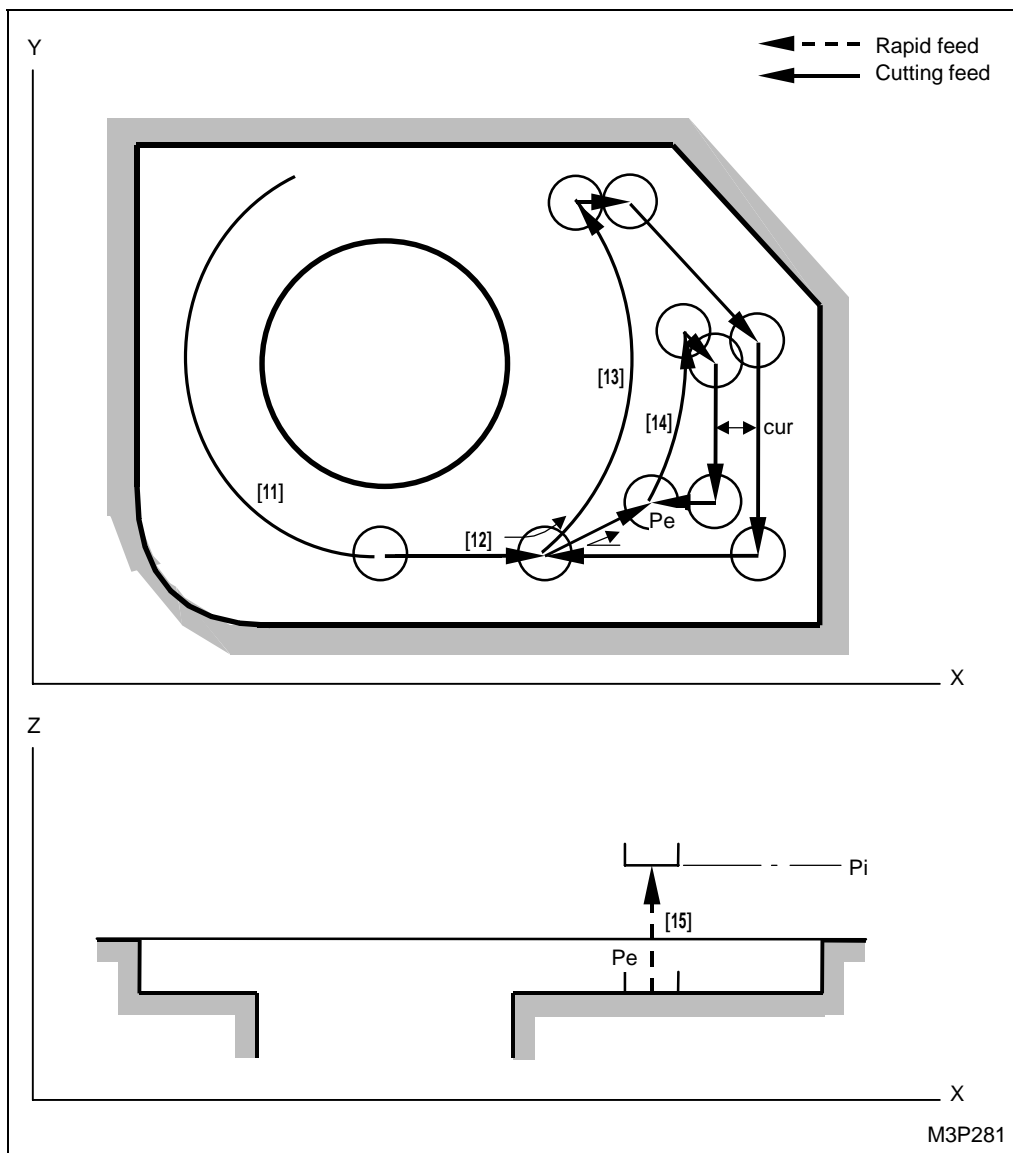
Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

- Machining the rest:



Pe: Escape point to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

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

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

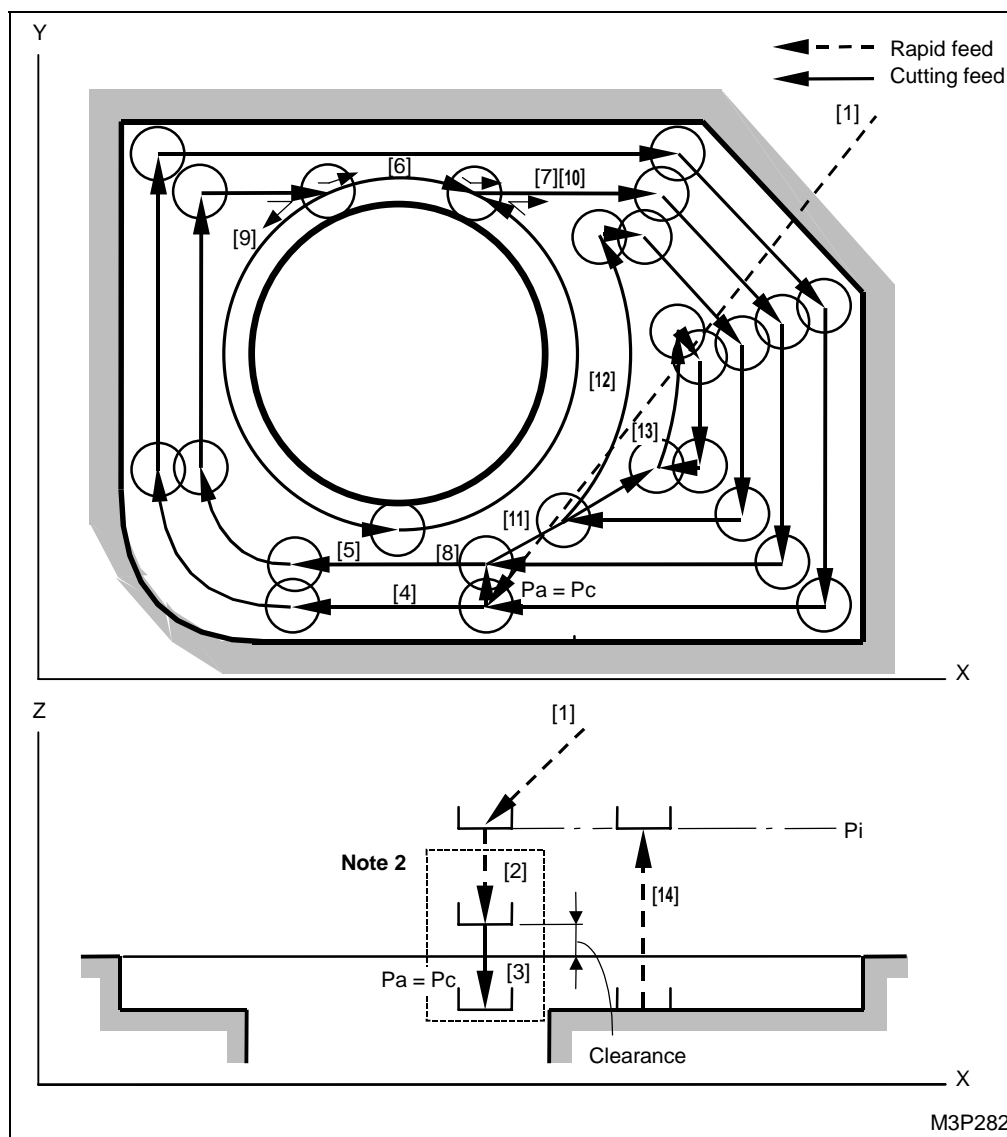
Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, the tool is positioned directly at the cutting start point and operation [2] and [3] are performed. In this case, the coordinates of cutting start point will be entered in these items.

Note 2: See Subsection 4-9-5, "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 ZFD 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 inwards 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)



The bold codes represent parameter addresses.

Pa, Pc: Approach point to be determined by the data APRCH-X, -Y 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.

Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 1: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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 4-9-5, "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.

E. Finishing

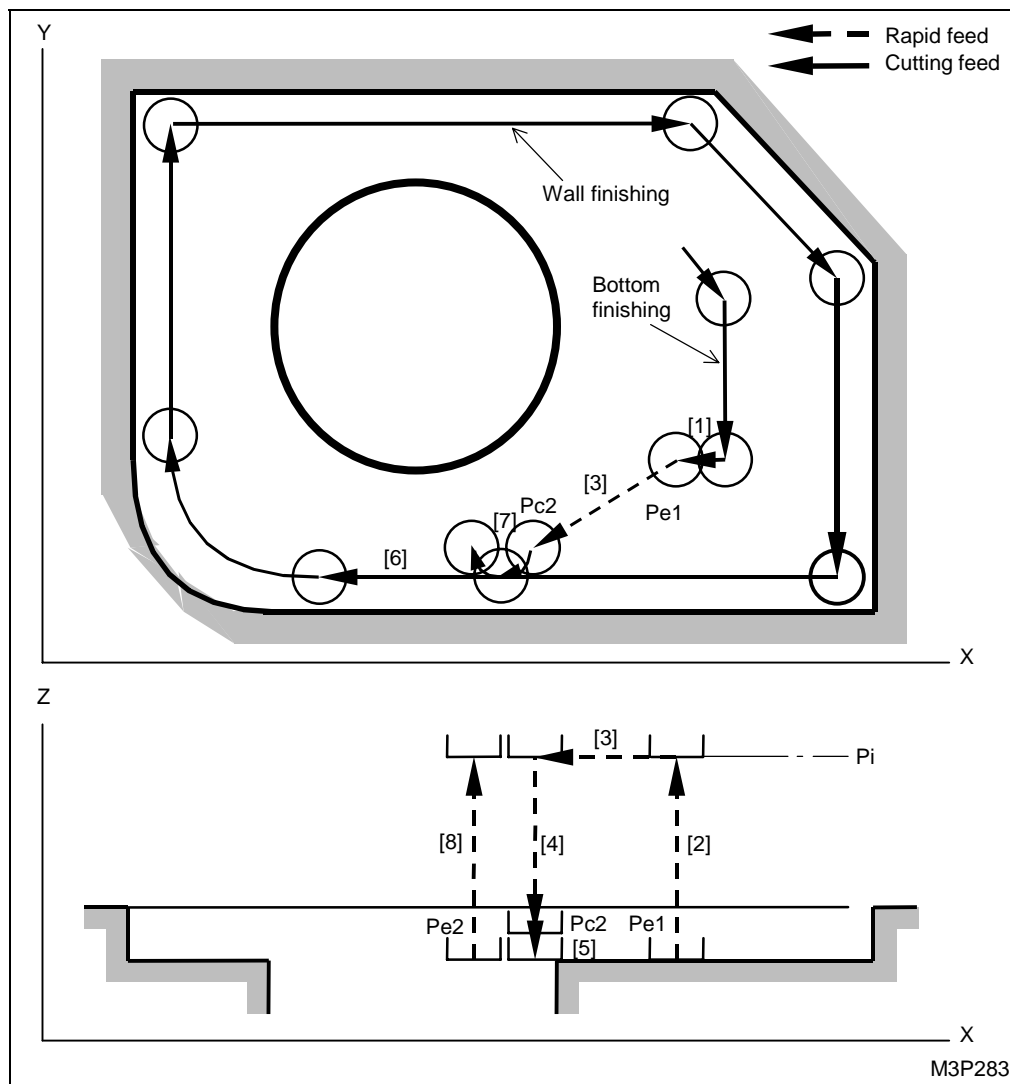
Finishing is performed, based on the entered data FIN-Z and FIN-R.

Bottom finishing is performed, with $0 < \text{FIN-Z}$.

Wall finishing is performed, with $0 < \text{FIN-R}$.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-X, -Y 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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

Note: When ? is displayed in the items APRCH-X, -Y by pressing the menu key **AUTO SET**, 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	DEPTH	SRV-Z	SLOT-WID	BTM	WAL	FIN-Z	FIN-R					
1	SLOT	999.999	999.999	99.999	9	9	999.999	999.999					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				

Remark 1: Data in unit represent a maximum input value.

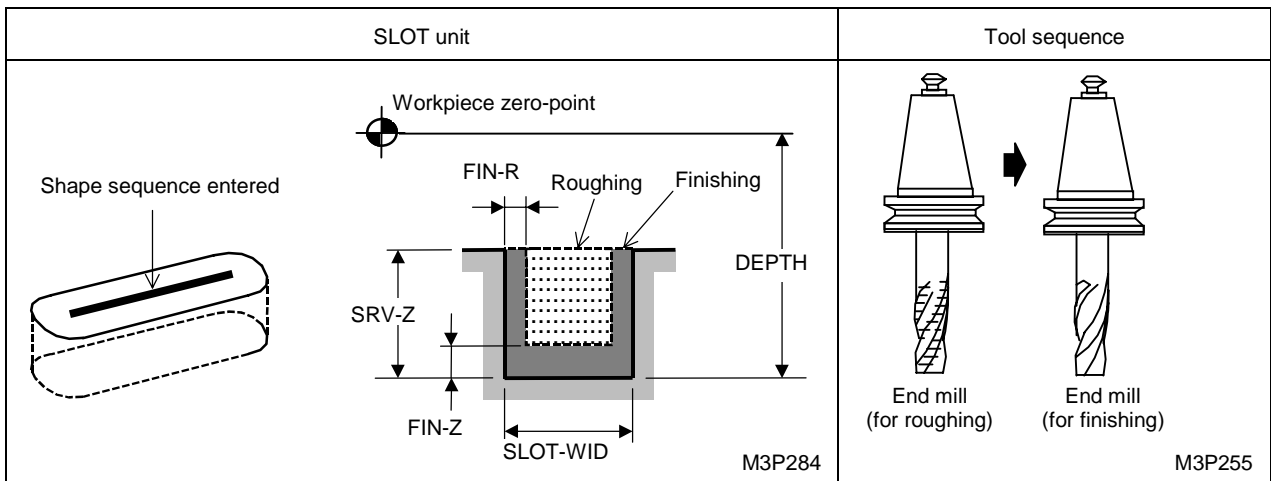
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z, SLOT-WID, FIN-Z and FIN-R.

Machining	Pattern
R1 (Roughing)	FIN-Z = 0 and FIN-R = 0 : One tool is selected.
F2 (Finishing)	SRV-Z ≤ FIN-Z or SLOT-WID ≤ (2 × FIN-R) : One tool is selected.

Remark 5: For the tool sequence data setting, see Subsection 4-9-4.



BTM: A bottom roughness code is selected out of the menu.

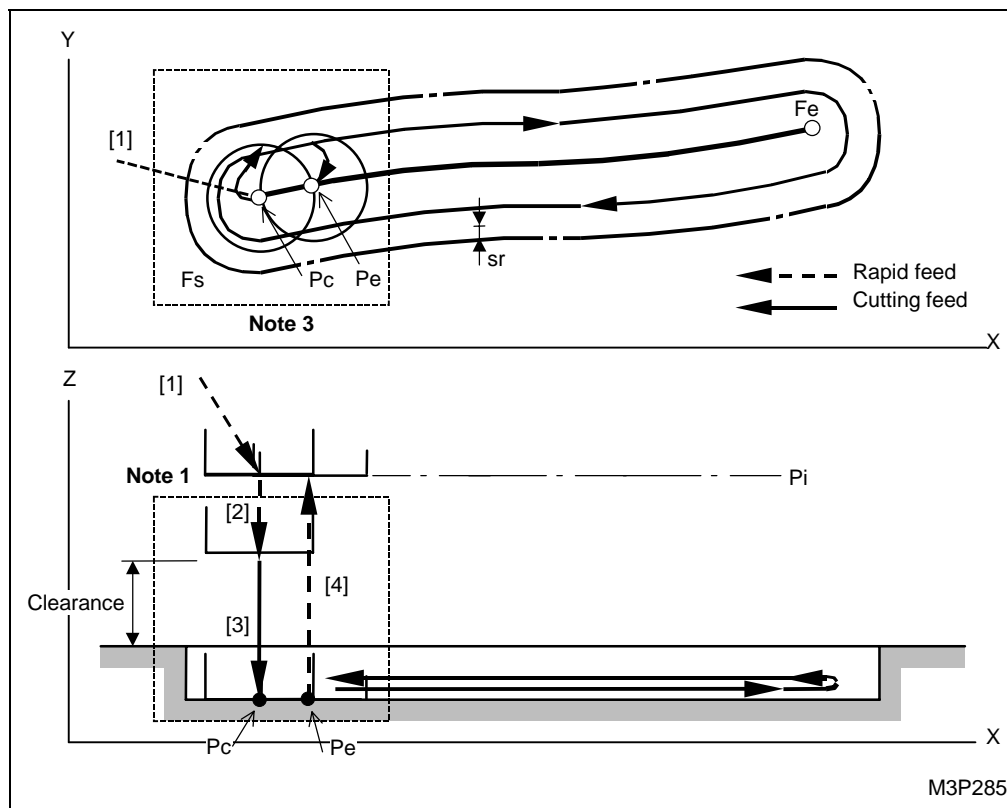
WAL: A wall roughness code is selected out of the menu.

FIN-Z: A Z-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. Tool path

$$\text{With tool radius} + \text{FIN-R} < \frac{\text{SLOT-WID}}{2} < \text{tool diameter} + \text{FIN-R}$$



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 Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

sr: Radial finishing allowance (FIN-R) to be entered in a machining unit

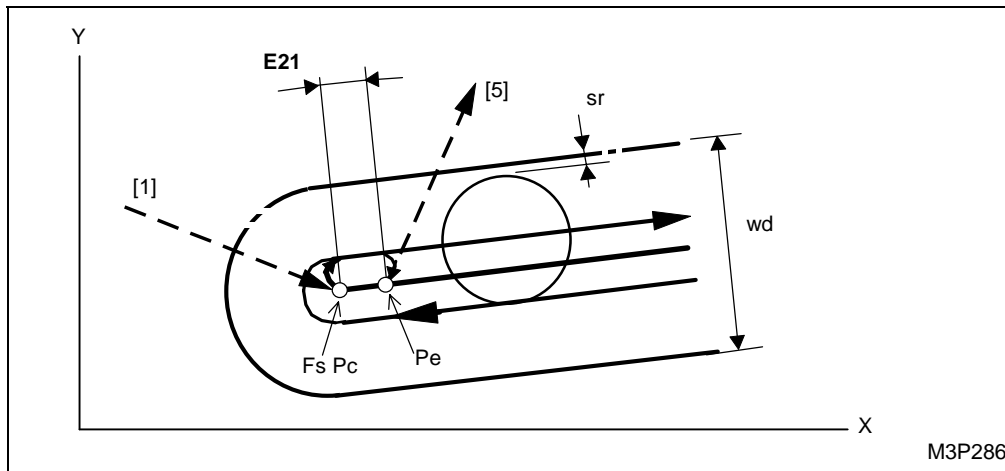
Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 1: See Subsection 4-9-5, "Precautions in face machining."

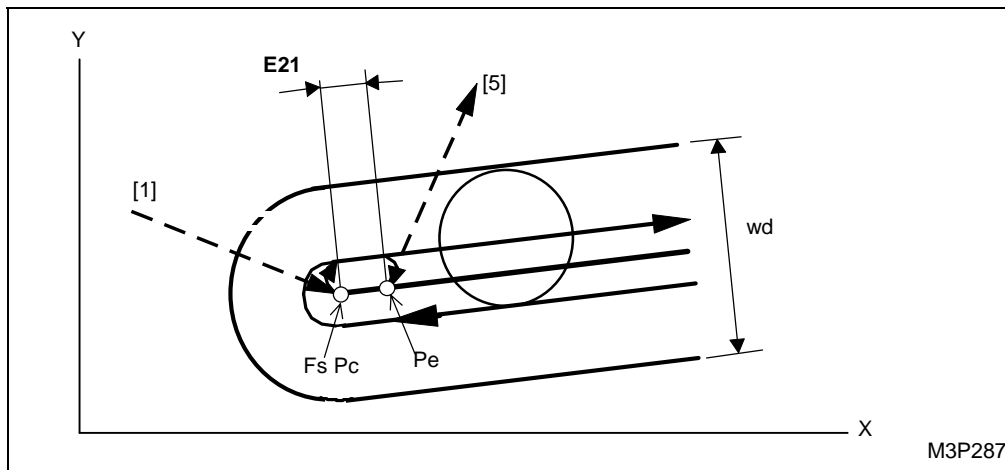
Note 2: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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>

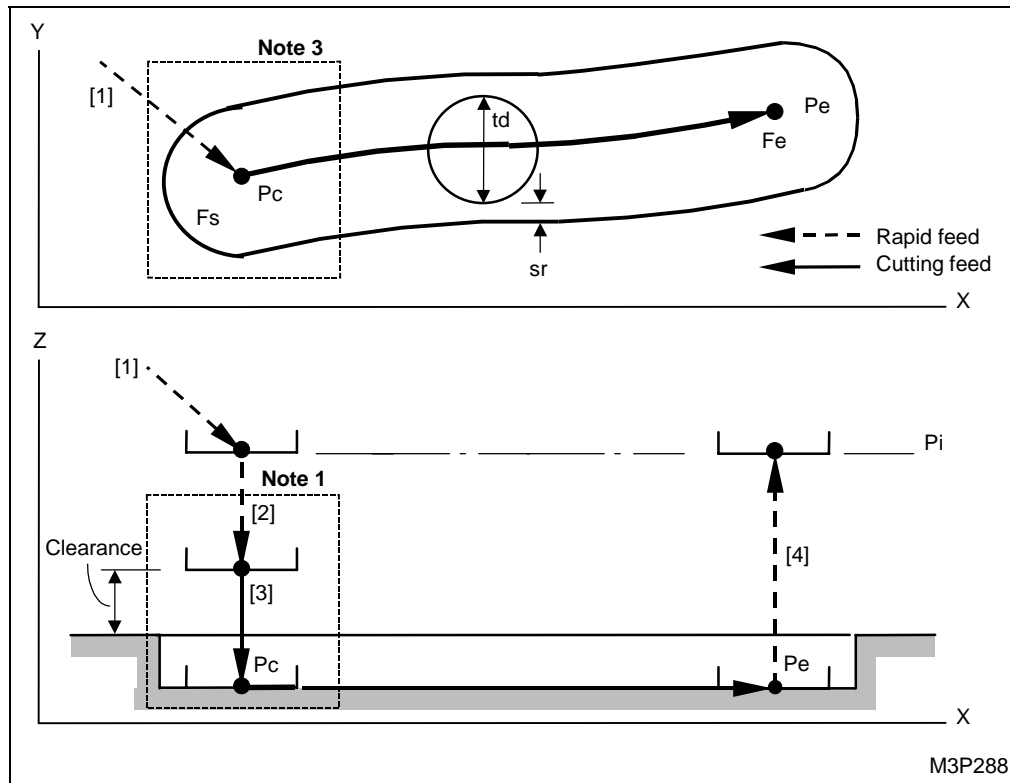
[1] The tool moves at a rapid feedrate to approach point (cutting start point).

[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.

$$\text{With tool radius + FIN-R} = \frac{\text{SLOT-WID}}{2}$$



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 to be automatically established

Pi: Initial point Z (See "Positions of the initial point and reference point (R)" in Section 4-6.)

td: Tool diameter to be registered in the **TOOL DATA** display

sr: Radial finishing allowance (FIN-R) to be entered in a machining unit

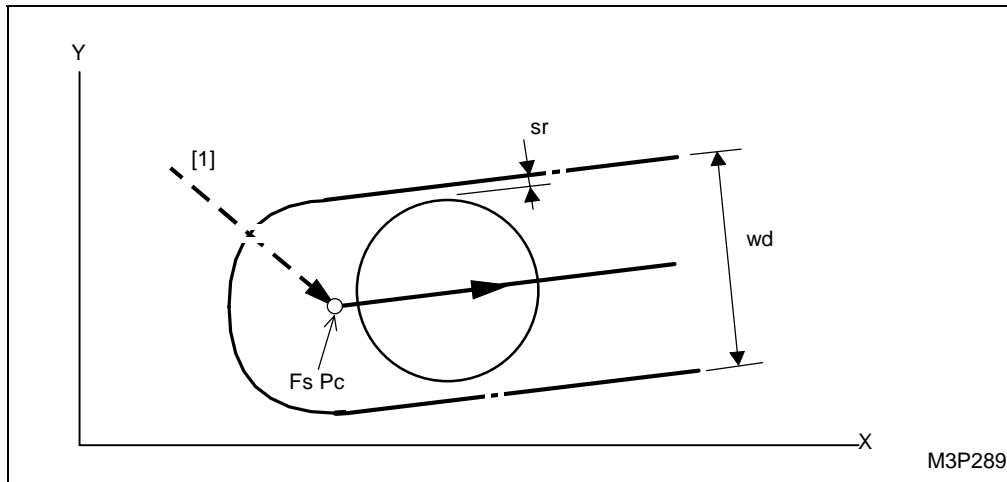
Remark: For the (safety) clearance, see "Positions of the initial point and reference point (R)" in Section 4-6.

Note 1: See Subsection 4-9-5, "Precautions in face machining."

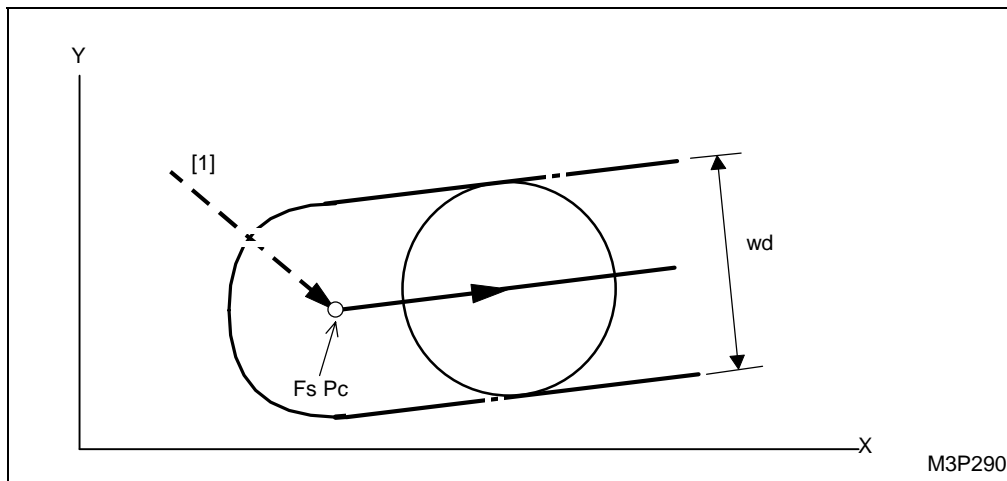
Note 2: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

Note 3: Detail of tool path near approach 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

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.

4-9-4 Face machining tool sequence data

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.

SN _o .	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL												
F2	END MILL							◆					
	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑
	1	2	3	4	5	6	7	8	9	10	10	11	11

◆: Not necessary to be set here.

1. Tool designation: TOOL

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

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	CENTER DRILL					
---------	----------	-------------------	-----------------	-----------------	--	--	--	--	--

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.

2. Nominal tool diameter: NOM-φ

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

3. Tool identification code

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

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

Press the menu key **HEAVY TOOL** 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. Priority number: No.

Enter the priority number for prior machining and subsequent machining. There are three following entry methods.

- Prior machining priority number

To be set by means of numeric keys. The number is displayed in yellow.

Setting range (1 to 99).

- Subsequent machining priority number

To be set by means of numeric keys after having reversed the display of **DELAY PRIORITY** by selecting its menu key.

The characters are displayed in reversed status.

Setting range (1 to 99).

- Without entry (Ordinary machining)

The priority number is not entered.

For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

5. Coordinates X and Y of the approach point: APRCH-X, APRCH-Y

Enter an X, Y coordinates of the position at which a tool is to cut in axially.

With menu key **AUTO SET** selected, ? is displayed. Check the tool path and ? will automatically change over to the coordinates of a cutting start point. (Refer to tool path by each unit.)

6. Machining method: TYPE

Machining method differs according to the units as follows.

A. Face milling unit

A tool path pattern is selected out of three: 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.

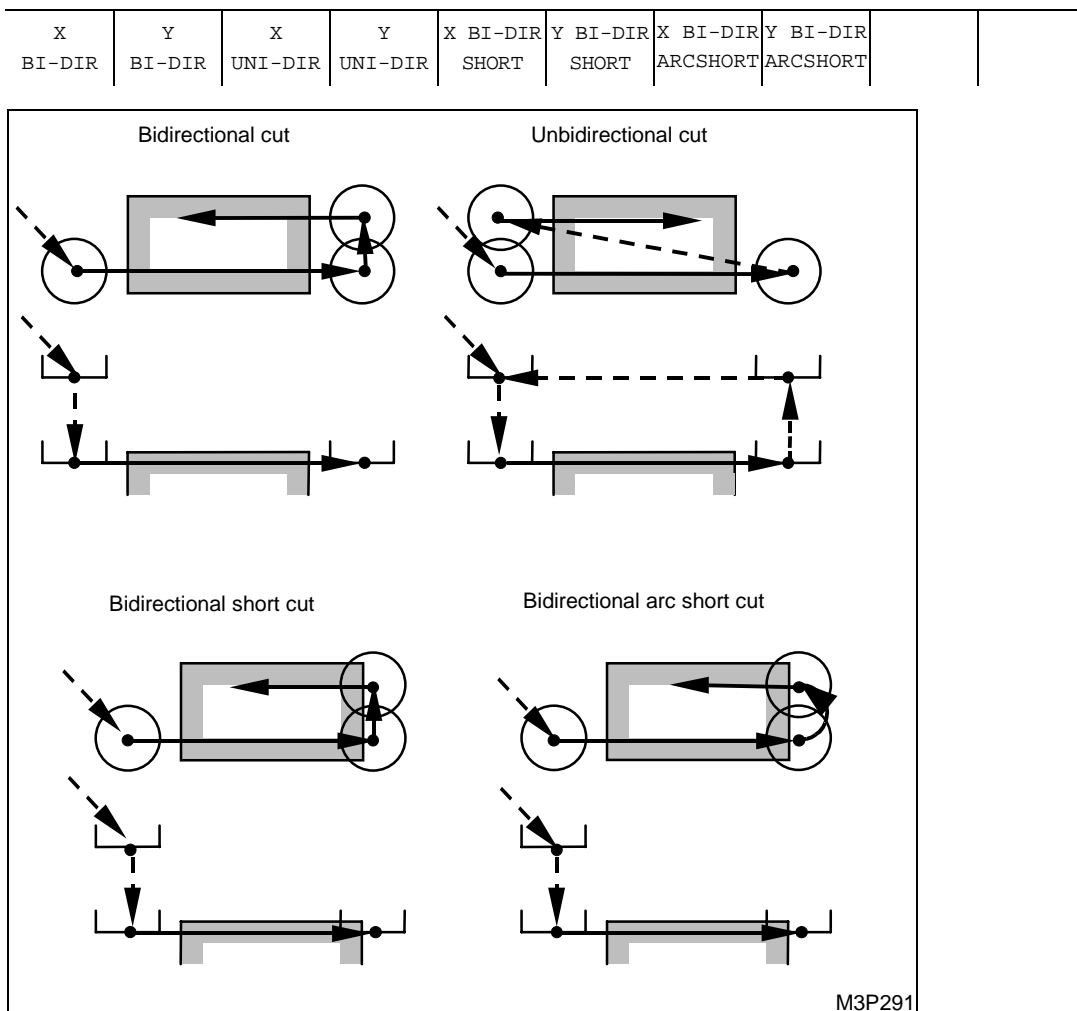
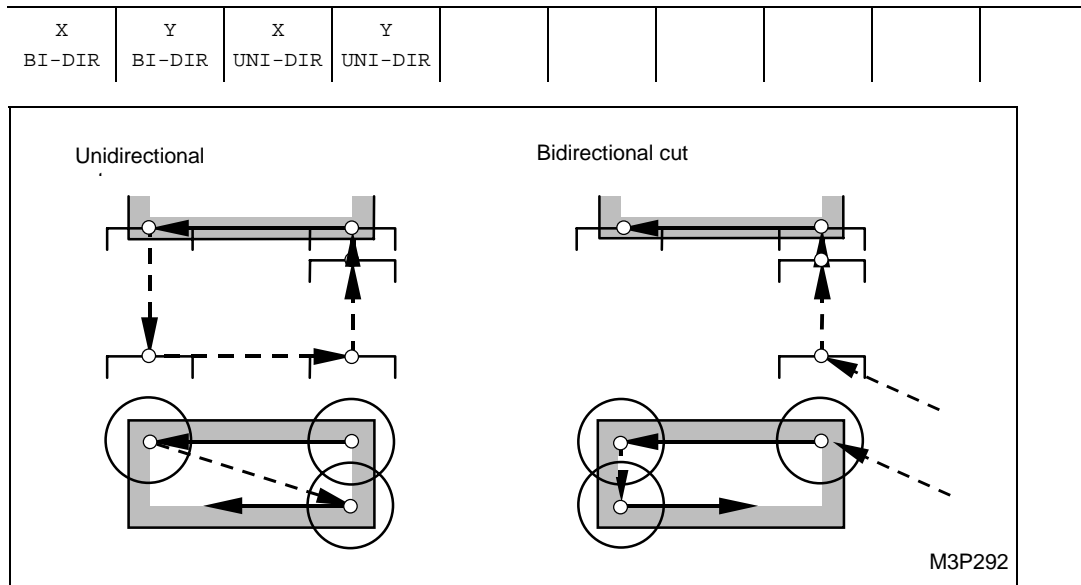


Fig. 4-20 Tool path patterns

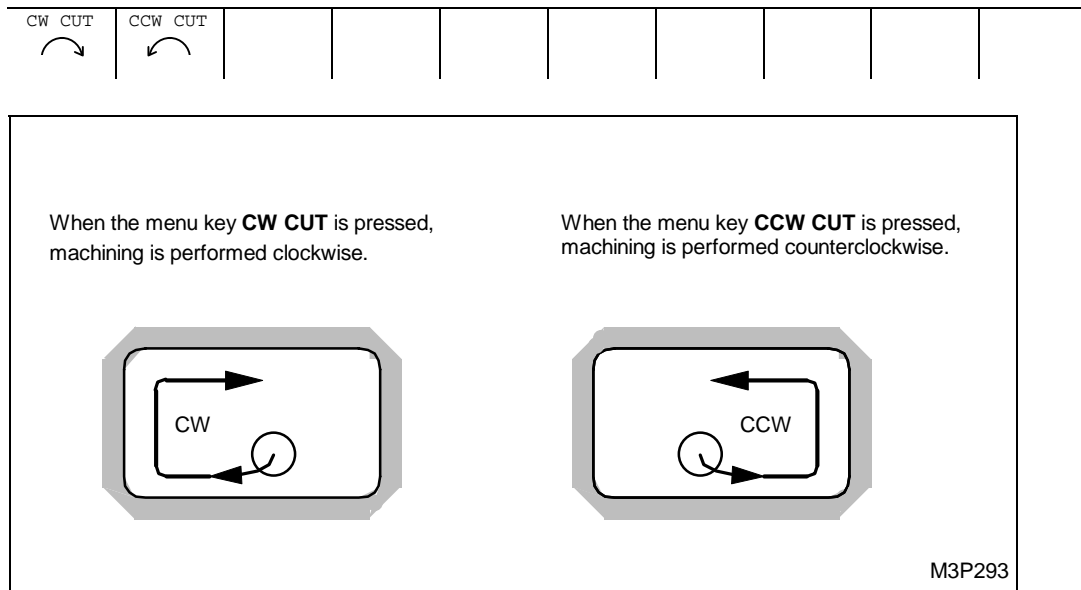
B. End milling-top unit

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.



C. Other units

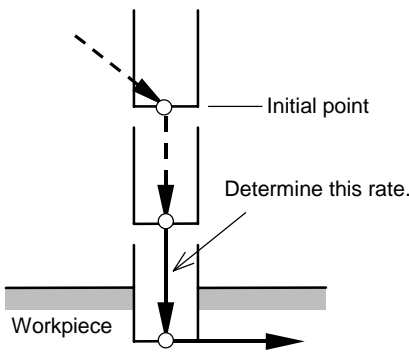
A machining (turning) direction is selected by the use of menu keys.



7. Z-axis feedrate: ZFD

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							
------------	--------------	--	--	--	--	--	--	--



ZFD	Feedrate
G00	Rapid feed
G01	Parameter E17 may be used to determine: $\text{Feed} \times \frac{\mathbf{E17}}{10}$
Numeric value (α)	$\text{Feed} \times \alpha$

M3P294

8. Cutting stroke-Z: DEP-Z

In roughing, a maximum axial cutting stroke in one pass is entered. With menu key **AUTO SET** pressed, a smaller value is entered, either the data SRV-Z 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-Z, SRV-Z and FIN-Z in the machining unit. (For calculation formula, see Subsection 4-9-5, "Precautions in face machining.")

9. Radial cutting depth: WID-R

A maximum cutting depth per pass is entered in roughing or bottom finishing.

With the menu key **AUTO SET** selected, the data WID-R is automatically calculated and determined by the parameter **E10** or **E14** and nominal tool diameter.

$$\text{WID-R} = \text{Nominal tool diameter} \times \frac{\mathbf{E10}}{10} : \text{FCE MILL, TOP EMIL and STEP}$$

$$\text{WID-R} = \text{Nominal tool diameter} \times \frac{\mathbf{E14}}{10} : \text{POCKET, PCKT MT and PCKT VLY}$$

10. Cutting conditions (circumferential speed, feedrate): C-SP, FR

A spindle speed and a cutting feedrate are entered.

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

11. M-codes: M

Enter an M-code which will be output immediately after mounting the tool onto the spindle in the ATC mode. A maximum of up to two codes may be entered. It is also possible, moreover, to select and enter a general M-code out of the menu. (See "List of M-codes" in the Operating Manual.)

4-9-5 Precautions in face machining

1. Tool path during rough-machining in the case of “Z-axial machining allowance (SRV-Z) > Z-axial cutting depth (DEP-Z)”

Cutting is performed at several passes. The tool path is determined by the parameter related to the following two factors, but these factors may not be all available in certain machining unit:

- A. Cutting start position along the Z-axis
- B. Type of routing through approach point

[Basic tool pat]

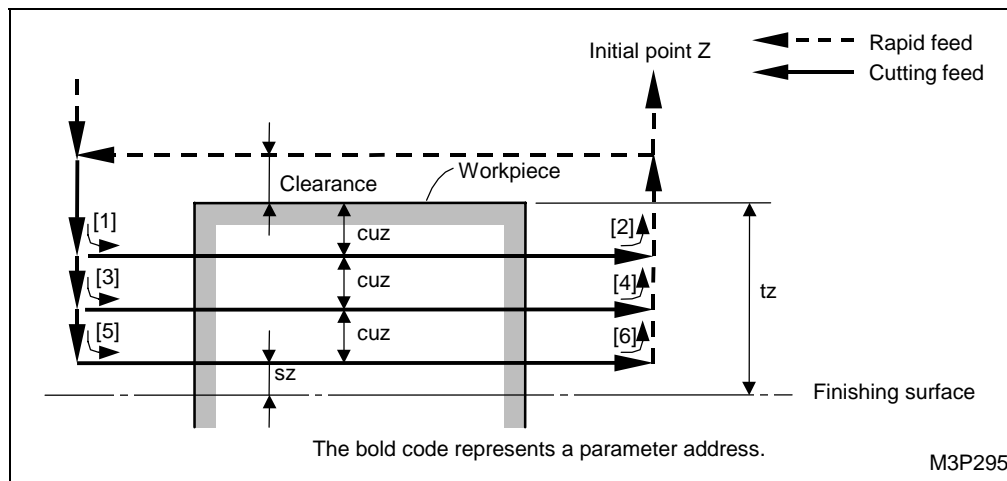


Fig. 4-21 Basic tool path

cuz: Z-axial cutting depth per pass

Calculation of cuz:

$$cuz = \frac{tz - sz}{n}$$

$$n = \frac{tz - sz}{cz}$$

tz: Z-axial machining allowance SRV-Z to be entered in the machining unit

sz: Z-axial finishing allowance FIN-Z to be entered in the machining unit

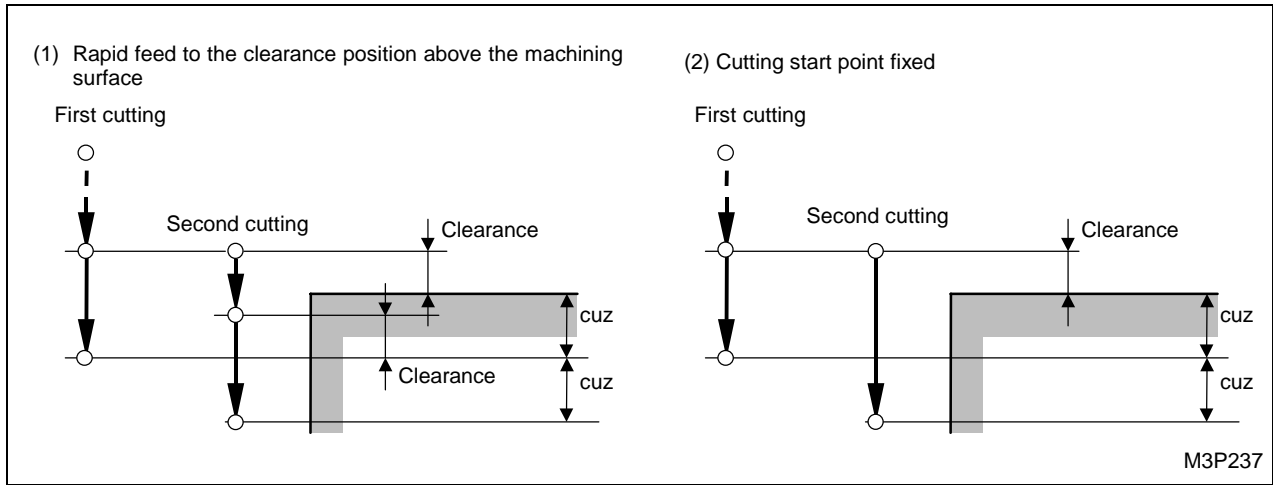
cz: Z-axial cutting depth DEP-Z to be entered in the tool sequence

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

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

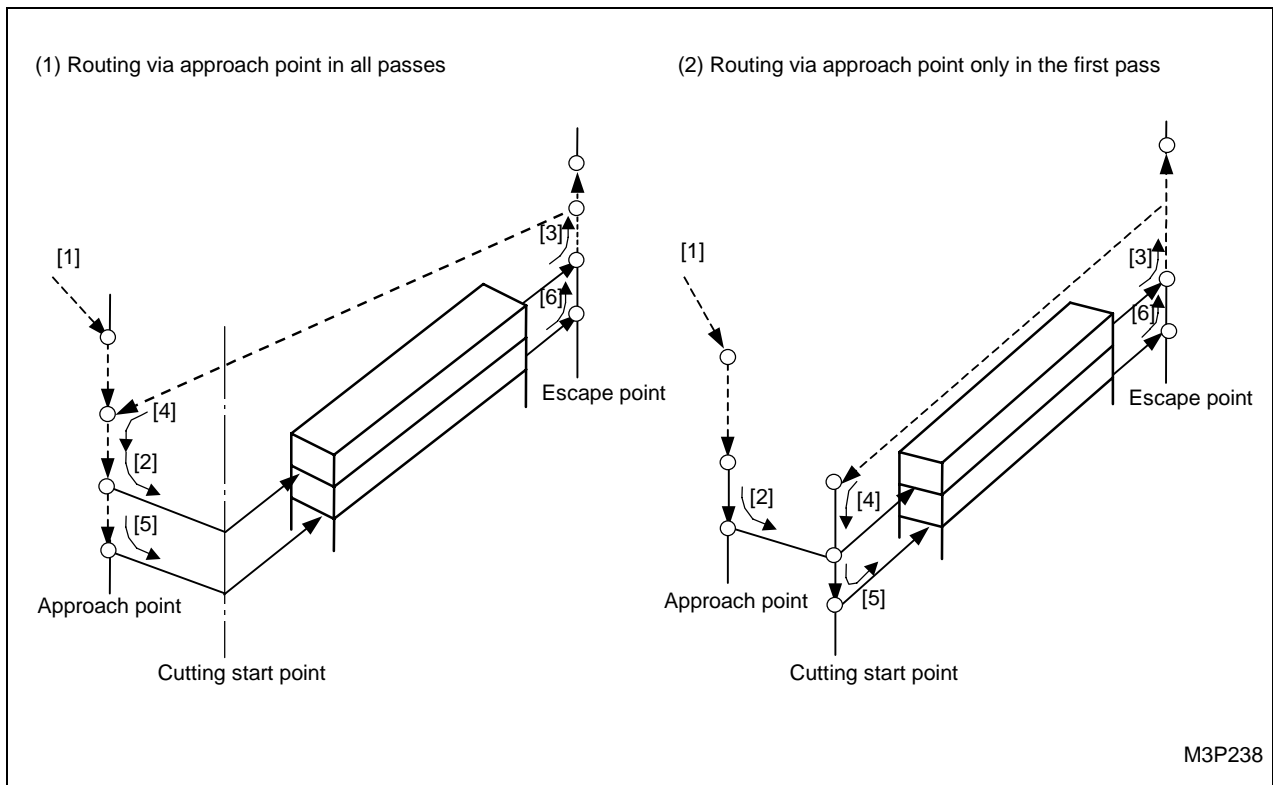
A. Setting the cutting start position along the Z-axis

Select one of the following two types:



B. Setting the type of routing via approach point

Select one of the following two types:



<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)

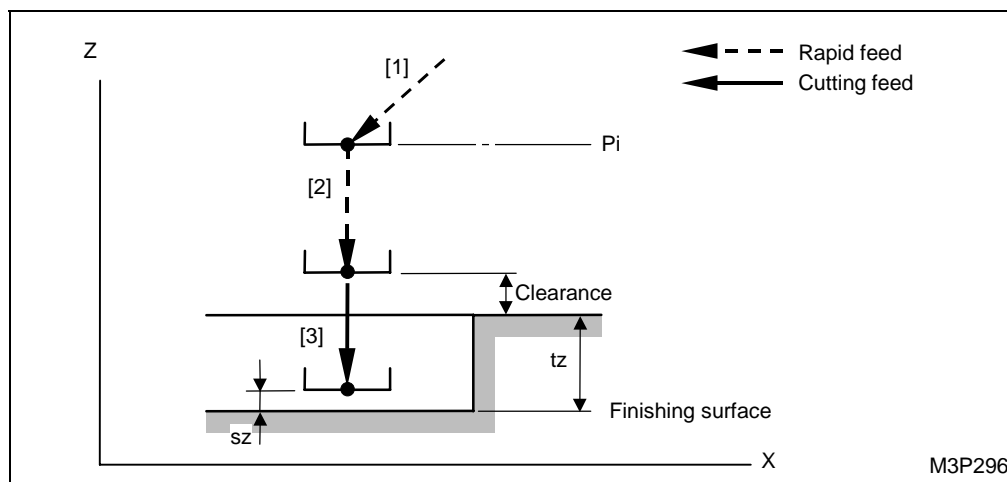
1: Routing via approach point in all passes, (1)

Note 1: B can only be used in the end milling-slot (SLOT) unit.

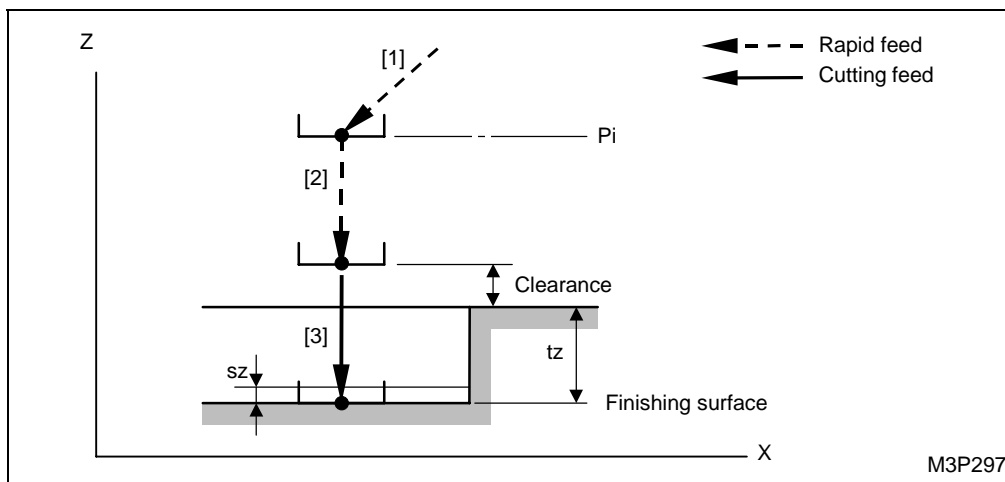
Note 2: The tool path shown at basic tool path above is selected automatically for face machining unit that is not assigned by these parameters.

2. Detail tool path of a Z-axial cutting

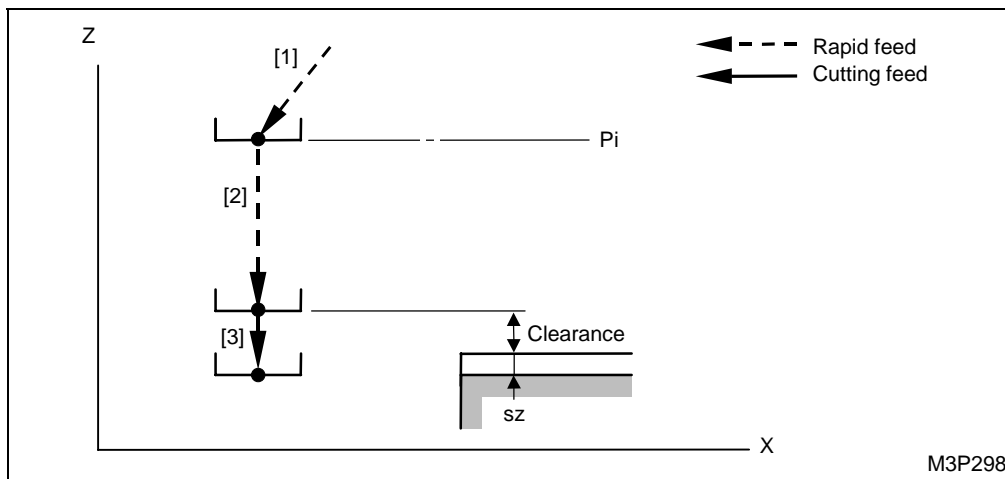
- Roughing



- 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 Z (See “Positions of the initial point and reference point (R)” in Section 4-6.)

tz: Z-axis machining allowance (SRV-Z) to be entered in a machining unit

sz: Z-axis finishing allowance (FIN-Z) to be entered in a machining unit

Remark: For the (safety) clearance, see “Positions of the initial point and reference point (R)” in Section 4-6.

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.

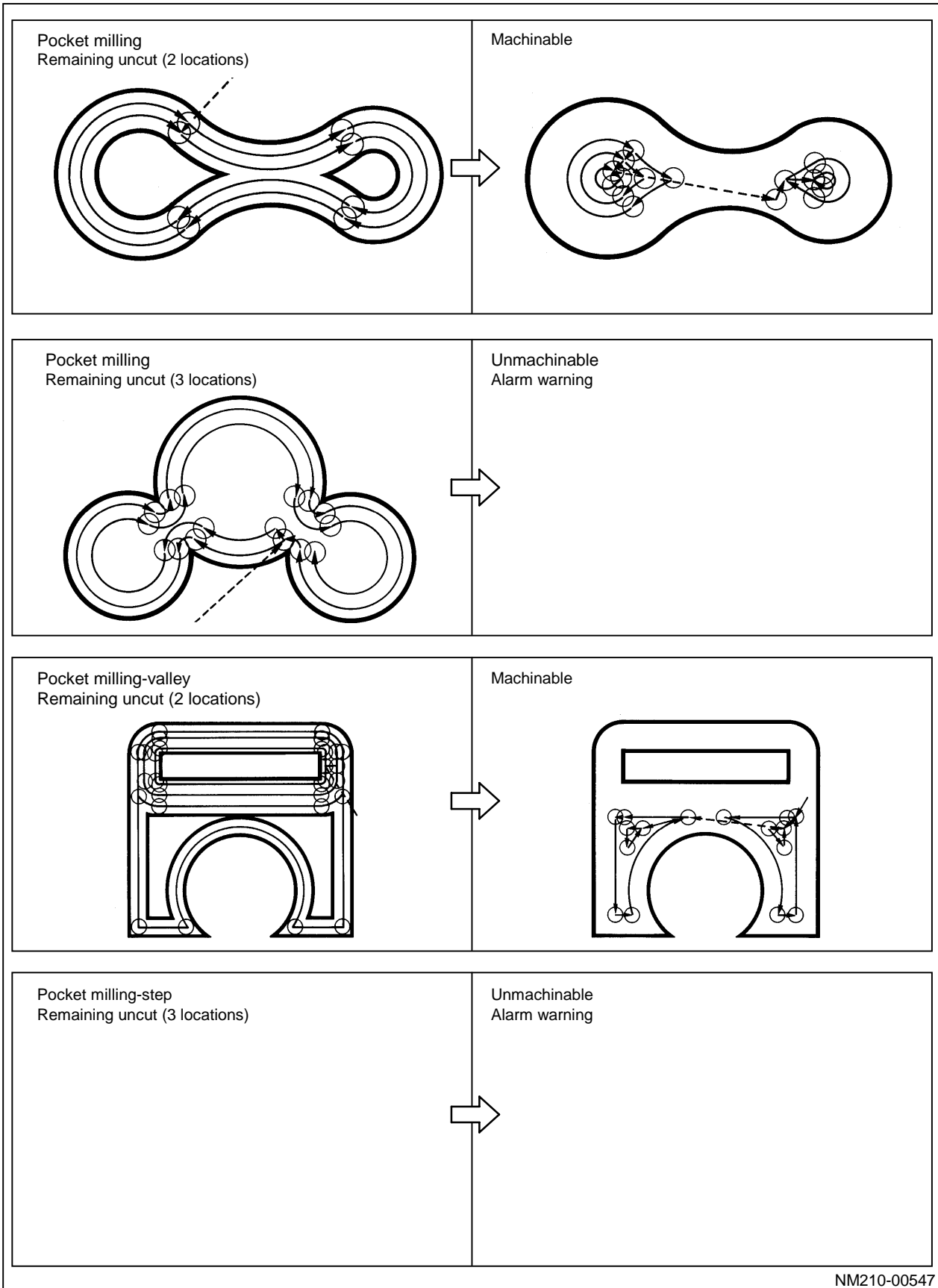
Unit	Parameter	Unit	Parameter
End milling-step	E91 , bit 3	Pocket milling-mountain	E93 , bit 3
Pocket milling	E92 , bit 3	Pocket milling-valley	E94 , bit 3

- A pre-machining tool is included in that tool sequence.

3. Other precautions

1. If shape data, tool data or parameters are modified after the automatic determination of the data APRCH-X, -Y in the tool sequence (displayed in yellow), the cutting start point will not be located on the same approach point and the tool path will be also modified.
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 two portions so remaining uncut, machining is automatically carried out. In excess of two, however, it will result in an alarm.

Example:



NM210-00547

4-9-6 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

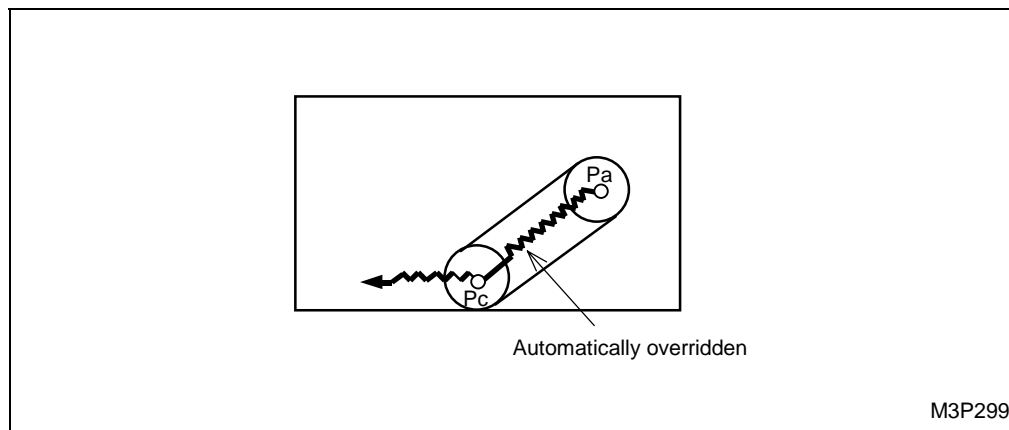
The overall width cutting override is valid in pocket milling on the following four cases on which the cutting width equals the tool diameter:

A. To move from the approach point to the cutting start point:

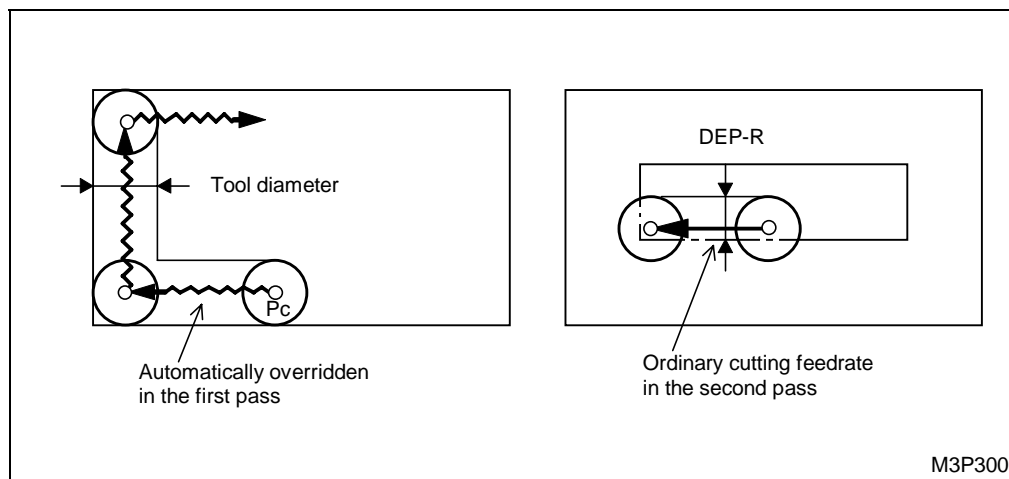
Illustration below is an example of the pocket milling unit.

Pa: Approach point

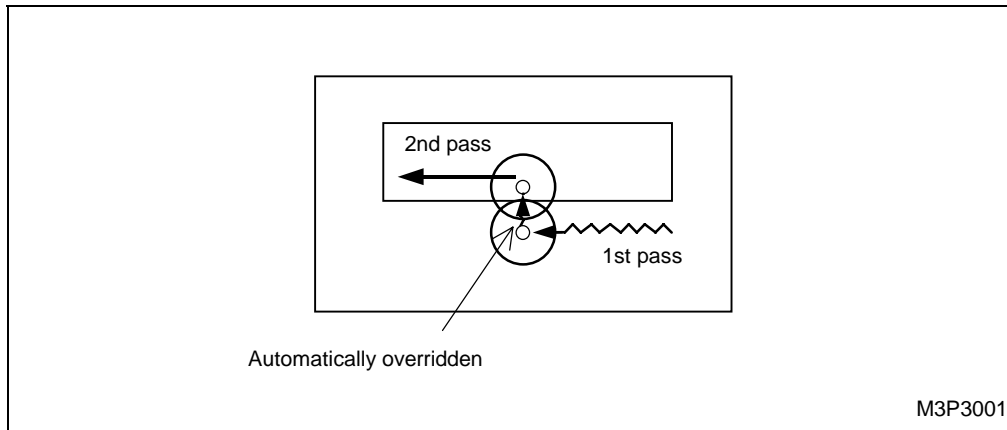
Pc: Cutting start point



B. First pass

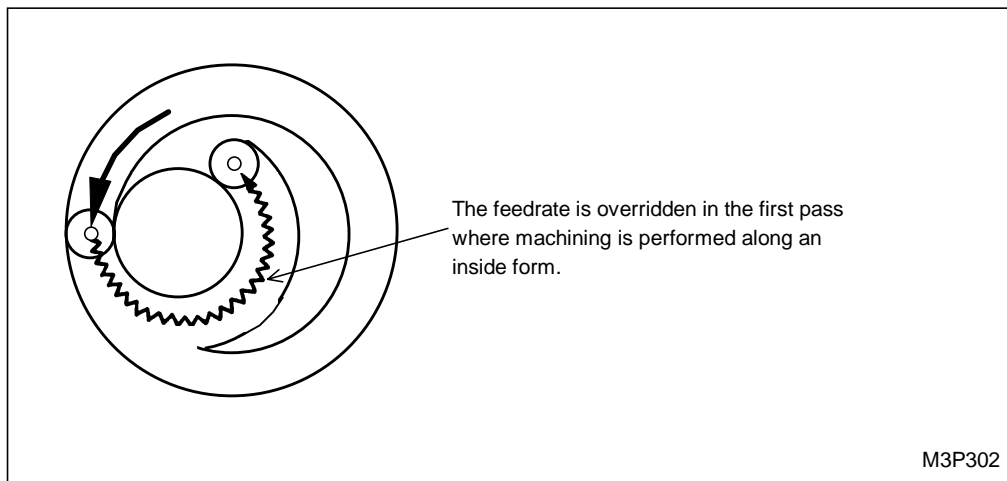


C. To move to the next cutting start point



D. First pass after the inversion of the tool path

Illustration below is an example of the pocket milling-mountain unit.



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**.

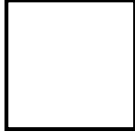
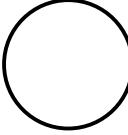
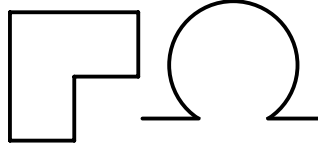
With the parameter set to 0, the overall width cutting override is invalid.

4-9-7 Definitions of forms in line machining and face machining units

After the data in the machining unit and the tool sequence have been entered, enter the data related to the machining form and dimensions in the shape sequence.




1. Definitions of forms

In line machining and face machining units one of the following three patterns can be selected.


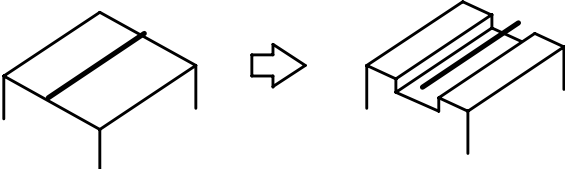
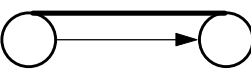
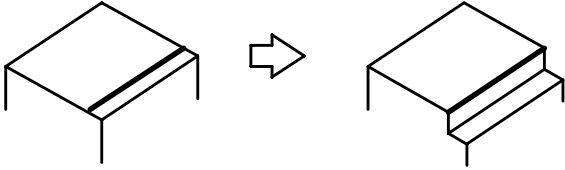
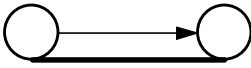
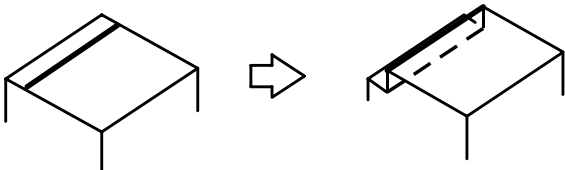
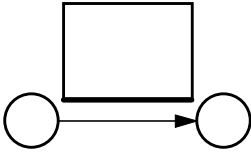
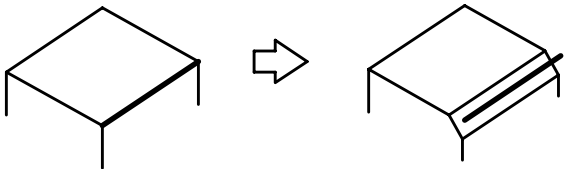
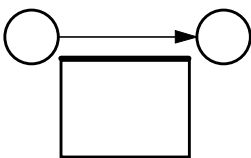
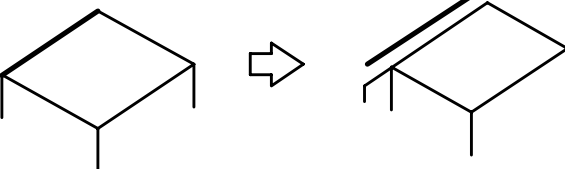
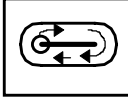
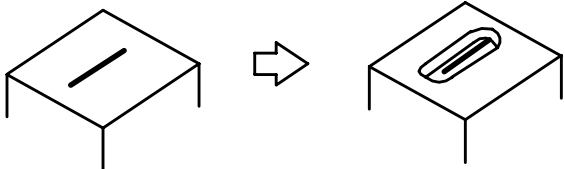
Fixed form		Arbitrary form
SQUARE	CIRCLE	ARBITRY
		
M3P303		

A. Closed form and open form

Depending on the machining units, machining form can be divided into the following two types:

	Closed form		Open form
	Fixed form	Arbitrary form	Arbitrary form
			
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 required	STEP, PCKT MT, PCKT VLY	
M3P304			

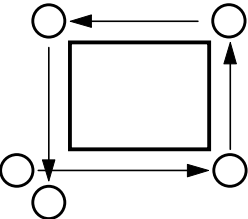
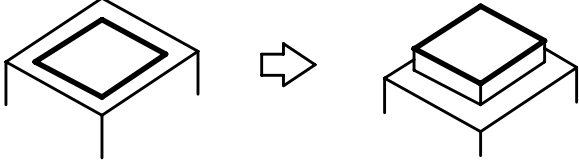
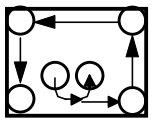
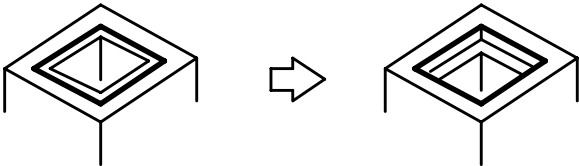
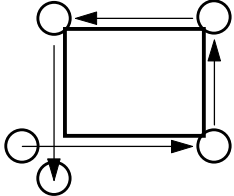
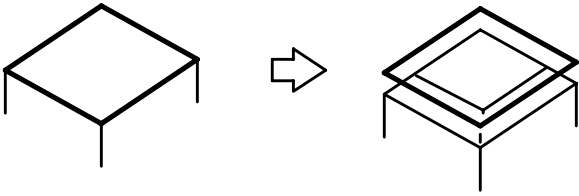
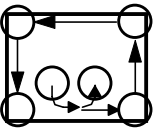
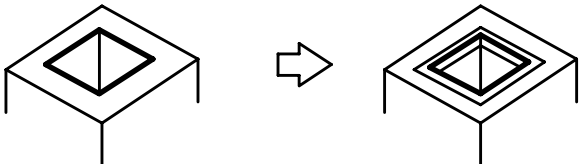
B. Machining units related to the open form

Machining unit		Tool path	Before machining → After machining
Line machining	LINE CTR		
	LINE RGT		
	LINE LFT		
	CHMF RGT		
	CHMF LFT		
Face machining	SLOT		

M3P305

C. Machining units related to the closed form

- Line machining

Machining unit	Tool path	Before machining → After machining
LINE OUT		
LINE IN		
CHMF OUT		
CHMF IN		

M3P306

- Face machining

Machining unit		Tool path	Before machining → After machining
One shape defined	FCE MILL		
	TOP EMIL		
	POCKET		
Two shapes defined	STEP		
	PCKT MT		
	PCKT VLY		

M3P307

2. To define an arbitrary form, take the following precautions:

1. Coordinates X, Y of the start and end points and coordinates I, J of the center of an arc should be entered in the workpiece coordinates system.
2. For an open form, do not fail to establish coordinates X, Y of its start and end points.
3. In an open form, it is impossible to select the corner (C or R) of its start or end point.
4. In end milling-step, pocket milling-mountain and pocket milling-valley, for which two forms, inside and outside, are required, always define the outside form, first.
(Entering the inside form first would lead to a tool path alarm, though the machining form could be drawn.)

Example:

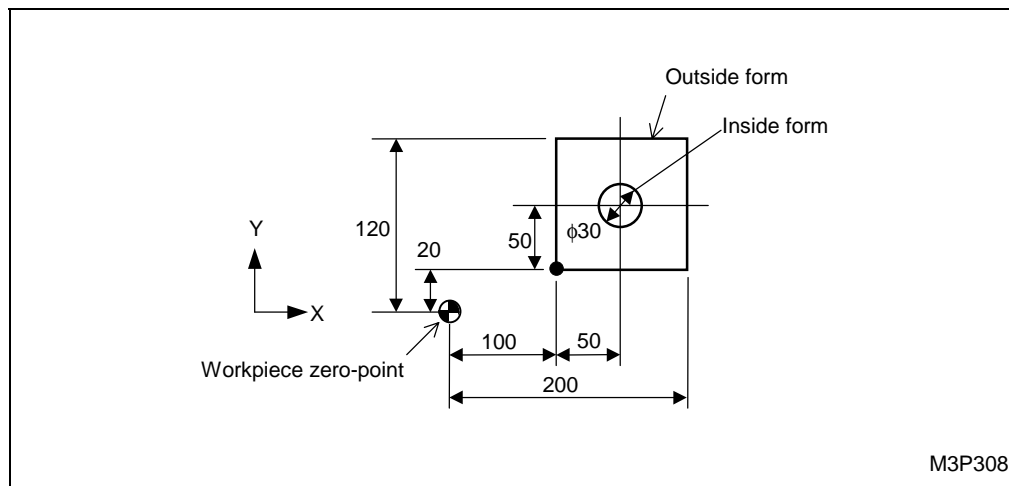


FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	100.	20.	200.	120.				
2	CIR	150.	70.	15.	◆	◆	◆	◆	◆

When both outside and inside forms are arbitrarily defined, it is necessary to use menu key **STARTING POINT** without fail at the head of the inside form. After pressing the menu key **STARTING POINT**, select an arbitrary form with menu key **LINE**, **CW ARC** or **CCW ARC**.

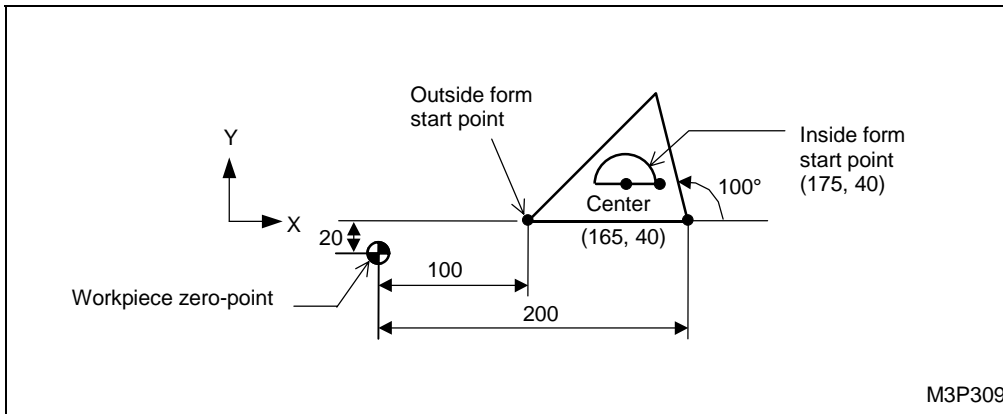


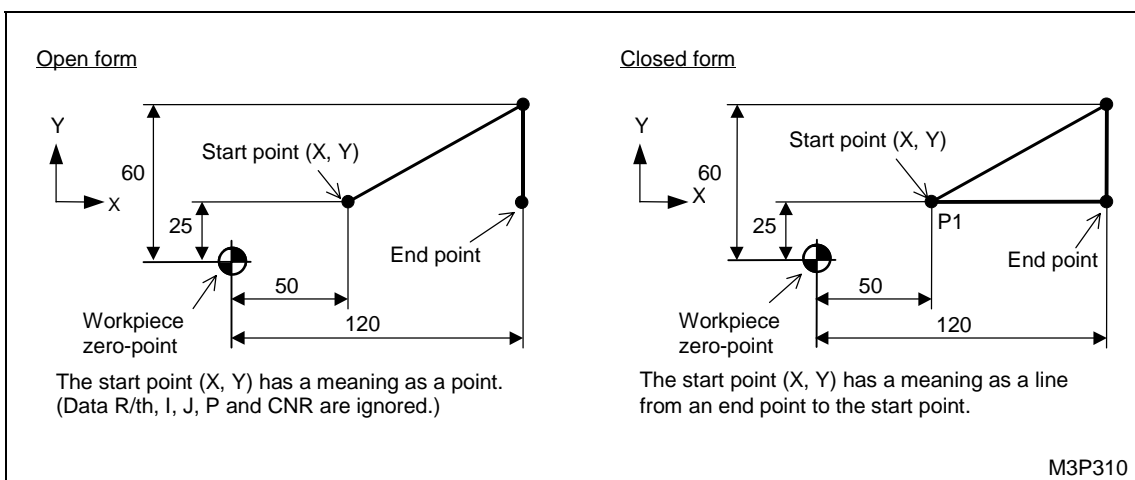
FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	100.	20.					
2	LINE	200.	20.					
3	LINE		80.	100.				
4	LINE	175.	40.					
5	CCW	155.	40.		165.	40.		

STARTING POINT → LINE The zone encircled is displayed in blue.

5. The meaning of a start point (X, Y) differs between an open form and a closed one.

FIG	PTN	Y	X	R/th	I	J	P	CNR
1	LINE	50.	25.					
2	LINE	120.	60.					
3	LINE	120.	25.					

The form in this program is indicated on a display as follows:

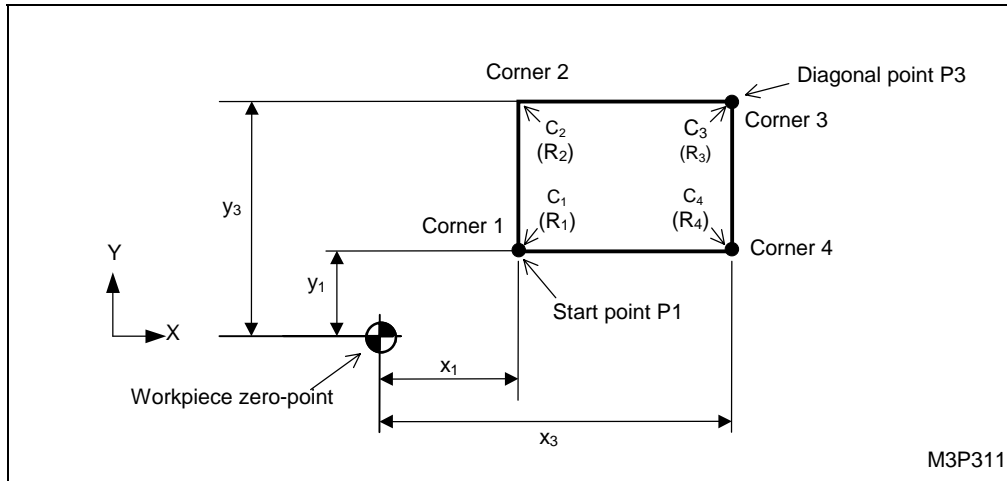


3. Entry of shape sequence data

The following explains the data entry method of the line/face machining sequence for each type of fixed/arbitrary form.

A. Fixed form

1. SQUARE



(a) Menu selection

After setting tool sequence data of the line/face machining unit, the following menu will be displayed.

PATTERN OF FIGURE <MENU>? ()									
SQUARE	CIRCLE		APBITRY				SHAPE	CHECK	
							END		

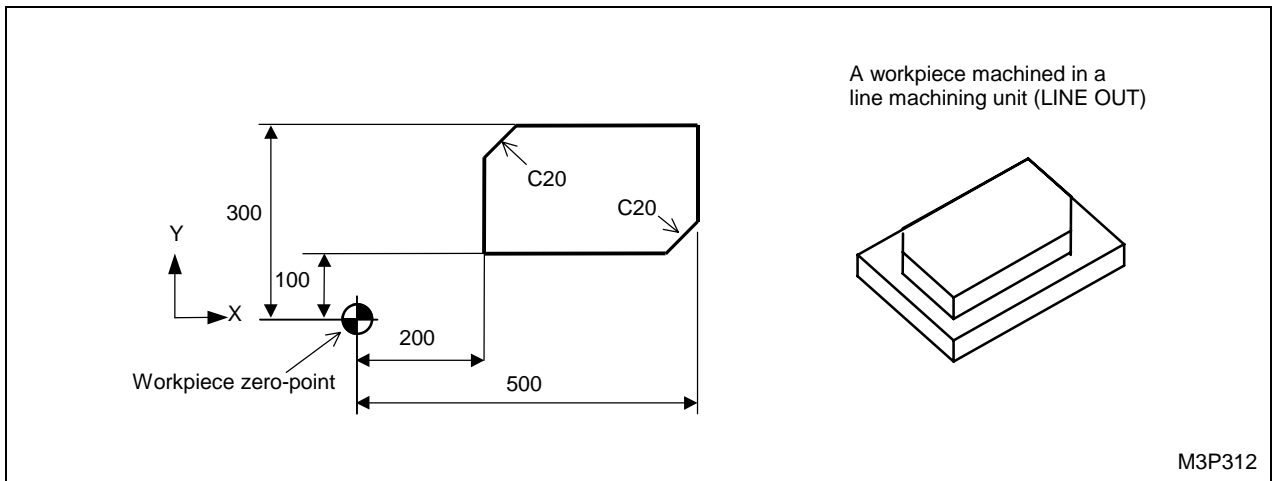
Press the menu key **SQUARE**.

(b) Data setting in shape sequence SQUARE (see figure above)

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	x ₁	y ₁	x ₃	y ₃	C ₁ (R ₁)	C ₂ (R ₂)	C ₃ (R ₃)	C ₄ (R ₄)

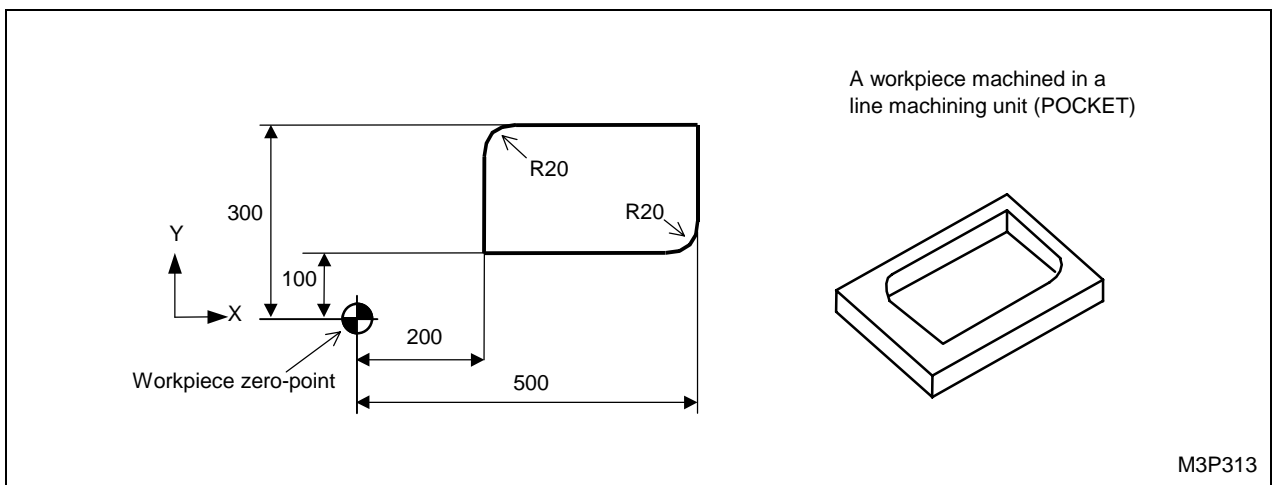
Cursor position	Description
P1X/CX	Enter the X coordinate of start point (x ₁).
P1Y/CY	Enter the Y coordinate of start point (y ₁).
P3X/R	Enter the X coordinate of diagonal point (x ₃).
P3Y	Enter the Y coordinate of diagonal point (y ₃).
CN1	Select a machining form at corner 1. Corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C Pressing the menu key CORNER CHAMFER makes the menu display reverse and entering a numeric value will cause that menu display status to become normal.
CN2	Enter the same data as those for corner 1.
CN3	
CN4	

Example 1:



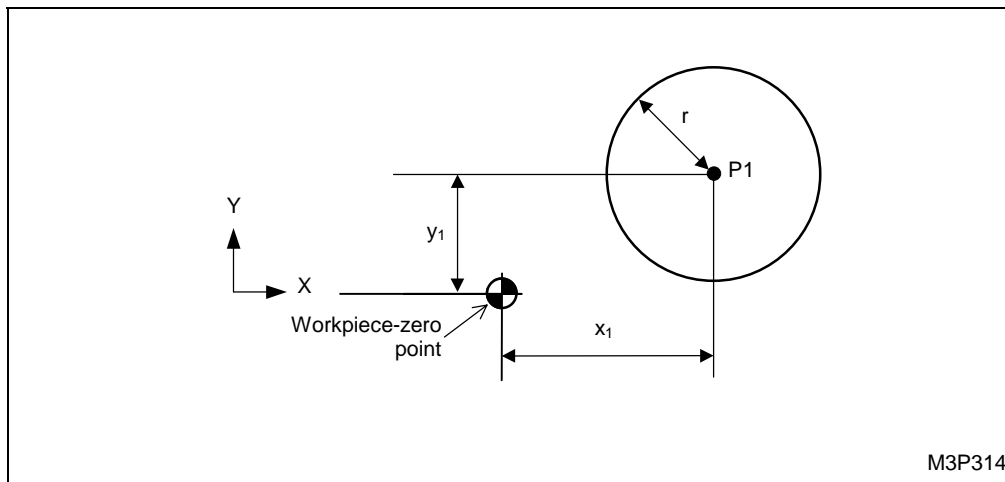
UNO.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE OUT	5.	5.	5.	1	0.	0.						
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	10.A		?	?	CW	G01	5.	◆	15	0.046		
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4	← Shape sequence data			
1	SQR	200.	100.	500.	300.	0.	C20.	0.	C20.				

Example 2:



UNO.	UNIT	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
1	POCKET	5.	5.	1	1	0.	0.						
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	10.A		?	?	CW	G01	5.	5.				
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4	← Shape sequence data			
1	SQR	200.	100.	500.	300.	0	R20.	0.	R20.				

2. CIRCLE (fixed form)



(a) Menu selection

After setting tool sequence data of the line/face machining unit, the following menu will be displayed.

PATTERN OF FIGURE <MENU>? ()									
SQUARE	CIRCLE		APBITRY				SHAPE END	CHECK	

Press the menu key **CIRCLE**.

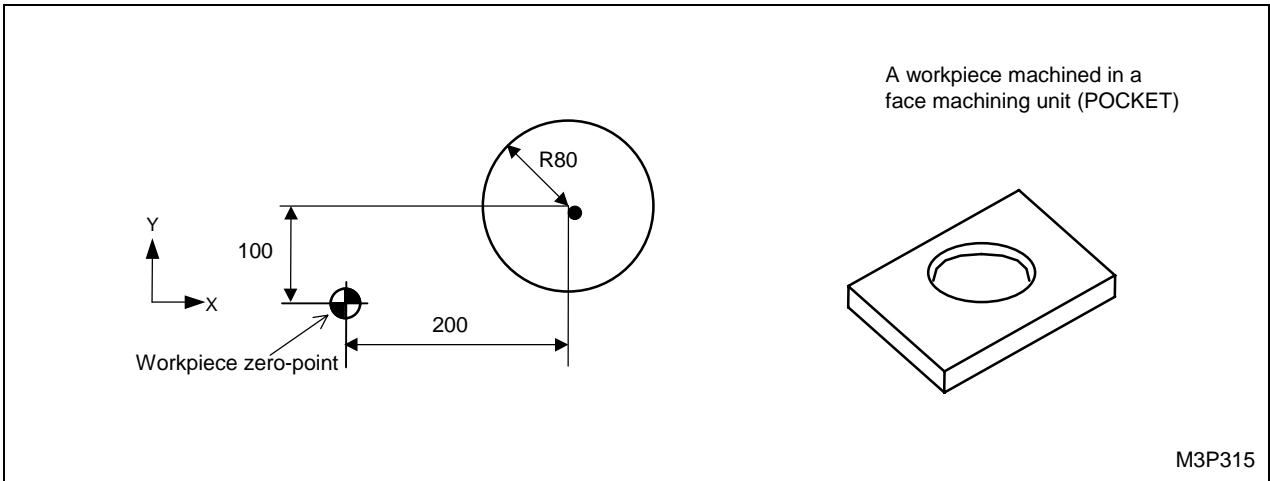
(b) Data setting in shape sequence CIRCLE (see figure above)

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
2	CIR	x_1	y_1	r	◆	◆	◆	◆	◆

◆: Data are not necessary to be set here.

Cursor position	Description
P1X/CX	Enter the X coordinate of the center of a circle (x_1).
P1Y/CY	Enter the Y coordinate of the center of a circle (y_1).
P3X/R	Enter the radius of the circle to be machined (r).

Example:



UNo.	UNIT	DEPTH	SRV-Z		BTM	WAL	FIN-Z	FIN-R					
1	POCKET	5.	5.		1	1	0.	0.					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	10.A		?	?	CW	G01	5.	5.				
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4	← Shape sequence data			
1	CIR	200.	100.	80.	◆	◆	◆	◆	◆				

B. Arbitrary

1. Selection of arbitrary form

After setting tool sequence data of the line/face machining unit, the following menu will be displayed.

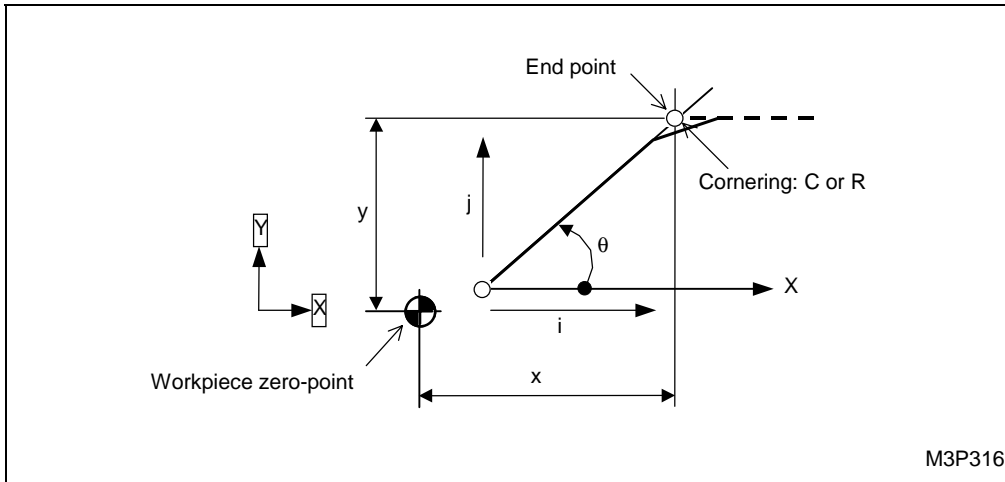
PATTERN OF FIGURE <MENU>? ()									
SQUARE	CIRCLE		APBITRY				SHAPE	CHECK	
							END		

Press the menu key **ARBITRY**.

➔ The following menu will be displayed.

LINE	CW ARC	CCW ARC	SHAPE	SHAPE	REPEAT	STARTING	SHAPE	CHECK	
			ROTATE	SHIFT	END	POINT	END		

2. LINE



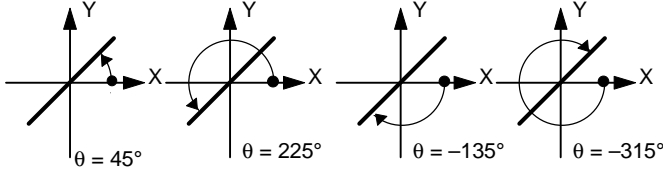
M3P316

(a) Menu selection

Press the menu key **LINE**.

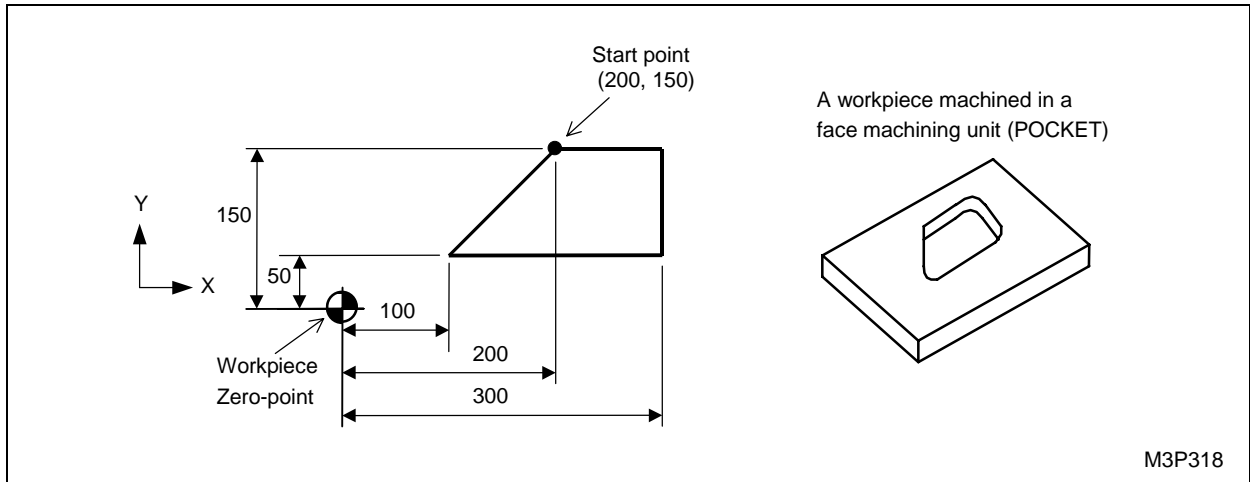
(b) Data setting in shape sequence LINE (see figure above)

FIG	PTN	X	Y	R/th	I	J	P	CNR
3	LINE	x	y	θ	i	j	p	C (R)

Cursor position	Description				
X	Enter the X coordinate of an end point for line machining (x). If it is unknown, select the menu key ?.				
Y	Enter the Y coordinate of an end point for line machining (y). If it is unknown, select the menu key ?.				
R/th	Enter the angle between X-axis and machining line (θ). Example: Four θs given below signify the identical line. 				
I	Enter the X-axial vector value (i).				
J	Enter the Y-axial vector value (j).				
P	Select the conditions of intersecting with the following forms (P): <table border="1" style="display: inline-table; vertical-align: middle;"> <tr> <td>UP</td> <td>DOWN</td> <td>LEFT</td> <td>RIGHT</td> </tr> </table> Four types given to the left are available. Note: For details refer to "C. Function of automatically calculating a point of intersection."	UP	DOWN	LEFT	RIGHT
UP	DOWN	LEFT	RIGHT		
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C				

M3P317

Example:



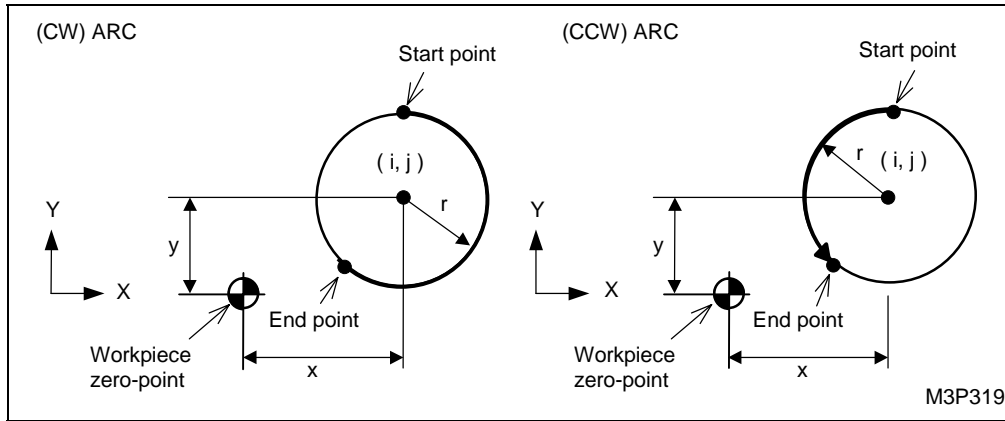
M3P318

UNo.	UNIT	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	POCKET	5.		5.		1	1	0.	0.				
R1	END MILL	10.A		?	?	CW	G01	5.	6.	12	0.03		
FIG	PTN	X		Y	R/th	I	J	P	CNR				
1	LINE	200.		150.									
2	LINE	300.		?		1.	0.						
3	LINE	?		50.	90.								
4	LINE	100.		?	↑	-1.	0.						

← Shape sequence data

(th = -90°, 270° or -270° is applicable.)

3. ARC (CW and CCW)



(a) Menu selection

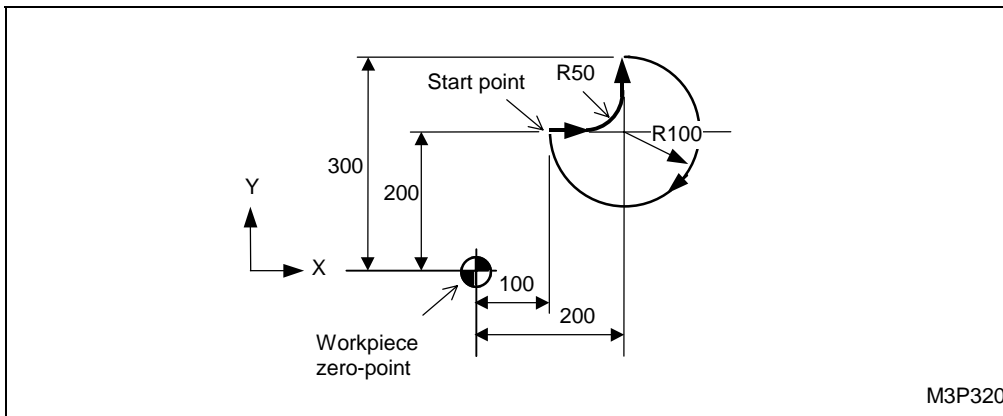
Press the menu keys **CW ARC** or **CCW ARC**.

(b) Data setting in shape sequence CW/CCW ARC. (see figure above)

FIG	PTN	X	Y	R/th	I	J	P	CNR
4	CW (CCW)	x	y	r	i	j	p	C (R)

Cursor position	Description				
X	Enter the X coordinate of an end point (x). If it is unknown, select the menu key ?.				
Y	Enter the Y coordinate of an end point (y). If it is unknown, select the menu key ?.				
R /th	Enter the radius of an arc (r). If it is unknown, select the menu key ?.				
I	Enter the X coordinate value of the center of an arc (i). If it is unknown, select the menu key ?.				
J	Enter the Y coordinate value of the center of an arc (j). If it is unknown, select the menu key ?.				
P	Select the conditions of intersecting with the following forms (P): <table border="1" style="display: inline-table; vertical-align: middle;"> <tr> <td style="text-align: center;">UP</td> <td style="text-align: center;">DOWN</td> <td style="text-align: center;">LEFT</td> <td style="text-align: center;">RIGHT</td> </tr> </table> Four types given to the left are available. Note: For details refer to "C. Function of automatically calculating a point of intersection."	UP	DOWN	LEFT	RIGHT
UP	DOWN	LEFT	RIGHT		
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly. Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value. Corner C				

Example 1:

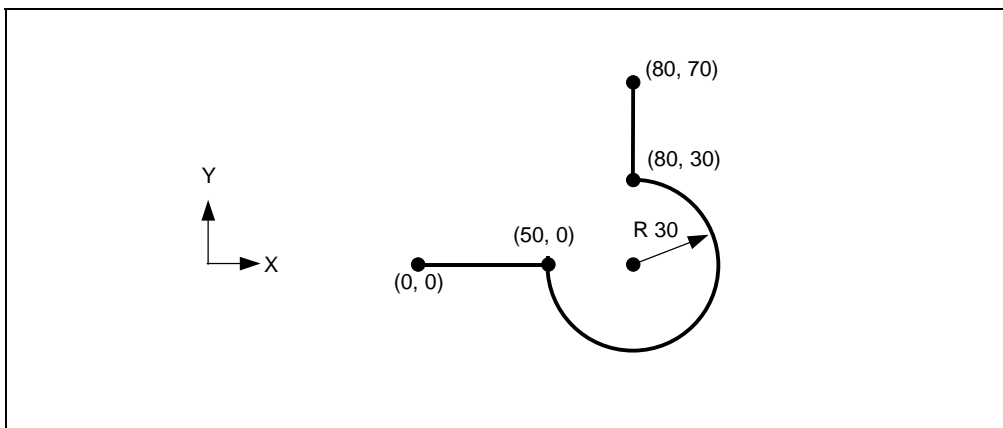


M3P320

UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE OUT	5.	5.	5.	1	0.	0.						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	10.A	?	?	CCW	G01	5.	◆	17	0.006			
FIG	PTN	X	Y	R/th	I	J	P	CNR					
1	CW	100.	200.	100.	200.	200.							
2	LINE	150.	200.	0.									
3	CCW	200.	250.	50.	150.	250.							
4	LINE	200.	300.	90.									

← Shape sequence data

Example 2:

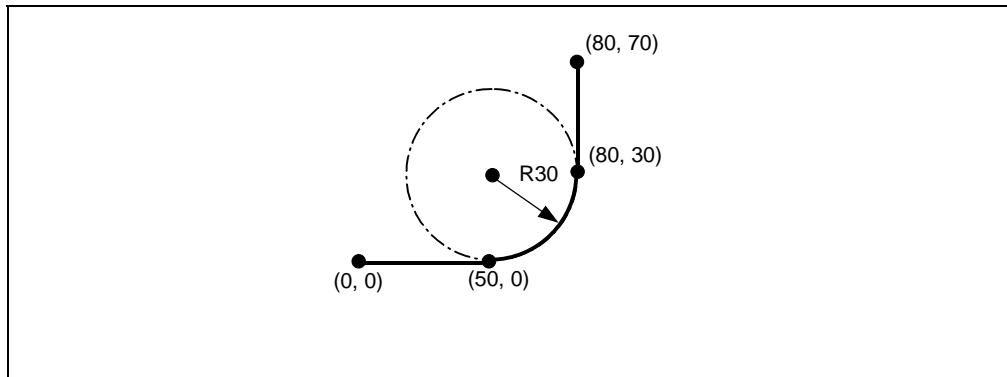


UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
2	LINE LFT	5.	5.	10.	3	0.1	0.1						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	20.A	?	?	◆	4.9	◆	76	0.123				
F2	END MILL	20.A	?	?	◆	◆	◆	76	0.313				
FIG	PTN	X	Y	R/th	I	J	P	CNR					
1	LINE	0.	0.										
2	LINE	50.	0.										
3	CCW	80.	30.	-30.*									
4	LINE	80.	70.										

← Shape sequence data

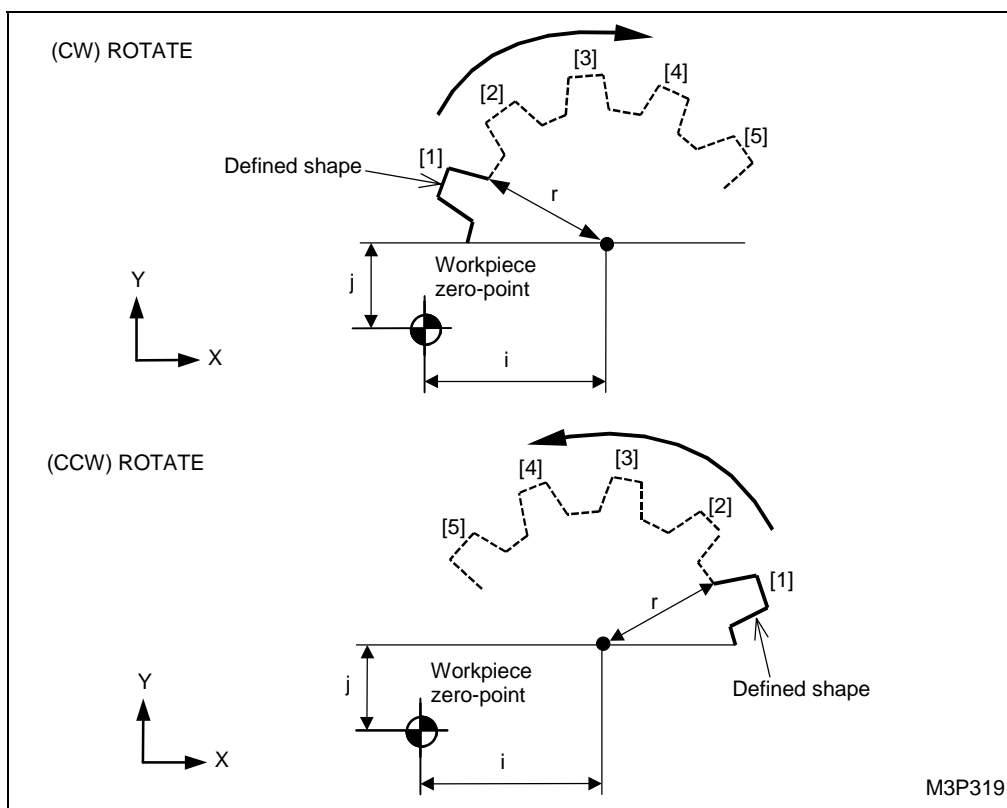
- For "CW ARC" and "CCW ARC," if the angle of the arc is more than 180 degrees, enter a minus (-) value as radius R. When the central coordinates (I, J) of the arc have been specified, however, radius R can be entered using a plus (+) sign.

- * If "30" is entered as R, the arc will appear as shown in the diagram below.



- With "CNR" specified, R can be entered using either a plus sign or a minus sign.

4. SHAPE ROTATE (CW and CCW)



(a) Menu selection

Press the menu keys **SHAPE ROTATE** and **CW SHIFT** or **CCW SHIFT** in this order.

(b) Data setting in shape sequence CW/CCW SHIFT(see figure above)

FIG	PTN	X	Y	R/θ	I	J	P	CNR
5	CW-SH (CCW-SH)	◆	◆	r	i	j	p	◆
Use LINE, CW ARC or CCW ARC to enter a defined shape.								
REP-EN		◆	◆	◆	◆	◆	◆	◆

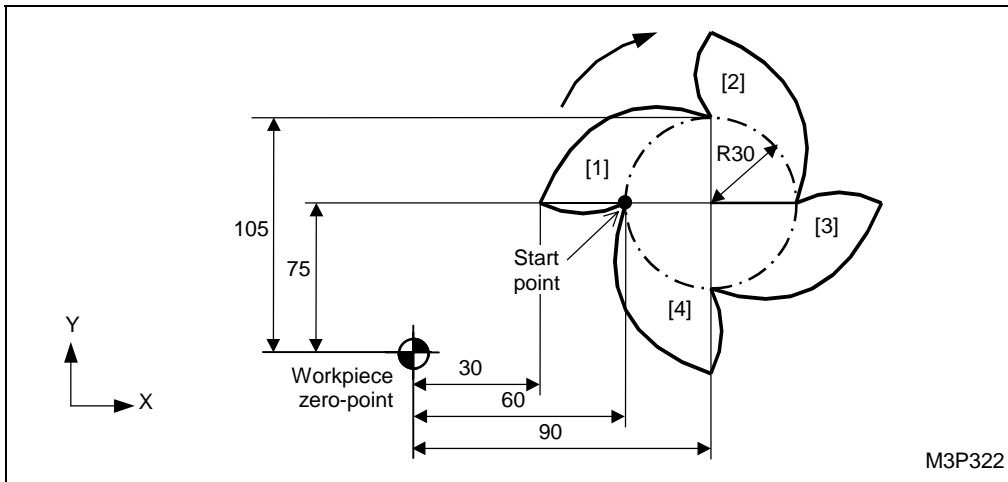
◆: Data are not necessary to be set here.

Cursor position	Description
R/th	Enter the radius to rotate a defined shape (r). If it is unknown, select the menu key ?.
I	Enter the X coordinate of the center to rotate a defined shape (i). If it is unknown, select the menu key ?.
J	Enter the Y coordinate of the center to rotate a defined shape (j). If it is unknown, select the menu key ?.
P	Enter the number of defined shape repetitions (p).

(c) REPEAT END

Press the menu key **REPEAT END** and a shape sequence of SHAPE ROTATE (CW or CCW) will be brought to the end.

Example: CW SHIFT

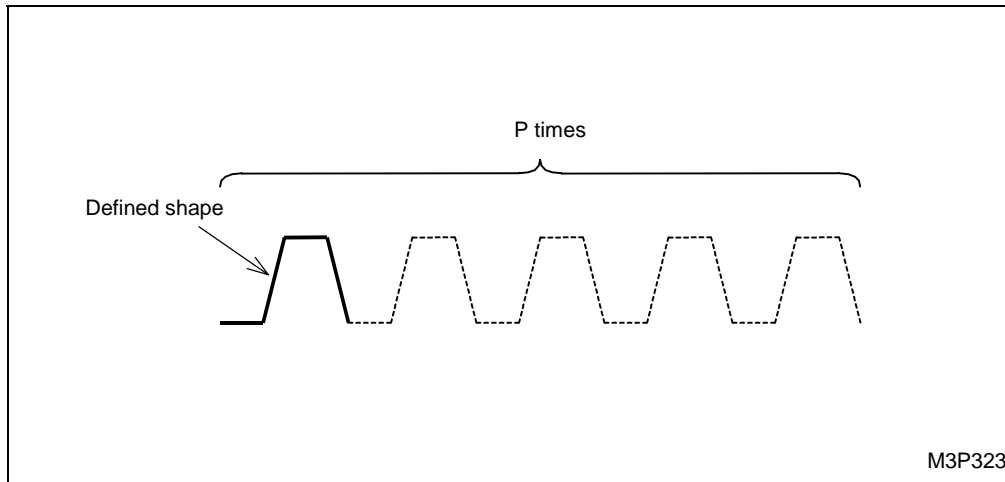


UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE OUT	10.	10.	10.	4	0.6	0.6						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	20.A	?	?	CW	G01	9.4	◆	14	0.043			
F2	END MILL	10.A	?	?	CW	G01	◆	◆	20	0.224			
FIG	PTN	X	Y	R/th	I	J	P	CNR					
1	CW-SH	◆	◆	30.	90.	75.	4	◆					
2	LINE	60.	75.										
3	CW	30.	75.	100.	45.	75.							
4	CW	90.	105.	50.									
5	REP-EN	◆	◆	◆	◆	◆	◆	◆					

← Shape sequence data

5. SHAPE SHIFT

The end point will be regarded as the next start point.



(a) Menu selection

Press the menu key **SHAPE SHIFT**.

(b) Data setting in shape sequence SHAPE SHIFT (see figure above)

FIG	PTN	X	Y	R/th	I	J	P	CNR
6	FIG-SH	◆	◆	◆	◆	◆	p	◆
Use LINE, CW ARC or CCW ARC to enter a defined shape.								
	REP-EN	◆	◆	◆	◆	◆	◆	◆

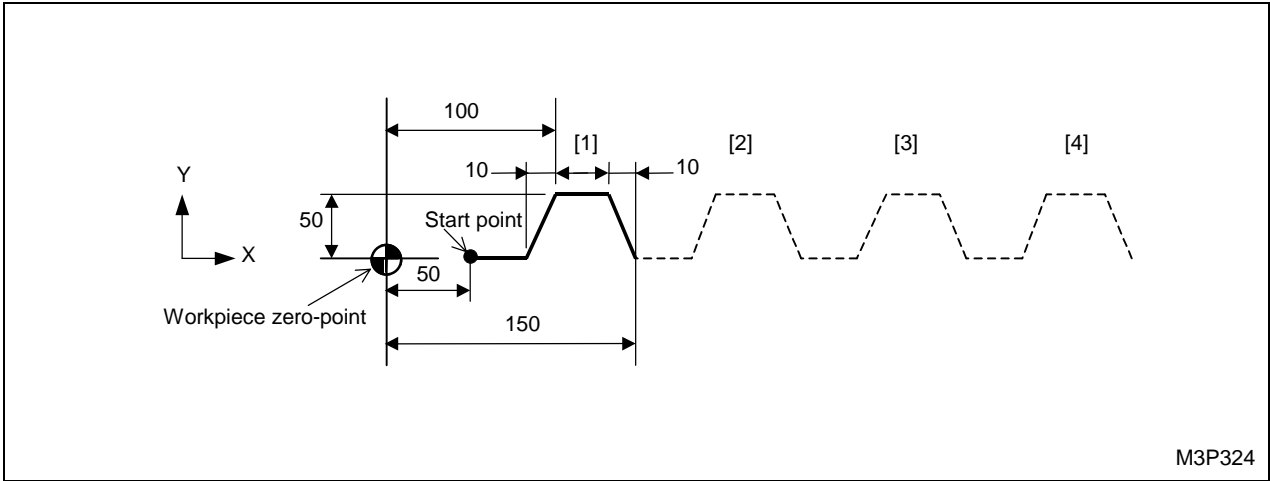
◆ : Data are not necessary to be set here.

Cursor position	Description
P	Enter the number of repetitions for a shape defined from the menu key SHAPE SHIFT to REPEAT END .

(c) REPEAT END

Press the menu key **REPEAT END** and a shape sequence of SHAPE SHIFT will be brought to the end.

Example:



UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R						
1	LINE CTR	5.	5.	5.	1	0.	◆						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL	10.A		?	?	◆	G01	5.	◆	17	0.066		
FIG	PTN	X	Y	R/th	I	J	P	CNR					
1	FIG-SH	◆	◆	◆	◆	◆	4	◆					
2	LINE	50.	0.										
3	LINE	90.	0.										
4	LINE	100.	50.										
5	LINE	140.	50.										
6	LINE	150.	0.										
7	REP-EN	◆	◆	◆	◆	◆	◆	◆	◆				

← Shape sequence data

C. Function of automatically calculating a point of intersection

Automatically calculating a point of intersection in the 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.

1. Point-of-Intersection X, Y coordinates (?, ?)

Even if X, Y coordinates of a point of intersection are unknown as illustrated above, the NC system will automatically obtain it from the coordinates of the start and end points and from angles involved.

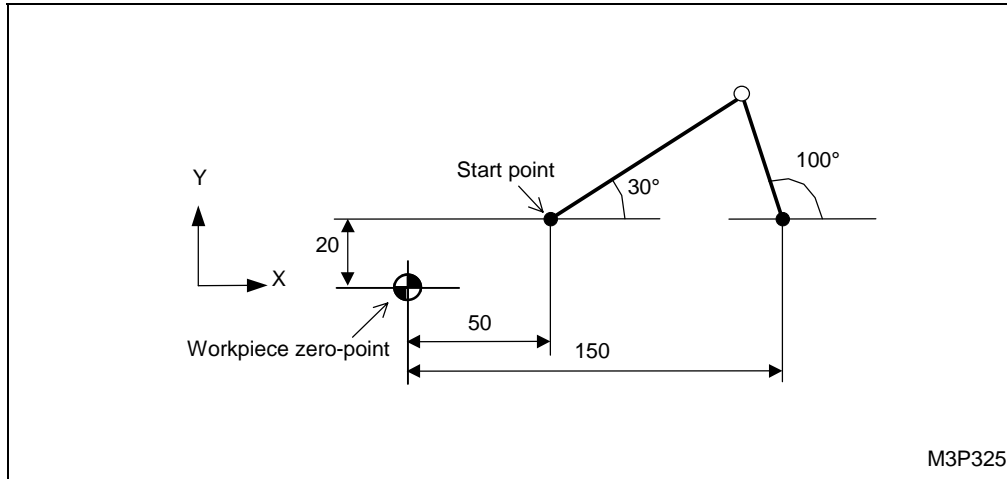


Fig. 4-22 Function of automatically calculating a point of intersection

FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	50.	20.					
2	LINE	?	?	30.				
3	LINE	150.	20.	100.				

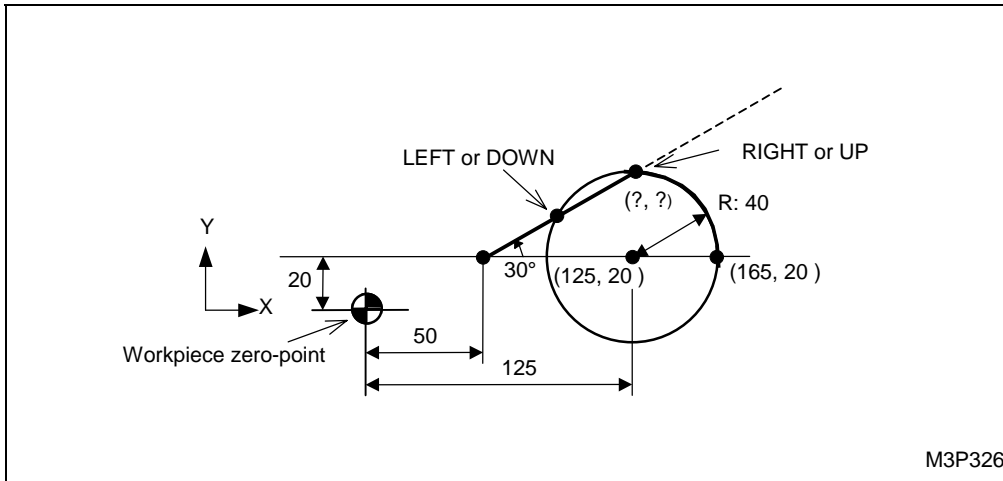
↓

FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	50.	20.					
2	LINE	140.76	72.4	30.				
3	LINE	150.	20.	100.				

Displayed in yellow

After checking the plane, return to the **PROGRAM** display again and the numeric value so automatically obtained as a point of intersection will be displayed in yellow.

Note: When unknown X, Y coordinates of a point of intersection 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 point of intersection.)



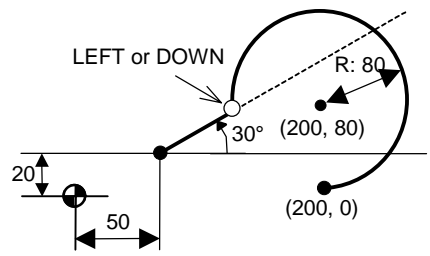
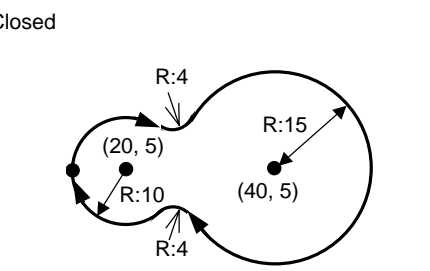
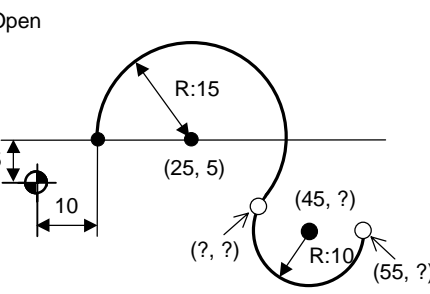
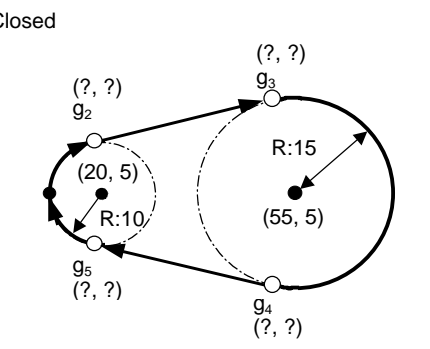
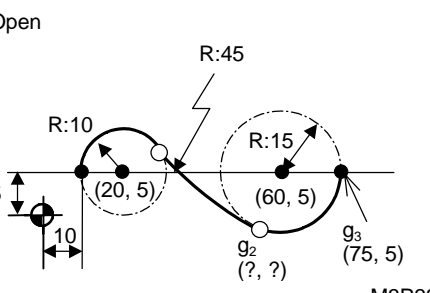
M3P326

FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	50.	20.					
2	LINE	?	?	30.				
3	CW	165.	20.	40.	125.	20.		

2 Examples of automatic intersection point calculation

A point of intersection 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		<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>50.</td> <td>20.</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>LINE</td> <td>?</td> <td>?</td> <td>30.</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>3</td> <td>LINE</td> <td>150.</td> <td>20.</td> <td>120.</td> <td></td> <td></td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	LINE	50.	20.						2	LINE	?	?	30.					3	LINE	150.	20.	120.				
FIG	PTN	X	Y	R/th	I	J	P	CNR																														
1	LINE	50.	20.																																			
2	LINE	?	?	30.																																		
3	LINE	150.	20.	120.																																		
LINE ARC (Contacting)		<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>50.</td> <td>20.</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>LINE</td> <td>?</td> <td>?</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>3</td> <td>CW</td> <td>150</td> <td>20.</td> <td>30.</td> <td>120.</td> <td>20.</td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	LINE	50.	20.						2	LINE	?	?						3	CW	150	20.	30.	120.	20.		
FIG	PTN	X	Y	R/th	I	J	P	CNR																														
1	LINE	50.	20.																																			
2	LINE	?	?																																			
3	CW	150	20.	30.	120.	20.																																

Pattern	Shape	Shape sequence																																													
LINE ARC (Intersecting)	 <p style="text-align: right;">M3P329</p>	<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>50.</td> <td>20.</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>LINE</td> <td>?</td> <td>?</td> <td>30.</td> <td></td> <td></td> <td>L</td> <td></td> </tr> <tr> <td>3</td> <td>CW</td> <td>200.</td> <td>0.</td> <td>80.</td> <td>200</td> <td>80.</td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	LINE	50.	20.						2	LINE	?	?	30.			L		3	CW	200.	0.	80.	200	80.											
FIG	PTN	X	Y	R/th	I	J	P	CNR																																							
1	LINE	50.	20.																																												
2	LINE	?	?	30.			L																																								
3	CW	200.	0.	80.	200	80.																																									
ARC ARC	<p>Closed</p>  <p style="text-align: right;">M3P330</p>	<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>CW</td> <td>?</td> <td>?</td> <td>10.</td> <td>20.</td> <td>5.</td> <td>U</td> <td>R4</td> </tr> <tr> <td>2</td> <td>CW</td> <td>?</td> <td>?</td> <td>15.</td> <td>40.</td> <td>5.</td> <td>D</td> <td>R4</td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	CW	?	?	10.	20.	5.	U	R4	2	CW	?	?	15.	40.	5.	D	R4																		
FIG	PTN	X	Y	R/th	I	J	P	CNR																																							
1	CW	?	?	10.	20.	5.	U	R4																																							
2	CW	?	?	15.	40.	5.	D	R4																																							
ARC ARC	<p>Open</p>  <p style="text-align: right;">M3P331</p>	<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>10.</td> <td>5.</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>CW</td> <td>?</td> <td>?</td> <td>15.</td> <td>25.</td> <td>5.</td> <td>D</td> <td></td> </tr> <tr> <td>3</td> <td>CCW</td> <td>55.</td> <td>?</td> <td>10.</td> <td>45.</td> <td>?</td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	LINE	10.	5.						2	CW	?	?	15.	25.	5.	D		3	CCW	55.	?	10.	45.	?											
FIG	PTN	X	Y	R/th	I	J	P	CNR																																							
1	LINE	10.	5.																																												
2	CW	?	?	15.	25.	5.	D																																								
3	CCW	55.	?	10.	45.	?																																									
ARC LINE ARC	<p>Closed</p>  <p style="text-align: right;">M3P332</p>	<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>?</td> <td>?</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>CW</td> <td>?</td> <td>?</td> <td>10.</td> <td>20.</td> <td>5.</td> <td></td> <td></td> </tr> <tr> <td>3</td> <td>LINE</td> <td>?</td> <td>?</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>4</td> <td>CCW</td> <td>?</td> <td>?</td> <td>15.</td> <td>55.</td> <td>5.</td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	X	Y	R/th	I	J	P	CNR	1	LINE	?	?						2	CW	?	?	10.	20.	5.			3	LINE	?	?						4	CCW	?	?	15.	55.	5.		
FIG	PTN	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	<p>Open</p>  <p style="text-align: right;">M3P333</p>	<table border="1"> <thead> <tr> <th>FIG</th> <th>PTN</th> <th>X</th> <th>Y</th> <th>R/th</th> <th>I</th> <th>J</th> <th>P</th> <th>CNR</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>LINE</td> <td>10.</td> <td>5.</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>2</td> <td>CW</td> <td>?</td> <td>?</td> <td>10.</td> <td>20.</td> <td>5.</td> <td></td> <td></td> </tr> <tr> <td>3</td> <td>CCW</td> <td>?</td> <td>?</td> <td>45.</td> <td></td> <td></td> <td></td> <td></td> </tr> <tr> <td>4</td> <td>CCW</td> <td>75.</td> <td>5.</td> <td>15.</td> <td>60.</td> <td>5.</td> <td></td> <td></td> </tr> </tbody> </table>	FIG	PTN	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.		
FIG	PTN	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.																																									

Remark: ● : Both X and Y coordinates are known (i, j in the case of the center of an arc).
○ : Both X and Y coordinates are not known (i, j in the case of the center of an arc).

4-10 End Unit (END)

This unit is set at the end of the program.

This unit indicates the end of the program concerned.

Note: When this unit is specified in the middle of a program, the part of the program after the unit thus specified is not executed.

1. Menu selection

Press the menu key **END**.

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

2. Data setting in end unit

UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE (5)
1	END	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Note: Instead of “4” or “5”, the axis name is indicated on the NC display.

Cursor position	Description
CONTI.	<p>Specify if the program is repeated or not.</p> <p>- No repetition <input type="checkbox"/> 0 </p> <p>- Repetition <input type="checkbox"/> 1 </p> <p>Note : Set 1 for the case of sub-program.</p>
NUMBER	<p>Specify if the number of executions of the program is displayed or not by the counter on the POSITION display.</p> <p>- No display <input type="checkbox"/> 0 </p> <p>- Display <input type="checkbox"/> 1 </p> <p>Note : The number of target times is entered in the POSITION display. Moreover, the cumulative value can be reset to zero.</p>
ATC	<p>Specify the ATC movement at the end of machining.</p> <p>The tool is not returned. <input type="checkbox"/> 0 </p> <p>After returning the tool, the axes are moved. <input type="checkbox"/> 1 </p> <p>After the axes movement, the tool is returned. <input type="checkbox"/> 2 </p> <p>Note 1: If no data is set, it is regarded that 1 has been set.</p> <p>Note 2: If 1 is set in the item CONTI, is displayed and no data can be entered.</p>

Cursor position	Description
X/Y/Z/4/5	<p>The axes are moved in rapid feed to the position entered in the items X, Y, Z, 4 and 5 in the machine coordinates system at the end of machining. The movement is done on Z-axis, X-axis, Y-axis, 4th axis or index table and 5th axis in this order. If 1 is set in the item ATC, X-, Y- and Z-axes are moved at the same time.</p> <p>Note 1: If the coordinates are not specified, the axes movement are not done.</p> <p>Note 2: If 1 is set in the item CONTI., ◆ are displayed in the items X, Y, Z, 4, 5 and ANGLE and no data can be set.</p>
X	<p>Enter the X-coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999)</p>
Y	<p>Enter the Y-coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999)</p>
Z	<p>Enter the Z-coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999)</p>
B	<p>Enter the B-coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±9999.999)</p>
C	<p>Enter the C-coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±9999.999)</p>

4-11 M-Code Unit (M-CODE)

This is for making the machine execute a special movement other than machining.

The respective movement (operations) are numbered. For details, refer to “List of M-codes” in the Operating Manual.

1. Menu selection

Press the menu key **M CODE**.

C-POINT MACH-ING	C-LINE MACH-ING	INDEX	M CODE	SUB PROGRAM	MMS	WORKPIECE MEASURE	TOOL MEASURE	WORKPIECE SHAPE	>>>
---------------------	--------------------	-------	--------	----------------	-----	----------------------	-----------------	--------------------	-----

2. Data setting in M-code unit

UNo.	UNIT	No.	M1	M2	M3	M4	M5	M6	M7	M8	M9	M10	M11	M12
1	M-CODE	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description																																				
No.	<p>Specify the priority machining number (prior machining, subsequent machining). There are the following three entry methods:</p> <p>(1) Prior machining number: To be entered by means of numeric keys. Permissible input range (0 to 99)</p> <p>(2) Subsequent machining number: To be entered by means of numeric keys after having pressed the menu key DELAY PRIORITY. Permissible input range (0 to 99)</p> <table border="1" style="width: 100%; text-align: center;"> <tr> <td colspan="10">MACHINING PRIORITY No. ? ()</td> </tr> <tr> <td>DELAY PRIORITY</td> <td></td> <td>PRI.No. CHANGE</td> <td>PRI.No. ASSIGN</td> <td></td> <td>PRI.No. ALL ERAS</td> <td>SUB PROG PROC END</td> <td></td> <td></td> <td></td> <td></td> </tr> </table> <p>(3) Without entry: ordinary machining For details, refer to Chapter 5, “PRIORITY FUNCTION FOR THE SAME TOOL.”</p>	MACHINING PRIORITY No. ? ()										DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END																			
MACHINING PRIORITY No. ? ()																																					
DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END																															
M1 to M12	<p>Specify the desired operation during the machining of a workpiece. Pressing the menu key >>> changes the menu a → b → c → a in this order.</p> <table border="1" style="width: 100%; text-align: center;"> <tr> <td>01 OPT. STOP</td> <td>03 SPNDL FWD</td> <td>04 SPNDL REV</td> <td>05 SPNDL STOP</td> <td>07 MIST COOLANT</td> <td>08 FLOOD COOLANT</td> <td>09 OFF COOLANT</td> <td>50 AIR BLAST</td> <td></td> <td></td> <td>>>></td> <td>a</td> </tr> <tr> <td>00 PROG STOP</td> <td>19 SPNDL ORIENT</td> <td>35 T-BRK DETECT</td> <td>38 SPNDL GEAR L/M</td> <td>39 SPNDL GEAR H</td> <td>33 OUT MSR UNIT</td> <td>34 IN MRS UNIT</td> <td></td> <td></td> <td></td> <td>>>></td> <td>b</td> </tr> <tr> <td>16 OPEN ATC CVR</td> <td>15 CLOSE ATC CVR</td> <td>11 TOOL UNCLAMP</td> <td>10 TOOL CLAMP</td> <td>53 CHIP VAC ON</td> <td>09 CHIP VAC OFF</td> <td></td> <td></td> <td></td> <td></td> <td>>>></td> <td>c</td> </tr> </table> <p>For the M-codes other than those of the displayed menu, enter the number of required M-code by referring to “List of M-codes” in the Operating Manual using the numeric key. Assigned M-codes are simultaneously sent in groups of four: M1 to M4 first, then M5 to M8, and M9 to M12, in that order.</p> <p>Note 1: When the M-code is not entered under the M-code unit, the alarm “616 DATA ERROR IN M CODE UNIT” will be displayed.</p> <p>Note 2: When M-code mirror image command is given, M91 and M92 command cannot be given in the same block. Give M91 command in M1 to M4, M92 command in M5 to M8 respectively.</p> <p>Note 3: The M-codes listed in the menu vary from machine to machine.</p>	01 OPT. STOP	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST COOLANT	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST			>>>	a	00 PROG STOP	19 SPNDL ORIENT	35 T-BRK DETECT	38 SPNDL GEAR L/M	39 SPNDL GEAR H	33 OUT MSR UNIT	34 IN MRS UNIT				>>>	b	16 OPEN ATC CVR	15 CLOSE ATC CVR	11 TOOL UNCLAMP	10 TOOL CLAMP	53 CHIP VAC ON	09 CHIP VAC OFF					>>>	c
01 OPT. STOP	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST COOLANT	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST			>>>	a																										
00 PROG STOP	19 SPNDL ORIENT	35 T-BRK DETECT	38 SPNDL GEAR L/M	39 SPNDL GEAR H	33 OUT MSR UNIT	34 IN MRS UNIT				>>>	b																										
16 OPEN ATC CVR	15 CLOSE ATC CVR	11 TOOL UNCLAMP	10 TOOL CLAMP	53 CHIP VAC ON	09 CHIP VAC OFF					>>>	c																										

4-12 Sub-Program 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 sub-program unit.

In this case, the calling side is referred to as the Main program and the called side is referred to as the Sub-program. Moreover, the call of the sub-program is called Nesting. The sub-program 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.

1. Menu selection

Press the menu key **SUB PROGRAM**.

C-POINT	C-LINE	INDEX	M CODE	SUB	MMS	WORKPIECE	TOOL	WORKPIECE	>>>
MACH-ING	MACH-ING			PROGRAM		MEASURE	MEASURE	SHAPE	

2. Data setting in sub-program unit

UNo.	UNIT	WORK No.	REPEAT			
	SUB PRO	<input type="checkbox"/>	<input type="checkbox"/>			
SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
WORK No.	Enter the workpiece number of the sub-program to be used by means of numeric keys. Use the menu key MEASURE MACRO only when the coordinates stored within the fundamental coordinates unit of the MAZATROL program is to be rewritten using a user macroprogram.
REPEAT	Enter the number of repetitions of the specified sub-program. Note : If no value has been entered, the sub-program will be performed only one time.
ARGM 1 to ARGM 6	Enter the address of the argument using the alphabet key. Enter the data corresponding to the address by means of numeric keys. Permissible input range (0 to ±99999.999) G, L, N, O and P cannot be used as address of the argument.

Note 1: The auxiliary coordinates system specified in the main program is also valid in the sub-program, but that specified in the sub-program is only valid in the said-sub-program. At the time of return to the main program, the auxiliary coordinates system which was valid before the use of the sub-program, becomes valid.

Note 2: When the fundamental coordinates system is entered again in the sub-program, the auxiliary coordinates system and fundamental coordinates system in the main program become void.

Note 3: Even when 0 is selected for the number of repetitions of the sub-program, it is executed once.

Note 4: For details concerning the following items, refer to the Programming Manual for EIA/ISO Programming.

- Calling up the EIA/ISO programs
- Description of the user macroprogram
- Rewriting the fundamental coordinates system of the MAZATROL program

4-13 Pallet Changing Unit (PALT CHG)

This is the unit for changing the pallet.

1. Menu selection

Press the menu key **PALLET CHANGE**.

SELECT HEAD	TRANSFER WORKPIECE	PROCESS END	PALLET CHANGE	WPC SHIFT					
----------------	-----------------------	----------------	------------------	--------------	--	--	--	--	--

2. Data setting in pallet changing unit

UNo.	UNIT	PALLET No.
	PALT CHG	<input type="checkbox"/> (<input type="checkbox"/>)

Cursor position	Description
PALLET No.	Enter the number of the pallet on which the machining will be performed, by means of numeric keys.
PALLET No. ()	Enter the number of pallet to be changed later, by means of numeric keys. However, this entering is only possible for the preparation specifications of the following pallet.

Note 1: The entering of this unit results in the delimiting of the range of priority numbers. In order to change the pallet in the program containing the priority function for the same tool, use the pallet changing unit in principle. For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

Note 2: The pallet number presently located on the table is displayed on the **POSITION** display.

3. Simultaneous movement of pallet change and ATC

With the parameter **L49 = 1**, the ATC for the tool to be used next is executed at the same time as the execution of the pallet change.

4-14 Process End Unit (PROC END)

This serves to delimit the useful scope of the priority number.

- (1) Move the cursor at the line (upper line) that delimits the useful scope of the priority number.

```

UNo.      UNIT
          DRILLING
SNo.      TOOL
  1       CTR-DR
          .
          .
UNo.      UNIT
          DRILLING

```

← Place the cursor on this line.
(When this line is to be the upper line of the useful scope.)

- (2) Press the menu selector key (key located at the right of the menu keys) and display the following editing menu.

PROGRAM COMPLETE	SEARCH	CALCULAT	TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP
---------------------	--------	----------	-----	--------	-------	---------------	--------------	-----------------	------

- (3) Press the menu key **INSERT**.

- (4) Press the menu key **PROCESS END** in the menu.

➔ The upper limit of the useful scope is set as follows.

```

UNo.      UNIT
          DRILLING
SNo.      TOOL
          CTR-DR
          .
          .
UNo.      UNIT
          PROC END
UNo.      UNIT
          DRILLING

```

- (5) Move the cursor at the line (lower line) that delimits the useful scope of the priority number.

```

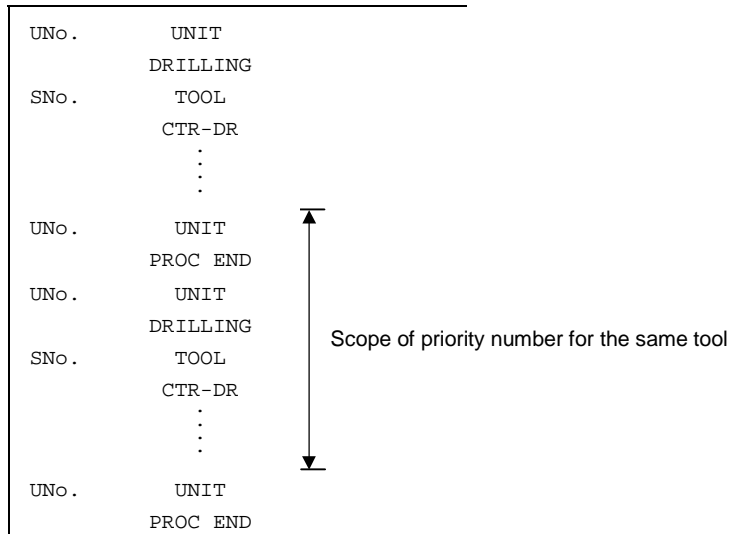
UNo.      UNIT
          DRILLING
SNo.      TOOL
  1       CTR-DR
          .
          .
UNo.      UNIT
          PROC END
SNo.      TOOL
  1       CTR-DR
          .
          .
UNo.      UNIT

```

← Place the cursor on this line.
(When this line is to be the upper line of the useful scope.)

(6) Insert the PROCESS END unit by the same operation as (2) to (5).

➔ The lower limit of the useful scope is set as follows.



Note: The scope of the priority number can also be delimited by entering of the pallet changing unit. For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

4-15 Manual Program Mode Unit (MANL PRG)

This unit serves for programming in detail each of the movement of the machine.

4-15-1 Menu selection
















Press the menu key **MANUAL PROGRAM**.

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

4-15-2 Unit data setting

1. Data setting in manual program mode unit

UNO.	UNIT	TOOL	NOM-φ	No.
	MANL PRG	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description										
TOOL	Select menu key corresponding to the tool used. Pressing the menu key >>> changes the menu a → b → c → a in this order.										
	<table border="1"> <tr> <td>ENDMILL</td> <td>FACEMILL</td> <td>CHAMFER CUTTER</td> <td>BALL ENDMILL</td> <td>OTHER TOOL</td> <td>TOUCH SENSOR</td> <td></td> <td></td> <td>>>></td> <td>a</td> </tr> </table>	ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR			>>>	a
	ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR			>>>	a	
<table border="1"> <tr> <td>CENTER DRILL</td> <td>DRILL</td> <td>BACKSPO T FACER</td> <td>REAMER</td> <td>TAP</td> <td>BACK BOR. BAR</td> <td>CHIP VACUUM</td> <td></td> <td>>>></td> <td>b</td> </tr> </table>	CENTER DRILL	DRILL	BACKSPO T FACER	REAMER	TAP	BACK BOR. BAR	CHIP VACUUM		>>>	b	
CENTER DRILL	DRILL	BACKSPO T FACER	REAMER	TAP	BACK BOR. BAR	CHIP VACUUM		>>>	b		
	<table border="1"> <tr> <td>CENERAL </td> <td>GROOVE </td> <td>THREAD </td> <td>T. DRILL </td> <td>T. TAP </td> <td></td> <td>SPECIAL</td> <td></td> <td>>>></td> <td>c</td> </tr> </table>	CENERAL 	GROOVE 	THREAD 	T. DRILL 	T. TAP 		SPECIAL		>>>	c
CENERAL 	GROOVE 	THREAD 	T. DRILL 	T. TAP 		SPECIAL		>>>	c		
	Note: The tool number will be 0 if no tool selection is made.										
NOM-φ	Enter the nominal diameter of the tool by means of numeric keys. Input range (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:										
	<table border="1"> <tr> <td>A</td> <td>B</td> <td>C</td> <td>D</td> <td>E</td> <td>F</td> <td>G</td> <td>H</td> <td>HEAVY TOOL</td> <td>>>></td> </tr> </table>	A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
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 menu key HEAVY TOOL .										
	Note: When one of the menu keys END MILL , FCE MILL , CHAMFER CUTTER and BALL ENDMILL is selected in the item TOOL, the alarm "434 NO ASSIGNED TOOL IN TOOL FILE" is displayed if the selected tool has not been previously recorded in the TOOL FILE display.										

Cursor position	Description																					
No .	<p>Determine the priority machining number (prior machining, subsequent machining). The input of data is done in accordance with the following 3 methods;</p> <ul style="list-style-type: none"> - Priority No. for prior machining: Enter the number by means of numeric keys. Input range (0 to 99) - Priority No. for subsequent machining: Press the menu key DELAY PRIORITY, then enter the number by means of numeric keys. Input range (0 to 99) <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td colspan="10" style="text-align: right;">MACHINING PRIORITY No. ? ()</td> </tr> <tr> <td style="width: 15%;"></td> <td style="width: 15%; text-align: center;">DELAY PRIORITY</td> <td style="width: 15%;"></td> <td style="width: 15%; text-align: center;">PRI.No. CHANGE</td> <td style="width: 15%; text-align: center;">PRI.No. ASSIGN</td> <td style="width: 15%;"></td> <td style="width: 15%; text-align: center;">PRI.No. ALL ERAS</td> <td style="width: 15%; text-align: center;">SUB PROG PROC END</td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> </tr> </table> <ul style="list-style-type: none"> - Without input: Normal machining. The machining order is not specified. <p>For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."</p>	MACHINING PRIORITY No. ? ()											DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END			
MACHINING PRIORITY No. ? ()																						
	DELAY PRIORITY		PRI.No. CHANGE	PRI.No. ASSIGN		PRI.No. ALL ERAS	SUB PROG PROC END															

4-15-3 Sequence data setting

The tool sequence is intended for selecting and specifying the optimal tool for the machining. In the manual program mode unit, one sequence data lines corresponds to one EIA/ISO program block.

For the details of each of the commands, refer to the Programming Manual for EIA/ISO Programming.

1. Sequence data setting in manual program mode unit

SNo.	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description																						
G1, 2	<p>Enter the G codes (preparatory function) by means of menu keys or numeric keys.</p> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 10%;"></td> <td style="width: 10%; text-align: center;">G00</td> <td style="width: 10%; text-align: center;">G01</td> <td style="width: 10%; text-align: center;">G02</td> <td style="width: 10%; text-align: center;">G03</td> <td style="width: 10%; text-align: center;">G40 CANCEL</td> <td style="width: 10%; text-align: center;">G41 LEFT</td> <td style="width: 10%; text-align: center;">G42 RIGHT</td> <td style="width: 10%; text-align: center;">MANUAL END</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> </tr> </table> <p>Note: Up to two G code commands can be designated in one sequence data line.</p>		G00	G01	G02	G03	G40 CANCEL	G41 LEFT	G42 RIGHT	MANUAL END													
	G00	G01	G02	G03	G40 CANCEL	G41 LEFT	G42 RIGHT	MANUAL END															
DATA 1 to DATA 6	<p>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.</p> <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 10%;"></td> <td style="width: 10%; text-align: center;">X</td> <td style="width: 10%; text-align: center;">Y</td> <td style="width: 10%; text-align: center;">Z</td> <td style="width: 10%; text-align: center;">4</td> <td style="width: 10%; text-align: center;">F</td> <td style="width: 10%; text-align: center;">R</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%; text-align: center;">>>></td> <td style="width: 10%; text-align: center;">a</td> </tr> <tr> <td style="width: 10%;"></td> <td style="width: 10%; text-align: center;">I</td> <td style="width: 10%; text-align: center;">J</td> <td style="width: 10%; text-align: center;">K</td> <td style="width: 10%; text-align: center;">P</td> <td style="width: 10%; text-align: center;">D</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%; text-align: center;">>>></td> <td style="width: 10%; text-align: center;">b</td> </tr> </table> <p>After the address has been set, enter and set data using the numeric keys.</p>		X	Y	Z	4	F	R			>>>	a		I	J	K	P	D				>>>	b
	X	Y	Z	4	F	R			>>>	a													
	I	J	K	P	D				>>>	b													
S	<p>Enter the rotational speed of the spindle by means of numeric keys. Input range (0 to 99999)</p>																						
M/B	<p>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 menu key B CODE INPUT and enter a numeric value. For M-code, refer to "List of M-codes" in the Operating Manual. If the sequence data contains G65 in the item "G1" or "G2", the data set here will be handled as an argument.</p>																						

2. 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 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. The data to be entered in the items DATA 1 to DATA 6 are limited to those indicated in the menu for address input. Moreover, all entries are made with a decimal point except the entry of the F function in G94 mode.
3. In the manual program mode, the offset of the tool (G44) is automatically performed at the head of the program on one tool length registered in the **TOOL FILE** display. It is necessary to retain this in case of a command concerning the correction of the tool length.
4. In the manual program unit, the tool is deviated on a distance entered in the item 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.
5. If the method of feed is modified from G94 to G95 or from G95 to G94, the designated feed value of F in the unit data will become blank (****).
6. A MAZATROL program cannot be called up during designation of a sub-program call such as G65.

4-16 Materials Shape Unit (MATERIAL)

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

Only the outside-diameter shape and inside-diameter shape of the intended workpiece can be defined using the materials shape unit. Data that has been set in this unit, as with common data, becomes base data for the entire program. This unit cannot be selected for round-bar materials. To use this unit, select the “workpiece sheme (MILL & TURN.)” for creating the common unit.

Press the menu key **WORKPIECE SHAPE** to select the materials shape unit.

C-POINT MACH-ING	C-LINE MACH-ING	INDEX	M CODE	SUB PROGRAM	MMS	WORKPIECE MEASURE	TOOL MEASURE	WORKPIECE SHAPE	>>>
---------------------	--------------------	-------	--------	----------------	-----	----------------------	-----------------	--------------------	-----

4-16-1 Setting unit data

UNo.	UNIT
*	MATERIAL [1]

[1] UNIT

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

--	--	--	--	--	--	--	--	--	--

- 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.

4-16-2 Setting sequence data

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







[1] SHP

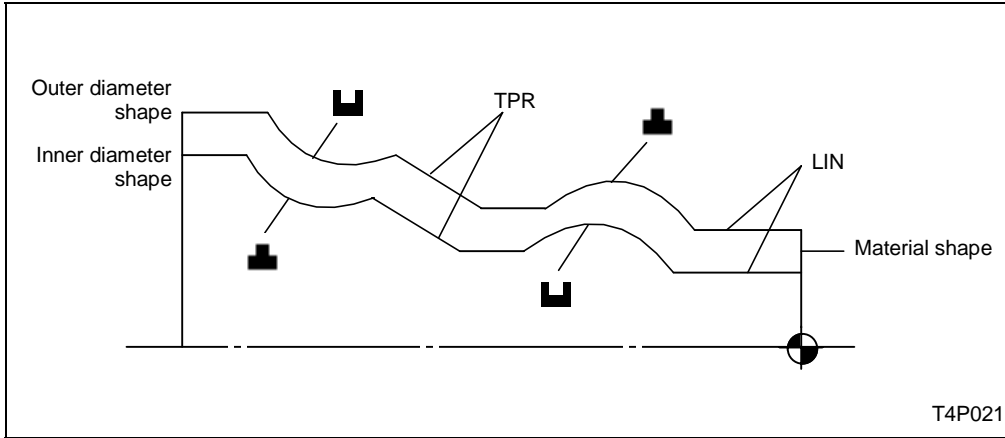
The following menu will be displayed when the cursor is placed at this item.

									SHAPE END
--	--	--	--	--	--	--	--	--	--------------



Select the type of shape from the above menu.

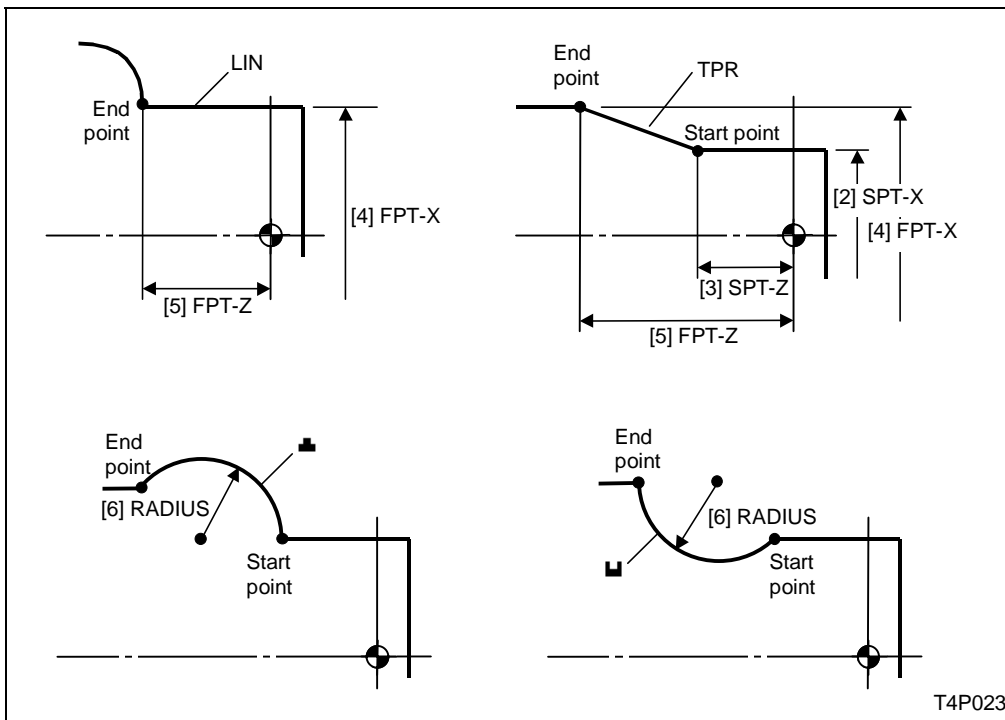
The data of the displayed menu denote the following shapes:

-  **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

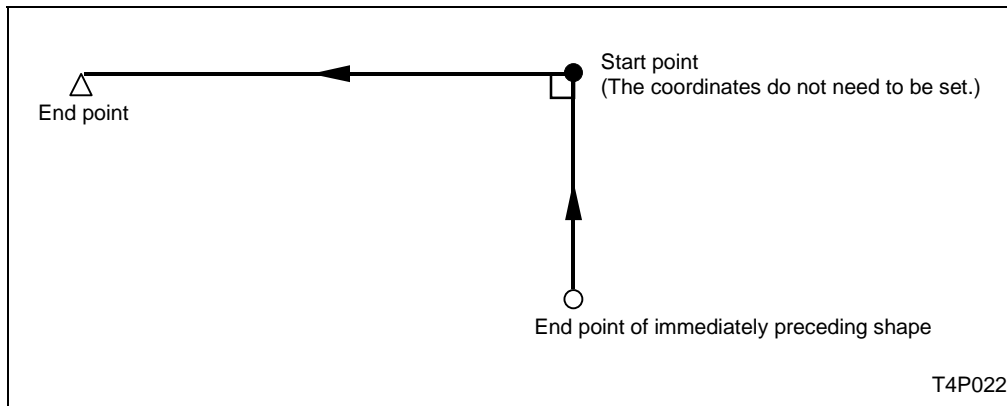


[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z, [6] RADIUS

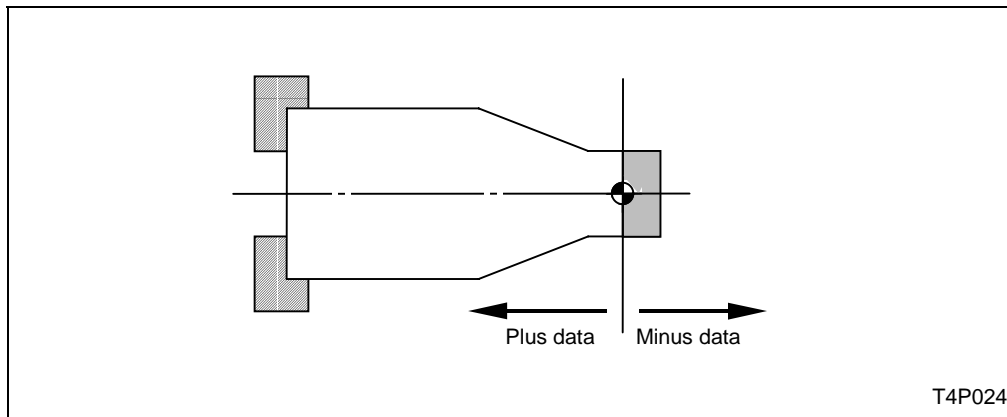
Set the coordinates of the intended start point and end point of the shape you selected for item [1]. Also set the radius of the desired circle if you have selected  or .



- 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 zero-point 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 menu key **CONTINUE**.

UNo.	UNIT	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
1	MATERIAL OUT					
FIG	SHF					
1	LIN	◆	◆	20.	30.	◆
2	TPR	◆	◆			◆

◆ ← Cursor

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

UNo.	UNIT	SPT-X	SPT-Z	FPT-X	FPT-Z	RADIUS
1	MATERIAL OUT					
FIG	SHF					
1	LIN	◆	◆	20.	30.	◆
2	TPR	20.	30.	◆		◆

These values are set automatically.

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

4-17 Head Selection Unit (HEAD)

Use the head selection unit to specify the turning spindle to be used. Set this unit in front of the machining unit. The specified status will be maintained until a turning spindle is selected again.

This unit is valid only when the machine has two turning spindles.

Press the menu key **SELECT HEAD** to select this unit.

SELECT HEAD	TRANSFER WORKPIECE	PROCESS END	PALLET CHANGE	WPC SHIFT						>>>
----------------	-----------------------	----------------	------------------	--------------	--	--	--	--	--	-----

4-17-1 Setting unit data

UNo.	UNIT	TYPE	HEAD
*	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].

Example: SINGLE and SYNCH. machining

	UNo.				
	0				
	UNo.	UNIT	TYPE	HEAD	
	1	HEAD	SIN	1	
A	UNo.	UNIT	~		
	2	FACING FCE			
	UNo.	UNIT	~		
	3	BAR OUT			
	UNo.	UNIT	~	HEAD	
	4	TRANSFER BAR		2	
	UNo.	UNIT	TYPE	HEAD	
	5	HEAD	SYNC	1	
B	UNo.	UNIT	~		
	6	T.GROOV OUT			
	UNo.	UNIT	~		
	7	BAR OUT			
	UNo.	UNIT	~	HEAD	
	8	TRANSFER BAR		1	
	UNo.	UNIT	TYPE	HEAD	
	9	HEAD	SIN	2	
C	UNo.	UNIT	~		
	10	FACING BAK			
	UNo.	UNIT	~		
	11	T.DRLL BAK			
	UNo.	UNIT	~		
	12	END			

The diagrams show the progression of a workpiece through three machining stages:

- Diagram A:** Shows the initial workpiece with a 'FACING FCE' operation on the right end and a 'BAR OUT' operation. The workpiece is shaded to show the current state.
- Diagram B:** Shows the workpiece after 'T.GROOV OUT' and 'BAR OUT' operations. The workpiece is shaded to show the current state.
- Diagram C:** Shows the workpiece after 'FACING BAK' and 'T.DRLL BAK' operations. The workpiece is shaded to show the current state.

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

4-18 Workpiece Transfer Unit (TRANSFER)

Select the workpiece transfer unit to change the chucking position of the workpiece, to deliver it from one turning spindle to the other turning spindle or to move the secondary spindle.

This unit is valid only when the machine has two turning spindles.

Press the menu key **TRANSFER WORKPIECE** to select this unit.

SELECT HEAD	TRANSFER WORKPIECE	PROCESS END	PALLET CHANGE	WPC SHIFT							>>>
----------------	-----------------------	----------------	------------------	--------------	--	--	--	--	--	--	-----

4-18-1 Setting unit data

UNo.	UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	MOVEMENT	C1	C2
*	TRANSFER	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]

[1] PAT.

From the following menu, select a transfer pattern.

CHUCK WORK	BAR LOOP	T-SPDL MOVE								
---------------	-------------	----------------	--	--	--	--	--	--	--	--

CHUCK WORK : To deliver a chuck work from No. 1 to No. 2 spindle or vice versa.

BAR LOOP : To rechuck a bar work.

T-SPDL MOVE : To move the No. 2 spindle on the Z-axis.

[2] HEAD

- If you have selected **CHUCK WORK** 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 LOOP** 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 **T-SPDL 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	1	2	3	4	5				
SPINLDE STOP	FORWARD	REVERSE	ORIENT	C-AXIS POSITION	KEEP				

0 SPINDLE STOP: The spindle does not rotate.

1 FORWARD: The spindle rotates forward at the speed set previously in parameter **U27**.

2 REVERSE: The spindle rotates backward at the speed set previously in parameter **U27**.

3 ORIENT: The spindle is orientated.

4 C-AXIS POSITION: The spindle undergoes C-axis positioning.

5 KEEP: The spindle status specified in the previous unit is held.

Note: Spindle status pattern 5 (spindle status hold) can be selected when bit 5 in parameter **P107** is "1."

[4] PUSH

Only when **CHUCK WORK** 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 LOOP** 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

- If you have selected **CHUCK WORK** for data item [1] above:

Position of the W-axis at the No. 2 spindle side in the machine coordinate system during workpiece transfer

- If you have selected **BAR LOOP** for data item [1] above:

Position of the W-axis at the No. 2 spindle side in the machine coordinate system during workpiece rechucking

Function of the menu key TRANSFER POSITION

A press of the menu key **TRANSFER POSITION** automatically assigns the coordinates of the current machine position to the items W1, C1, and C2, as shown below. Automatic assignment to C1 and C2, however, is valid only in milling mode and does not occur in turning mode.

Item	Value that is set automatically
W1	Position of the W-axis at the No. 2 spindle side in the machine coordinate system
C1	Position of the C-axis at the No. 1 spindle side in the machine coordinate system
C2	Position of the C-axis at the No. 2 spindle side in the machine coordinate system

This function can be used in listing mode as well.

[7] W2

- If you have selected **CHUCK WORK** for data item [1] above:
Position of the W-axis at the No. 2 spindle side in the machine coordinate system after workpiece transfer
- If you have selected **BAR LOOP** for data item [1] above:
Position of the W-axis at the No. 2 spindle side in the machine coordinate system after workpiece rechucking
- If you have selected **T-SPDL MOVE** for data item [1] above:
Movement position of the W-axis at the No. 2 spindle side in the machine coordinate system

Function of the menu key **TRANSFER POSITION**

A press of the menu key **TRANSFER POSITION** automatically assigns the coordinates of the current machine position to the item W2, as shown below.

Item	Value that is set automatically
W2	Position of the W-axis at the No. 2 spindle side in the machine coordinate system

This function assigns also automatically the movement stroke in the item MOVEMENT as follows according to the particular combination of an operation pattern and a spindle head:

Operation pattern	Head	Value that is set automatically
CHUCK	1→2	Movement stroke = W2 – W1
CHUCK	2→1	Manually enter the movement stroke since the position of the workpiece before being transferred is unknown.
BAR	1	Movement stroke = W2 – Previous W1 Previous W1: The workpiece transfer unit is searched for in the opposite direction from the current unit, and if the unit that has thus been detected first is for rechucking and head 2, the corresponding W1 value will be used. Subprograms are excluded from the search. Automatic setting will not occur if the corresponding unit is not detected.
BAR	2	—
MOVE	—	Manually enter the movement stroke since the position of the workpiece before being transferred is unknown.

Even if the movement stroke is changed manually, neither W1 nor W2 will be auto-set.
This function can be used in listing mode as well.

[8] MOVEMENT

Set the movement stroke of the workpiece.

The WPC zero-point automatically shifts according to the setting value of the movement stroke. Otherwise, there is no need to re-create the WPC SHIFT unit after the workpiece has been moved in the current unit.

Based of the entered movement stroke in this item, the NC unit recognize the movement of the workpiece, and after the recognition, the path is created so that the tool does not interfere with the workpiece.

[9] C1

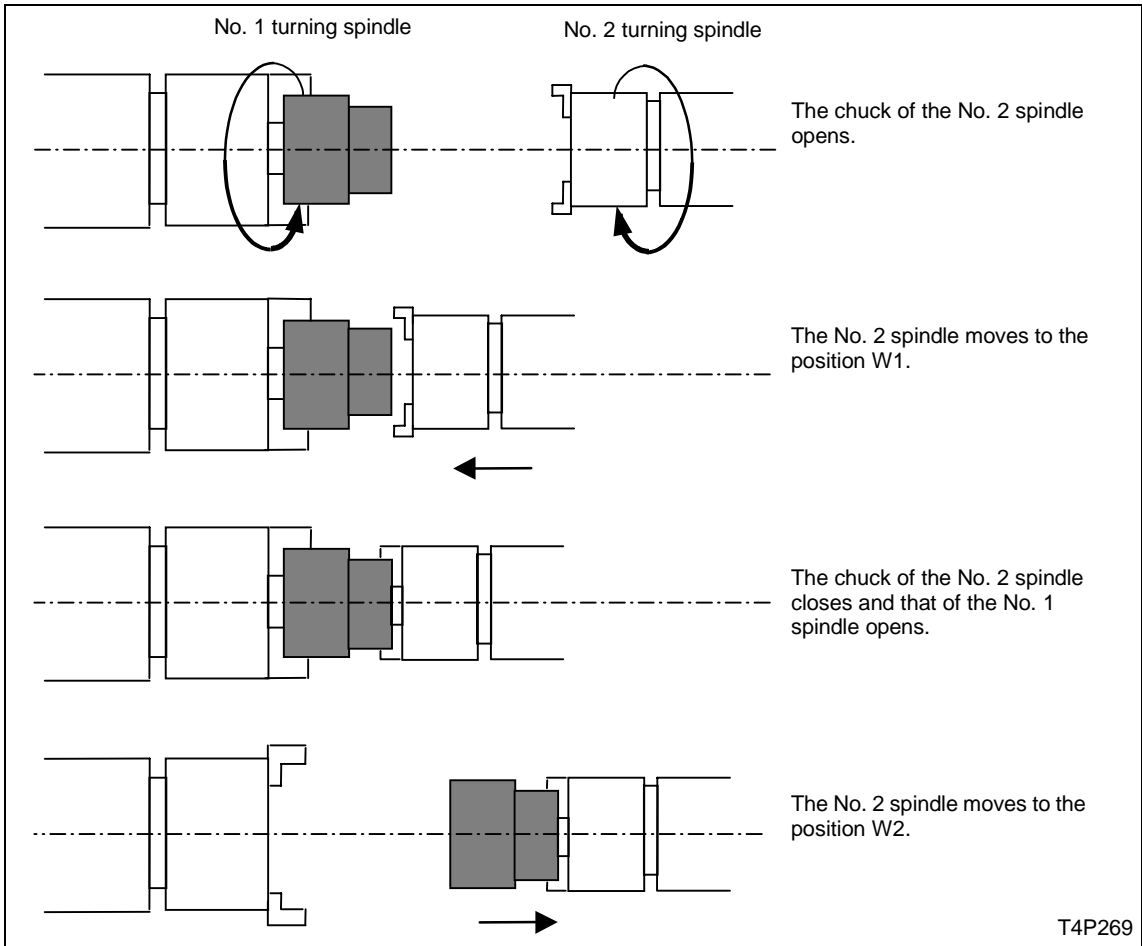
C-coordinate of the workpiece transfer position at the No. 1 spindle side (C-coordinate in the machine coordinate system)

[10] C2

C-coordinate of the workpiece transfer position at the No. 2 spindle side (C-coordinate in the machine coordinate system)

Example 1: TRANSFER CHUCK

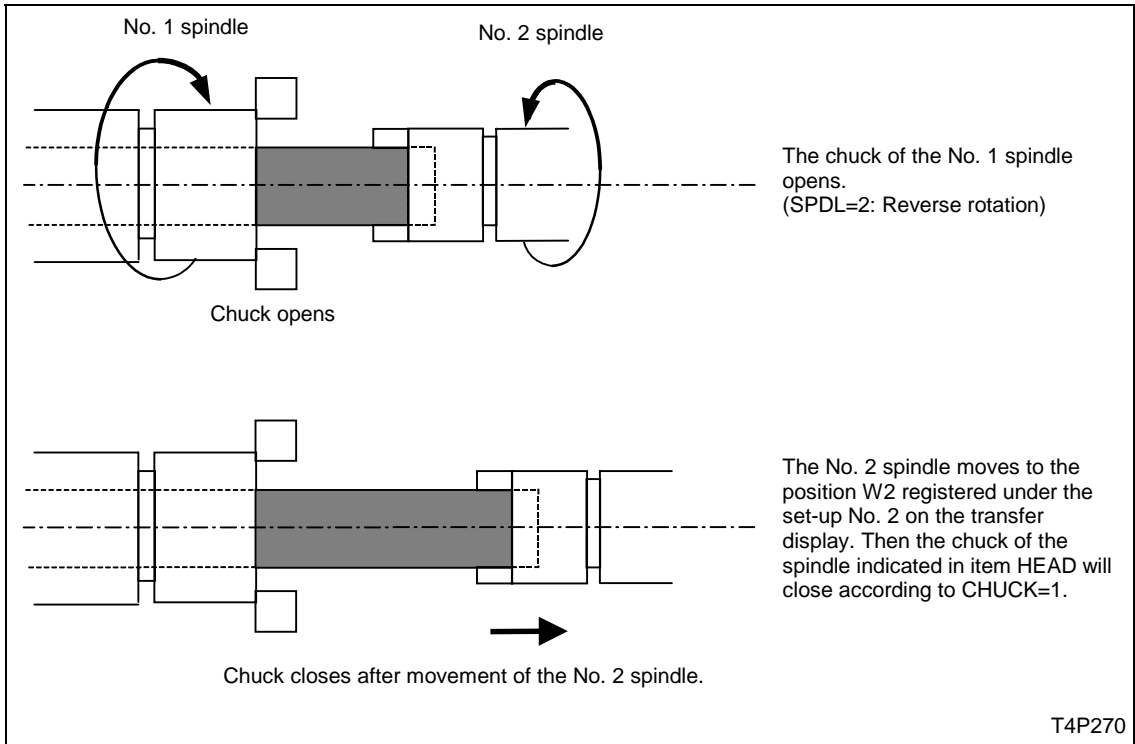
UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	MOVEMENT	C1	C2
TRANSFER	CHUCK	1→2	1	1	◆					



Note: The first transfer operation in a program must be that from the No.1 to the No. 2 spindle. Otherwise, an alarm occurs.

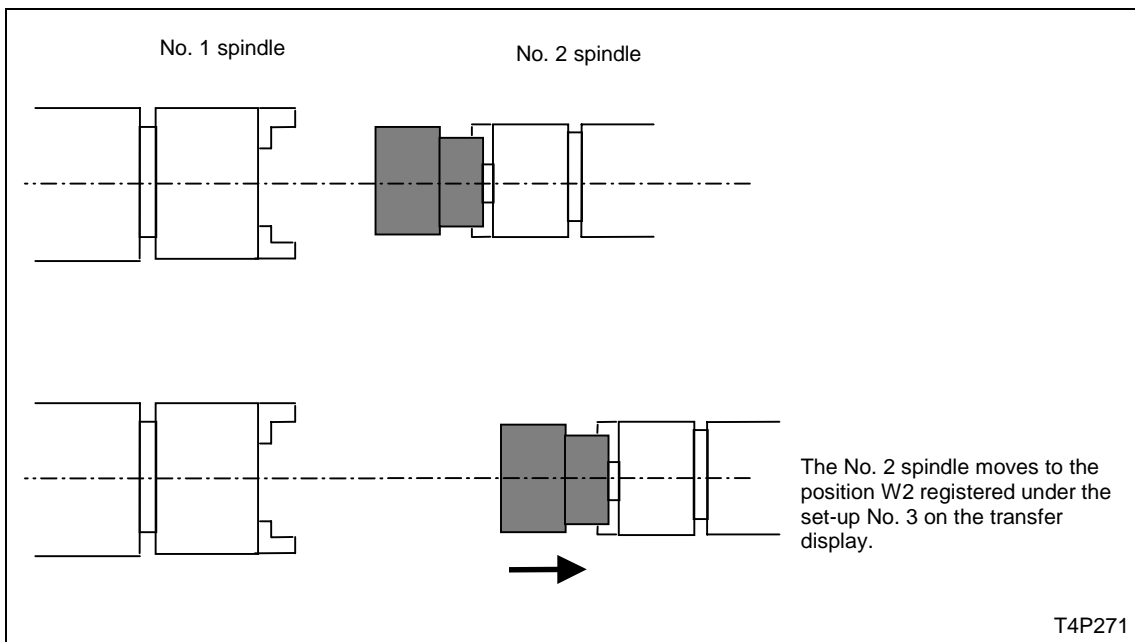
Example 2: TRANSFER BAR

UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	MOVEMENT	C1	C2
TRANSFER	BAR	1	2	◆	1					



Example 3: TRANSFER MOVE

UNIT	PAT.	HEAD	SPDL	PUSH	CHUCK	W1	W2	MOVEMENT	C1	C2
TRANSFER	MOVE	◆	◆	◆	◆					



4-19 C-Axis Point Machining Unit

The C-axis point machining unit is intended to specify data on the machining method to be used when drilling holes after positioning with the C-axis and other axes (X, Z), and data on the shape of the section to be machined.

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.

4-19-1 Types of C-axis point machining units

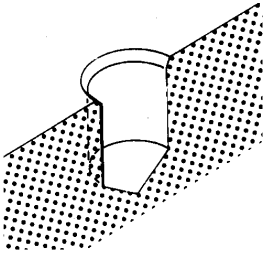
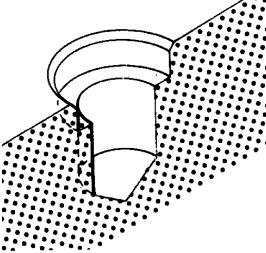
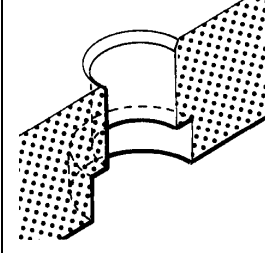
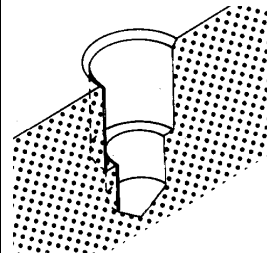
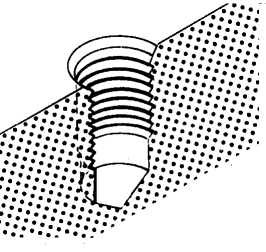
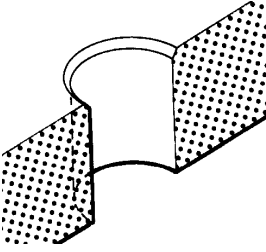
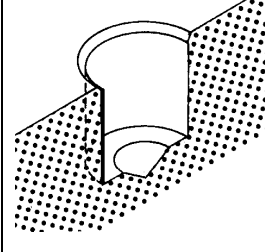
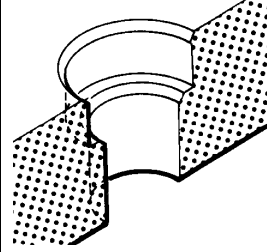
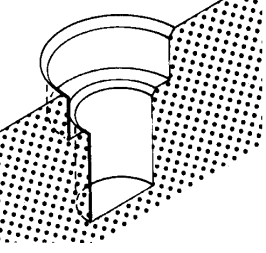
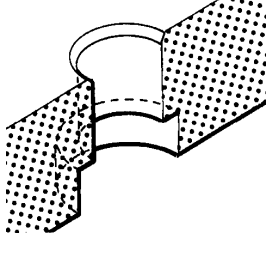
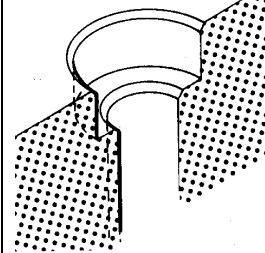
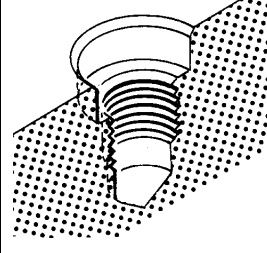
1. Drilling  NM210-00532	2. RGH CBOR machining  NM210-00533	3. RGH BCB machining  NM210-00534	4. Reaming  NM210-00535
5. Tapping  NM210-00536	6-(1) Boring of through hole  NM210-00537	6-(2) Boring of non-through hole  NM210-00538	6-(3) Boring of stepped through hole  NM210-00539
6-(4) Boring of stepped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

Fig. 4-23 Types of C-axis point machining units

4-19-2 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	WPC	OFFSET	END	SHAPE CHECK	>>>
-------------------	------------------	------------------	---------	-------------------	-----	--------	-----	----------------	-----





- (2) Pressing on the menu key >>> displays the following unit menu.

C-POINT MACH-ING	C-LINE MACH-ING	INDEX	M CODE	SUB PROGRAM	MMS	WORKPIECE MEASURE	TOOL MEASURE	WORKPIECE SHAPE	>>>
---------------------	--------------------	-------	--------	----------------	-----	----------------------	-----------------	--------------------	-----

- (3) Pressing on the menu key **C-POINT MACH-ING** displays the following unit menu.

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

- (4) Press the appropriate menu key of the desired machining unit.
 - When the menu key **BORING** is pressed, the menu of the four following machining sub-units is displayed.

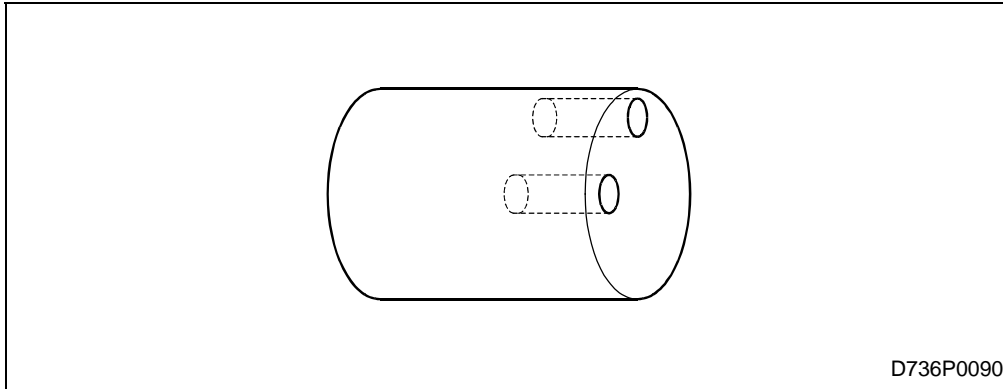
BORING 	BORING 	BORING 	BORING 						HI SPD. DRL.USE
--	--	--	--	--	--	--	--	--	--------------------

Note: For menu item **HI SPD DRL. USE**, refer to Subsection 4-7-4 “Automatic tool development for cemented carbide drill.”

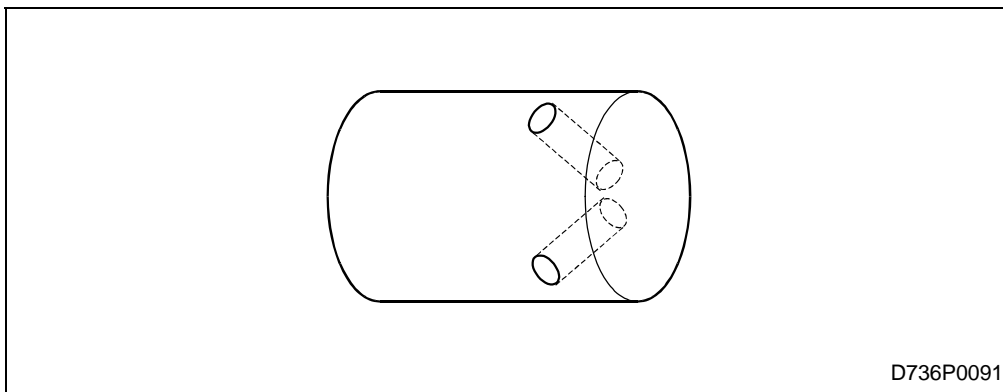
4-19-3 C-axis point machining unit data and automatic tool development

For C-axis point-machining, specify the desired face (**FACE**, **CYLIND** or **SLANT**) in the item C-FACE.

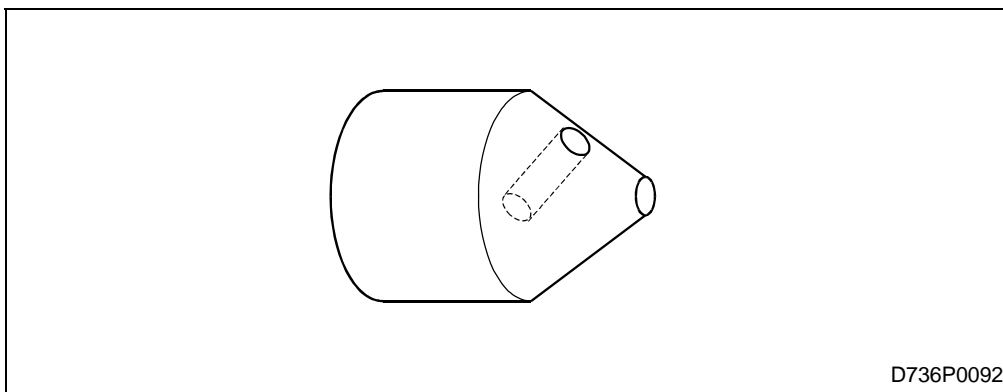
- **FACE**..... Select to drill holes at the edge of the workpiece.



- **CYLIND** Select to drill holes on the cylindrical surface of the workpiece.



- **SLANT** Select to drill holes on the inclined surface of the workpiece.



When the cursor is present in the item C-FACE, the following menu is displayed. Specify the desired face by pressing the menu key:

FACE	CYLIND	SLANT							
------	--------	-------	--	--	--	--	--	--	--

1. Drilling unit (DRILLING)

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

A. Data setting

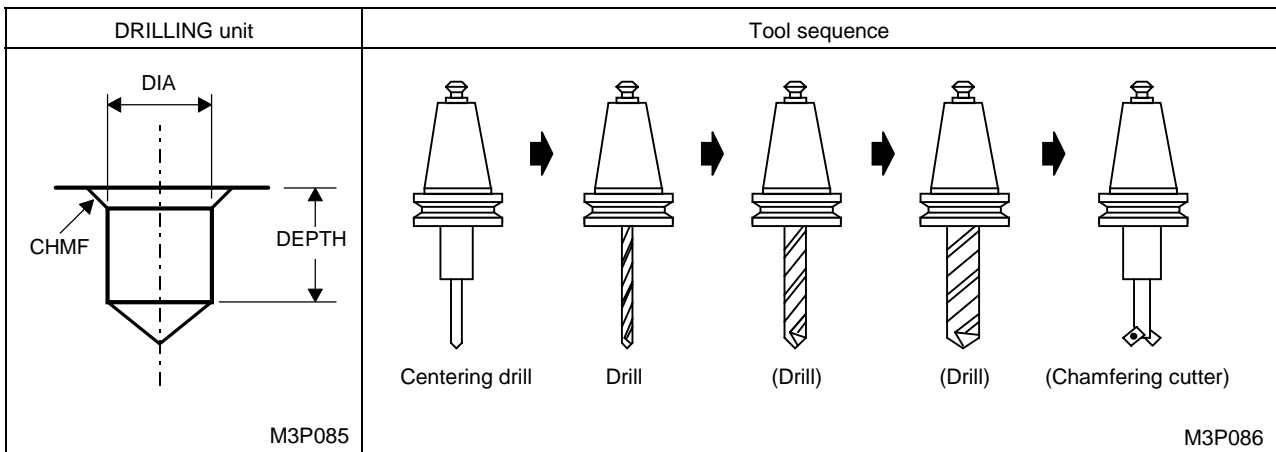
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF								
2	DRILLING		999.999	999.999	99.9								
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum entry value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the drilling unit of the point-machining.

2. RGH CBOR machining unit (RGH CBOR)

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

A. Data setting

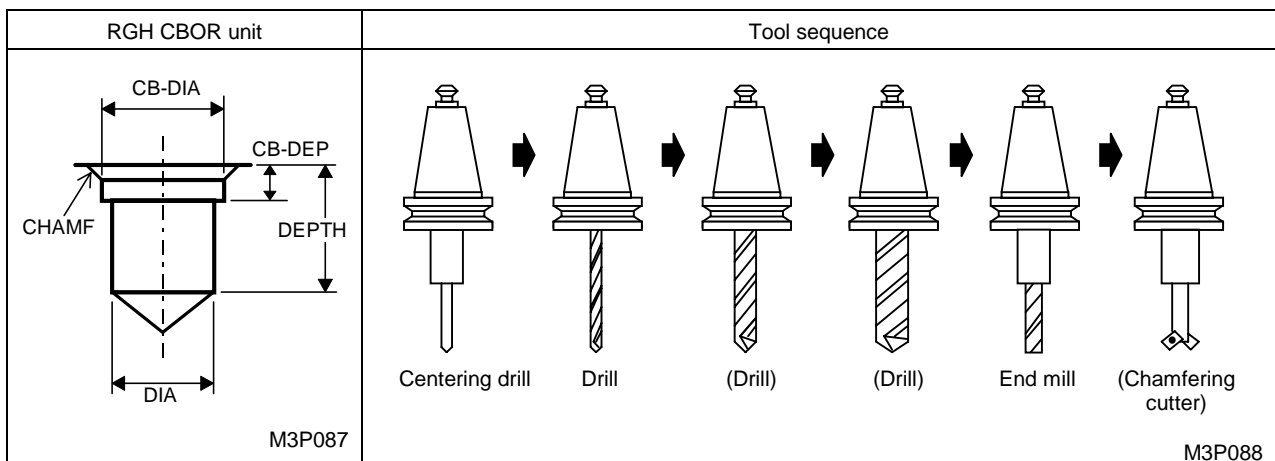
UNo.	UNIT	C-FACE	CB-DIA	CB-DEP	CHMF	BTM	DIA	DEPTH					
2	RGH CBOR		999.999	999.999	99.9		999.999	999.999					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	END MILL			○	○	○	○	◆	○				
6	CHAMFER			○	○	○	○	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the RGH CBOR machining unit of the point-machining.

3. RGH BCB machining unit (RGH BCB)

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

A. Data setting

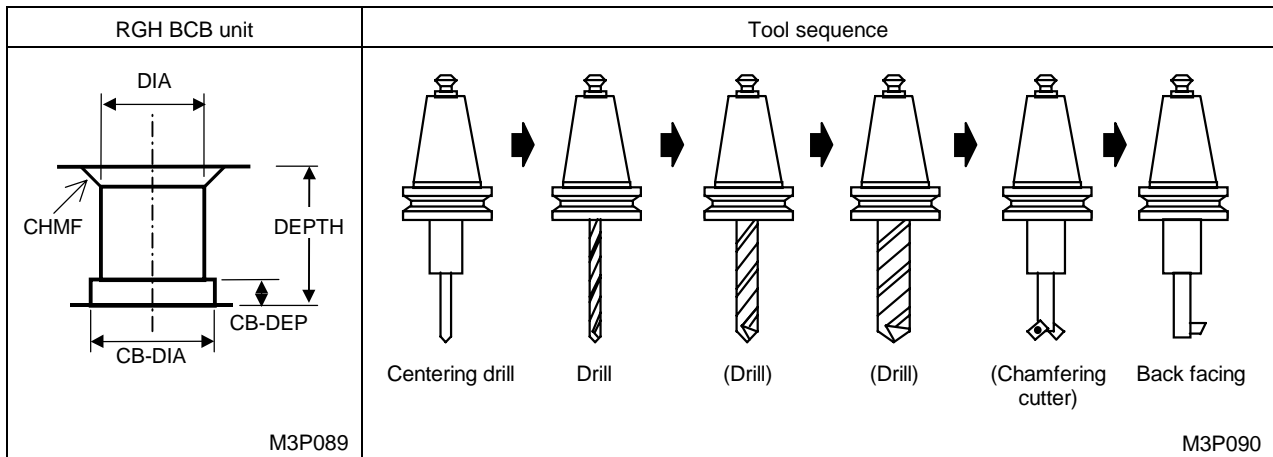
UNo.	UNIT	C-FACE	CB-DIA	CB-DEP		DIA	DEPTH	CHMF					
2	RGH BCB		999.999	999.999		999.999	999.999	99.9					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				
6	BCK FACE	○		○	○	◆	○	◆	◆				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the RGH BCB machining unit of the point-machining.

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.

Case of preceding process = drilling

A. Data setting

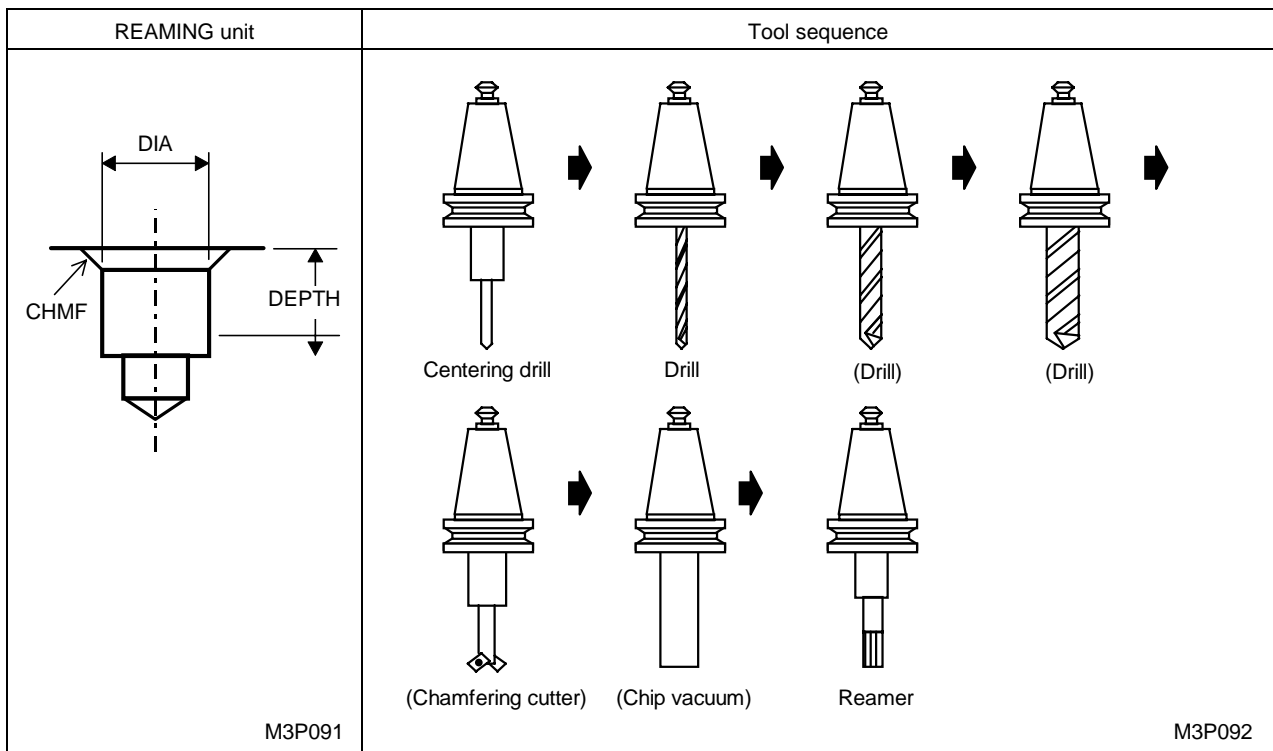
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING		999.999	999.999	99.9								
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				
6	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
7	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the reaming unit of the point-machining.

Case of preceding process = boring

A. Data setting

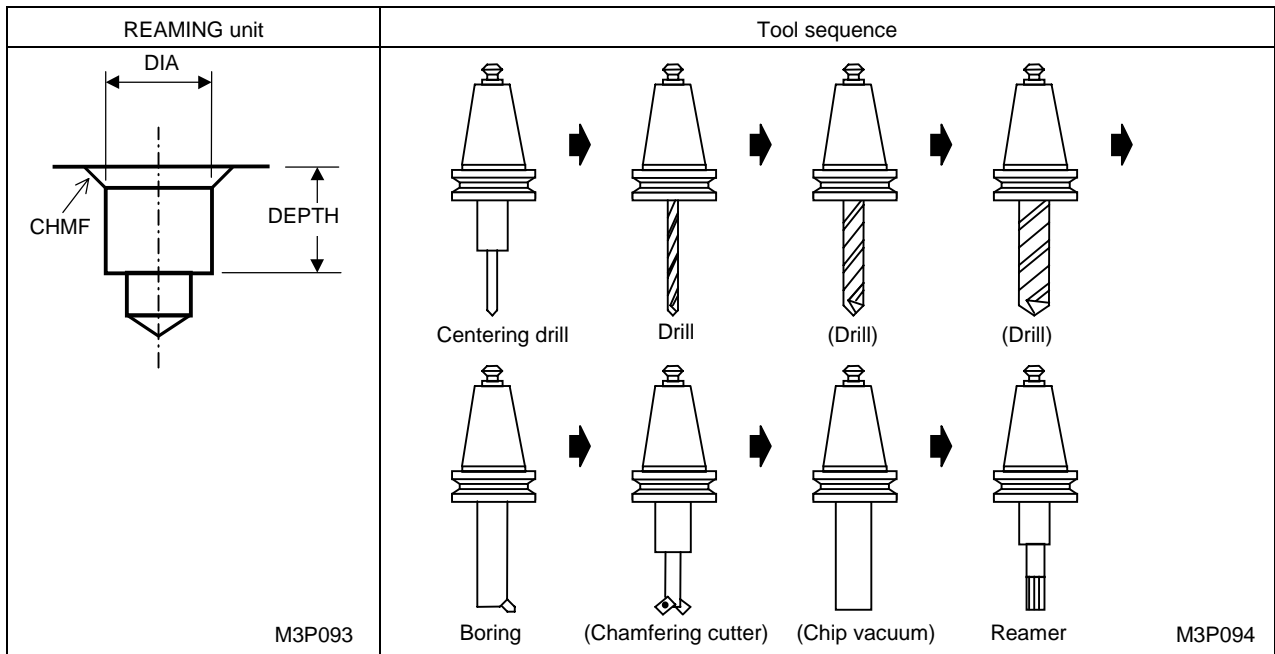
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING		999.999	999.999	99.9								
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
8	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the reaming unit of the point-machining.

Case of preceding process = end mill

A. Data setting

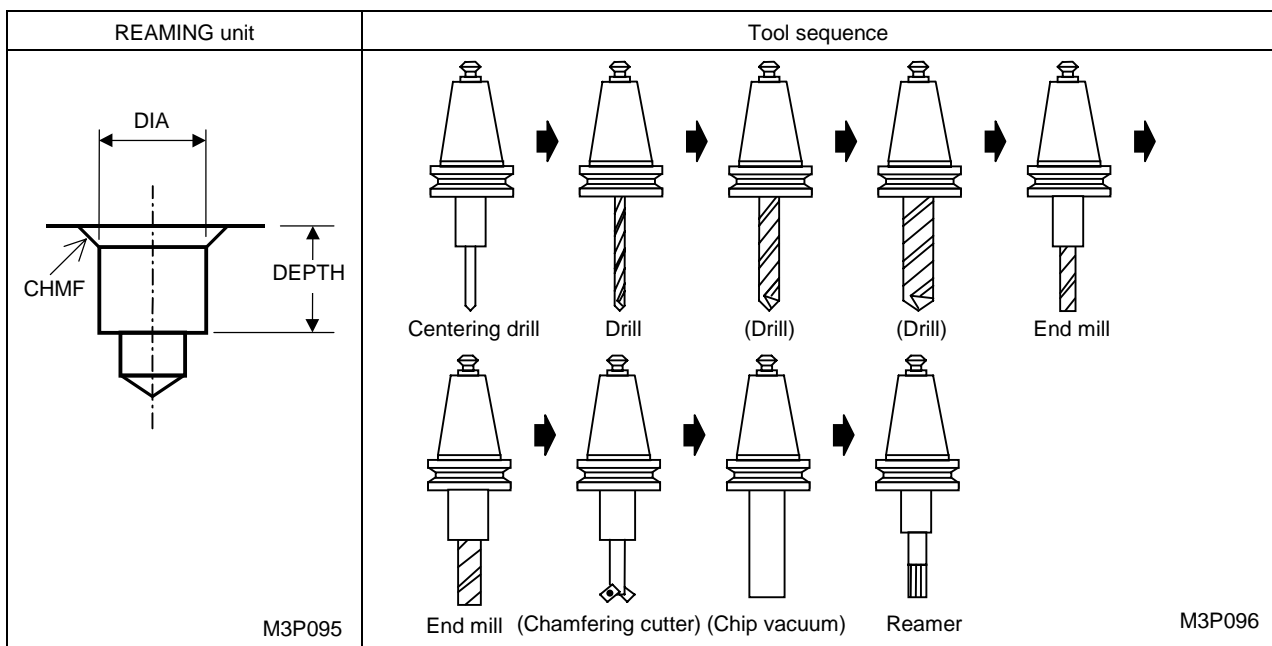
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING		999.999	999.999	99.9								
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	END MILL			○	○	○	○	◆	○				
6	END MILL			○	○	○	○	◆	○				
7	CHAMFER			○	○	○	○	◆	○				
8	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
9	REAMER	○		○	○	◆	◆	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The data of the unit represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the reaming unit of the point-machining.

5. Tapping unit (TAPPING)

Select this unit for performing tapping.

A. Data setting

UNo.	UNIT	NOM-	C-FACE	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP					
2	TAPPING					999.999	99.9						
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHAMFER			○	○	○	○	◆	○				
6	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
7	TAP	○		○	○	○	◆	○	○		○		

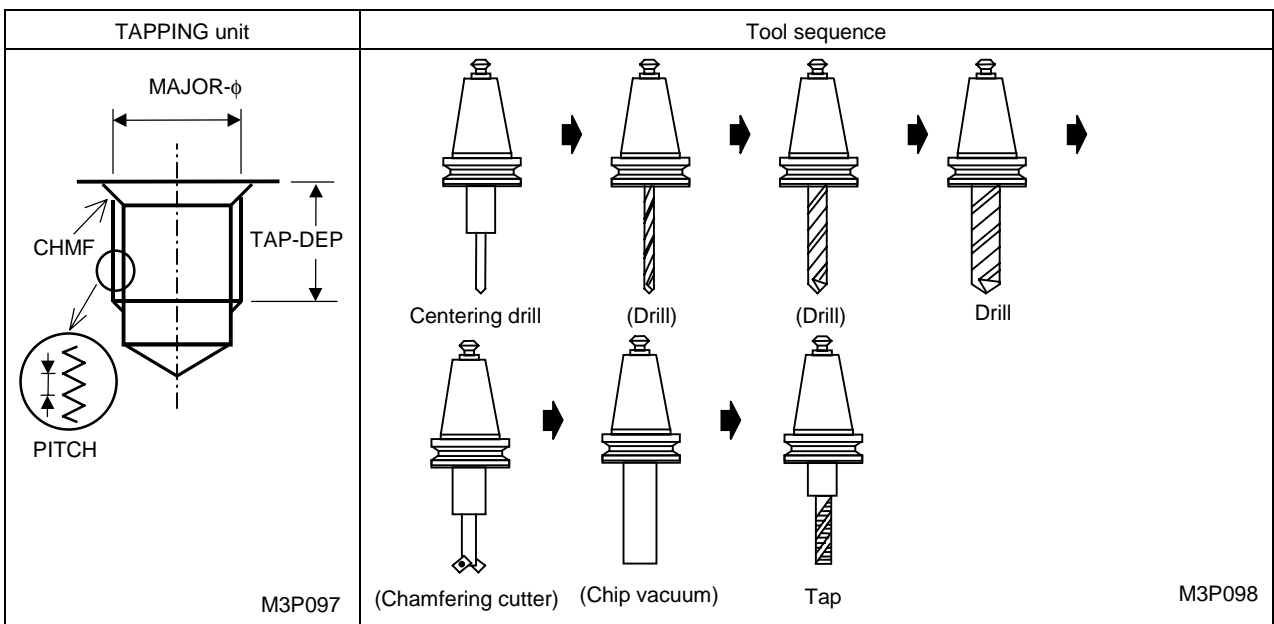
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.

Note 3: If **TAPPING CYCLE** menu item is selected for PRE-DIA, there is no need to set data in PRE-DEP.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the tapping unit of the point-machining.

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.

Through hole boring unit (BORING T1)

Select this unit for performing through-hole boring.

A. Data setting

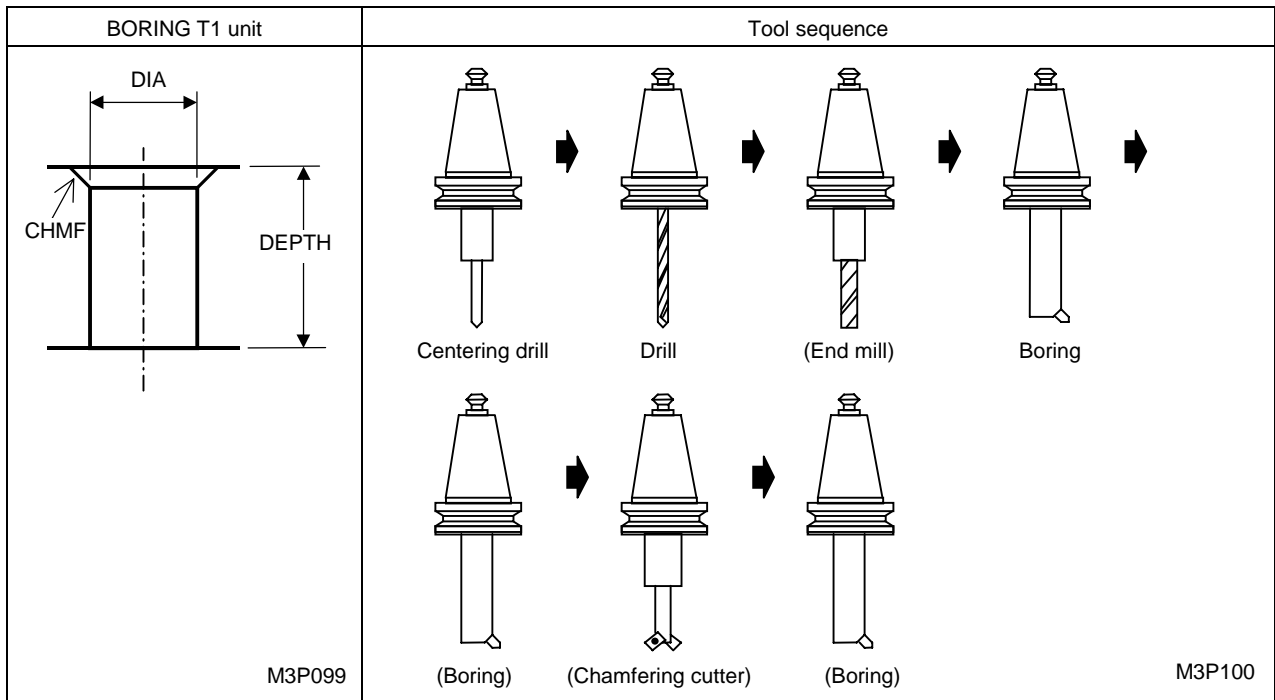
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF	WAL							
2	BORE T1		999.999	999.999	99.9								
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	◆	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the through hole boring unit of the point-machining.

Non-through hole boring unit (BORING S1)

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

A. Data setting

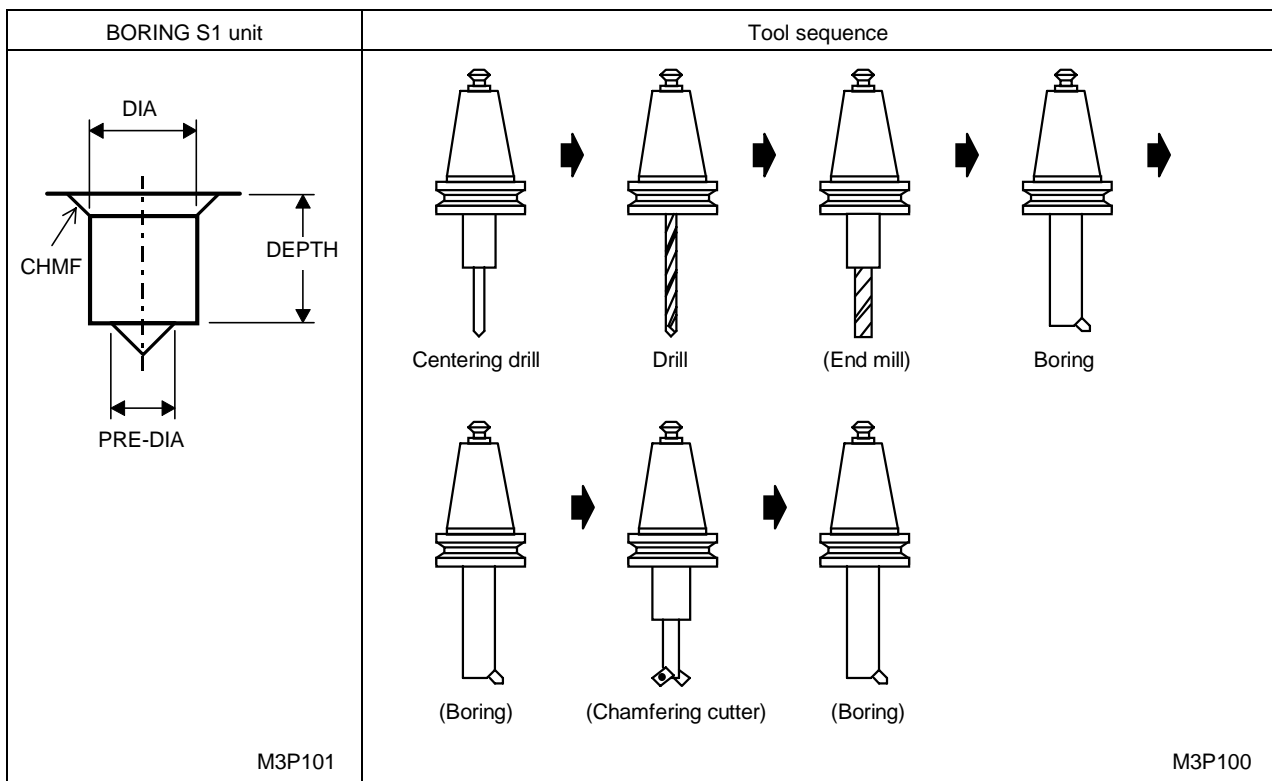
UNo.	UNIT	C-FACE	DIA	DEPTH	CHMF	BTM	WAL	PRE-DIA					
2	BORE S1		999.999	999.999	99.9			999.999					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆		90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	◆	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the non-through hole boring unit of the point-machining.

Stepped through hole boring unit (BORING T2)

Select this unit for performing stepped through hole boring.

A. Data setting

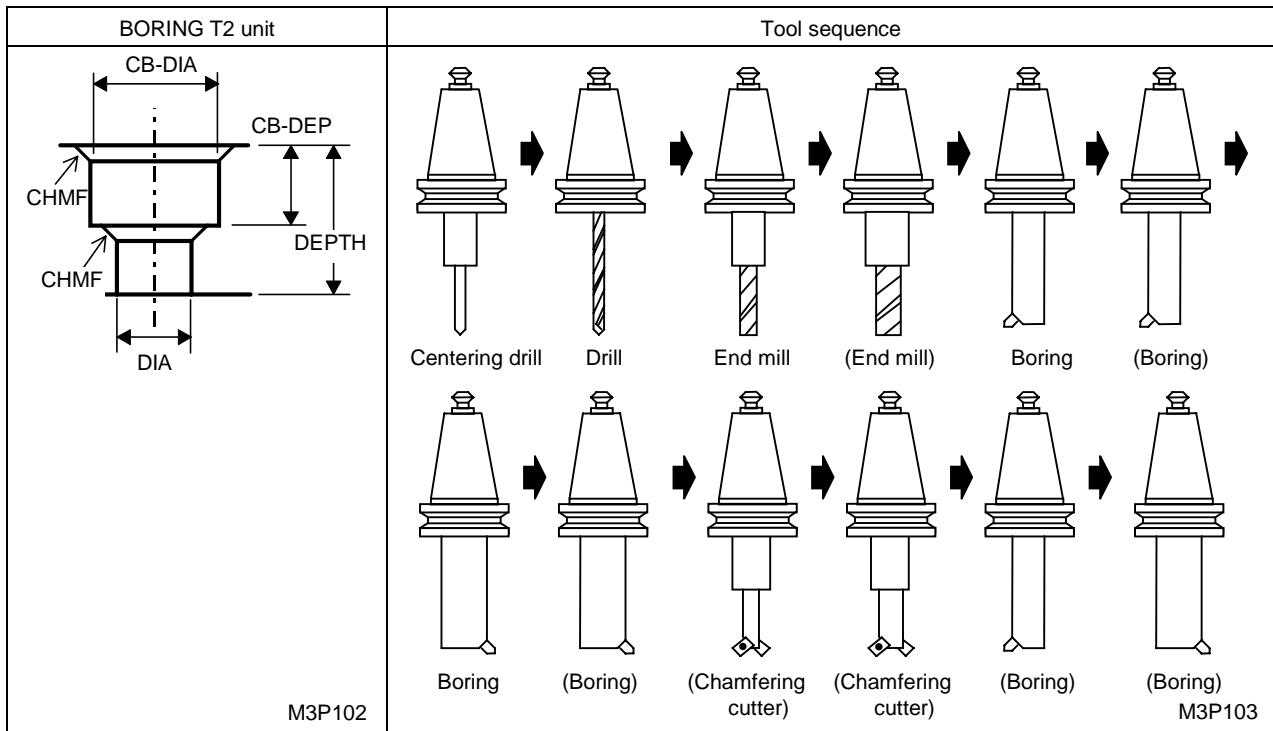
UNo.	UNIT	C-FACE	CB-DIA	CB-DEP	CHMF	BTM	WAL	DIA	DEPTH	CHMF	WAL		
2	BORE T2		999.999	999.999	99.9			999.999	999.999	99.9			
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	◆	○				
4	END MILL			○	○	○	○	◆	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	BOR BAR	○		○	○	○	○	○	○				
7	BOR BAR	○		○	○	○	○	○	○				
8	BOR BAR	○		○	○	○	○	○	○				
9	CHAMFER			○	○	○	○	◆	○				
10	CHAMFER			○	○	○	○	◆	○				
11	BOR BAR	○		○	○	○	○	○	○				
12	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the stepped through hole boring unit of the point-machining.

Stepped non-through hole boring unit (BORING S2)

Select this unit for performing stepped non-through boring.

A. Data setting

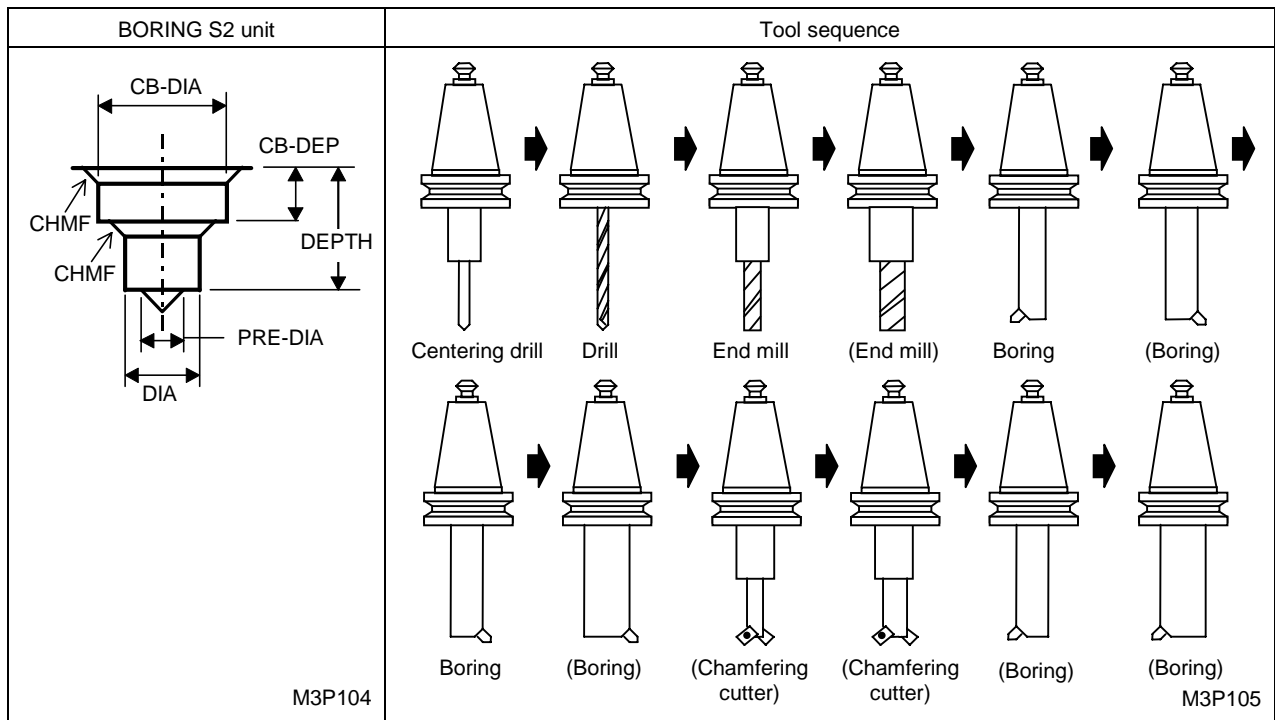
UNo.	UNIT	C-FACE	CB-DIA	CB-DEP	CHMF	BTM	WAL	PRE-DIA	DIA	DEPTH	CHMF	BTM	WAL
2	BORE S2		999.999	999.999	99.9			999.999	999.999	999.999	99.9		
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	◆	○				
4	END MILL			○	○	○	○	◆	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	BOR BAR	○		○	○	○	○	○	○				
7	BOR BAR	○		○	○	○	○	○	○				
8	BOR BAR	○		○	○	○	○	○	○				
9	CHAMFER			○	○	○	○	◆	○				
10	CHAMFER			○	○	○	○	◆	○				
11	BOR BAR	○		○	○	○	○	○	○				
12	BOR BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the stepped non-through hole boring unit of the point-machining.

7. Back boring unit (BK-CBORE)

Select this unit for performing back boring.

A. Data setting

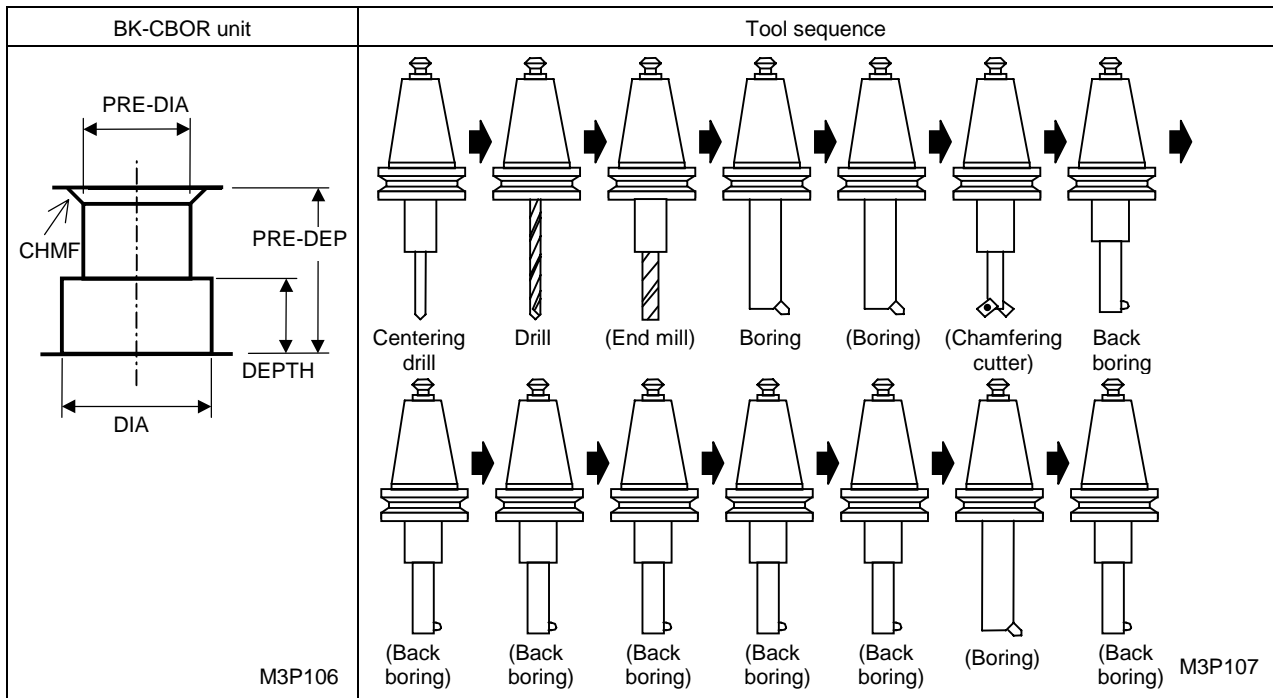
UNo.	UNIT	C-FACE	DIA	DEPTH	BTM	WAL	PRE-DIA	PRE-DEP	CHMF	WAL			
2	BK-CBORE		999.999	999.999			999.999	999.999	99.9				
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	END MILL			○	○	○	○	◆	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	B-B BAR	○		○	○	○	○	○	○				
8	B-B BAR	○		○	○	○	○	○	○				
9	B-B BAR	○		○	○	○	○	○	○				
10	B-B BAR	○		○	○	○	○	○	○				
11	B-B BAR	○		○	○	○	○	○	○				
12	B-B BAR	○		○	○	○	○	○	○				
13	BOR BAR	○		○	○	○	○	○	○				
14	B-B BAR	○		○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the back boring unit of the point-machining.

8. Circular milling unit (CIRC MIL)

Select this unit for performing drilling with the end mill.

According to the set value in the 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

However, the function TORNA. 2 (precision rapid boring tornado cycle) requires the shape correction option and the MAZAK precision rapid boring tornado option.

Circular milling cycle

A. Data setting

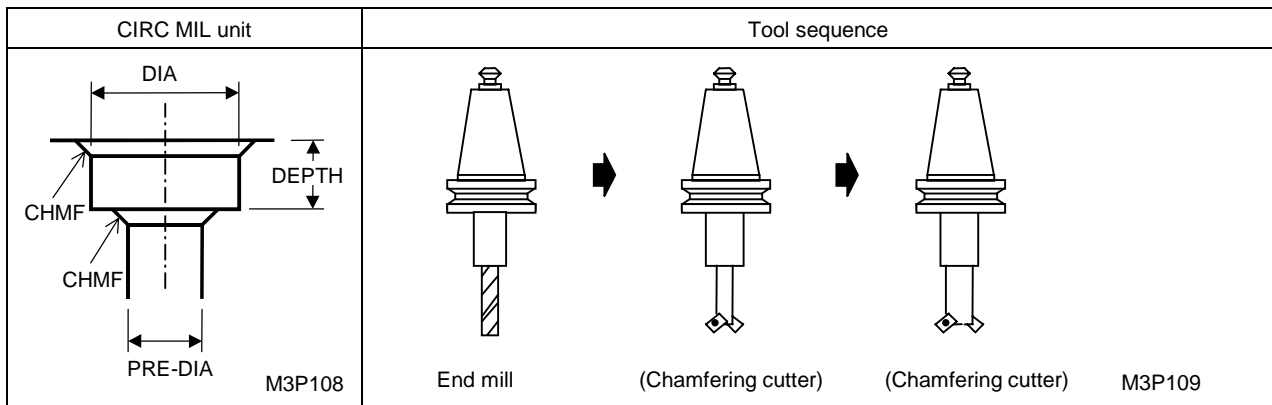
UNo.	UNIT	C-FACE	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2		
2	CIRC MIL		0	999.999	999.999	99.9	9	999.999	99.9	◆	◆		
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				
2	CHAMFER			○	○	○	○	◆	○				
3	CHAMFER			○	○	○	○	◆	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the circular milling unit of the point-machining.

Circular tornado milling cycle

A. Data setting

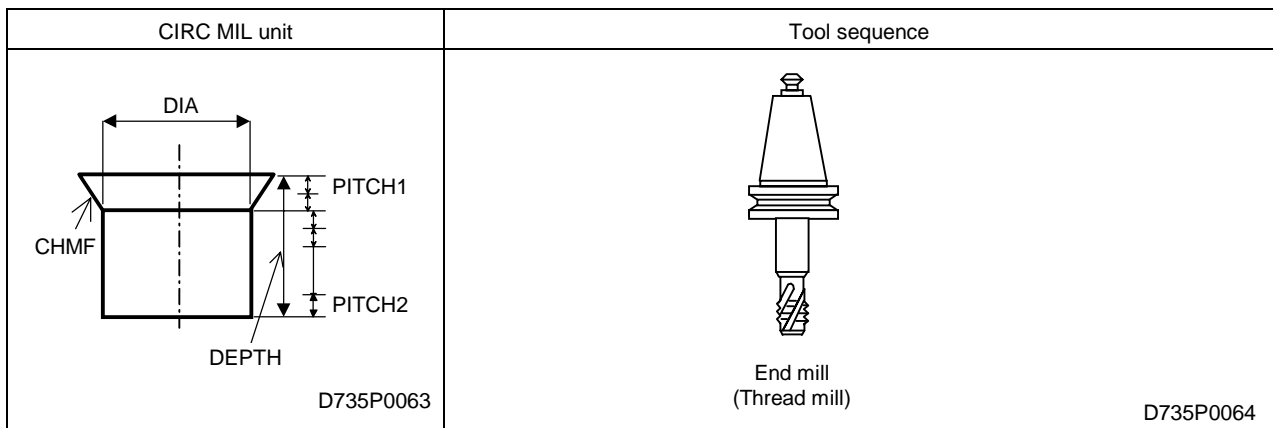
UNo.	UNIT	C-FACE	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2		
2	CIRC MIL		1	999.999	999.999	99.9	9	◆	◆	999.999	999.999		
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the circular tornado milling unit (circular tornado milling cycle) of the point-machining.

Precision rapid boring tornado cycle

A. Data setting

UNo.	UNIT	C-FACE	TORNA.	DIA	DEPTH	CHMF	BTM	PRE-DIA	CHMF	PITCH1	PITCH2		
2	CIRC MIL		2	999.999	999.999	99.9	9	999.999	99.9	999.999	999.999		
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	END MILL			○	○	○	○	○	○				

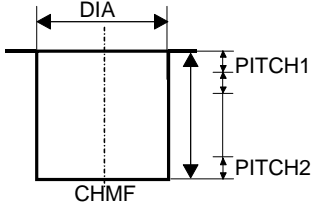
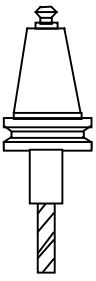
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value (except for TORNA.).

Note 2: To set 2 for the unit data item TORNA., the shape correction option and the MAZAK precision rapid boring tornado option are required.

Note 3: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.

CIRC MIL unit	Tool sequence
	 End mill
	D735P0071

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the circular milling unit (precision rapid boring tornado cycle) of the point-machining.

9. Counterbore-tapping unit (CBOR-TAP)

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

A. Data setting

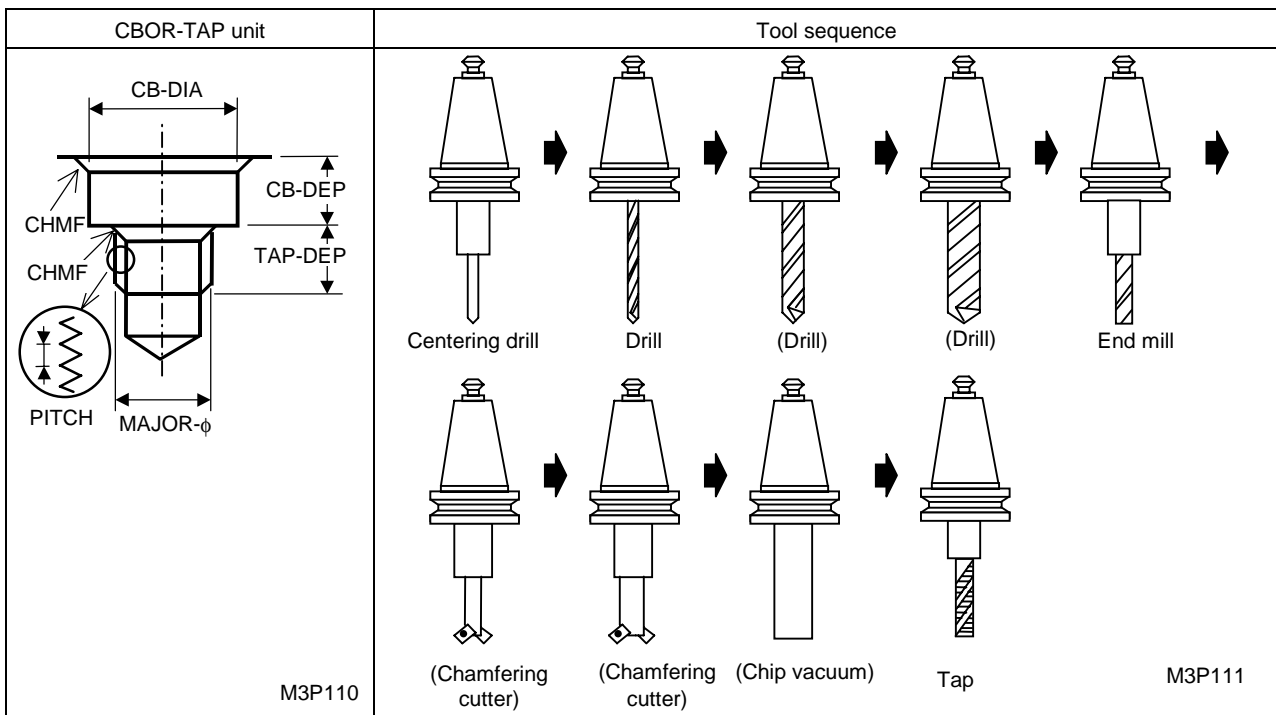
UNo.	UNIT	NOM-	C-FACE	MAJOR-φ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM	CHP	
2	CBOR-TAP					999.999	99.9	999.999	999.999	99.9			
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	END MILL			○	○	○	○	○	○				
6	CHAMFER			○	○	○	○	◆	○				
7	CHAMFER			○	○	○	○	◆	○				
8	CHIP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
9	TAP	○		○	○	○	◆	○	○				

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

◆ : Data are not necessary to be set here.

Note 1: The unit data represent the maximum input value.

Note 2: The tool sequence represents the case of the maximum tools developed. For data setting in the tool sequence refer to Subsection 4-19-4.



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

Remark: Instead of the chamfering cutter, a centering drill can be used for chamfering. For data setting in the tool sequence refer to Subsection 4-19-4.

B. Automatic tool development

The tool data is automatically developed in various patterns on the basis of the data entered in the unit. The machining is executed using the tool sequence data and the unit data is not used for the machining. If developed tool data is not suitable for machining, the tool data can be edited by changing or deleting it.

The automatic development pattern for tool data is the same as that of the counterbore-tapping unit of the point-machining.

4-19-4 C-axis point machining tool sequence data

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.

Data to be set in the tool sequence are the same as those of the point-machining. See Section 4-7-6 "Tool sequence data."

4-19-5 Tool path

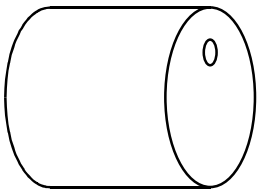
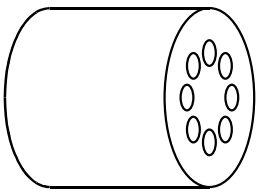
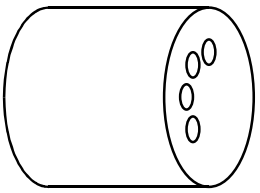
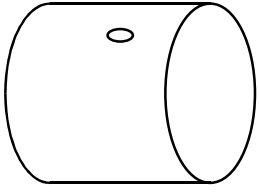
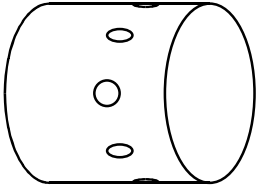
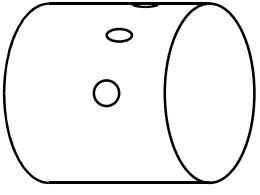
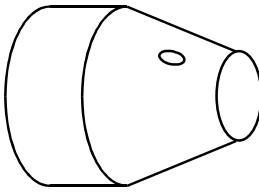
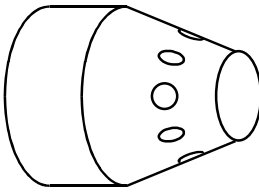
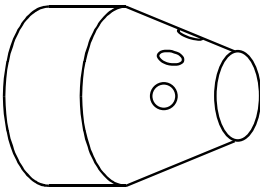
The tool paths for the C-axis point machining are the same as those of the point-machining. See Section 4-7-7 "Tool path."

4-19-6 C-axis point machining shape sequence

Once the entering of the data of the machining unit and of the tool sequence is completed, the shape sequence is entered.

1. Types of C-axis point machining shape

The C-axis point machining shape sequence is available in the following three types.

C-FACE	POINT	CIRCLE	ARC
FACE			
CYLIND			
SLANT			

D736P0093

2. Entry of shape sequence data

A. Machining at the edge of the workpiece (FACE)

1. POINT

FIG	PTN	SPT-R/x	SPT-th/y	SPT-Z	NUM.	ANG	Q	R
1	PT	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	◆	<input type="checkbox"/>

Cursor positon	Description
<p>SPT-R/x SPT-th/y SPT-Z</p>	<p>Specify the start point of hole to be machined.</p> <ul style="list-style-type: none"> - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT. <p style="text-align: right;">D736P0094</p>
R	<p>Specify the position to which the tool returns after the machining.</p> <p>0: Initial point 1: Reference point (R)</p>

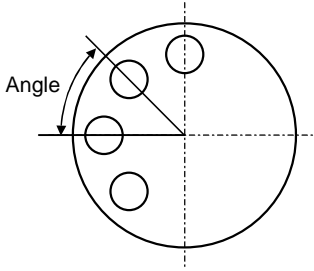
2. CIRCLE

FIG	PTN	SPT-R/x	SPT-th/y	SPT-Z	NUM.	ANG	Q	R
1	CIR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	<input type="checkbox"/>

Cursor positon	Description
<p>SPT-R/x SPT-th/y SPT-Z</p>	<p>Specify the start point of hole to be machined.</p> <ul style="list-style-type: none"> - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT. <p>For further details, see "1. POINT."</p>
NUM.	Specify the number of holes to be drilled
R	<p>Specify the position to which the tool returns after the machining.</p> <p>0: Initial point 1: Reference point (R)</p>

3. ARC

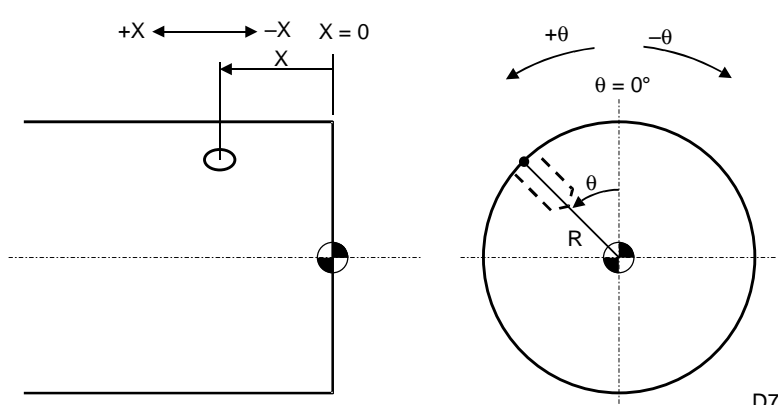
FIG	PTN	SPT-R/x	SPT-th/y	SPT-Z	NUM.	ANG	Q	R
1	ARC	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
SPT-R/x SPT-th/y SPT-Z	Specify the start point of hole to be machined. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT . For further details, see "1. POINT."
NUM.	Specify the number of holes to be drilled
ANG	Specify the angle of the hole with respect to the next hole.  D736P0095
Q	Specify whether the machining at the start point is executed or not. 0: Execution 1: No execution
R	Specify the position to which the tool returns after the machining. 0: Initial point 1: Reference point (R)

B. Machining on the cylindrical surface (CYLIND)

1. POINT

FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R
1	PT	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	◆	<input type="checkbox"/>

Cursor position	Description
SPT-R SPT-th SPT-X	Specify the start point of hole to be machined.  D736P0096
R	Specify the position to which the tool returns after the machining. 0: Initial point 1: Reference point (R)

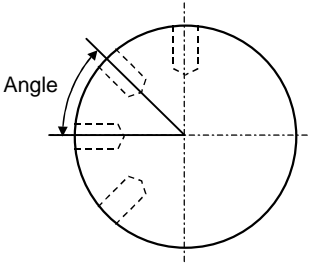
2. CIRCLE

FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R
1	CIR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	<input type="checkbox"/>

Cursor positon	Description
SPT-R	Specify the start point of hole to be machined.
SPT-th	For further details, see "1. POINT."
SPT-X	
NUM.	Specify the number of holes to be drilled
R	Specify the position to which the tool returns after the machining. 0: Initial point 1: Reference point (R)

3. ARC

FIG	PTN	SPT-R	SPT-th	SPT-X	NUM.	ANG	Q	R
1	ARC	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor positon	Description
SPT-R	Specify the start point of hole to be machined.
SPT-th	For further details, see "1. POINT."
SPT-X	
NUM.	Specify the number of holes to be drilled
ANG	Specify the angle of the hole with respect to the next hole. 
Q	Specify whether the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
R	Specify the position to which the tool returns after the machining. 0: Initial point 1: Reference point (R)

D736P0097

C. Machining on the inclined surface (SLANT)

1. POINT

FIG	PTN	SPT-R	SPT-th	SHIFT	NUM.	ANG	Q	R
1	PT	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	◆	<input type="checkbox"/>

Cursor positon	Description
SPT-R SPT-th SHIFT	<p>Specify the start point of hole to be machined.</p> <p style="text-align: right;">D736P0098</p>
R	<p>Specify the position to which the tool returns after the machining.</p> <p>0: Initial point 1: Reference point (R)</p>

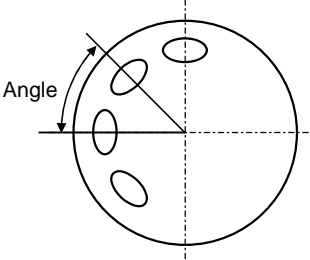
2. CIRCLE

FIG	PTN	SPT-R	SPT-th	SHIFT	NUM.	ANG	Q	R
1	CIR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	<input type="checkbox"/>

Cursor positon	Description
SPT-R SPT-th SHIFT	<p>Specify the start point of hole to be machined.</p> <p>For further details, see "1. POINT."</p>
NUM.	Specify the number of holes to be drilled
R	<p>Specify the position to which the tool returns after the machining.</p> <p>0: Initial point 1: Reference point (R)</p>

3. ARC

FIG	PTN	SPT-R	SPT-th	SHIFT	NUM.	ANG	Q	R
1	ARC	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

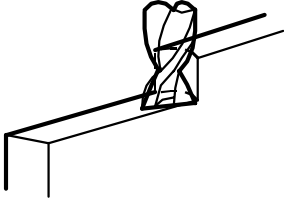
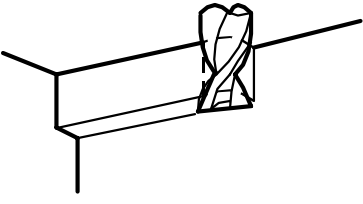
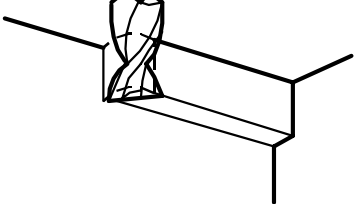
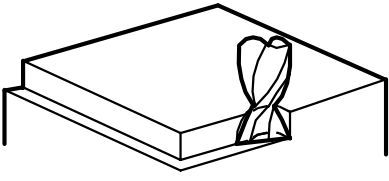

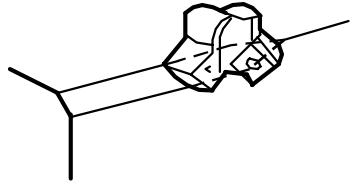
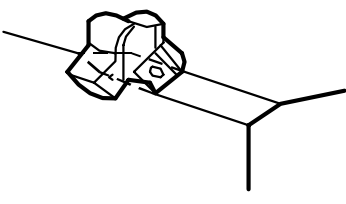
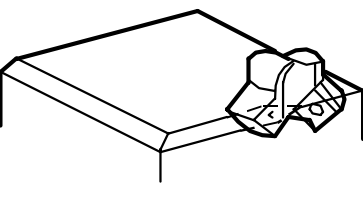
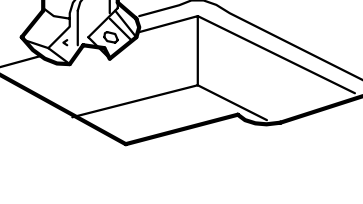
Cursor positon	Description
SPT-R	Specify the start point of hole to be machined.
SPT-th	For further details, see "1. POINT."
SHIFT	
NUM.	Specify the number of holes to be drilled
ANG	Specify the angle of the hole with respect to the next hole.  <p style="text-align: right;">D736P0135</p>
Q	Specify whether the machining at the start point is executed or not. 0: Actual execution of machining 1: Only positioning without machining
R	Specify the position to which the tool returns after the machining. 0: Initial point 1: Reference point (R)

4-20 C-Axis Line Machining Unit

The C-axis line machining unit is intended to specify data on the machining method to be used when counter machining after positioning with the C-axis and other axes (X, Z), and data on the shape of the section to be machined.

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.

4-20-1 Types of C-axis line machining units

1. Central linear machining	2. Right-hand linear machining	3. Left-hand linear machining
		
4. Outside linear machining	5. Inside linear machining	6. Right-hand chamfering
		
7. Left-hand chamfering	8. Outside chamfering	9. Inside chamfering
		

M3P171

Fig. 4-24 Types of C-axis line machining units

4-20-2 Procedure for selecting C-axis line 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	WPC	OFFSET	END	SHAPE CHECK	>>>
-------------------	------------------	------------------	---------	-------------------	-----	--------	-----	----------------	-----

(2) Pressing on the menu key >>> displays the following unit menu.

C-POINT MACH-ING	C-LINE MACH-ING	INDEX	M CODE	SUB PROGRAM	MMS	WORKPIECE MEASURE	TOOL MEASURE	WORKPIECE SHAPE	>>>
---------------------	--------------------	-------	--------	----------------	-----	----------------------	-----------------	--------------------	-----

(3) Pressing the menu key **C-LINE MACH-ING** displays the following line machining unit menu.

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

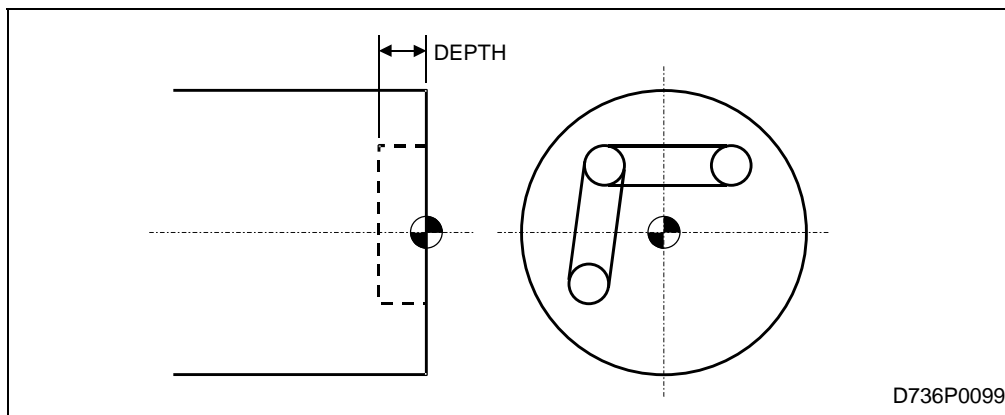
(4) Press the appropriate menu key of the desired machining unit.

4-20-3 C-axis line machining unit data and automatic tool development

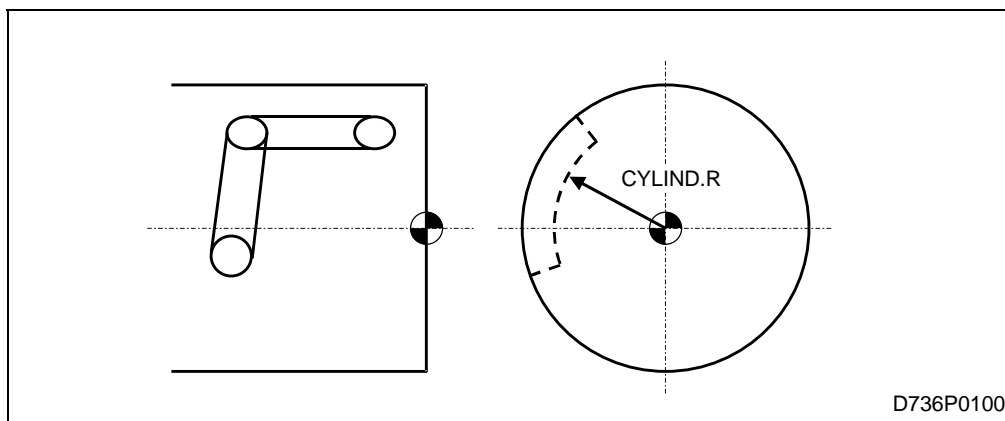
For C-axis line-machining, specify the desired face (**FACE** or **CYLIND**) in the item C-FACE.

The data items to be set in the unit differ between edge machining and cylindrical surface machining.

- If the edge (FACE) is selected: Enter a value in DEPTH.



- If the cylindrical (CYLIND) surface is selected: Enter a value in CYLIND.R.



Each tool path corresponding to the C-axis line-machining unit is also shown here.

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	C-FACE	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R					
1	LINE CTR	FACE	999.999	999.999	999.999	9	99.999	◆					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R					
1	LINE CTR	CYLIND	999.999	999.999	999.999	9	99.999	◆					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

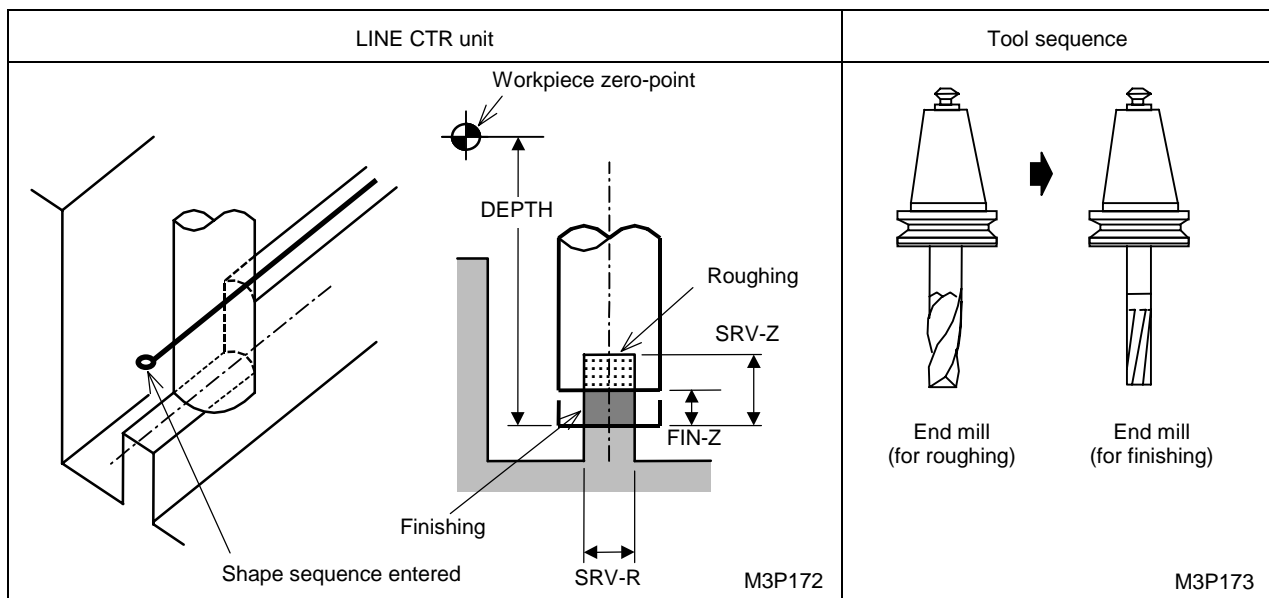
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to either face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed based on SRV-Z and on FIN-Z.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-20-4.



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

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

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	C-FACE	DEPTH	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE RGT	FACE	999.999	999.999	999.999	9		99.999	99.999				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE RGT	CYLIND	999.999	999.999	999.999	9		99.999	99.999				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL					◆			◆				
F2	END MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

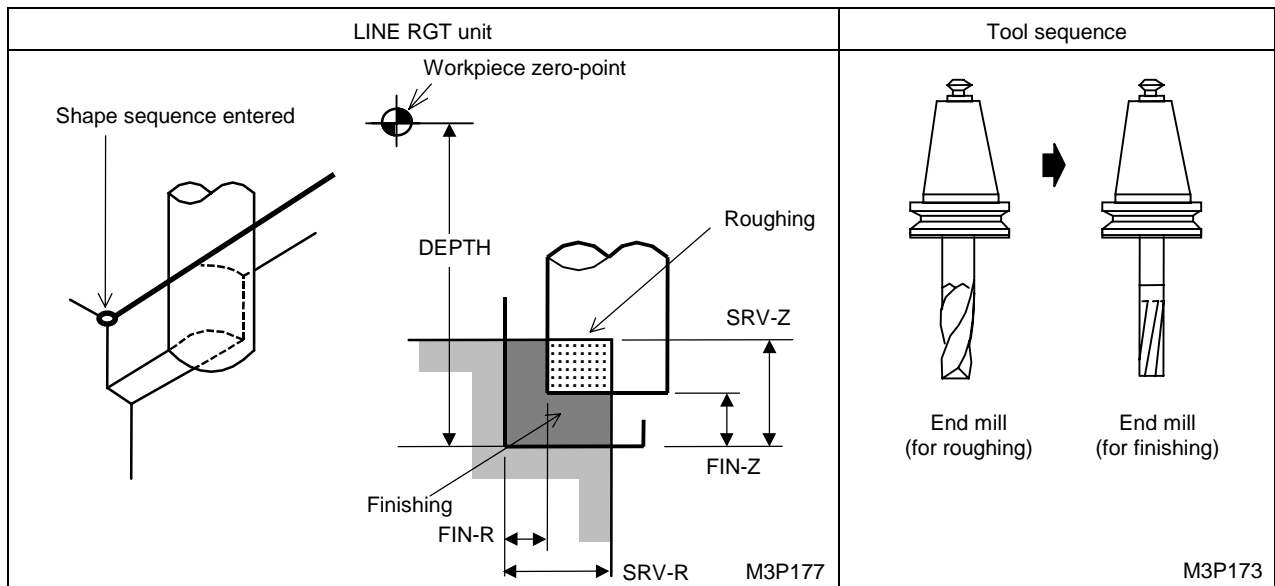
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-20-4.



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

FIN-Z: A Z-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.

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	F-FACE	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R				
1	LINE LFT	FACE	999.999	999.999	999.999	9	99.999	99.999				
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL				◆			◆				
F2	END MILL				◆		◆	◆				

UNo.	UNIT	F-FACE	CYLIN.R	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R				
1	LINE LFT	CYLIND	999.999	999.999	999.999	9	99.999	99.999				
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL				◆			◆				
F2	END MILL				◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

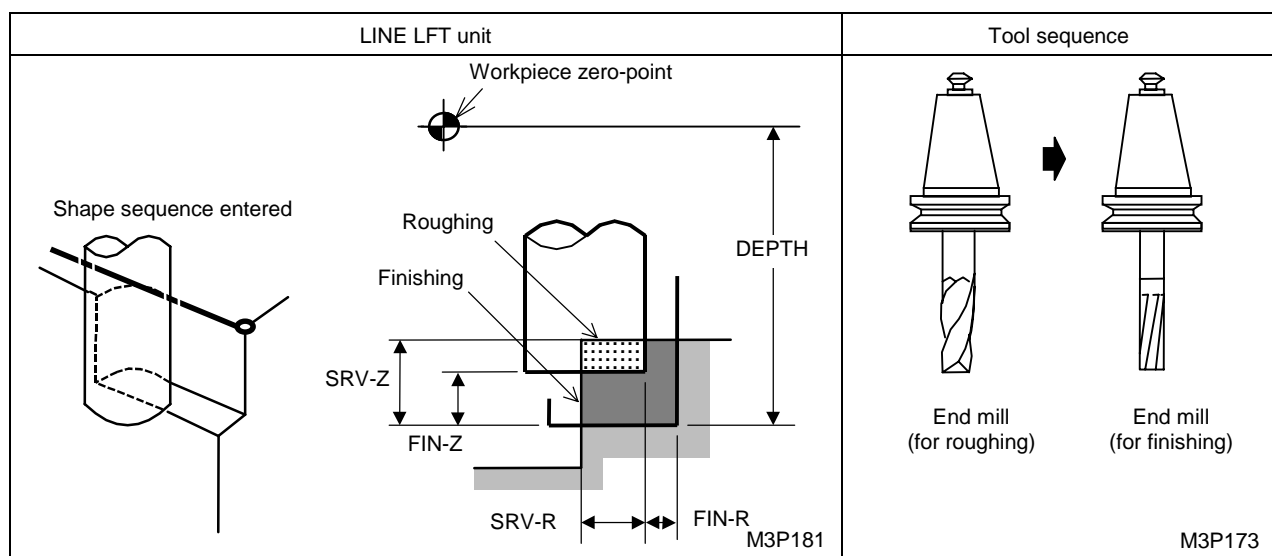
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-20-4.



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

FIN-Z: A Z-axis 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.

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	C-FACE	DEPTH	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE OUT	FACE	999.999	999.999	999.999	9		99.999	99.999				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL								◆				

UNo.	UNIT	C-FACE	CYLIN.R	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE OUT	CYLIND	999.999	999.999	999.999	9		99.999	99.999				
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL								◆				

Remark 1: Data in unit represent a maximum input value.

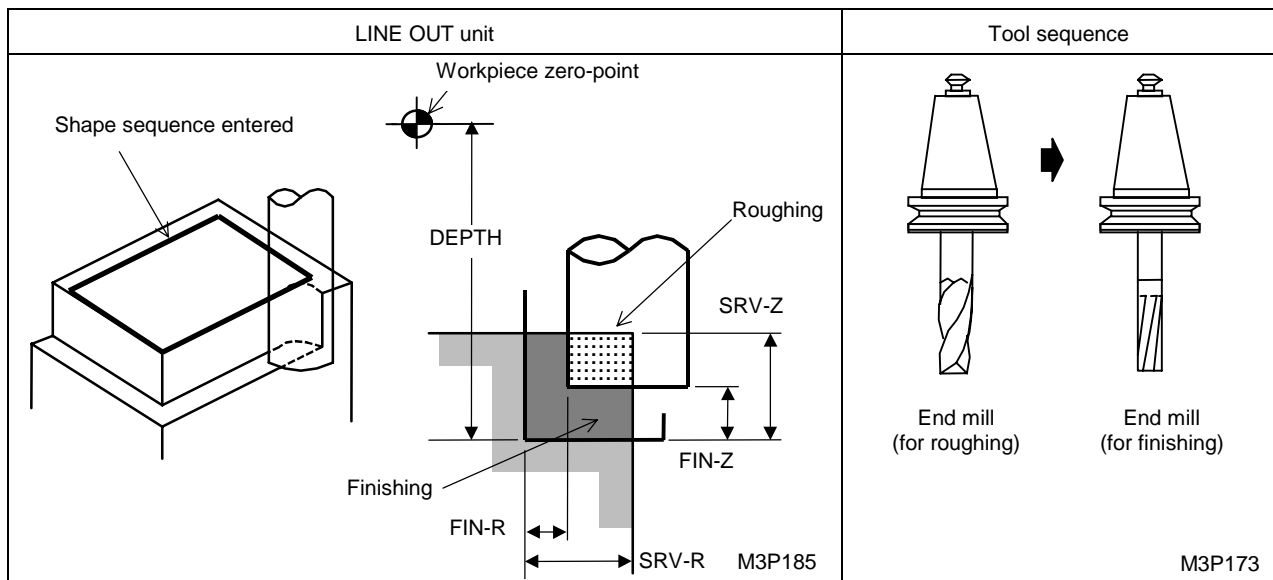
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, refer to Subsection 4-20-4.



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

FIN-Z: A Z-axis 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.

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	C-FACE	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R					
1	LINE IN	FACE	999.999	999.999	999.999	9	99.999	99.999					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R					
1	LINE IN	CYLIND	999.999	999.999	999.999	9	99.999	99.999					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	END MILL								◆				
F2	END MILL							◆	◆				

Remark 1: Data in unit represent a maximum input value.

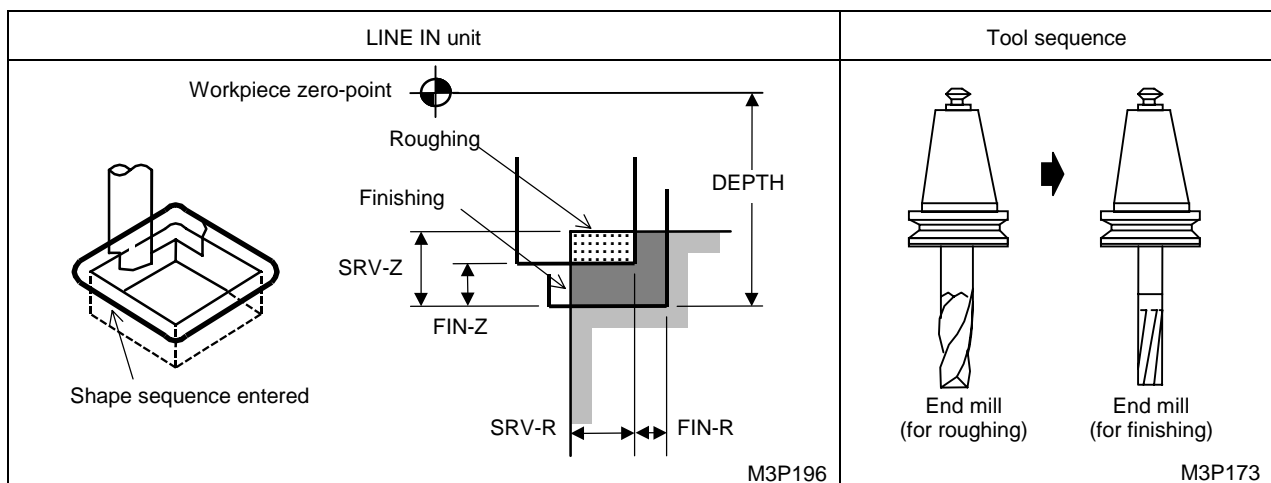
Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, end mills are automatically developed. Nevertheless, they may be switched over to face mill or ball end mill.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed though dependent upon the data SRV-Z, SRV-R, FIN-Z and FIN-R.

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

Remark 5: For the tool sequence data setting, see Subsection 4-20-4.



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

FIN-Z: A Z-axis 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.

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	C-FACE	DEPTH	INTER-Z	INTER-R	CHMF						
1	CHMF RGT	FACE	999.999	99.999	99.999	99.99						
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER				◆		◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	INTER-Z	INTER-R	CHMF						
1	CHMF RGT	CYLIND	999.999	99.999	99.999	99.99						
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER				◆		◆	◆				

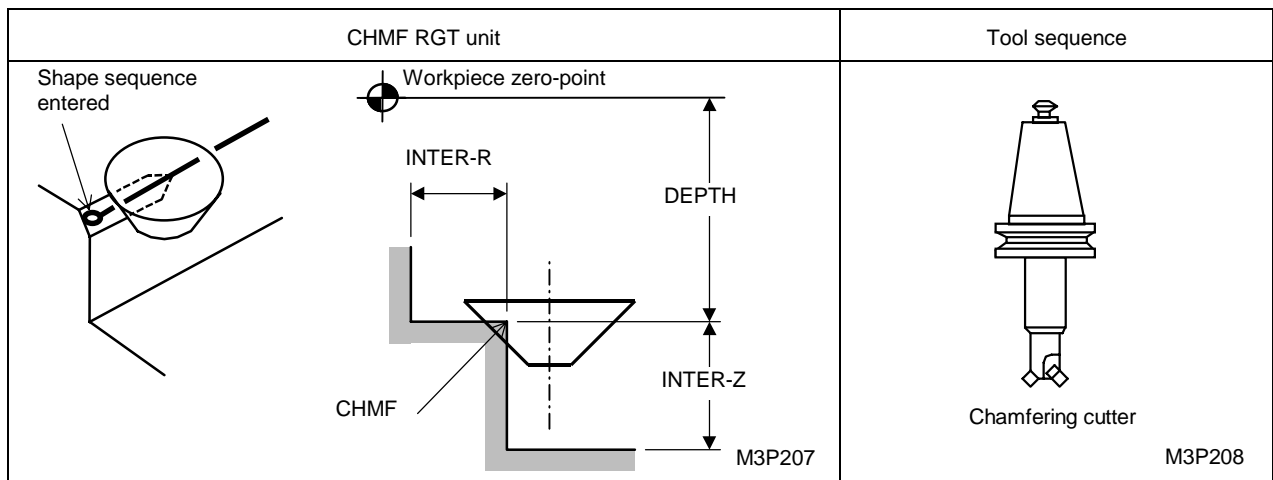
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For the tool sequence data setting, refer to Subsection 4-20-4.

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



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	C-FACE	DEPTH	INTER-Z	INTER-R	CHMF							
1	CHMF LFT	FACE	999.999	99.999	99.999	99.9							
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
1	CHAMFER				◆		◆	◆					

UNo.	UNIT	C-FACE	CYLIN.R	INTER-Z	INTER-R	CHMF							
1	CHMF LFT	CYLIND	999.999	99.999	99.999	99.9							
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
1	CHAMFER				◆		◆	◆					

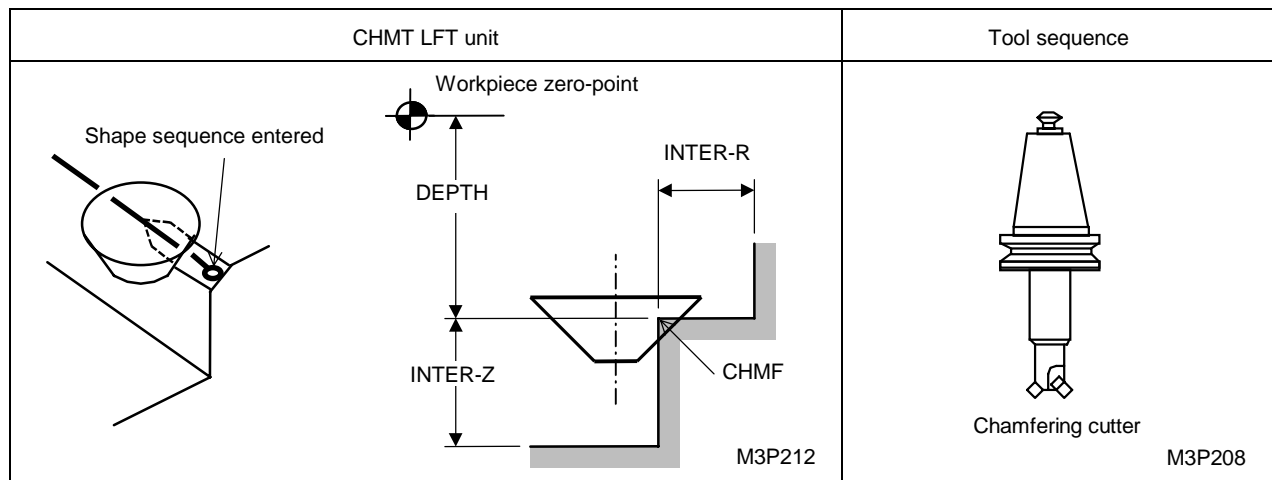
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For the tool sequence data setting, refer to Subsection 4-20-4.

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



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	C-FACE	DEPTH	INTER-Z	INTER-R	CHMF						
1	CHMF OUT	FACE	999.999	99.999	99.999	99.99						
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER						◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	INTER-Z	INTER-R	CHMF						
1	CHMF OUT	CYLIND	999.999	99.999	99.999	99.99						
SNo.	TOOL	NOM-φ No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER						◆	◆				

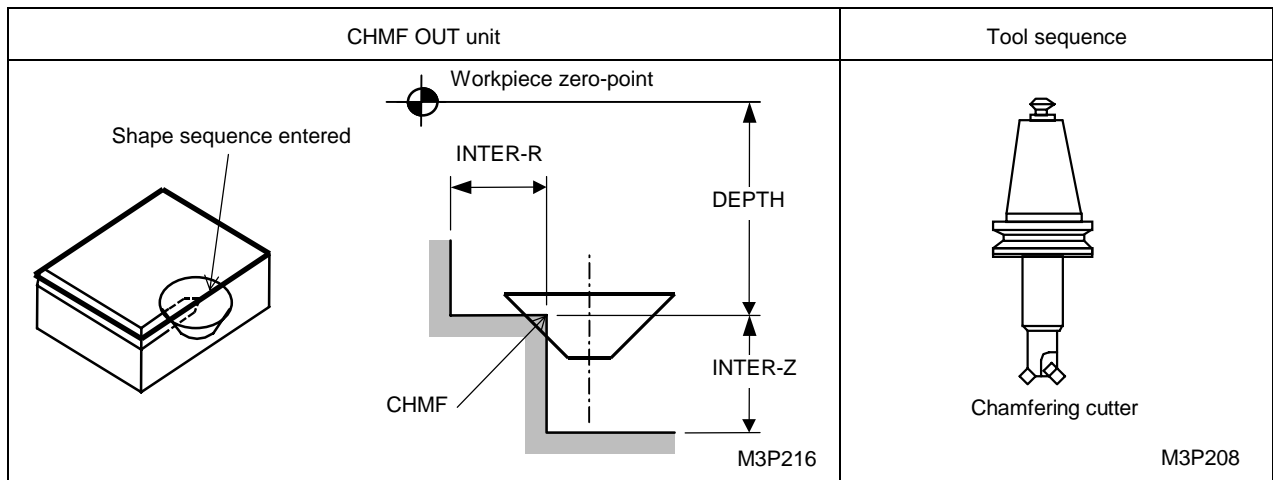
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be here.

Remark 3: In this unit, chamfering tools are automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For tool sequence data setting, refer to Subsection 4-20-4.

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



9. Inside chamfering unit (CHMF IN)

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

A. Data setting

UNo.	UNIT	C-FACE	DEPTH	INTER-Z	INTER-R	CHMF							
1	CHMF IN	FACE	999.999	99.999	99.999	99.9							
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER							◆	◆				

UNo.	UNIT	C-FACE	CYLIN.R	INTER-Z	INTER-R	CHMF							
1	CHMF IN	CYLIND	999.999	99.999	99.999	99.9							
SNo.	TOOL	NOM- ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER							◆	◆				

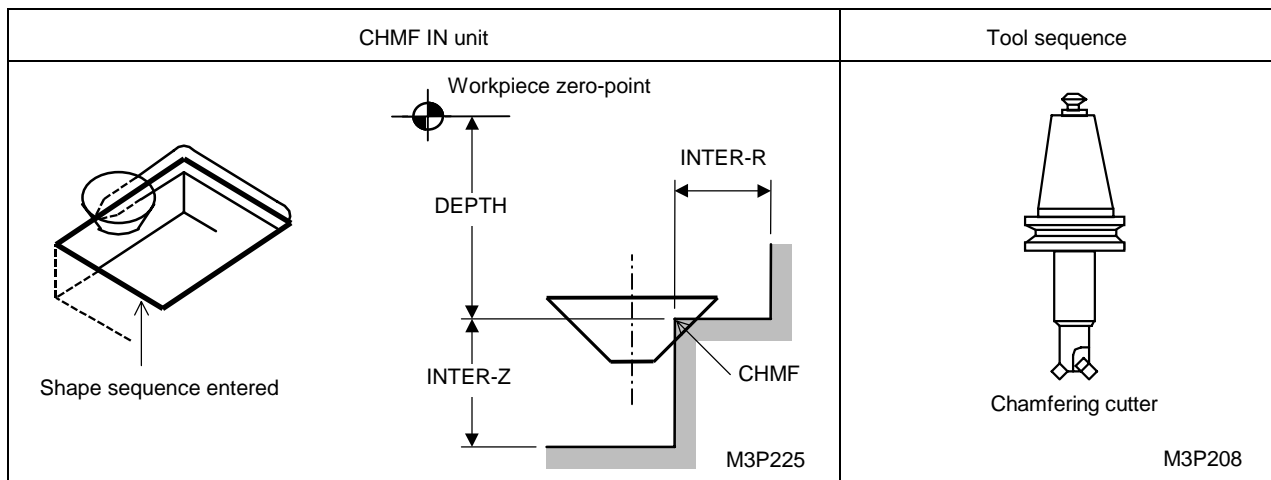
Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

Remark 3: In this unit, chamfering cutter is automatically developed. Instead of the chamfering cutter, a centering drill can be used.

Remark 4: For sequence data setting, refer to Subsection 4-20-4.

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



4-20-4 C-axis line machining tool sequence data

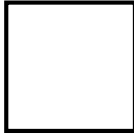
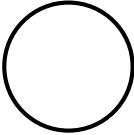
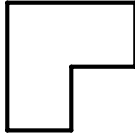
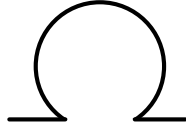
For line machining tool sequence data only a tool name is automatically selected once a machining unit has been entered. Other data should be entered by use of menu keys or numeric keys according to a form of the workpiece to be machined or to the procedure for machining. Data to be set in the tool sequence are the same as those of the point-machining. See Section 4-8-4 “Line machining tool sequence data.”

4-20-5 Definitions of forms in C-axis line machining and face machining units

After the data in the machining unit and the tool sequence have been entered, enter the data related to the machining form and dimensions in the shape sequence.



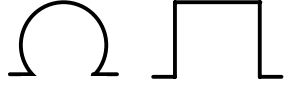
1. Definitions of forms

In line machining and face machining units one of the following three patterns can be selected.

Fixed form		Arbitrary form	
SQUARE	CIRCLE	ARBITRY	
			

A. Closed form and open form

Depending on the machining units, machining form can be divided into the following two types:



	Closed form		Open form
	Fixed form	Arbitrary form	Arbitrary form
			
Line machining	LINE OUT, LINE IN, CHMF OUT, CHMF IN		LINE CTR, LINE RGT, LINE LFT, CHMF RGT, CHMF LFT

2. To define an arbitrary form, take the following precautions:



1. For an open form, do not fail to establish coordinates X, Y of its start and end points.
2. In an open form, it is impossible to select the corner (C or R) of its start or end point.
3. The meaning of a start point (X, Y) differs between an open form and a closed one.
 Open form.....The start point (X, Y) has a meaning as a point.
 Closed form.....The start point (X, Y) has a meaning as a line from an end point to the start point.

3. Setting shape sequence data

For C-axis point-machining, specify the desired shape (**SQUARE**, **CIRCLE** or **ARBITRY**) by pressing the menu key in the item PTN.

SQUARE 	CIRCLE 		ARBITRY 				SHAPE END	
---	---	--	--	--	--	--	--------------	--

If the shape **ARBITRY** is selected, the following menu will be displayed. Select the desired shape from the menu.

LINE	CW ARC 	CCW ARC 	SHAPE ROTATE	SHAPE SHIFT	REPEAT END	STARTING POINT	SHAPE END	
------	---	--	-----------------	----------------	---------------	-------------------	--------------	--

The shape data to be set differs according to the face that has been selected in the item C-FACE in the machining unit. Shape data is described below for each face that is to be machined.

A. Machining at the edge of the workpiece (FACE)

1. SQUARE (fixed form)

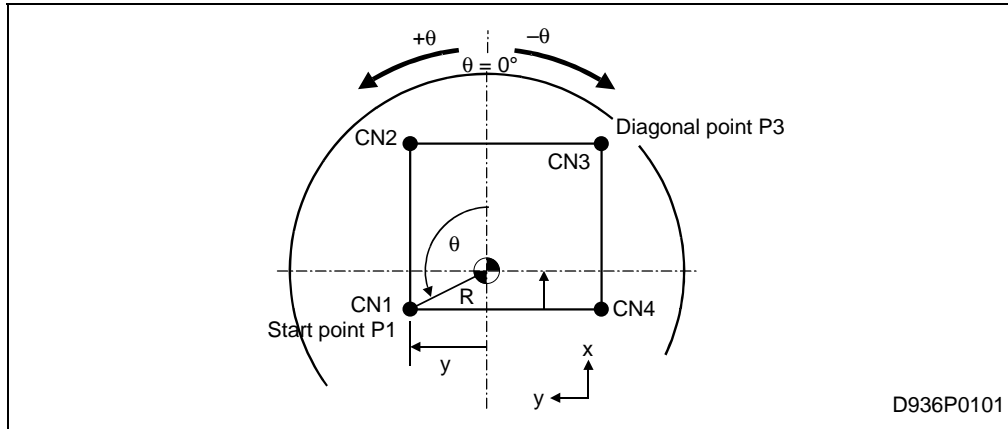


FIG	PTN	P1Rx/CRx	P1thy/Cthy	P3Rx/R	P3thy	CN1	CN2	CN3	CN4
	SQR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor positon	Description
P1Rx/CRx P1thy/Cthy	Enter the coordinates of start point. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT .
P3Rx/R P3thy	Enter the coordinates of diagonal point. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT .
CN1	Select a machining form at corner 1. Set corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly. Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C Pressing the menu key CORNER CHAMFER makes the menu display reverse and entering a numeric value will cause that menu display status to become normal.
CN2 CN3 CN4	Select a machining form at corners 2, 3 and 4. Enter the same data as those for corner 1.

2. CIRCLE (fixed form)

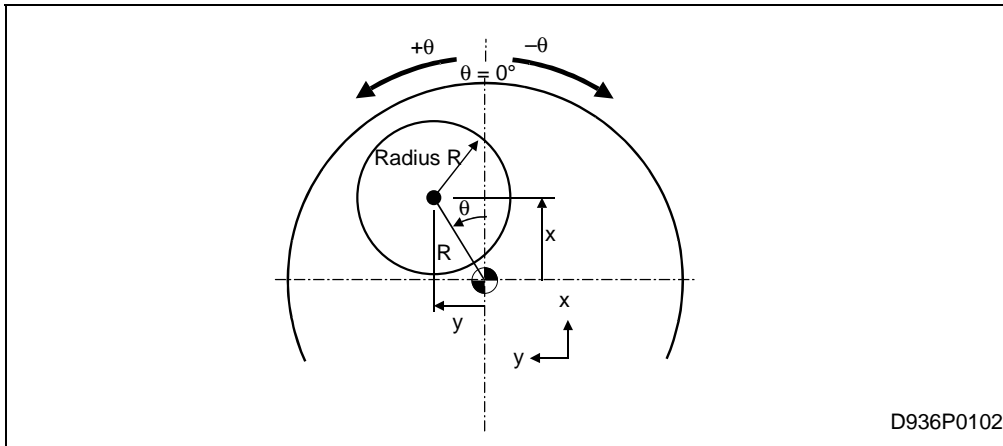


FIG	PTN	P1Rx/CRx	P1thy/Cthy	P3Rx/R	P3thy	CN1	CN2	CN3	CN4
	CIR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	◆	◆	◆

Cursor positon	Description
CRx Cthy	Enter the coordinates of the center of a circle. - To specify the start point in the R- θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT .
R	Enter the radius of the circle to be machined.

3. ARBITRARY (arbitrary form) - LINE

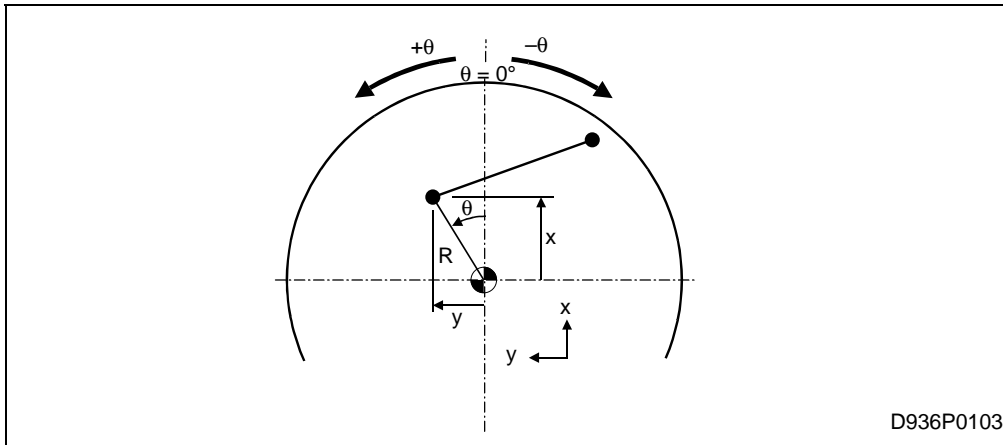


FIG	PTN	R/x	th/y	RADIUS/th.	I	J	P	CNR	R-FEED
	LIN	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor positon	Description
R/x th/y	Enter the coordinates of an end point for line machining. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT . - If the coordinate is unknown, select the menu key ?.
RADIUS/th.	Enter the angle between X-axis and machining line (θ).
I J	Enter the X- and Y-axial vector values.
P	Select the conditions of intersecting using the menu key. Note: For details, refer to “Function of automatically calculating a point of intersection” in Subsection 4-9-7.
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C
R-FEED (RGH)	Specify the feed rate for roughing. If this value is not specified, the feed rate preset for the tool sequence will be used. Note: If bit 0 in parameter E99 is “1”, specify the feed rate for finishing. In this case, “RGH” will be displayed in the item.

4. ARBITRY (arbitrary form) – CW ARC, CCW ARC

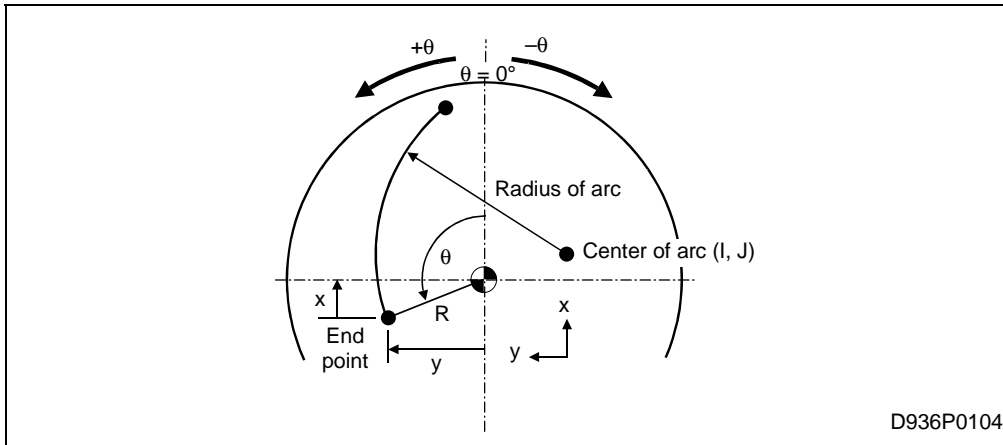


FIG	PTN	R/x	th/y	RADIUS/th.	I	J	P	CNR	R-FEED
	CW ARC	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
R/x th/y	Enter the coordinates of an end point. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT . - If the coordinate is unknown, select the menu key ?.
RADIUS/th.	Enter the radius of an arc. - If the coordinate is unknown, select the menu key ?.
I J	Enter the coordinates value of the center of an arc. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT . - If the coordinate is unknown, select the menu key ?.
P	Select the conditions of intersecting using the menu key. Note: For details, refer to “Function of automatically calculating a point of intersection” in Subsection 4-9-7.
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C
R-FEED (RGH)	Specify the feed rate for roughing. If this value is not specified, the feed rate preset for the tool sequence will be used. Note: If bit 0 in parameter E99 is “1”, specify the feed rate for finishing. In this case, “RGH” will be displayed in the item.

5. ARBITRY (arbitrary form) - SHAPE ROTATE (CW and CCW), SHAPE SHIFT
For setting of shape data to machine the arbitrary form by the fonctions **SHAPE ROTATE** (CW and CCW), **SHAPE SHIFT**, see Section 4-9-7 “Definitions of forms in line machining and face machining units”.

B. Machining on the cylindrical surface (CYLIND)

1. SQUARE (fixed form)

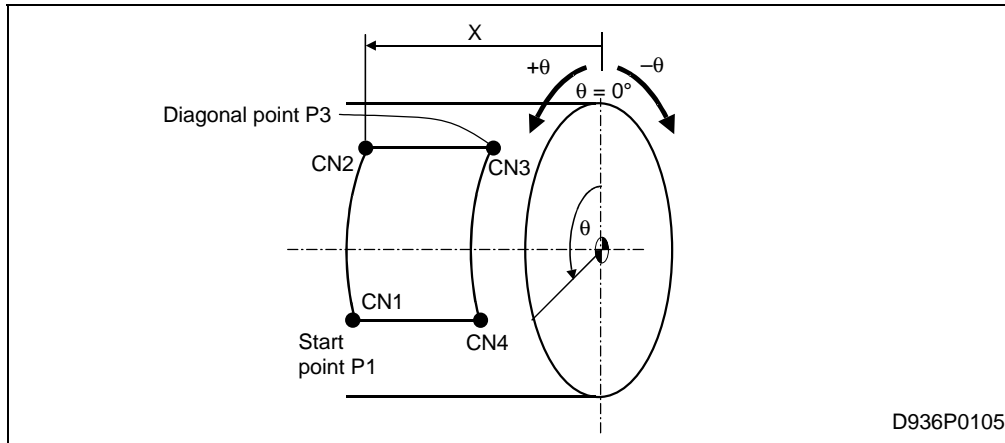


FIG	PTN	P1X/CX	P1th/Cth	P3X/R	P3th	CN1	CN2	CN3	CN4
	SQR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
P1X/CX P1th/Cth	Enter the coordinates of start point. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT .
P3X/R P3th	Enter the coordinates of diagonal point. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT .
CN1	Select a machining form at corner 1. Set corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C Pressing the menu key CORNER CHAMFER makes the menu display reverse and entering a numeric value will cause that menu display status to become normal.
CN2 CN3 CN4	Select a machining form at corners 2, 3, and 4. Enter the same data as those for corner 1.

2. CIRCLE (fixed form)

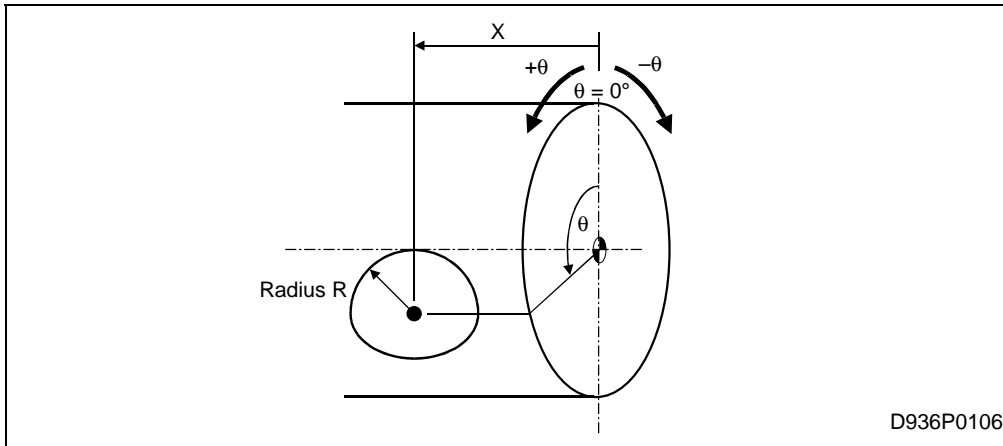


FIG	PTN	P1X/CX	P1th/Cth	P3X/R	P3th	CN1	CN2	CN3	CN4
	CIR	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	◆	◆	◆	◆	◆

Cursor positon	Description
CX Cth	Enter the coordinates of the center of a circle.
R	Enter the radius of the circle to be machined.

3. ARBITRARY (arbitrary form) – LINE

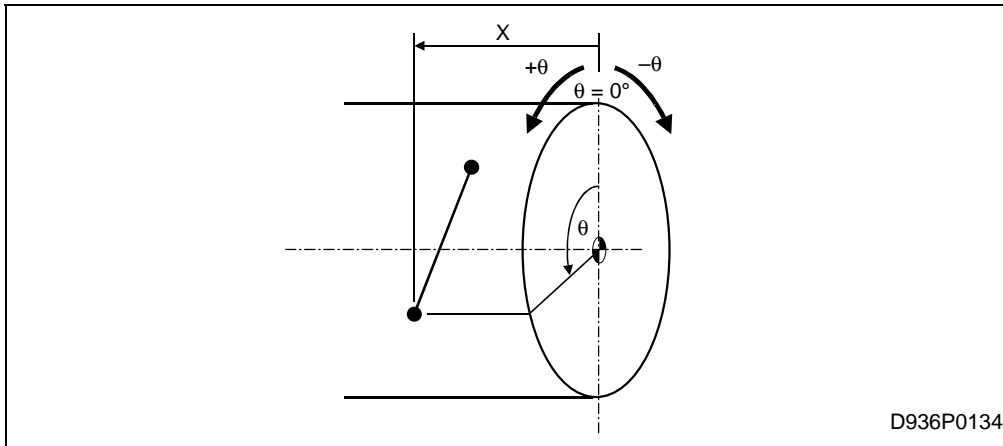
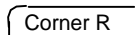
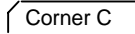


FIG	PTN	X	th	RADIUS/th.	I	J	P	CNR	R-FEED
	LIN	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
X th	Enter the coordinates of an end point for line machining. - If the coordinate is unknown, select the menu key ?.
RADIUS/th.	Enter the angle between X-axis and machining line (θ).
I J	Enter the X- and Y-axial vector values.
P	Select the conditions of intersecting using the menu key. Note: For details, refer to "Function of automatically calculating a point of intersection" in Subsection 4-9-7.
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly..... Corner R C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value..... Corner C
R-FEED (RGH)	Specify the feed rate for roughing. If this value is not specified, the feed rate preset for the tool sequence will be used. Note: If bit 0 in parameter E99 is "1", specify the feed rate for finishing. In this case, "RGH" will be displayed in the item.

4. ARBITRY (arbitrary form) – CW ARC, CCW ARC

FIG	PTN	X	th	RADIUS/th.	I	J	P	CNR	R-FEED
	CW ARC	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Cursor position	Description
X th	Enter the coordinates of an end point. - If the coordinate is unknown, select the menu key ?.
RADIUS/th.	Enter the radius of an arc. - If the coordinate is unknown, select the menu key ?.
I J	Enter the coordinates value of the center of an arc. - To specify the start point in the R-θ coordinate format, enter the radius and the angle as they are. - To specify the start point in the x-y coordinate format, enter data after highlighting the menu key x-y INPUT . - If the coordinate is unknown, select the menu key ?.
P	Select the conditions of intersecting using the menu key. Note: For details refer to “Function of automatically calculating a point of intersection.” in Subsection 4-9-7.
CNR	Specify the corner chamfering distance (C) or corner radius of a circular arc (R). R-machining: Enter the numeric value directly.  C-chamfering: Press the menu key CORNER CHAMFER and enter a numeric value. 
R-FEED (RGH)	Specify the feed rate for roughing. If this value is not specified, the feed rate preset for the tool sequence will be used. Note: If bit 0 in parameter E99 is “1”, specify the feed rate for finishing. In this case, “RGH” will be displayed in the item.

5. ARBITRY (arbitrary form) – SHAPE ROTATE (CW and CCW), SHAPE SHIFT
For setting of shape data to machine the arbitrary form by the functions **SHAPE ROTATE** (CW and CCW), **SHAPE SHIFT**, see Section 4-9-7 “Definitions of forms in line machining and face machining units.”

4-20-6 Precautions in C-axis line machining

See Section 4-8-6 “Precautions in line machining.”

4-21 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. See Section 4-3, Index Unit, for further details.

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.

4-21-1 Types of turning units

Eight types of turning units are provided.









- Bar-materials machining unit (BAR)
- Copy-machining unit (CPY)
- Corner-machining unit (CORNER)
- Edge-machining unit (FACING)
- Threading unit (THREAD)
- Grooving unit (T.GROOVE)
- Turning drilling unit (T.DRILL)
- Turning tapping unit (T.TAP)

4-21-2 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	WPC	OFFSET	END	SHAPE CHECK	>>>
-------------------	------------------	------------------	---------	-------------------	-----	--------	-----	----------------	-----









- (2) Pressing on the menu key **TURNING** displays the following unit menu.

BAR 	CPY 	CORNER 	FACING 	THREAD 	T.GROOVE 	T.DRILL 	T.TAP 		FINISH ALLOW
--	--	---	---	---	---	--	--	--	-----------------

- (3) Press the menu key corresponding to the desired machining unit.

Notes on the menu option FINISH ALLOW:

When the menu key **FINISH ALLOW** is selected from the turning unit selection menu and then a turning unit is created with the menu item **FINISH ALLOW** highlighted as follows, if another turning unit already exists in front of that created unit, the operator can automatically set the same values as the finishing allowances specified in the existing turning unit.

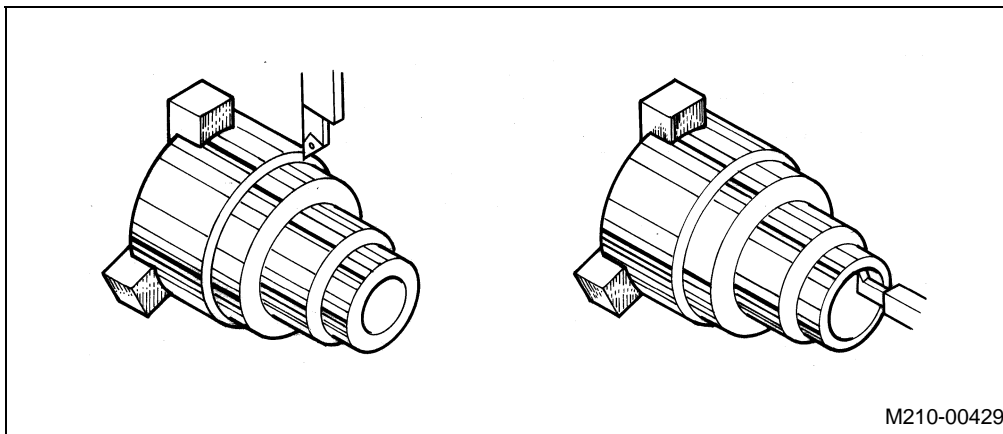
BAR 	CPY 	CORNER 	FACING 	THREAD 	T.GROOVE 	T.DRILL 	T.TAP 		FINISH ALLOW
--	--	---	---	---	---	--	--	--	-------------------------

- For "FIN-X" and "FIN-Z" each, independent finishing allowance values are automatically determined from the values that have been specified in the existing turning unit.
- When no finishing allowances are specified in the previous turning unit, older turning units will be referred to in sequence and if the settings of finishing allowances are not detected in any turning units down to the beginning of the program, finishing allowances will not be auto-set.

- The highlighted status of the **FINISH ALLOW** menu option is maintained, even after power has been turned off.

4-21-3 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.



Press the menu key **BAR**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
*	BAR	[1]	[2]	[3]	[4]	[5]

[1] PART

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

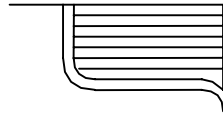
OUT	OUT	IN	IN	FCE	FCE	BAX	BAX		
-----	------------	----	-----------	-----	------------	-----	------------	--	--

From the menu, select the section to be machined.

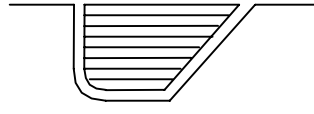
Sections to be machined that correspond to each menu item are as follows:

- 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
- FCE** : Front face (Cutting from the outer or inner periphery)
..... Front-face open type
- FCE** : Front face (Cutting from the middle of the front face)
..... Front-face middle type
- FCE** : Back face (Cutting from the outer or inner periphery)
..... Back-face open type
- BAK** : Back face (Cutting from the middle of the back face)
..... Back-face middle type

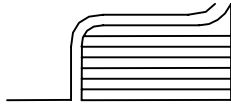
<OUT : Oustside-diameter open type>



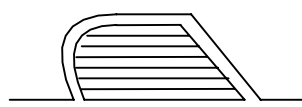
< **OUT** : Oustside-diameter middle type>



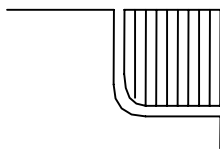
<IN : Inside-diameter open type>



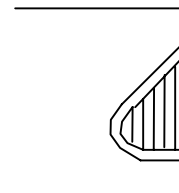
< **IN** : Inside-diameter middle type>



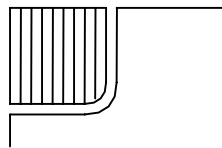
<FCE : Front-face open type>



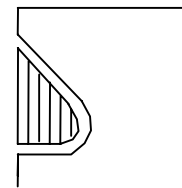
< **FCE** : Front-face middle type>



<BAK : Back-face open type>



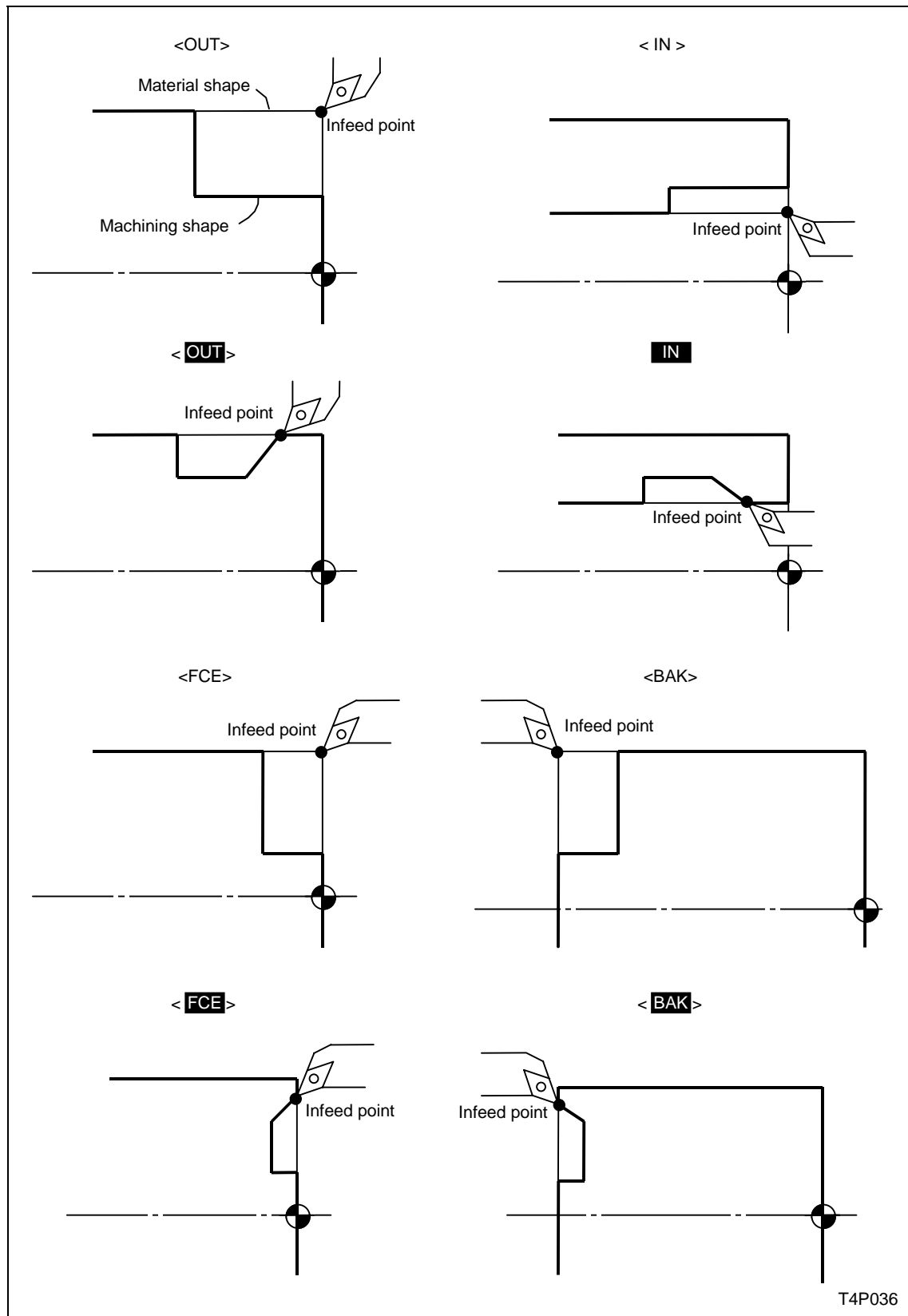
< **BAK** : Back-face middle type>



T4P030

[2] CPT-X, [3] CPT-Z

Set the X- and Z-coordinates of the desired infeed point.

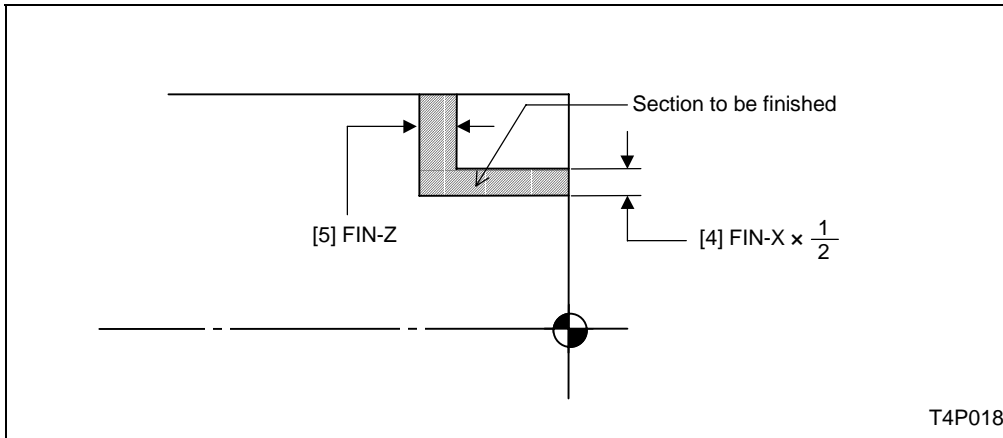


T4P036

- The infeed point refers to the cutting start point of the tool tip.
Data thus set and sequence data automatically determine the actual cutting area.

[4] FIN-X, [5] FIN-Z

Set the finishing allowances for the X-axis and Z-axis directions (removal allowances during finishing).



- Set the finishing allowance for the X-axis direction in terms of diameter data.
- These values are used for bar-machining units (BAR), copy-machining units (CPY), Edge-machining 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 Section 4-21-2.

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
R1						◆	◆	◆	◆				
F2				◆	◆	◆	◆						
	↑	↑	↑ ↑	↑	↑			↑	↑	↑	↑	↑	↑
	[1]	[2]	[3] [4]	[5]	[6]	[7]		[8]	[9]	[10]	[11]	[12]	[12]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	One tool for roughing is automatically selected.
F2 (Finishing)	FIN > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T. DRILL	T. TAP	SPECIAL			

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item “[1] TOOL (Name)” is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				
--------------------------	-------------------------	-------------	--	-----------------------------	-----------------------------	--	--	--	--

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				
--	--	-------------	--	--	-----------------------------	--	--	--	--

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	
------	------	------	------	------	------	------	------	------	--

When creating a bar-materials machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
OUT		OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
IN		IN INNER DIAMETER, IN INNER (BAK)
FCE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
FCE		EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)
BAK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys. “NOM.” is a data item that identifies tools of the same type. The tools of the same type that match in item “NOM-φ” and “Tool identification code” registered on the **TOOL DATA** display are used during actual machining.

Enter numeric data in item “NOM.” for the purpose of identifying tools. Although the numeric data can be either the “Nose R”, “Nose angle”, and/or any other characteristic factor of the tools, the data must be the same as that of the desired tools registered on the **TOOL DATA** display.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

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

With the menu key **HEAVY TOOL** 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 size.

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining. There are three following entry methods.

- Prior machining priority number

To be set by means of numeric keys. The number is displayed in yellow.

Setting range (1 to 99).

- Subsequent machining priority number

To be set by means of numeric keys after having reversed the display of **DELAY PRIORITY** by selecting its menu key. The characters are displayed in reversed status.

Setting range (1 to 99).




- Without entry (ordinary machining)

The priority number is not entered.

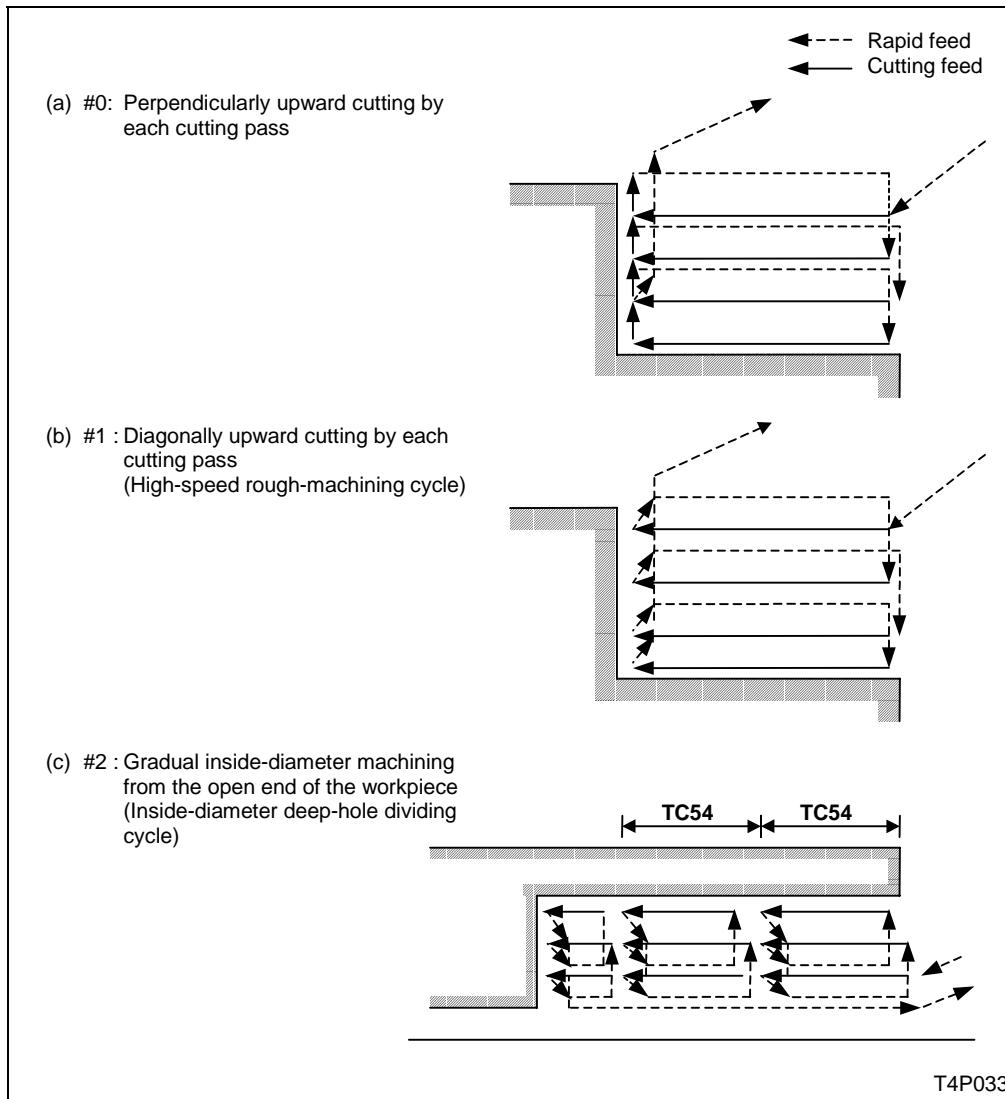
For details, refer to Chapter 5, "PRIORITY FUNCTION FOR THE SAME TOOL."

[6] PAT. (Machining pattern)

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

#0 	#1 	#2 							
(a)	(b)	(c)							

Select the machining pattern from (a) through (c) above. The data of the displayed menu denote the following machining patterns:



#2 machining can be used only when **IN** is selected for item [1] of the unit.

The #2 cycle, however, cannot be selected to turn inside diameter for a shape of which the size increases with the depth.

Chips may clog the hole bottom during conventional inside-diameter machining of deep holes. No such problems occur with this machining pattern (#2) since gradual cutting from the open end of a workpiece ensures highly efficient, automatic removal of chips.

Use parameter **TC54** to specify the depth of cut per pass.

[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.

For automatic setting of items [7] DEP-1, [10] C-SP, and [11] 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 Section 8-1, "CUTTING CONDITION (W.-MAT./T.-MAT.) Display" of the Operating Manual.

Example of display:

CARBIDEL	UNINTRPT	COATINGL	CERMET L	CERAMICL	CBN L	HSS D	CARBIDED	>>>	TOOL DAT
AUTO	AUTO	AUTO	AUTO	AUTO	AUTO	AUTO	AUTO		WINDOW

Specification using the numeric data keys is also possible. In addition, using th menu key **TOOL DAT WINDOW** the tools of the same type that are registered on the **TOOL DATA** display can be listed in a window display format.

[8] FIN-X, [9] 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 Subsection 2-2-3 "Insertion" for the insertion of tool sequences.

Note 1: The allowance to be left for the tool sequence corresponding to the automatically developed finishing tool data is set to 0 automatically.

Note 2: If a value other than 0 is specified in the final finishing tool sequence as the allowance to be left, the as-finished shape of the workpiece will differ from the shape that has been specified in the shape sequence.

[10] C-SP

Specify the peripheral speed for the turning spindle.

This peripheral speed, as with item [7] DEP-1 (Maximum cutting depth), can be selected from the menu or entered using the numeric data keys.

[11] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution. Use the numeric data keys to enter the value.

For the roughing tool sequence, this feedrate, as with items [7] DEP-1 (Maximum cutting depth) and [10] C-SP, can be selected from the menu or entered using the numeric data keys.

[12] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).

Select the desired code from the following menu (the codes displayed vary from model to model) or enter the desired code using the numeric data keys:





01 OPT.	03 SPNDL	04 SPNDL	05 SPNDL	07 MIST	08 FLOOD	09 OFF	50 AIR	51 THR	>>>
STOP	FWD	REV	STOP	COOLANT	COOLANT	COOLANT	BLAST	COOLANT	
00 PROG	19 SPNDL	35 T-BRK	38 SPNDL	39 SPNDL	33 OUT	34 IN			>>>
STOP	ORIENT	DETECT	GEAR L/M	GEAR H	MSR UNIT	MSR UNIT			
16 OPEN	15 CLOSE	11 TOOL	10 TOOL	53 CHIP	09 CHIP				>>>
ATC CVR	ATC CVR	UNCLAMP	CLAMP	VAC ON	VAC OFF				

3. Setting shape sequence data

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/\$	RADIUS/th	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

[1] PTN

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

					CENTER				SHAPE END
(a)	(b)	(c)	(d)		(e)				(f)

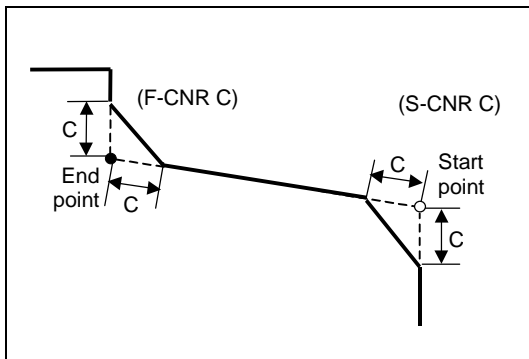
Select the type of machining shape pattern from the above four types (a) to (d).

Menu function	Description
(a)	Select to specify the straight line parallel to the center line of the workpiece.
(b)	Select to specify a straight line not parallel to the center line of the workpiece (namely, a taper).
(c)	Select to specify a bulged arc.
(d)	Select to specify a recessed arc.
(e)	Select to use the automatic crossing-point calculation function on the bulged (convex) arc or recessed (concave) arc drawn on the previous sequence line.
(f)	Select to proceed to the next unit after entering all shape data.

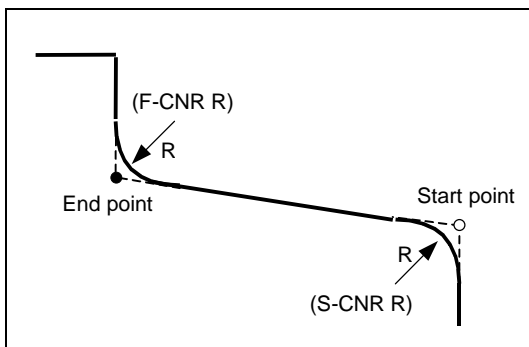
[2] S-CNR

Set data for this item when C-chamfering (cornering) or R-chamfering (rounding) is to be done at the start point of the shape.

- If C-chamfering is to be done:
Set the amount of chamfering (C in the diagram).

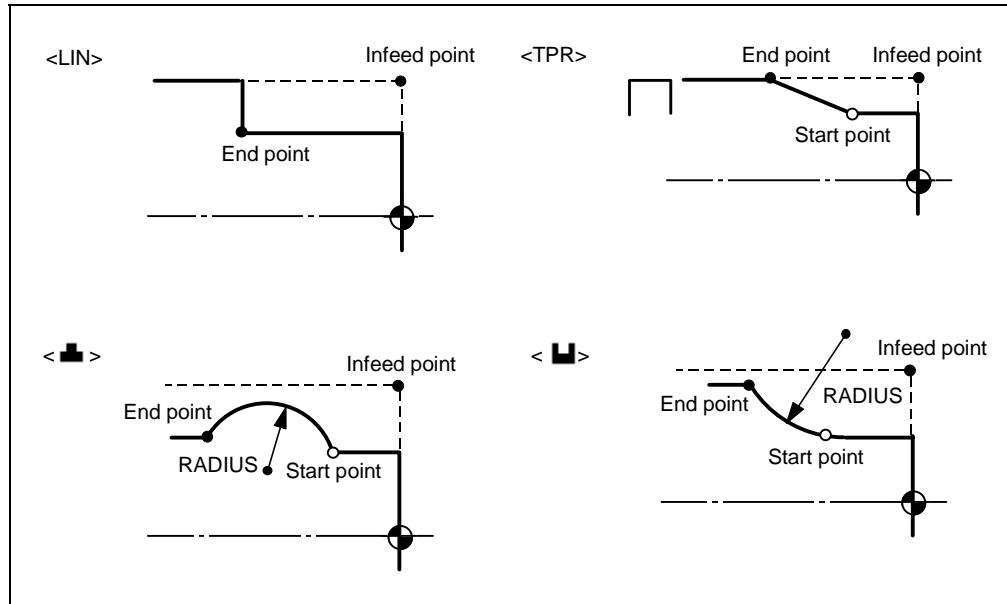


- If R-chamfering is to be done:
After pressing the menu key **CORNER R**, set the radius of rounding (R in the diagram).

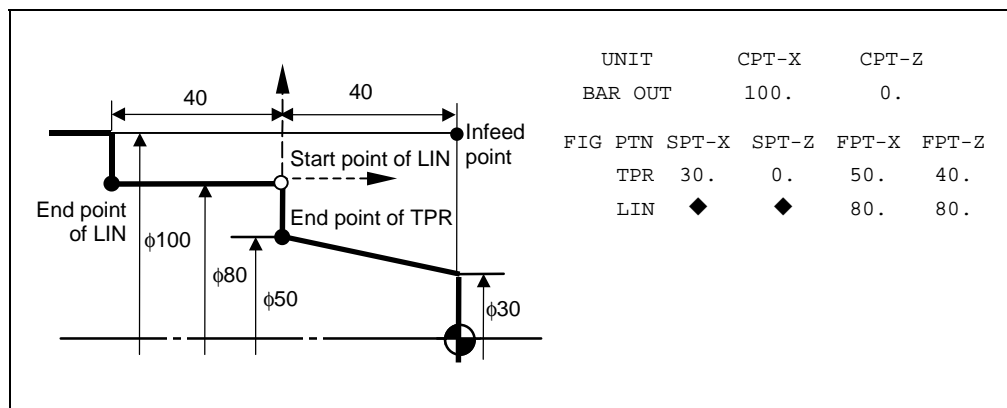


[3] SPT-X, [4] SPT-Z, [5] FPT-X, [6] FPT-Z

- 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 menu key **I. POINT?** or **C. POINT?** $\overline{\circ}$.
See "Automatic Crossing-Point Calculation Function" for further details.



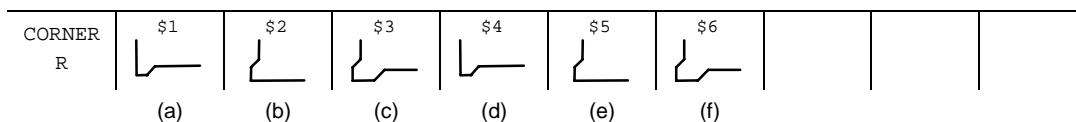
- If the selected shape type is LIN, the coordinates of the start point need not be set. The NC unit will auto-set those coordinates. A horizontal line will be drawn from the end point of LIN towards the infeed point, and the crossing point of this line and the line that is perpendicularly drawn from the end point of the preceding SEQ (or from the infeed point for an LIN as the first SEQ) will be set as the start point of the relevant LIN.



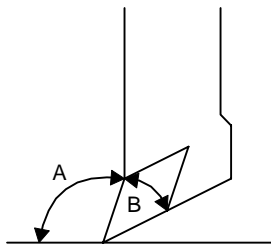

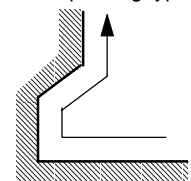
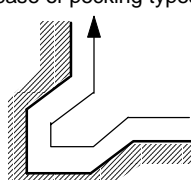
[7] F-CNR/\$

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:



- 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 Parameter List for further details.)
- Pecking can be done only if the following conditions are satisfied:
 - 1) During finishing.
 - 2) The selected shape and the next shape are linear and orthogonal.
 - 3) The cutting-edge angle and tip angle of the tool to be used satisfy the conditions listed in the table below.

 <p>A: Tool blade angle B: Tool tip angle</p> <p>T4P047</p>	<p>In the case of pecking types \$1, \$4:</p> 	<p>$A \geq 93^\circ$ $B \leq 57^\circ$ $A + B \leq 150^\circ$</p>
	<p>In the case of pecking types \$2, \$5:</p> 	<p>$A \geq 120^\circ$ $B \leq 57^\circ$ $A + B \leq 177^\circ$</p>
	<p>In the case of pecking types \$3, \$6:</p> 	<p>$A \geq 120^\circ$ $B \leq 30^\circ$ $A + B \leq 150^\circ$</p>

[8] RADIUS/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 “Function of automatically calculating a point of intersection” for details.
- Data setting is not required in any other cases ( mark will be displayed for this item).

[9] RGH

Set a finishing feedrate appropriate for particular surface roughness.

For setting a finishing feedrate, two methods are available: selection of a surface roughness code (for this case, the NC unit automatically calculates the appropriate feedrate for the selected surface roughness code), and direct setting of any desired feedrate.

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

ROUGHNES	FEEEDRATE							
	/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 menu key **ROUGHNES**. 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	
↓	↓	↓	↓	↓	↓	↓	↓	↓	Surface roughness (μm)
100	50	25	12.5	6.3	3.2	1.6	0.8	0.4	
(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 menu key **FEEDRATE/rev**, 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 [11] 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 [11] 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.

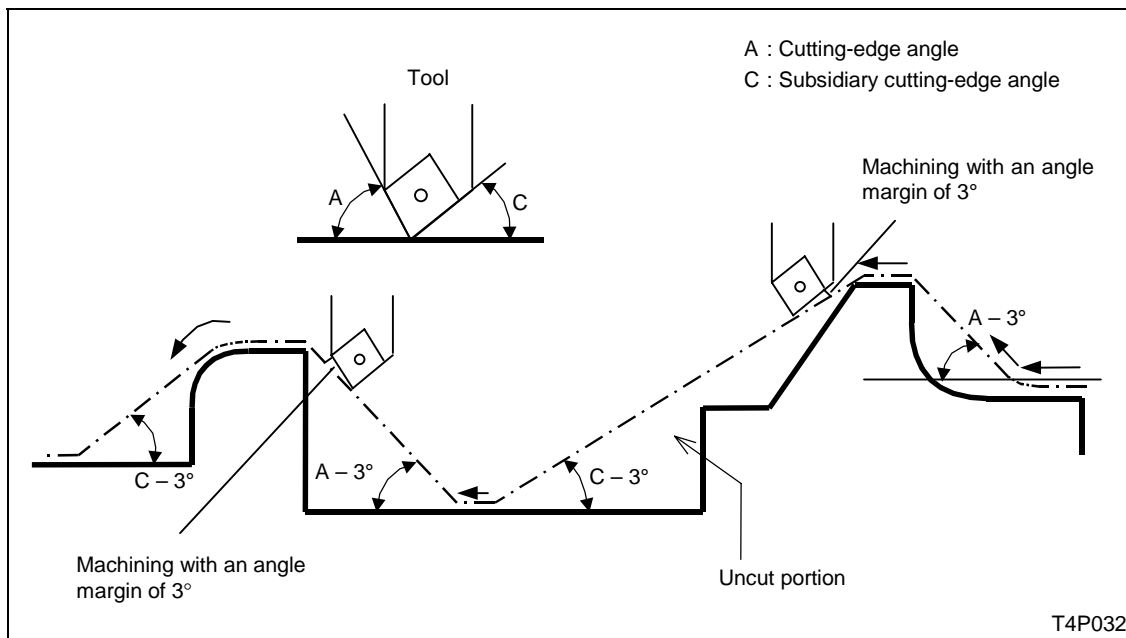
Note 3: If no data is entered in this item, the settings of feedrates in item [11] FR of each tool sequence will be used for each machining operation.

Note 4: The same value will automatically be set here if the preceding sequence has a set data of RGH.

<Precautions for BAR unit>

Some parts may remain uncut because of the tool shape.

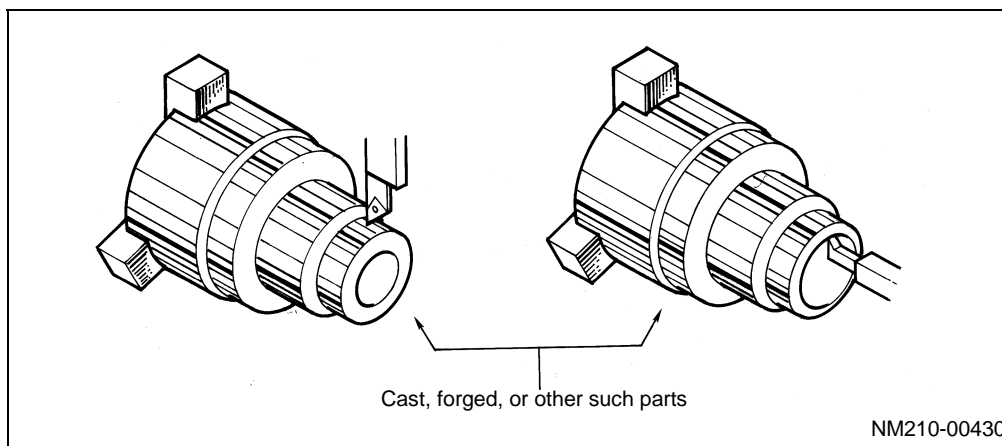
For a tool having a cutting-edge angle of A and a subsidiary cutting-edge angle of C, uncut portions occur at an angle of $A - 3^\circ$ in the machining direction and an angle of $C - 3^\circ$ in an opposite direction because machining will occur generally with an angle margin of 3° .



* The description given above also applies to CPY unit.

4-21-4 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 menu key **CPY**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	FIN-X	FIN-Z
*	CPY	[1]	[2]	[3]	[4]	[5]	[6]	[7]

[1] PART

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

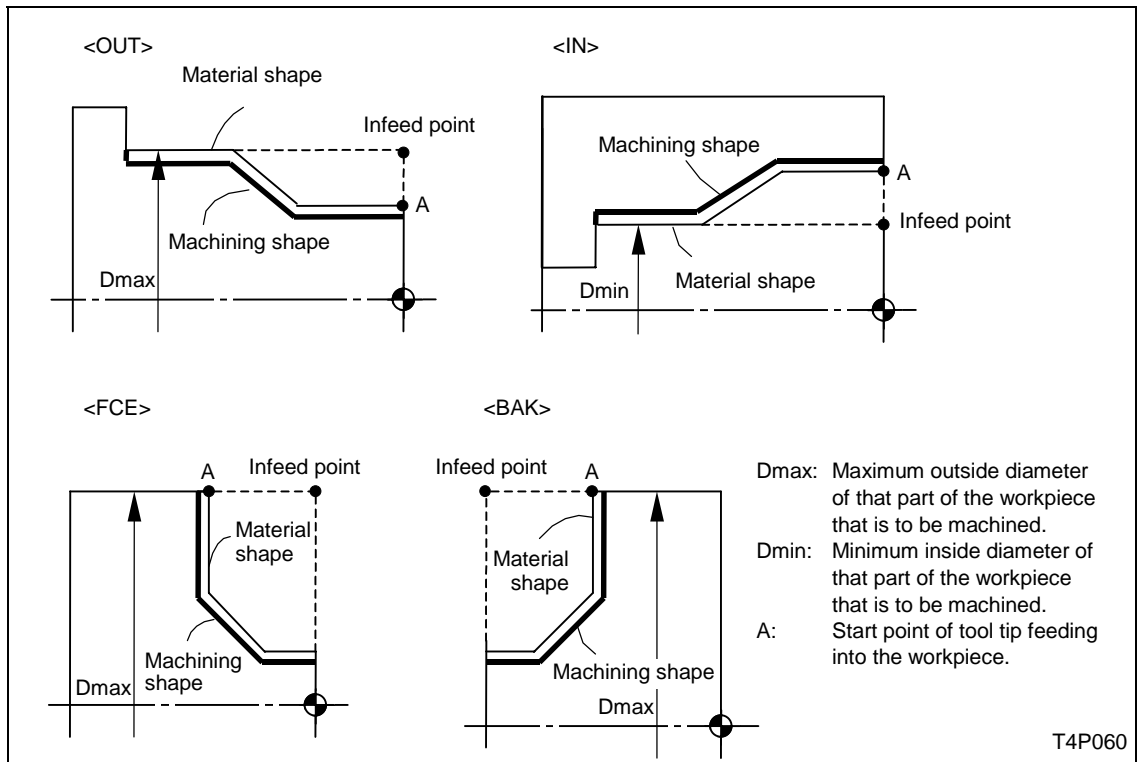
OUT	OUT	IN	IN	FCE	FCE	BAX	BAX
							

From the menu, select the section to be machined. The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[2] CPT-X, [3] CPT-Z

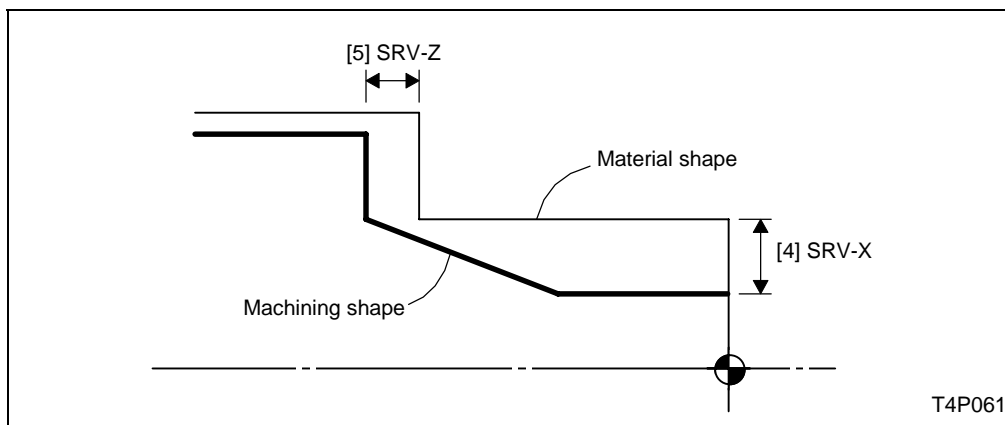
Set the X- and Z-coordinates of the desired infeed point.

In general, the infeed point means the start point of tool tip feeding into a workpiece. For copy-machining units, however, the points shown in the diagrams below are infeed points.



[4] SRV-X, [5] SRV-Z

Set the maximum removal allowances in the X-axis and Z-axis directions (removal allowances for the section that is to be cut most deeply).
The removal allowance in the X-axis direction must be set with the radius value (half the workpiece thickness).



[6] FIN-X, [7] FIN-Z

Set the finishing allowances for the X-axis and Z-axis directions (removal allowances during finishing). The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

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
R1				◆		◆	◆	◆	◆				
F2				◆	◆	◆	◆						
	↑	↑	↑ ↑	↑		↑		↑	↑	↑	↑	↑	↑
	[1]	[2]	[3] [4]	[5]		[6]		[7]	[8]	[9]	[10]	[11]	[11]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	SRV > FIN : One tool for roughing is automatically selected.
F2 (Finishing)	FIN > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			
---------	--------	--------	---------	-------	--	---------	--	--	--

[1] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				
--------------------------	-------------------------	-------------	--	-----------------------------	-----------------------------	--	--	--	--

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				
--	--	-------------	--	--	-----------------------------	--	--	--	--

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	
------	------	------	------	------	------	------	------	------	--

When creating a copy-machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
OUT		OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
IN		IN INNER DIAMETER, IN INNER (BAK)
FCE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
FCE		EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)
BAK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] 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.

[7] FIN-X, [8] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

[9] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for BAR unit.

[10] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for BAR unit.

[11] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for BAR unit.

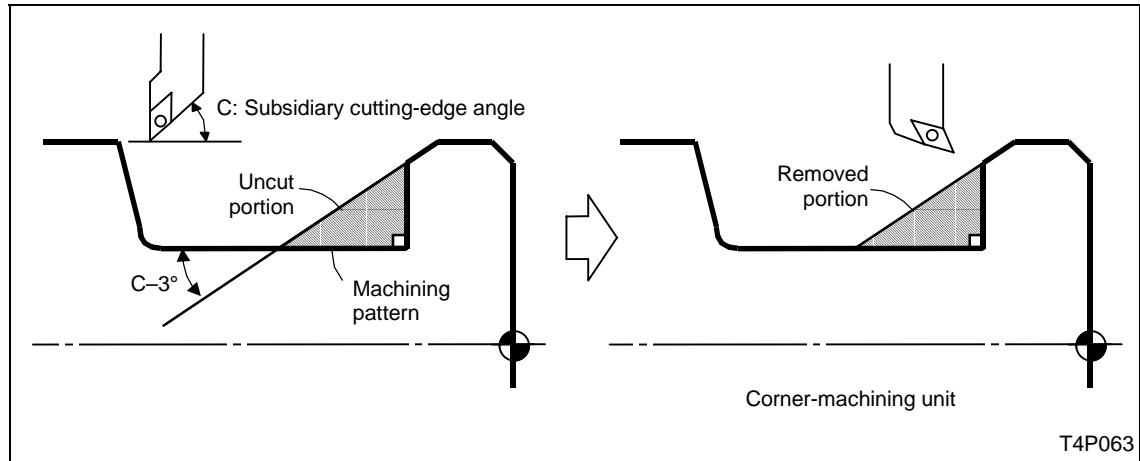
3. Setting shape sequence data

FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/\$	RADIUS/th	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]

The shape sequence data for the copy-machining unit is the same as that for the bar-materials machining unit. See the description of the relevant item in Section 4-21-3 "Bar-materials machining unit (BAR)."

4-21-5 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 menu key **CORNER**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	FIN-X	FIN-Z
*	CORNER	[1]	[2]	[3]

[1] PART

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

OUT	IN	FCE	BAK					
								

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
- FCE:** Uncut portion on front face
- BAK:** Uncut portion on back face

[2] FIN-X, [3] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

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
R1				◆		◆	◆	◆	◆				
F2				◆	◆	◆	◆						
	↑	↑	↑ ↑	↑	↑			↑	↑	↑	↑	↑	↑
	[1]	[2]	[3] [4]	[5]	[6]	[7]		[8]	[9]	[10]	[11]	[12]	[12]

Remark 1: ◆: Data are not necessary to be set here.






Remark 2: In the tool sequence, a maximum of up to two tools are automatically developed as follows.

Machining	Pattern
R1 (Roughing)	One tool for roughing is automatically selected.
F2 (Finishing)	FIN > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			
									

[1] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				
--------------------------	-------------------------	-------------	--	----------------------	----------------------	--	--	--	--

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				
--	--	-------------	--	--	----------------------	--	--	--	--

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	
------	------	------	------	------	------	------	------	------	--

When creating a copy-machining unit, usually select tools as follows according to the machining section that has been selected for the unit:

PART in the unit (section to be machined)	TOOL (Name)	TOOL (Section to be machined)
OUT	GENERAL	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FCE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----



The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] 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.

[7] DEP-1 (Maximum cutting depth)

Specify the maximum cutting depth per roughing pass. The maximum cutting depth in the X-axial direction is to be specified in terms of radius.
See the description of the relevant item for BAR unit.

[8] FIN-X, [9] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

[10] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for BAR unit.

[11] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.
See the description of the relevant item for BAR unit.

[12] M

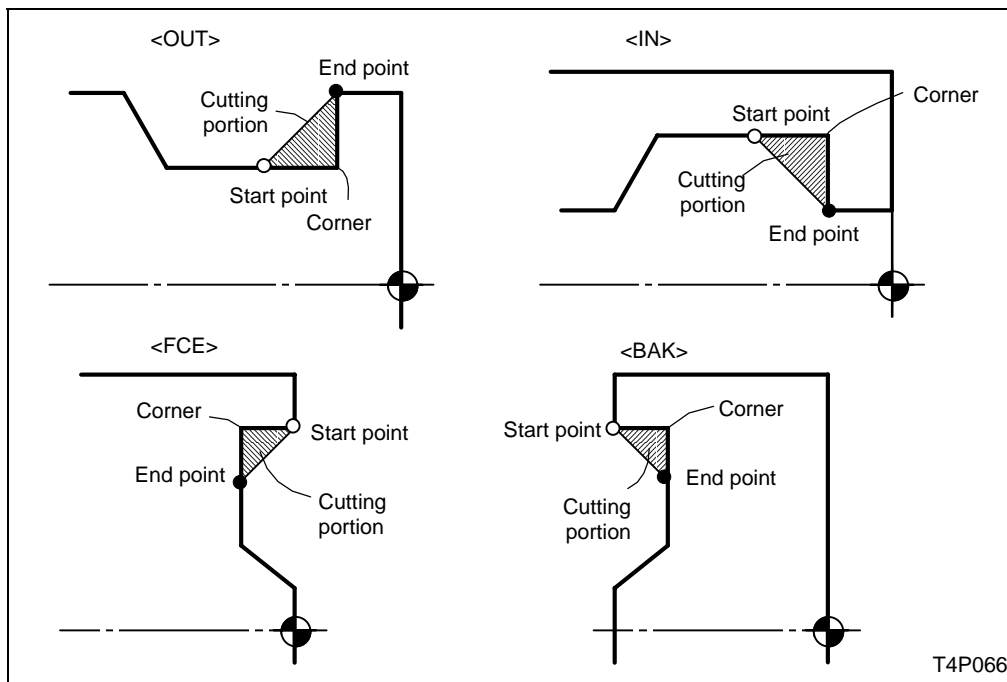
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for BAR unit.

3. Setting shape sequence data

FIG	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/\$	RGH
1	[1]	[2]	[3]	[4]	[5]	[6]

[1] SPT-X, [2] SPT-Z, [3] FPT-X, [4] FPT-Z

Set the coordinates of the desired start and end points of cornering.
The position of the start point and the end point are shown below.



[5] F-CNR/\$

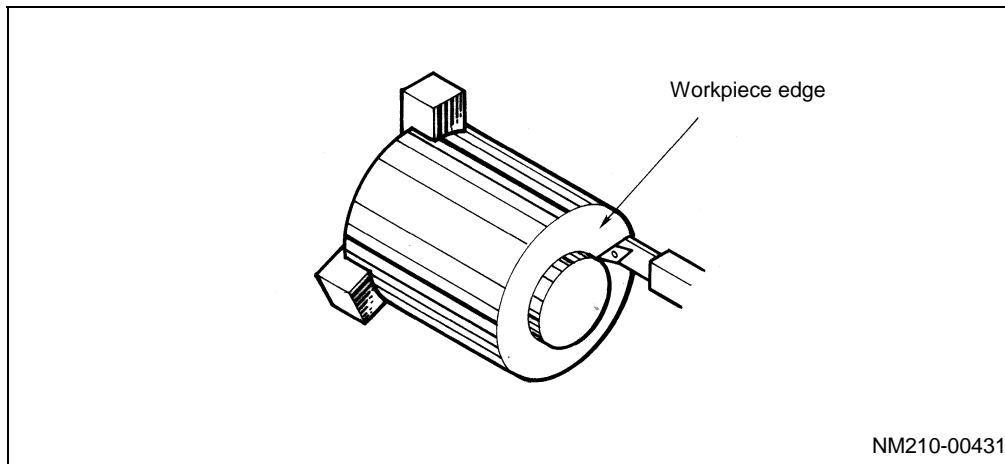
This item applies to the corners shown in the figure below, not the ending-point section. Enter data in this item to perform C-chamfering, R-chamfering, or polish-necking operations. For data setting method, see the relevant items for BAR unit.

[6] RGH

Set the appropriate, finish-machining feedrate for particular finishing surface roughness. For data setting method, see the relevant items for BAR unit.

4-21-6 Edge-machining unit (FACING)

Select the edge-machining unit (FACING) when chipping off any protrusions of the workpiece edges (front face or back face).





Press the menu key **FACING**  to select this unit.

1. Setting unit data

UNO.	UNIT	PART	FIN-Z
*	FACING	[1]	[2]

[1] PART

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

FCE	BAK								
									

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FCE: Right edge of the workpiece

BAK: Left edge of the workpiece

Note: The menu item **BAK** may not be selectable for special machine specifications.

[2] FIN-Z

Set the finishing allowances for the Z-axis directions (removal allowances during finishing).

See the description of the relevant item for BAR unit.

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
R1				◆		◆	◆	◆	◆				
F2				◆	◆	◆	◆	◆					
	↑	↑	↑ ↑	↑	↑				↑	↑	↑	↑	↑
	[1]	[2]	[3] [4]	[5]	[6]				[7]	[8]	[9]	[10]	[10]

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)	FIN > 0 : One tool for finishing is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			
									

[1] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				
--------------------------	-------------------------	-------------	--	-----------------------------	-----------------------------	--	--	--	--

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				
--	--	-------------	--	--	-----------------------------	--	--	--	--

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	
------	------	------	------	------	------	------	------	------	--

When creating a edge-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)
FCE	GENERAL	OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] 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.

[7] FIN-Z

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

[8] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for BAR unit.

[9] 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.

[10] 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.

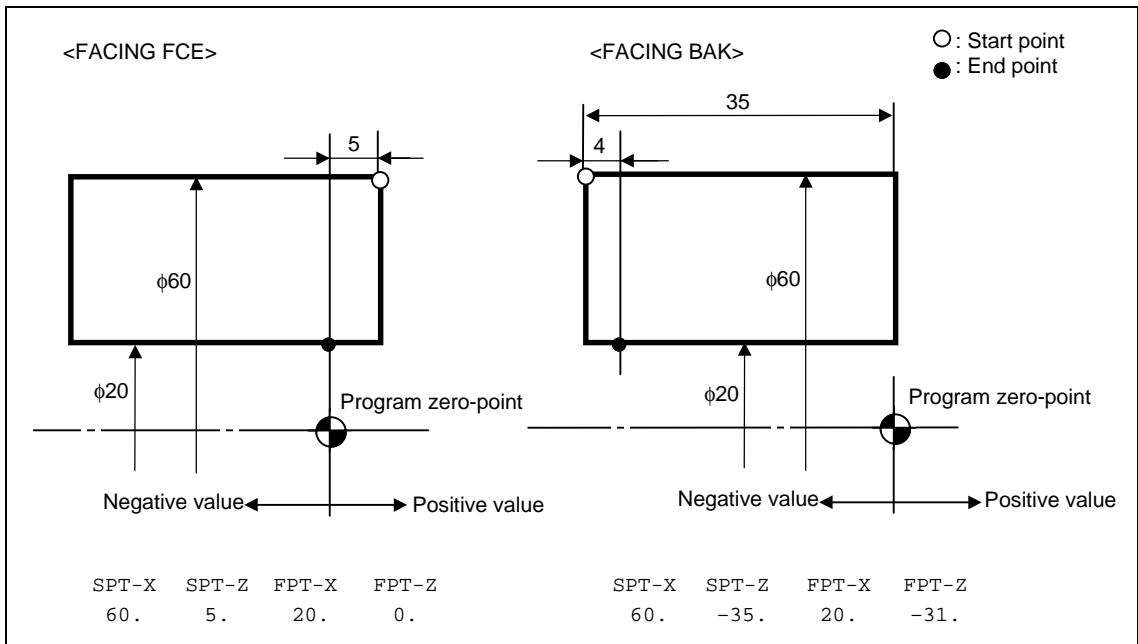
3. Setting shape sequence data

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 edge-machining 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 zero-point.

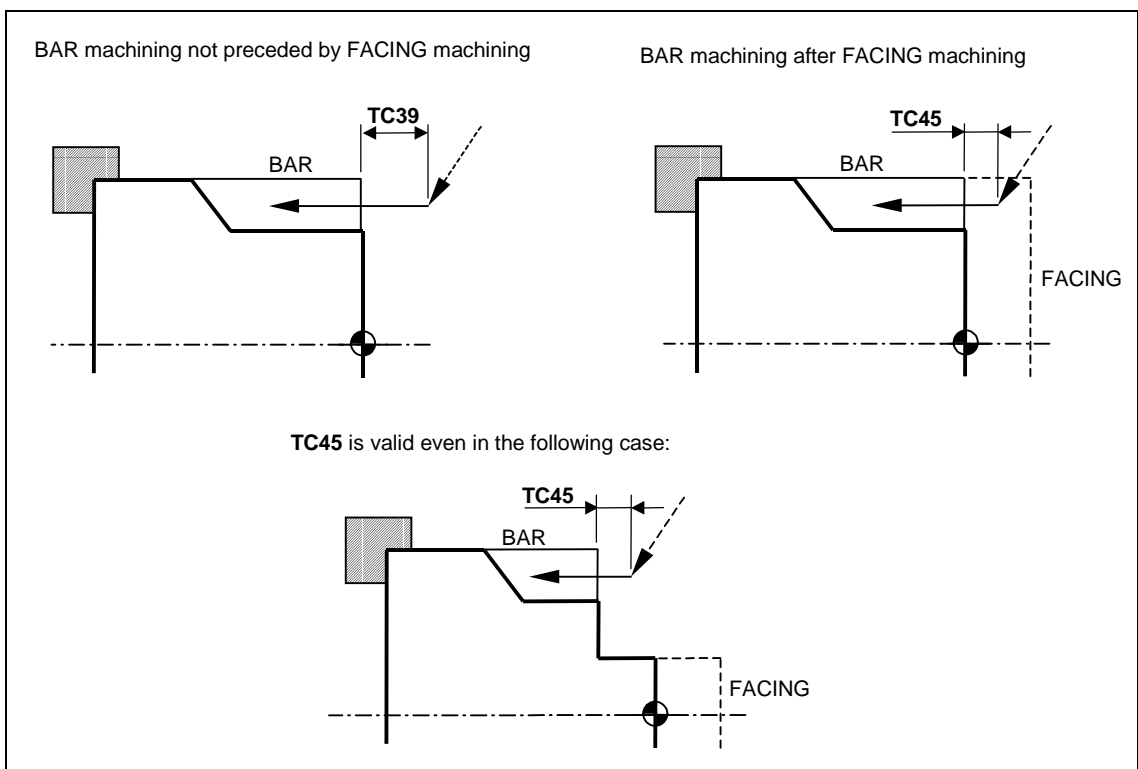


[5] RGH

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.

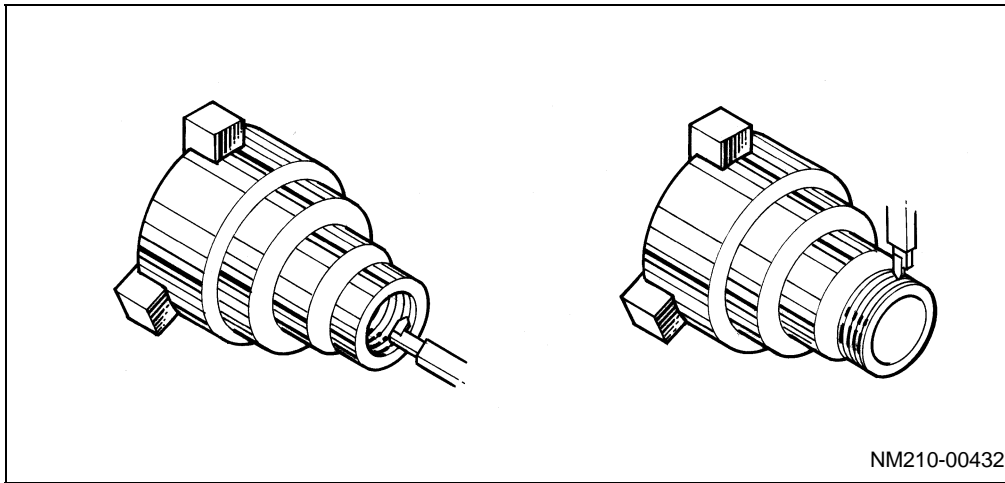
See related items of “Bar-materials machining unit (BAR).”

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 an FACING unit, in particular, the value of parameter **TC45** is used as the end-face clearance.



4-21-7 Threading unit (THREAD)

Select the threading unit to thread the outer peripheries, inner peripheries or front faces or back faces of a workpiece.




Press the menu key **THREAD**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	CHAMF	LEAD	ANG	MULTI	HGT
*	THREAD	[1]	[2]	[3]	[4]	[5]	[6]

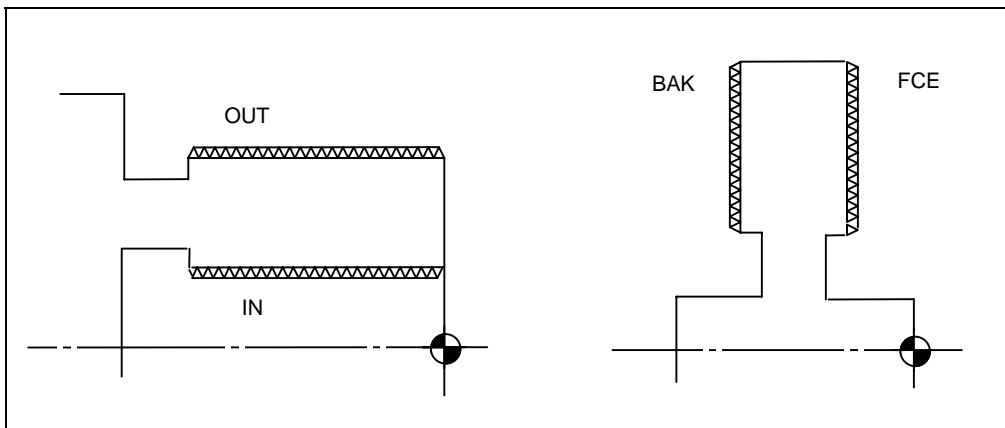
[1] PART

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

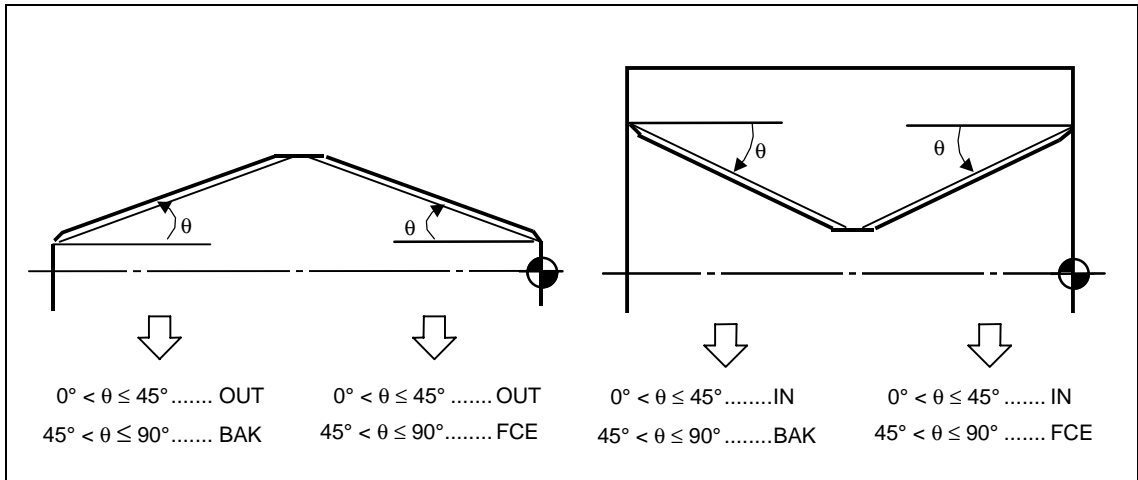
OUT	IN	FCE	BAK						
									

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)
- FCE** : Right edge of the workpiece (front face)
- BAK** : Left edge of the workpiece (back face)



- For taper threading, select the appropriate machining section as follows according to the desired taper angle:

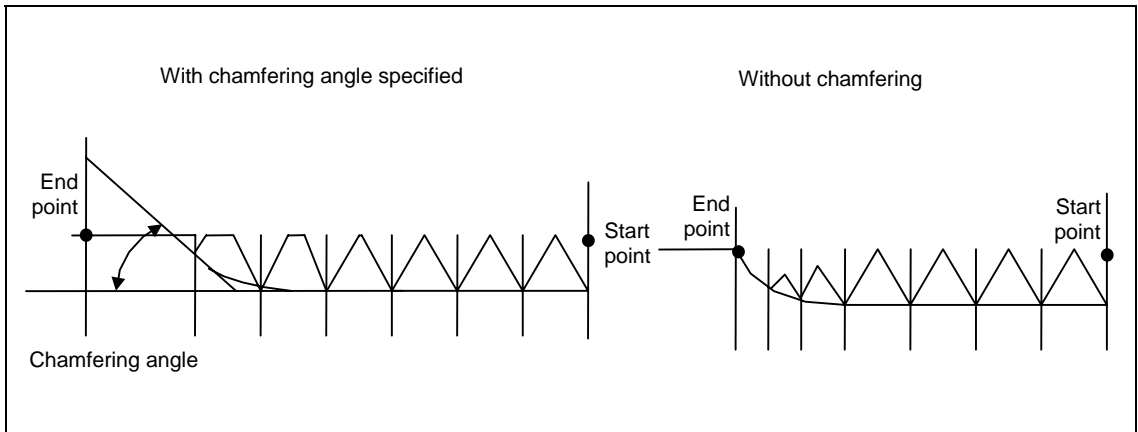


[2] CHAMF

Set a chamfering angle for the section you want to thread.

- Set 0 if chamfering is not required.
- Set 1 for a chamfering angle of 45 degrees.
- Set 2 for a chamfering angle of 60 degrees.

Designate chamfering to maintain the lead up to the ending point of threading.



Use the parameter **TC82** to specify the chamfering amount.

[3] LEAD

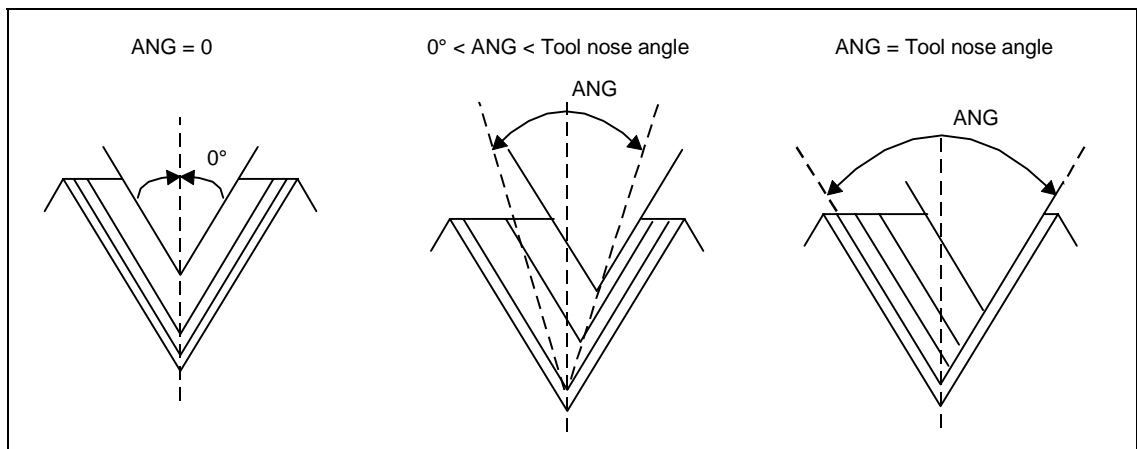
Set the threading lead given by the following expression:

$$(\text{Lead}) = (\text{Pitch}) \times (\text{Number of threads})$$

[4] ANG

Set a threading angle.

- Usually, set a several degrees smaller value than the nose angle of the tool.



[5] MULTI

Set the desired number of threads.

[6] HGT

Set the threading height.

Pressing the menu key **AUTO SET** with the cursor at item [7] will automatically set data into items [7].

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
1	↑	↑ ↑	↑	↑	↑	↑	◆	◆	◆	↑	◆	↑	↑
	[1]	[2]	[3] [4]	[5]	[6]	[7]	[8]			[9]		[10]	[10]

Remark 1: ◆: Data are not necessary to be set here.

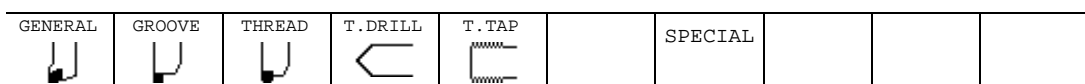
Remark 2: In the tool sequence, one tool is automatically developed as follows.

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:



[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 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	THREAD	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FCE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.
See the description of the relevant item for BAR unit.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] PAT. (Machining pattern)

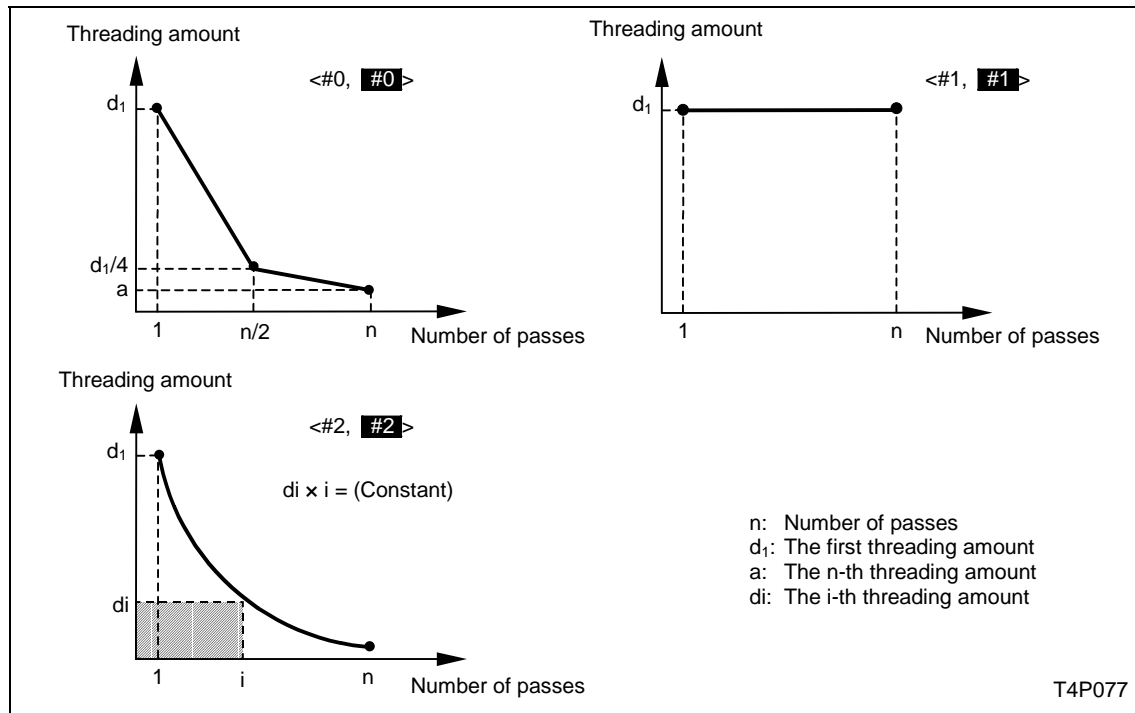
The following menu will be displayed when the cursor is placed at this item:

#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA	#0 STANDARD	#1 CONST. DEPTH	#2 CONST. AREA				
(a)	(b)								

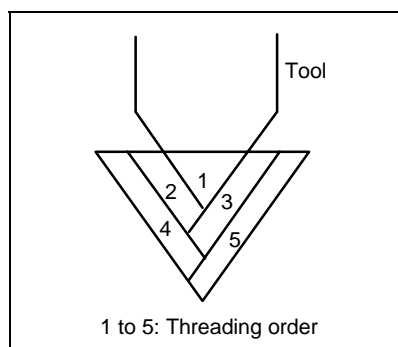
From the menu, select the threading pattern to be machined.

Patterns to be machined that correspond to the data of the displayed menu are as follows.

- #0, **#0**: Standard patterns
The threading amount gradually decreases as the number of passes increases.
- #1, **#1**: Constant-threading pattern
The threading amount always stays constant, irrespective of the number of passes.
- #2, **#2**: Area-constant pattern
The threading amount is inversely proportional to the number of passes.



Note: If you select **#0**, **#1** or **#2** zigzag threading (alternate threading with the left and right cutting edges) will occur unless you set a value of 30 or less in item [4], ANG in the unit data.



[7] DEP-1 (First cutting depth)

Enter the first cutting depth during the threading pass. For X-axial cutting, enter this value in terms of radius. The above value can likewise be auto-set by pressing the menu key **AUTO SET**.

[8] DEP-2 (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.

[9] C-SP

Specify the peripheral speed for the tool in terms of turning spindle.
See the description of the relevant item for BAR unit.

[10] M

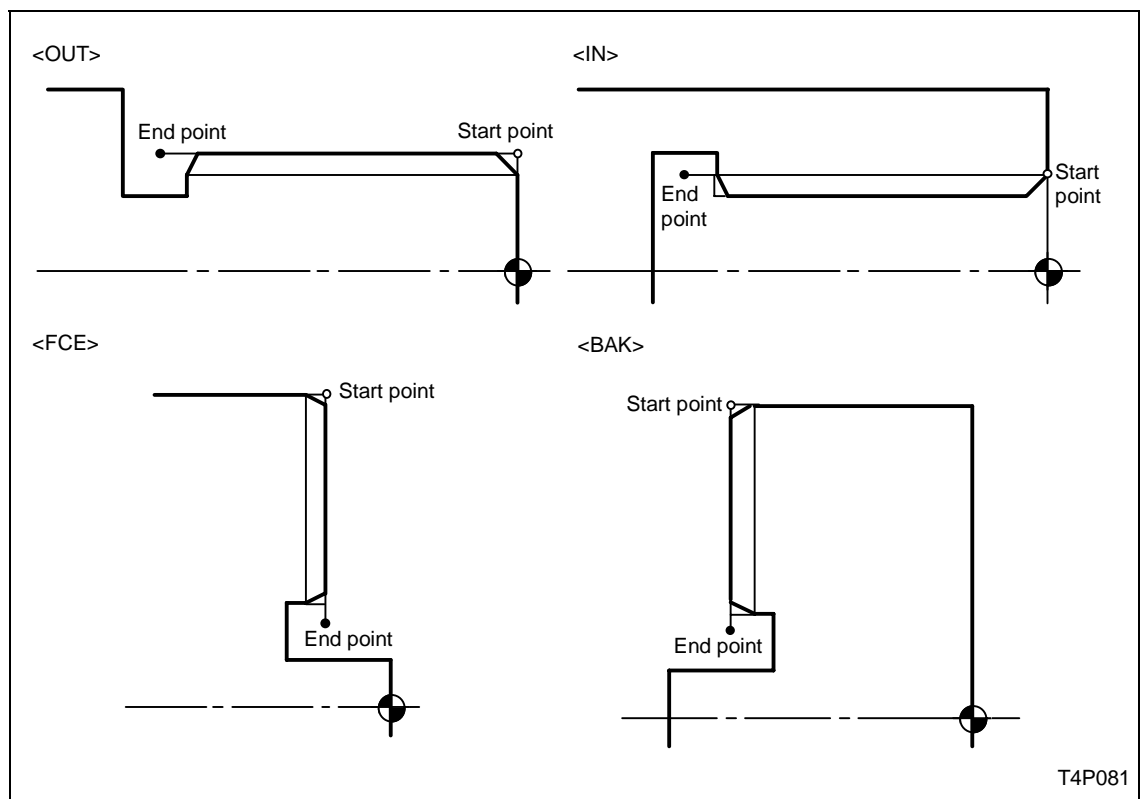
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for BAR unit.

3. 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.

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 **◆**).

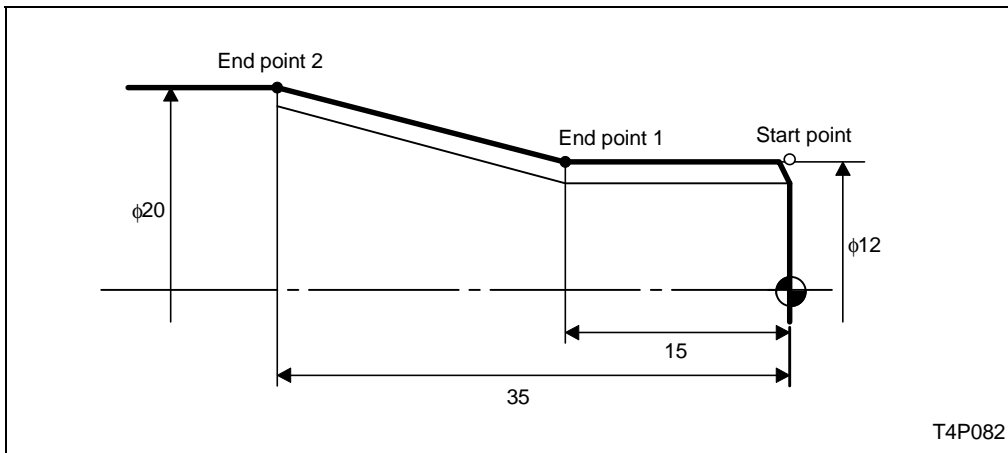
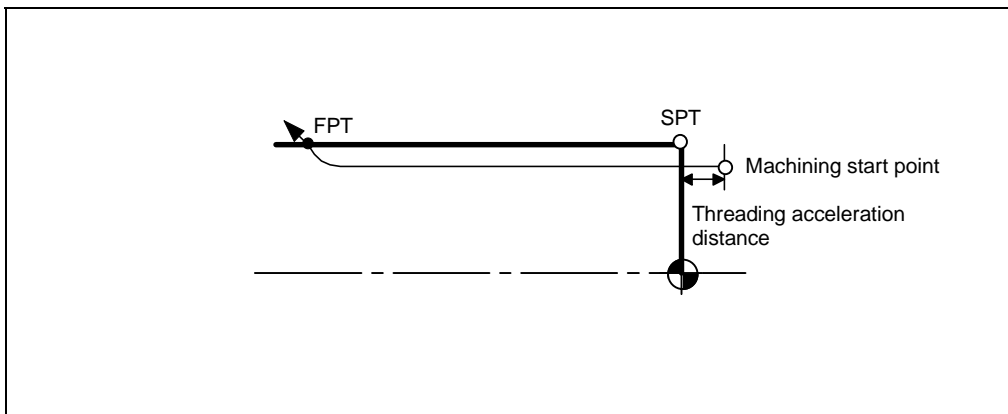


FIG	SPT-X	SPT-Z	FPT-X	FPT-Z
1	12.	0.	12.	15.
2	◆	◆	20.	35.

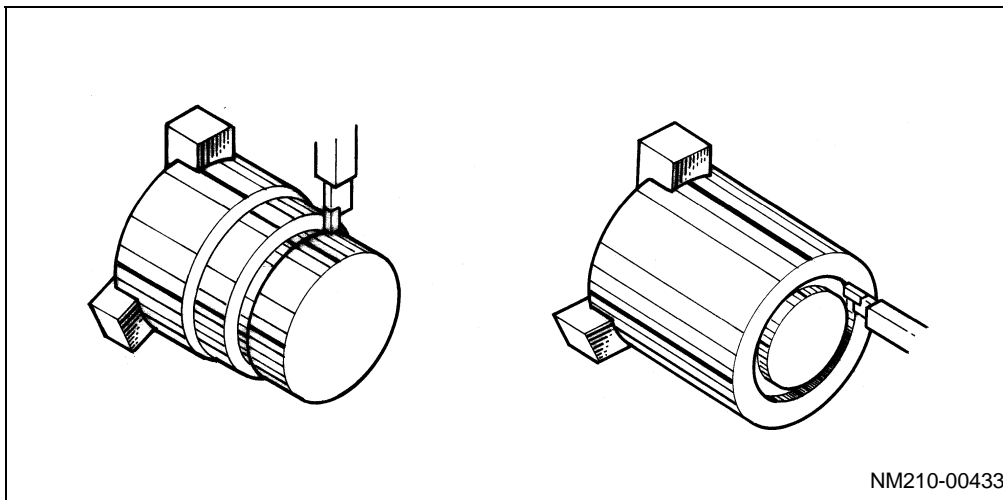
Note 2: Threading action begins at a position that is away from the start point specified in the program by the acceleration distance. Before carrying out a threading operation, therefore, check for possible interference with the tailstock or the workpiece during threading.



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.

4-21-8 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.







Press the menu key **T. GROOVE**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	PAT.	No.	PITCH	WIDTH	FINISH
*	T.GROOVE	[1]	[2]	[3]	[4]	[5]	[6]

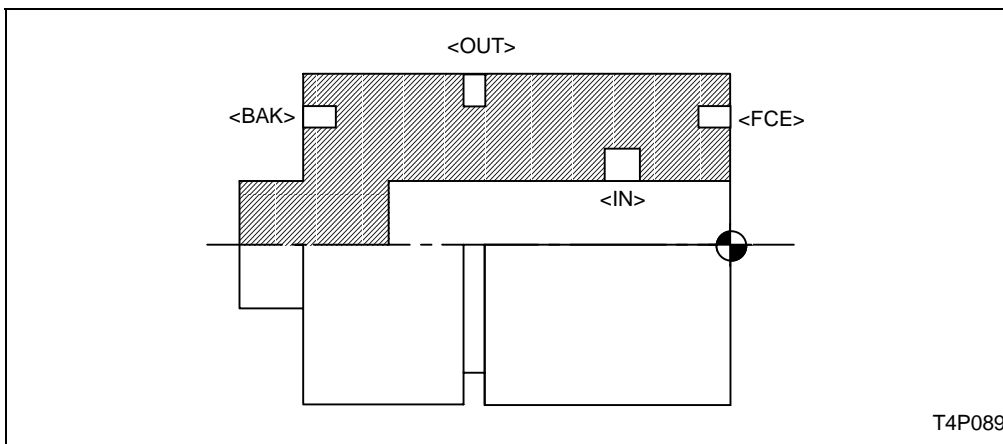
[1] PART

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

OUT	IN	FCE	BAX						
									

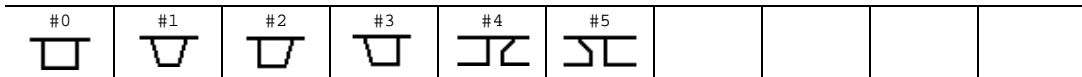
From the menu, select the section to be machined. Sections to be machined that correspond to the data of the displayed menu are as follows.

- OUT** : Outer periphery
- IN** : Inner periphery
- FCE** : Right edge of the workpiece (front face)
- BAK** : Left edge of the workpiece (back face)

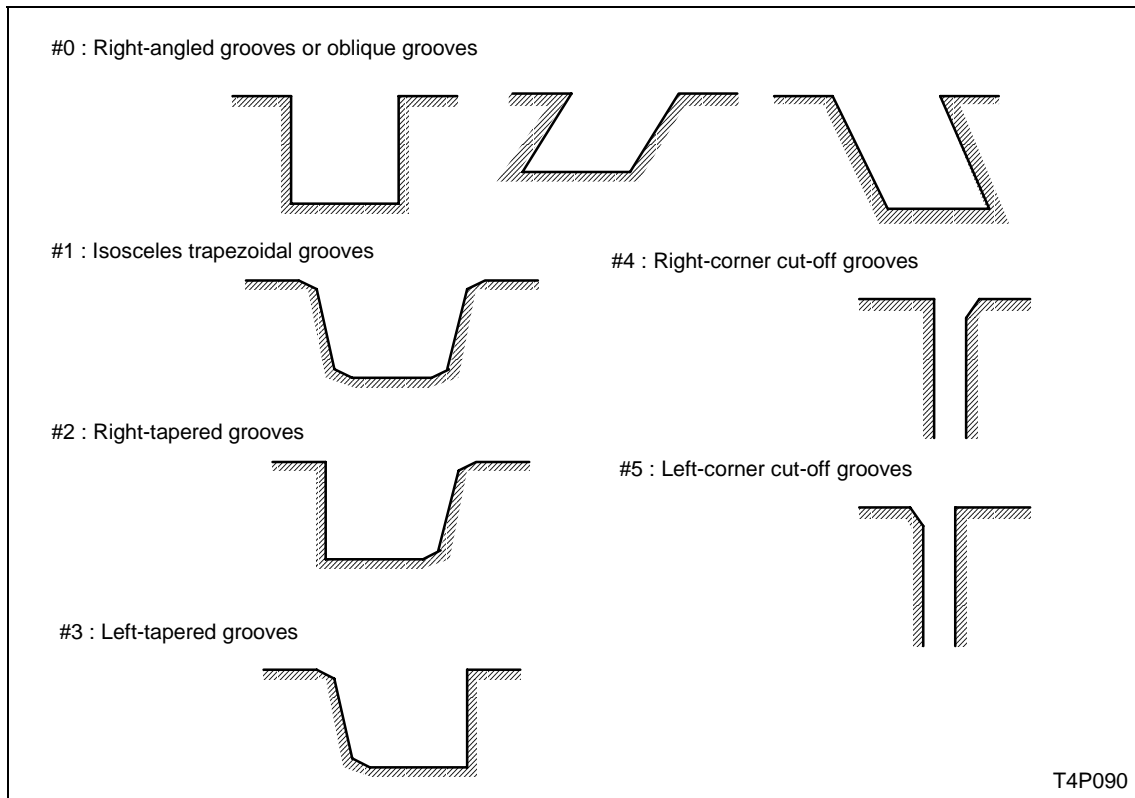


[2] PAT.

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



Select a grooving pattern from the menu. The data of the displayed menu denote the following grooving patterns:



Note 1: Grooving patterns #4 and #5 (both, cutting-off) are available only when outside diameter (OUT) is selected for item [1] PART.

Note 2: For grooving patterns #4 and #5, the feed reduction count can be changed using parameter **TC50**.

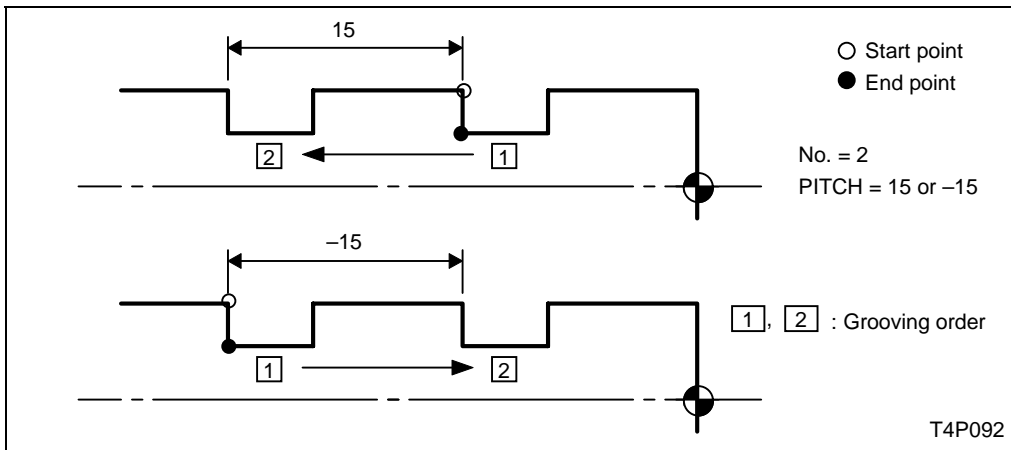
[3] No.

Set the number of grooves when multiple grooves of the same shape are to be machined at fixed spacings.

[4] PITCH

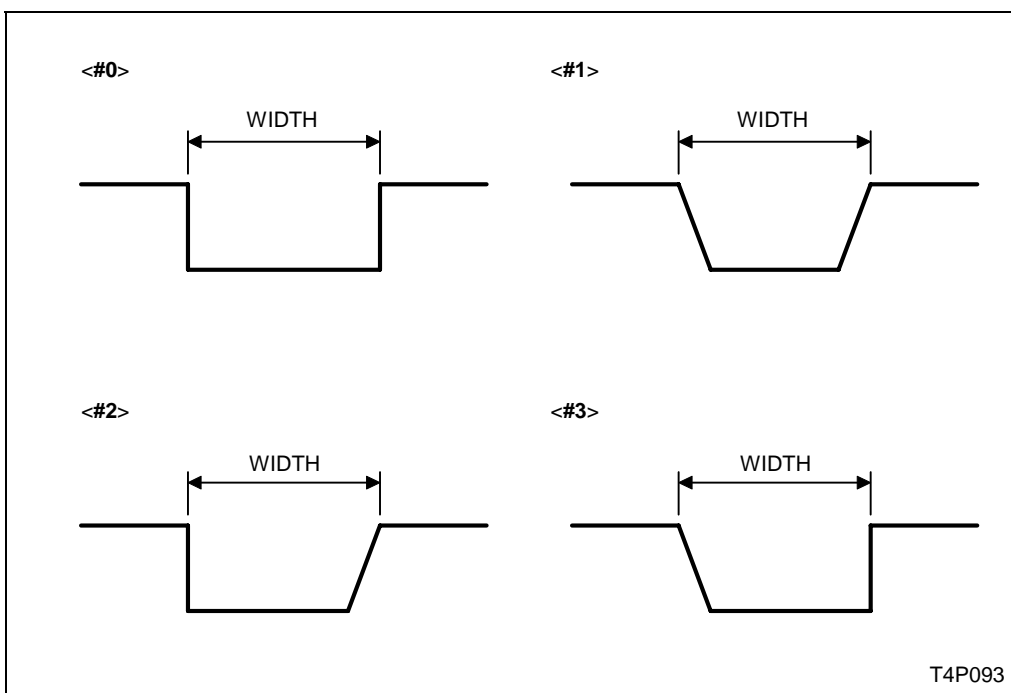
Set a pitch when multiple grooves of the same shape are to be made at fixed spacings.

The pitch can be set as either a plus value or a minus value. Setting the pitch as a plus value causes sequential grooving in a forward direction. Setting the pitch with a minus sign causes sequential grooving in a reverse direction.



[5] WIDTH

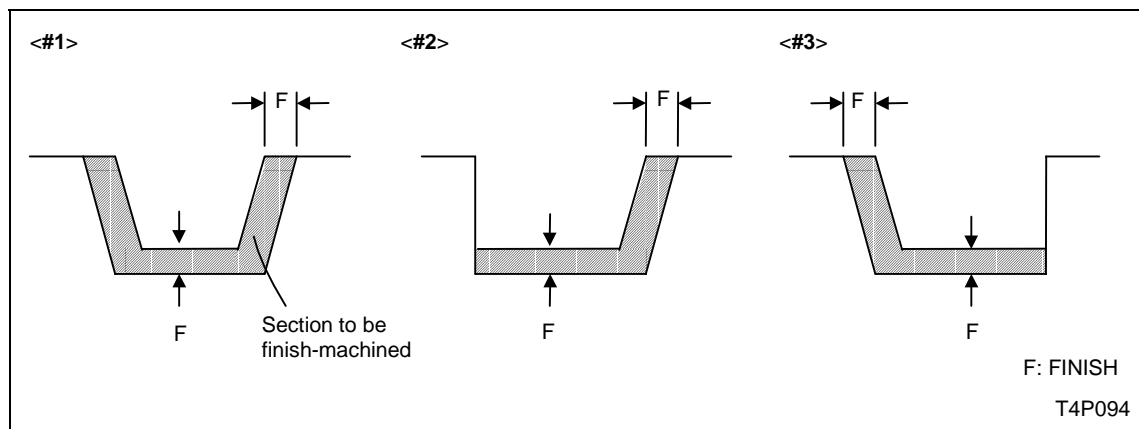
Set a grooving width.



- If you have selected grooving pattern #4 or #5, a cutting-off tool tip width is considered as a grooving width.

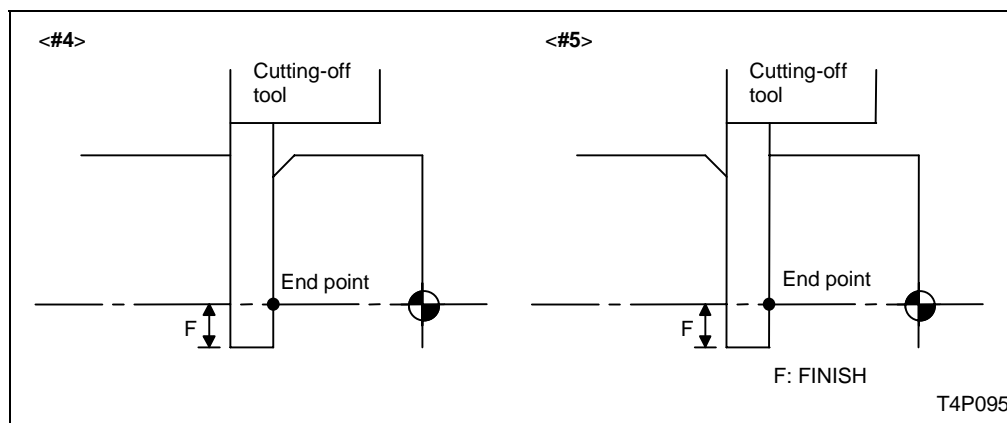
[6] FINISH

- No data can be set if you have selected grooving pattern #0.
- Set a finish-machining removal allowance if you have selected grooving pattern #1, #2 or #3.



Note: No finishing allowance will be provided to right-angled walls if you have selected grooving pattern #2 or #3.

- Set a cutting-off tool overshoot if you have selected grooving pattern #4 or #5.



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
R1				◆		◆	◆	◆	◆				
F2				◆		◆	◆		◆				
	↑	↑	↑ ↑	↑	↑			↑		↑	↑	↑	↑
	[1]	[2]	[3] [4]	[5]	[6]			[7]		[8]	[9]	[10]	[10]

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 set here.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL 	GROOVE 	THREAD 	T.DRILL 	T.TAP 		SPECIAL			
--	---	---	--	--	--	---------	--	--	--

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				
--------------------------	-------------------------	-------------	--	-----------------------------	-----------------------------	--	--	--	--

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				
--	--	-------------	--	--	-----------------------------	--	--	--	--

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	
------	------	------	------	------	------	------	------	------	--

When creating a 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	GROOVE	OUT OUTER DIAMETER
IN		IN INNER DIAMETER, IN INNER (BAK)
FCE		OUT OUTER DIAMETER, EDG EDGE, EDG EDGE (BAK)
BAK		OUT OUTER DIAMETER, EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal size)

Enter the nominal size of tools using the numeric data keys.

See the description of the relevant item for BAR unit.

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] DEP-1(Max. depth of cut), **[8] C-SP**, **[9] FR**

- For each grooving pattern, set data into these items as follows:

Pattern	Sequence	DEP-1 (Max. depth of cut)	C-SP	FV
#0	F (Finishing)	Max. depth of cut per pass (Designate in radius for OUT or IN)	Peripheral speed during grooving	Feedrate during grooving (Shape seq. data RGH ineffective)
#1, #2 or #3	R (Roughing)		Peripheral speed during roughing	Feedrate during roughing
	F (Finishing)	—	Peripheral speed during finishing	Feedrate during finishing
#4 or #5 (Parameter TC50 = 0, 1)	F (Finishing)	Max. depth of cut per pass (Designate in radius; without pecking if 0 is set)	Peripheral speed during grooving (limited by the rotational speed specified by the parameter TC49)	Feedrate during grooving (Shape seq. data RGH effective for cutting-off area specified by the parameter TC9)
#4 or #5 (Parameter TC50 ≥ 0, 1)	F (Finishing)		Number of revolutions during grooving (*)	Starting feedrate for grooving (**)

* 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.

[7] FIN-X

Specify the allowance to be left for the next finishing tool sequence.
See the description of the relevant items for BAR unit.

[10] 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.

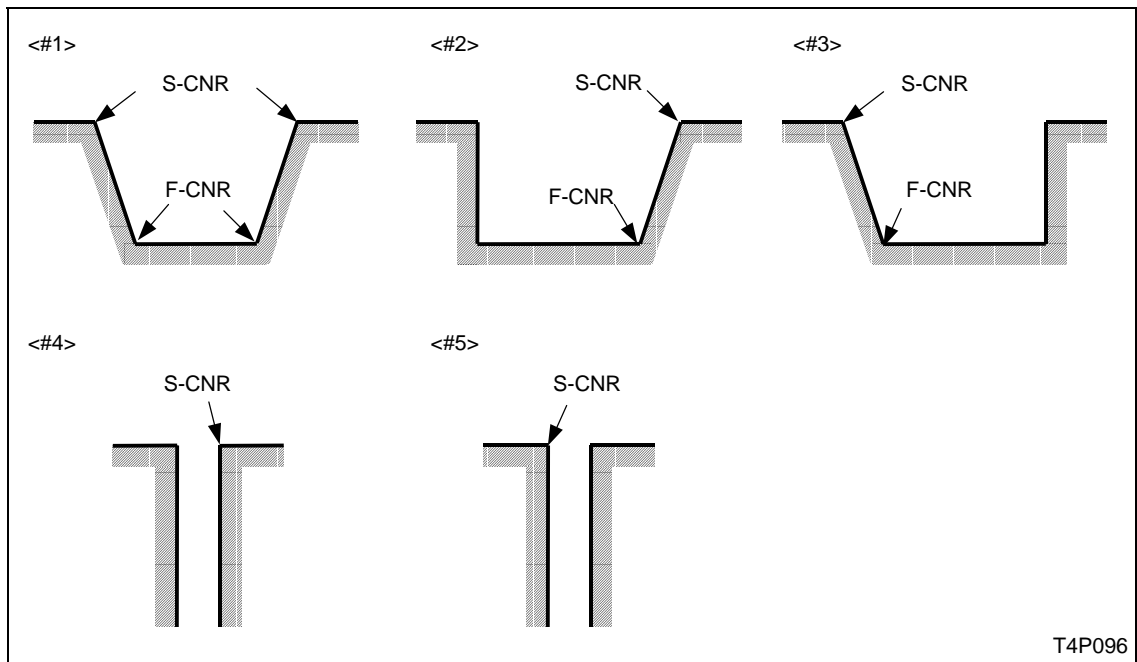
3. Setting shape sequence data

FIG	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR	ANGLE	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 menu key **CORNER R**.
- If you have selected grooving pattern #0, the data specified here will become invalid.

- If you have selected a grooving pattern other than #0, set data in this item when C-chamfering or R-chamfering (rounding) is to be done on the corners shown in the diagrams below.

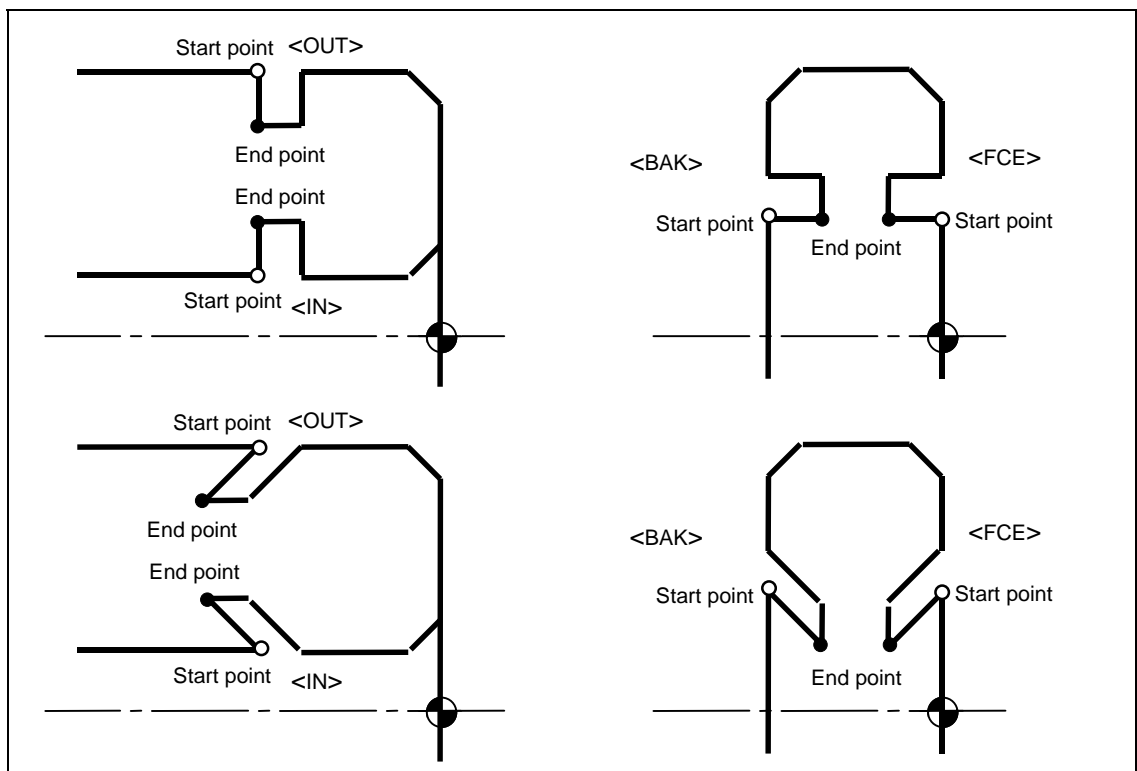


[2] SPT-X, [3] SPT-Z, [4] FPT-X, [5] FPT-Z

Set the coordinates of the start point and end point of grooving.

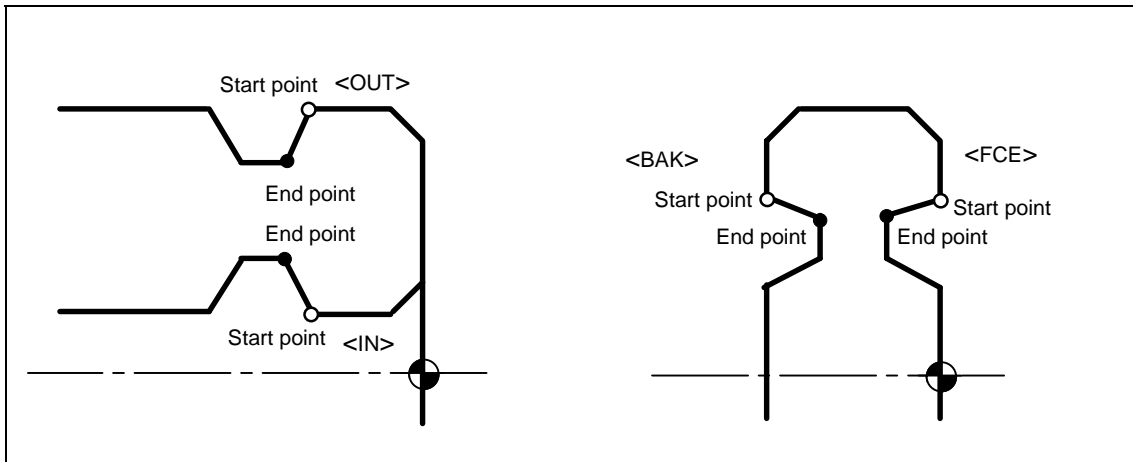
Set the position of the start point and the end point as follows according to the selected grooving pattern.

- For grooving pattern #0:

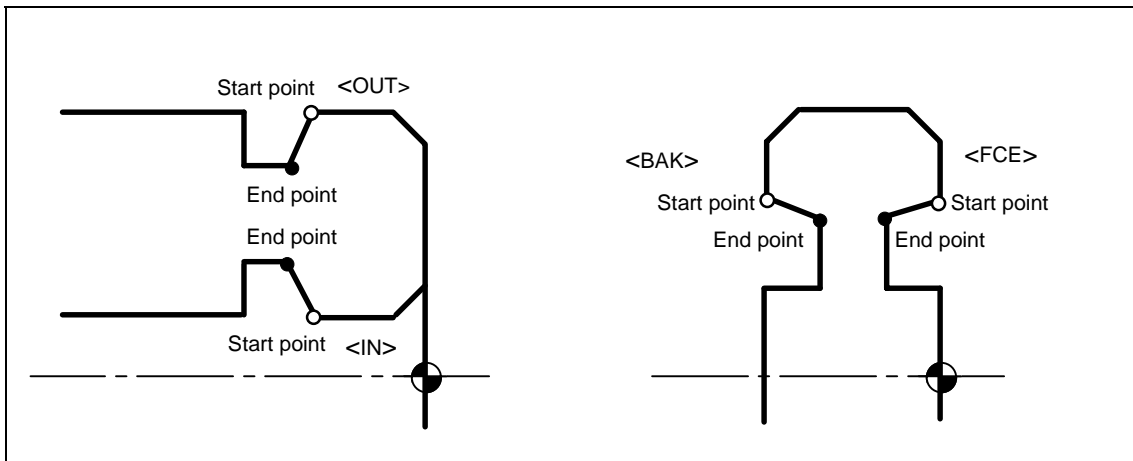


For pattern #0, oblique grooves can be machined by setting the start and end points as shown above.

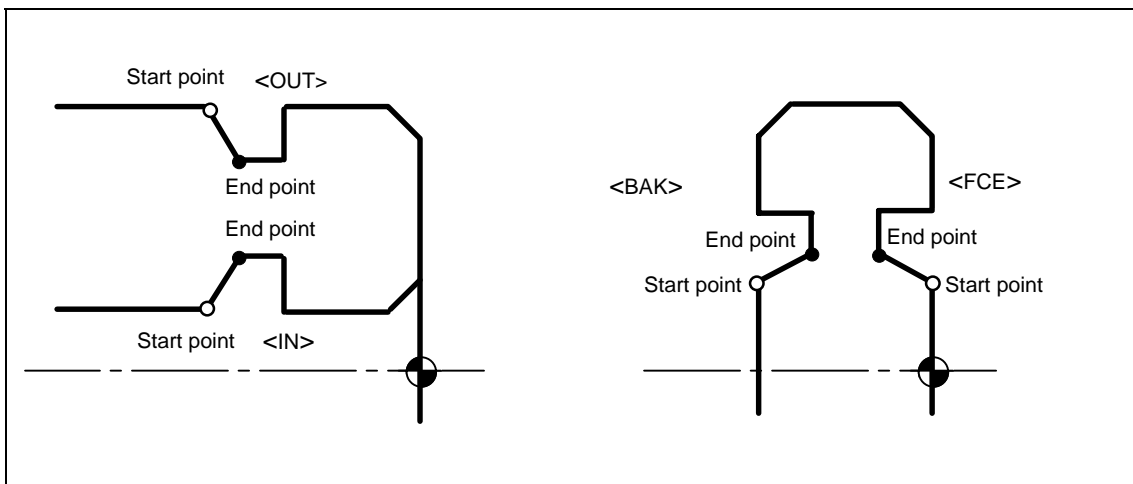
- For grooving pattern #1



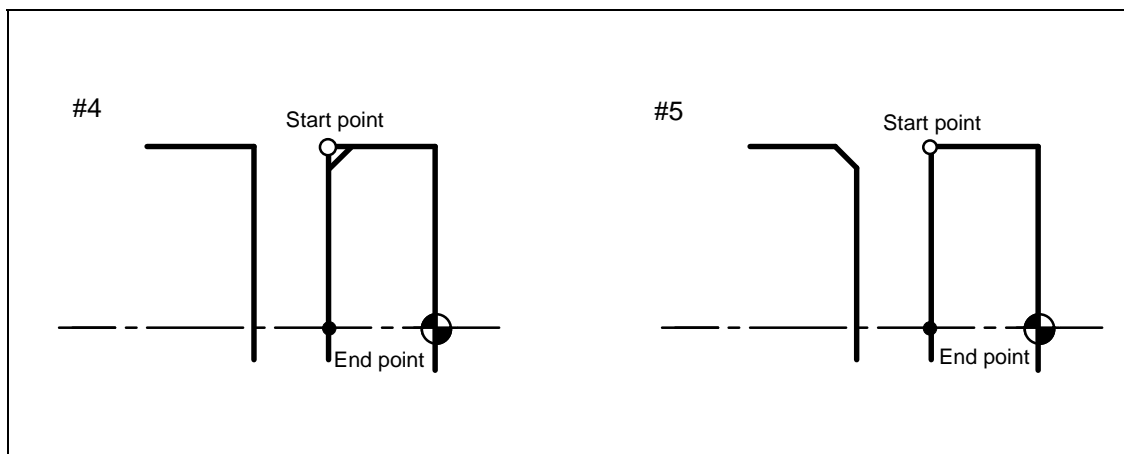
- For grooving pattern #2:



- For grooving pattern #3:



- For grooving pattern #4 or #5:



Note 1: For grooving pattern #0, #1, #2 or #3, the machining drawing may have an indicated taper angle but not have a clearly indicated position for the start point or the end point. In such cases, temporarily type the question mark “?” in all unclear items by pressing the menu key ?. You will be able to set data automatically at a later time using the automatic calculation function of the crossing-point. See “Automatic Crossing-Point Calculation Function” for further details.

Note 2: If multiple grooves of the same shape are to be machined (according to the setting of unit data “No.”), set the coordinates of the start point and end point of grooving of the first groove.

Note 3: If grooving pattern #4 or #5 is selected, no data setting is required for “FPT-Z.”

[6] 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] ANGLE

Set a taper angle if you have typed the question mark “?” in one of the four items from [2] to [5] above. See “Function of automatically calculating a point of intersection” for details of setting a taper angle.

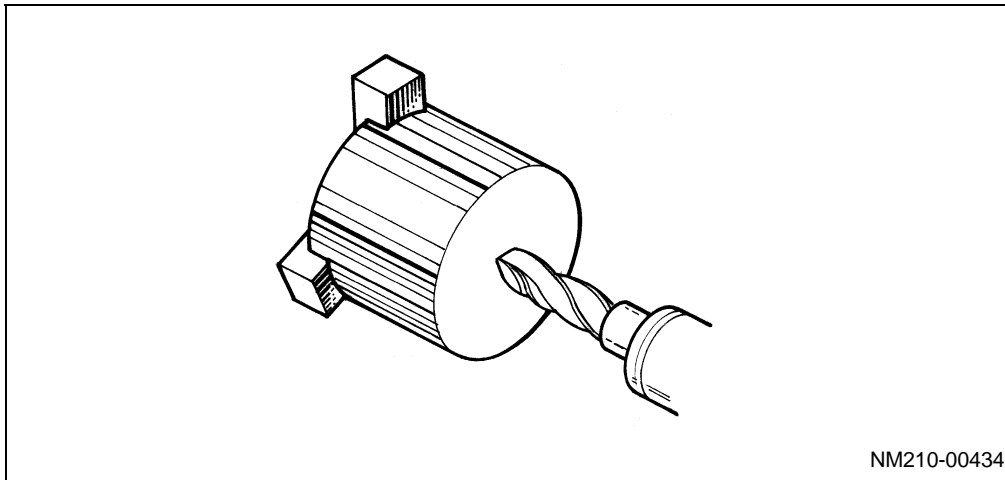
[8] RGH

For each grooving pattern, set data into this item as follows:

Pattern	Description of RGH data
#0	Invalid. (Set the feedrate in item FEED of sequence data.)
#1, #2, #3	Set the feedrate during finishing. (Finishing will also be executed at the 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 FEED rate of tool sequence data if no data is designated here.)

4-21-9 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.





Press the menu key **T. DRILL**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	DIA
*	T.DRILL	[1]	[2]

[1] PART

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

FCE	BAK						
							

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FCE: Right edge of the workpiece

BAK: Left edge of the workpiece

Note: The **BAK** menu item may not be selectable for special machine specifications.

[2] DIA

Set the diameter of the hole to be drilled (nominal diameter of the turning drill).

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
1	↑	↑ ↑	↑	↑	↑	↑	◆	◆	↑	↑	↑	↑
	[1]	[2]	[3][4]	[5]	[6]	[7]	[8]	[9]	[10]	[11]	[12]	[12]

Remark 1: ◆: Data are not necessary to be set here.

Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP	SPECIAL

[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 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)
FCE	T.DRILL	EDG EDGE
BAK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal diameter)

Set the diameter of the hole to be drilled (nominal diameter of the turning drill).

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] PAT. (Machining pattern)

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

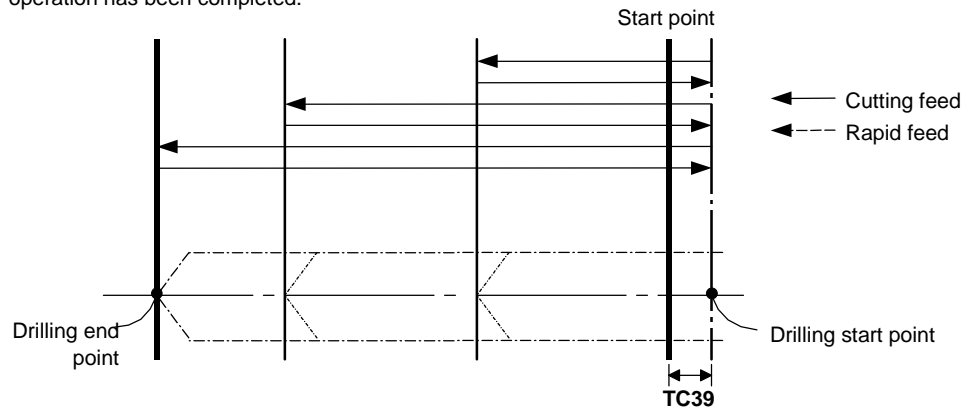
#0	#1	#2	#3	#4					>>>
DRILLING THROUGH	PECKING BOTOMED	PECKING BOTTOMED	REAMER BOTTOMED	PECKING BOTTOMED					

#0	#1	#2	#3	#4					>>>
DRILLING THROUGH	PECKING THROUGH	PECKING THROUGH	REAMER THROUGH	PECKING THROUGH					

From the menu, select the turning drilling pattern.

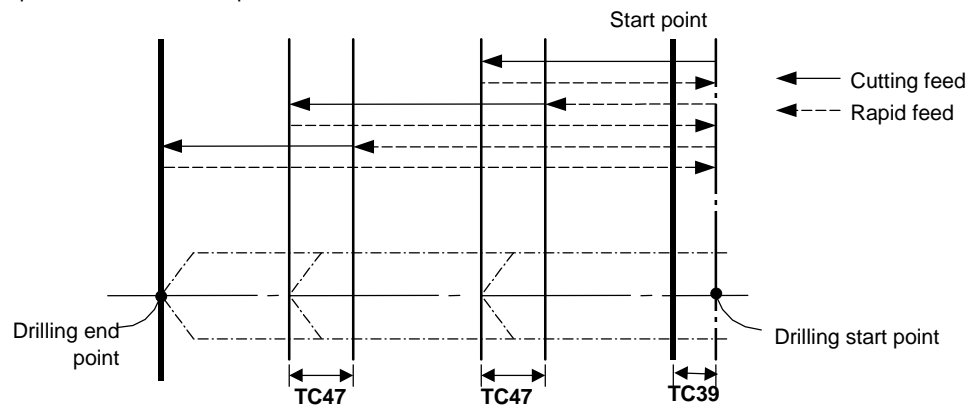
The data of the displayed menu represent following drill patterns.

#0, **#0** : Conventional type of drilling cycle
 The drill returns to the drilling start point at a cutting feedrate after each infeed operation has been completed.



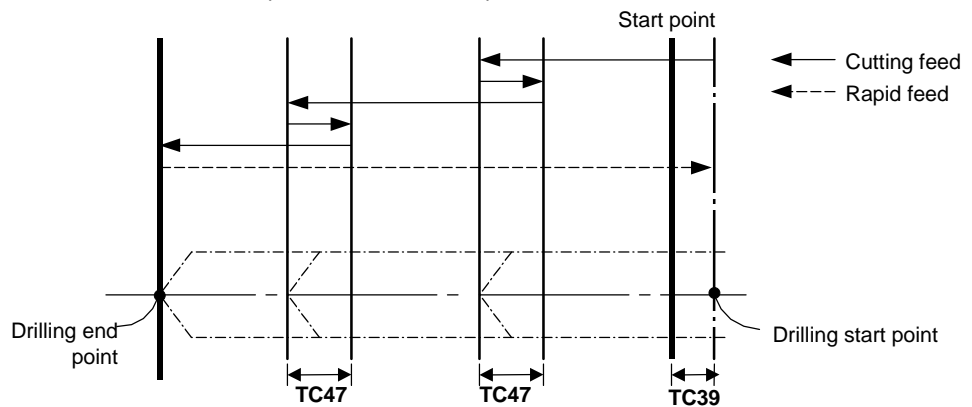
T4P114

#1, **#1** : Deep-hole drilling cycle
 The drill returns to the drilling start point at a rapid feedrate after each infeed operation has been completed.



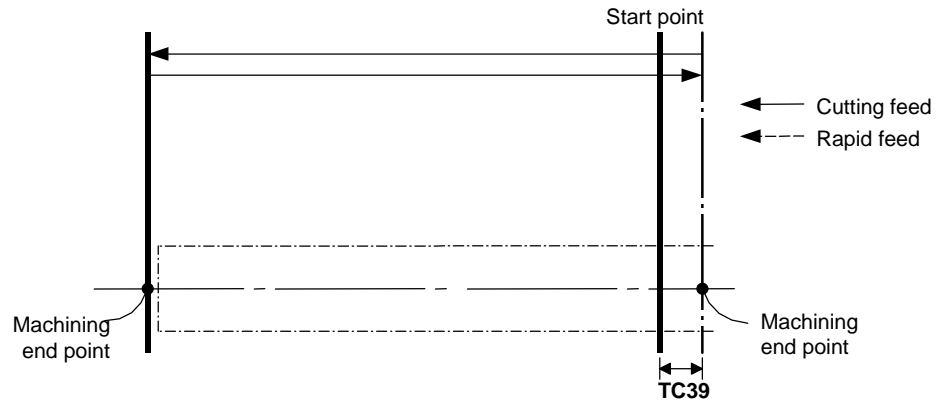
T4P115

#2, **#2** : High-speed deep-hole drilling cycle
 The drill returns through the specified distance (data specified in parameter TC47) at a cutting feedrate after each infeed operation has been completed.



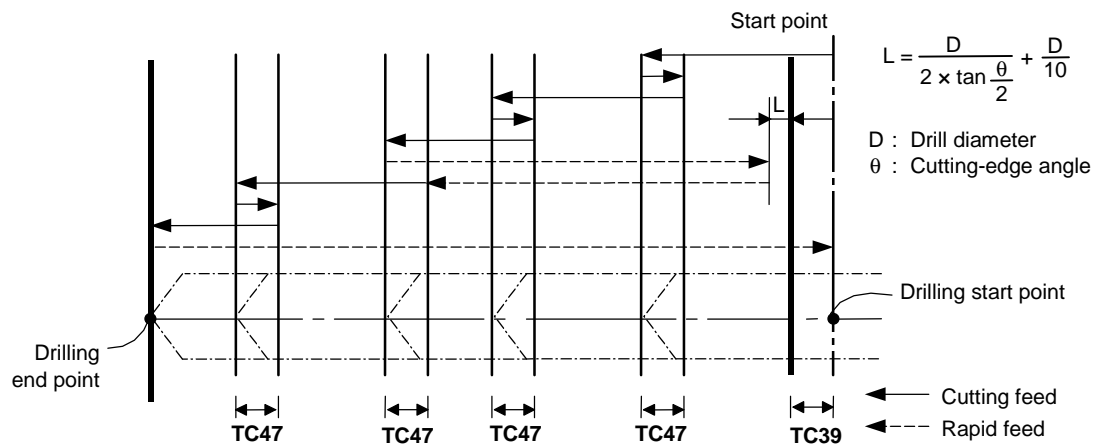
T4P116

#3, #3 : Reaming cycle
Drilled holes are finished with a reamer.



T4P117

#4, #4 : Very-deep-hole drilling cycle
Each time the normal pecking has been executed the parameter-designated number of times, the tool returns at a rapid feedrate to a position near the drilling start point.



T4P118

Note 1: Select either #0, #1, #2, #3 or #4 to drill stop-holes. Select either #0, #1, #2, #3 or #4 to drill through-holes.

Note 2: For patterns #0 to #4, the tool dwells at the bottom of the hole while the spindle rotates in accordance with the parameter-designated value. For patterns #4 and #4 the tool dwells for the same while after it has returned to a position near the drilling start point.

Note 3: With patterns #4 and #4, the rapid feedrate during the cycle can be reduced to the value designated in parameter K76.

[7] DEP-1, [8] DEP-2/NUM., [9] DEP-3

You can automatically set data in these items.

When machining pattern other than #3 or #3 is selected.

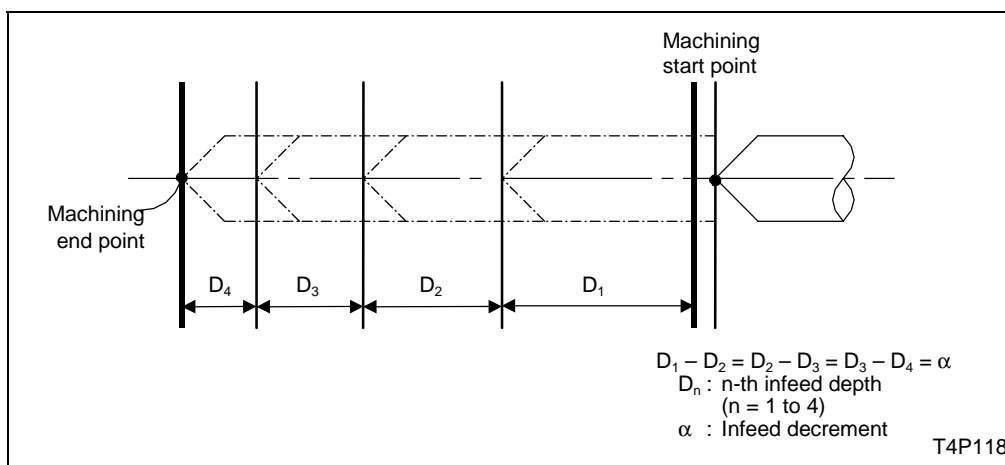
Pressing the menu key **AUTO SET** with the cursor at item [7] and setting a tool will automatically set the data that has been calculated by the NC unit into items [7] through [9].

When machining pattern #3 or #3 is selected.

Items [7] through [9] 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

$$D_n (= D_{n+1} = D_{n+2} = \dots) = D_{\min}$$

[10] C-SP

Specify the peripheral speed for the turning spindle.

See the description of the relevant item for BAR unit.

[11] FR

Enter the desired feedrate of the tool in terms of turning spindle speed per revolution.

See the description of the relevant item for BAR unit.

[12] M

Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).

See the description of the relevant item for BAR unit.

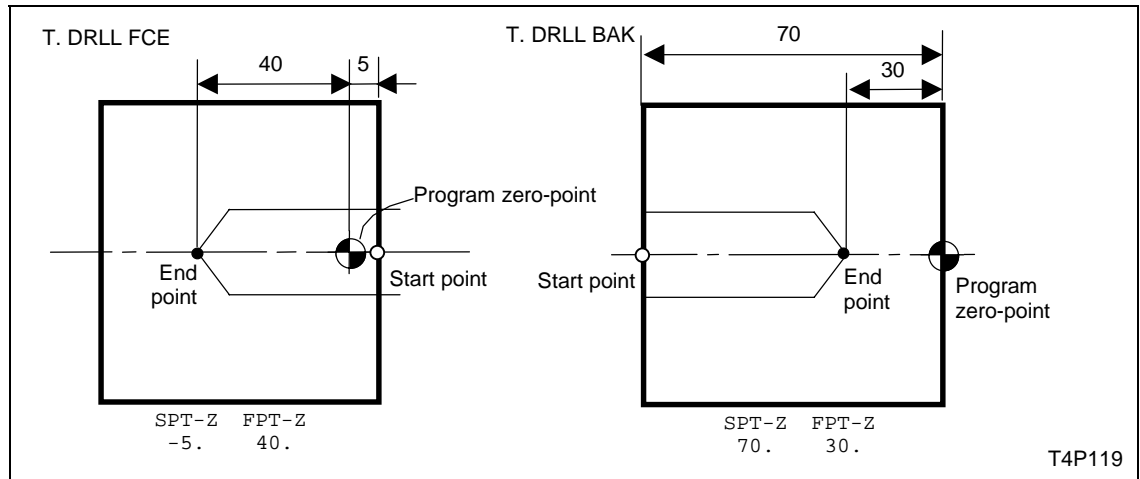
3. Setting shape sequence data

FIG	SPT-Z	FPT-Z
1	[1]	[2]

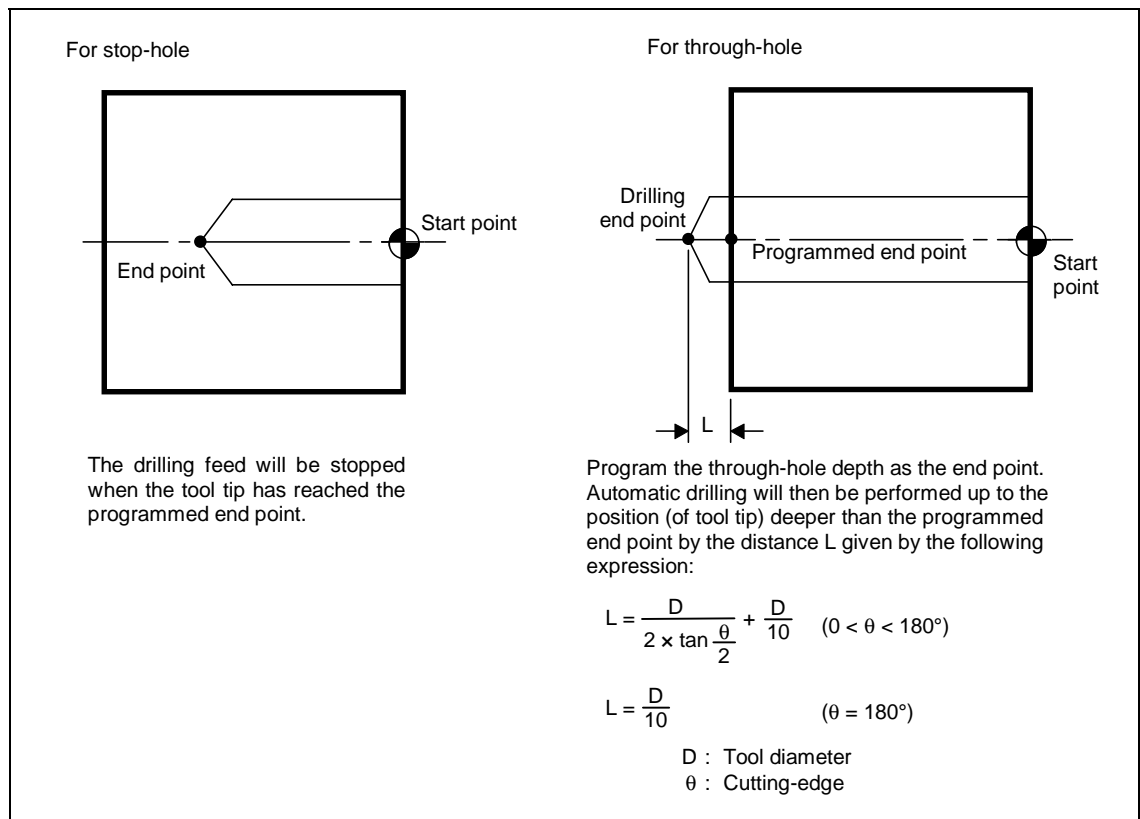
[1] SPT-Z, [2] FPT-Z

Set the coordinates of the start point and end point of the intended drilling pattern.

- The start point and end point for drilling stop-holes, for example, are positioned as shown below.

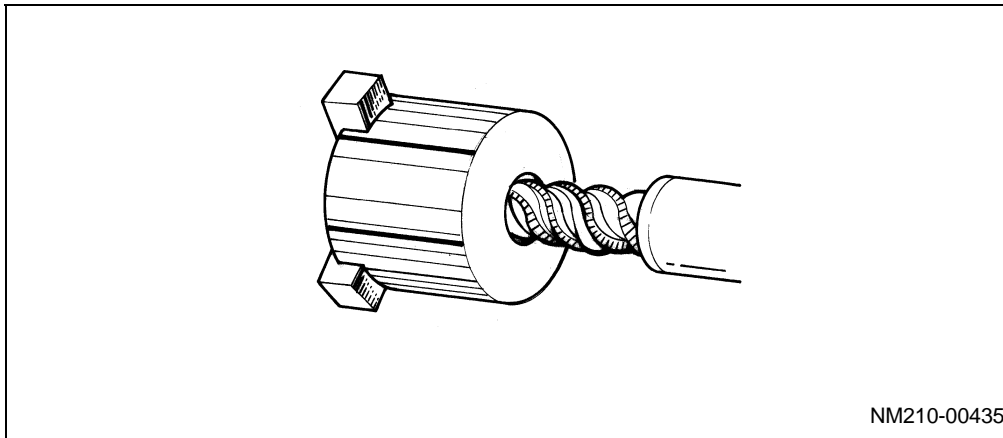


- The position of the end point differs from stop-hole to through-hole as follows:



4-21-10 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.





Press the menu key **T. TAP**  to select this unit.

1. Setting unit data

UNo.	UNIT	PART	NOM-DIA	PITCH
*	T. TAP	[1]	[2]	[3]

[1] PART

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

FCE 	BAK 								
--	--	--	--	--	--	--	--	--	--

From the menu, select the section to be machined.

Sections to be machined that correspond to each menu item are as follows:

FCE: Right edge of the workpiece

BAK: Left edge of the workpiece

Note: The menu item **BAK** may not be selectable for special machine specifications.

[2] NOM-DIA

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

METRIC THR(D M)	UNFY THR(D UN)	IPE THR(D PT)	PIPE THR(D PF)	PIPE THR(D PS)					OTHER
(a)	(b)	(c)	(d)	(e)					(f)

From (a) through (f) above, select the type of threads to be tapped. Then, set the nominal diameter of the threads.

The data of the displayed menu denote the following types of threads:

- (a)

METRIC THRD(M)

 : Metric threads
- (b)

UNFY THRD(UN)

 : Unified threads
- (c)

PIPE THRD(PT)

 : Tapered pipe threads (PT)
- (d)

PIPE THRD(PF)

 : Parallel pipe threads (PF)
- (e)

PIPE THRD(PS)

 : Tapered pipe threads (PS)
- (f)

OTHER

 : Other thread types

- If you select metric threads, the message “NOMINAL SIZE OF TAP?” will be displayed. In that case, set the nominal diameter of the threads to be tapped.

Example: To tap M8 metric threads:

Press the keys

METRIC THRD(M)

,

8

, and

↔

 in this order.

- If you select unified thread types, the message “TAPNOMINAL SIZE <MENU → INPUT>?” will be displayed and then the menu will change over to:

No.	H(1/2)	Q(1/4)	E(1/8)	S(1/16)		NOM-φ SELECT			
	HALF	QUARTER	EIGHTH	SIXTENTH					

Using the menu, set the nominal diameter of the threads to be tapped.

Example 1: To tap unified thread of the 3/4-16UN size:

Press the keys

UNFY THRD(UN)

,

Q(1/4) QUARTER

,

3

,

-

,

1

,

6

, and

↔

 in this order.

Example 2: To tap unified thread of the 1-1/8-7UN size:

Press the keys

UNFY THRD(UN)

,

E(1/8) EIGHTH

,

9

,

-

,

7

, and

↔

 in this order.

Example 3: To tap unified thread of the No. 1-16UN size:

Press the keys

UNFY THRD(UN)

,

No.

,

1

,

-

,

1

,

6

, and

↔

 in this order.

- Also, a press of the menu key **NOM-φ SELECT** 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 "TAPNOMINAL SIZE <MENU → INPUT>?" will be displayed and then the menu will change over to:

H(1/2) HALF	Q(1/4) QUARTER	E(1/8) EIGHTH	S(1/16) SIXTENTH		NOM-φ SELECT			
----------------	-------------------	------------------	---------------------	--	-----------------	--	--	--

Using the menu, set the nominal diameter of the threads to be tapped.


Example 1: To tap pipe thread of the PT3/8 size:

Press the keys

PIPE THRD(PT)

,

E(1/8) EIGHTH

, **3**, and  in this order.


Example 2: To tap pipe thread of the PF1/4 size:

Press the keys

PIPE THRD(PF)

,

Q(1/4) QUARTER

, **1**, and  in this order.


Example 3: To tap pipe thread of the PS1/8 size:

Press the keys

PIPE THRD(PS)

,

E(1/8) EIGHTH

, **1**, and  in this order.

- Also, a press of the menu key **NOM-φ SELECT** displays the following nominal tap diameter window to allow the desired nominal thread diameter to be entered by selecting it using the cursor keys.

[Tapered pipe thread (PT)]

PIPE THREAD PT	
PT 1/16	PT 1
PT 1/8	PT 1 [1/8]
PT 1/4	PT 1 [1/4]
PT 3/8	PT 1 [1/2]
PT 1/2	PT 2
PT 5/8	
PT 3/4	
PT 7/8	

[Tapered pipe thread (PF)]

PIPE THREAD PF	
PF 1/8	PF 1 [1/8]
PF 1/4	PF 1 [1/4]
PF 3/8	PF 1 [1/2]
PF 1/2	PF 2
PF 5/8	
PF 3/4	
PF 7/8	
PF 1	

[Tapered pipe thread (PS)]

PIPE THREAD PS	
PS 1/8	PS 1 [1/8]
PS 1/4	PS 1 [1/4]
PS 3/8	PS 1 [1/2]
PS 1/2	PS 2
PS 5/8	
PS 3/4	
PS 7/8	
PS 1	

[3] PITCH

Set the pitch of the threads to be tapped (pitch of turning tapping tool to be used).

When the appropriate nominal tool diameter is entered in item [2] NOM-DIA, data will be auto-set, except for special threads. Any data, however, can also be entered instead.

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
1			◆	◆	◆	◆	◆	◆		◆		
	↑	↑	↑	↑	↑				↑		↑	↑
	[1]	[2]	[3]	[4]	[5]				[6]		[7]	[7]

Remark 1: ◆: Data are not necessary to be set here.






Remark 2: In the tool sequence, one tool is automatically developed as follows.

Machining	Pattern
1	One tool for machining is automatically selected.

[1] TOOL (Name)

The name of the tool to be used for machining is set automatically.

When the cursor is present at this item, the following menu is displayed to allow the tool to be changed:

GENERAL	GROOVE	THREAD	T.DRILL	T.TAP		SPECIAL			
									

[2] TOOL (Section to be machined)

When the cursor is present at this item, the appropriate menu according to the tool name that was selected at item "[1] TOOL (Name)" is displayed as shown below.

- If either **GENERAL**, **GROOVE**, or **THREAD** has been selected

OUT OUTER DIAMETER	IN INNER DIAMETER	EDG EDGE		IN INNER (BAK)	EDG EDGE (BAK)				

- If either **T.DRILL**, or **T.TAP** has been selected

		EDG EDGE			EDG EDGE (BAK)				

- If **SPECIAL** has been selected

0001	0002	0003	0004	0005	0006	0007	0008	0009	

When creating a 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)
FCE	T. TAP	EDG EDGE
BAK		EDG EDGE (BAK)

Note: The above example applies when the tools best suited to a general machining shape pattern are to be used. Tools other than those shown in the above example may be suitable for the shape actually specified.

[3] NOM. (Nominal diameter)

Set the diameter of the hole to be tapped (nominal diameter of the turning tap).

[4] NOM. (Tool identification code)

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

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>
---	---	---	---	---	---	---	---	---------------	-----

The meaning of each data of the displayed menu is the same as for the bar-materials machining unit (BAR).

[5] No. (Priority number: No.)

Enter the priority number for prior machining and subsequent machining.
See the description of the relevant item for BAR unit.

[6] C-SP

Specify the peripheral speed for the turning spindle.
See the description of the relevant item for BAR unit.

[7] M

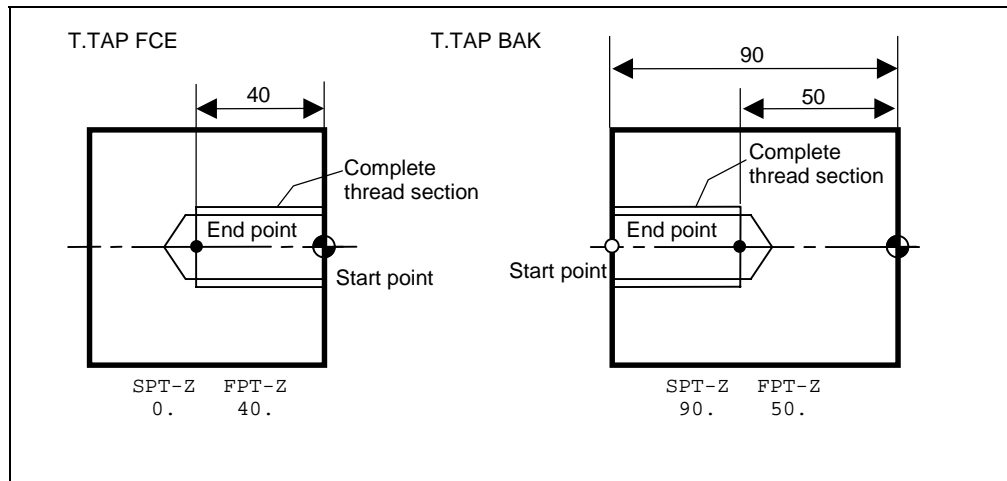
Specify the M-code to be issued for the tool immediately after its ATC (automatic tool change).
See the description of the relevant item for BAR unit.

3. 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.



4-22 Coordinate Measuring Unit (MMS)

The coordinate measuring unit automatically compensates the workpiece coordinates system by performing the centering measurements.

The measuring movement is performed by specifying the feeler (touch sensor) as a tool and the type of measurement.

Note: The coordinate 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, as the coordinates in the sequence data, the position from the WPC zero-point in the axial direction of the machine coordinate system.

4-22-1 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	LINE	FACE	TURNING	MANUAL	WPC	OFFSET	END	SHAPE	>>>
MACH-ING	MACH-ING	MACH-ING		PROGRAM				CHECK	

- (2) Press the menu key >>>. The following menu will be displayed.

C-POINT	C-LINE	INDEX	M CODE	SUB	MMS	WORKPICE	TOOL	WORKPICE	>>>
MACH-ING	MACH-ING			PROGRAM		MEASURE	MEASURE	SHAPE	

- (3) Press the menu key **MMS**.

4-22-2 Composition of the MMS unit

1. Data setting in MMS unit

UNo.	UNIT	TOOL	NOM-φ	No.	U. SKIP	\$
	MMS	TOL SENS				

Cursor position	Description
NOM-φ	Specify the nominal diameter of the feeler. Enter the approximate diameter of the point of feeler by means of numeric keys.
No.	Enter the machining priority number.
U.SKIP	Specify whether the MMS unit is executed or not. Execution Non-execution

4-22-3 Composition of the MMS sequence

1. Setting MMS sequence data

SNO.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1										

Cursor position	Description																																	
PTN	<p>Select the type of measurement from the following menus. Pressing the menu key >>> changes the menu a → b → a in this order.</p> <table border="1"> <tr> <td>X FACE</td> <td>Y FACE</td> <td>Z FACE</td> <td>C FACE</td> <td>X GROOVE</td> <td>Y GROOVE</td> <td>Z GROOVE</td> <td>C GROOVE</td> <td>PTN END</td> <td>>>></td> <td>a</td> </tr> <tr> <td>X STEP</td> <td>Y STEP</td> <td>Z STEP</td> <td>C STEP</td> <td>X-Y BORE</td> <td>Y-Z BORE</td> <td>X-Y BOSS</td> <td>Y-Z BOSS</td> <td>PTN END</td> <td>>>></td> <td>b</td> </tr> <tr> <td>X-Y-th. CNR</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>CALIBR.</td> <td>PTN END</td> <td>>>></td> <td>c</td> </tr> </table> <p>For details, refer to Subsection 4-22-6, "Type of measurement."</p>	X FACE	Y FACE	Z FACE	C FACE	X GROOVE	Y GROOVE	Z GROOVE	C GROOVE	PTN END	>>>	a	X STEP	Y STEP	Z STEP	C STEP	X-Y BORE	Y-Z BORE	X-Y BOSS	Y-Z BOSS	PTN END	>>>	b	X-Y-th. CNR							CALIBR.	PTN END	>>>	c
X FACE	Y FACE	Z FACE	C FACE	X GROOVE	Y GROOVE	Z GROOVE	C GROOVE	PTN END	>>>	a																								
X STEP	Y STEP	Z STEP	C STEP	X-Y BORE	Y-Z BORE	X-Y BOSS	Y-Z BOSS	PTN END	>>>	b																								
X-Y-th. CNR							CALIBR.	PTN END	>>>	c																								
X, Y, Z, C	Specify the measurement starting position by means of numeric keys.																																	
5	Specify the positioning coordinate of the 5th axis for measurement movement by means of numeric keys.																																	
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.																																	
D/L	Specify the diameter of the hole, the diameter of the boss, the width of the groove, the width of the projection, etc. by means of numeric keys. The contents of the entry differ according to the type of measurement. For details, see Subsection 4-22-6, "Type of measurement."																																	
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.																																	
DIR.	<p>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.</p> <table border="1"> <tr> <td>←Z</td> <td>←X</td> <td>↓Z</td> <td>↓X</td> <td>→Z</td> <td>→X</td> <td></td> <td></td> <td></td> <td></td> </tr> </table> <p>Selection of the menu key ← X specifies the milling spindle head facing in the direction of the arrow (left) and the approche/escape in X-axial direction during the measurement.</p> <p style="text-align: right;">D736P0107</p>	←Z	←X	↓Z	↓X	→Z	→X																											
←Z	←X	↓Z	↓X	→Z	→X																													

Note 1: The measurement movement is executed after the movement, if provided, on 5th axis.

Note 2: 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 3: The MMS unit performs the movement in the initial fundamental coordinates system until the completion of all the types of measurement specified in the unit.

Example:

[1]	UNo.	UNIT	ADD WPC		X	Y	th	Z
	1	WPC-0			-300.	-400	0.	-500.
[2]	UNo.	UNIT	TOOL	NOM-φ	No.	U.SKIP		\$
	2	MMS	TOL SENS	5.		0.		
[2]	SNo.	PTN	X	Y	Z	C	5	R
	1	X-FACE	-20.	0.	-10.	0.	0.	0.
[2]		Y-FACE	0.	-20.	-10.	0.	0.	0.
[3]	UNo.	UNIT	DIA	DEPTH	CHMF			
	3	T.DRILL	12.	20.	0.5			

In the above example, all the types of measurement in the MMS unit [2] are executed according to the data of the fundamental coordinates unit [1].

From unit [3], the machining is performed according to the new system of fundamental coordinates (workpiece zero-point) corrected in the MMS unit [2].

But the data of the fundamental coordinates unit [1] are rewritten for each type of measurement.

Note 4: When the common unit contains the workpiece multi-piece machining command, the MMS unit is only executed a first time.

Note 5: The compensation is made correctly even when the fundamental coordinates unit and the MMS unit are entered separately in the main program and in the sub-program.

Note 6: Except for measurement of inclinations, no measurements can be correctly performed if an angle has been set in item th of the fundamental coordinates unit.

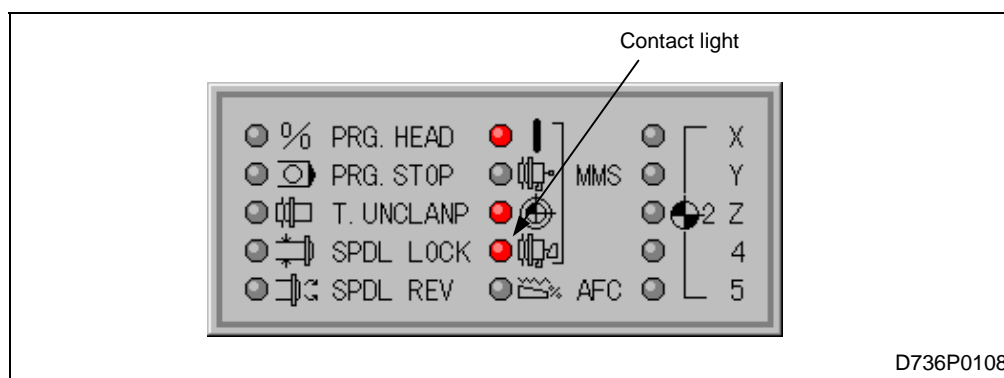
Note 7: Before execution of the MMS unit turn off the symmetrical image function.

The measurement movement after contact with the workpiece may not be correctly performed if the symmetrical image function is valid.

4-22-4 Measurement of length of feeler

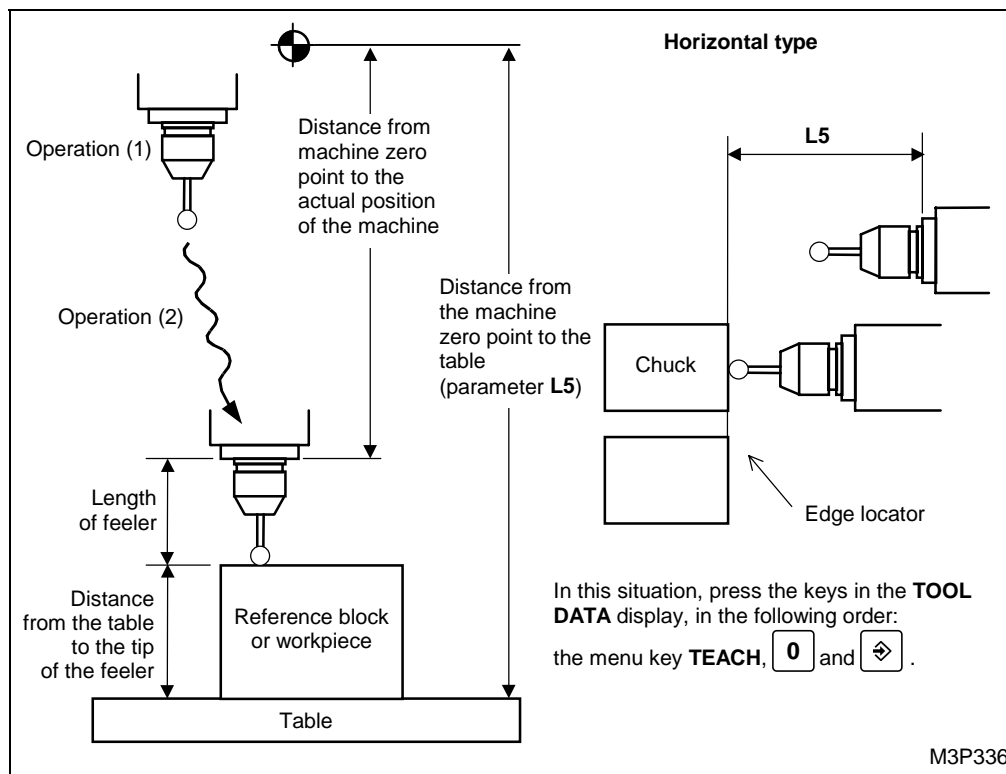
It is necessary to measure the length of the feeler in order to enter the data into the item LENGTH on the **TOOL DATA** display. This section describes the tool length measurement in manual mode using the TEACH function on the **TOOL DATA** display.

- (1) Mount the feeler on the spindle.
- (2) In manual mode, move the axes in order to bring the point of feeler to the upper face of the reference block or to a workpiece of which the height is known.
- (3) In manual handle speed mode, slowly move the Z-axis until the contact light (MMS) in the CNC signal window goes on.



D736P0108

- (4) Call up the **TOOL DATA** display.
- (5) Press the menu key **TEACH**.
 - ➔ The cursor appears in the item **LENGTH** on the display.
- (6) Enter the height of the reference block or of the workpiece by means of numeric keys.
 - ➔ The NC equipment calculates the length of the feeler and the value thus obtained is automatically entered in the item **LENGTH**.



Since the NC equipment memorizes the actual position of the machine, the length of the feeler (distance from the table to the point of feeler) is automatically calculated if the height of the reference block or of the workpiece is specified.

4-22-5 Feeler calibration measurement

The calibration measurement function serves to automatically record in the parameters the main measurement correction of the point of feeler such as the alignment deviation of the point of feeler and the tool shank (on X, Y), the functional clearance and the inertia feed of the machine itself, etc.

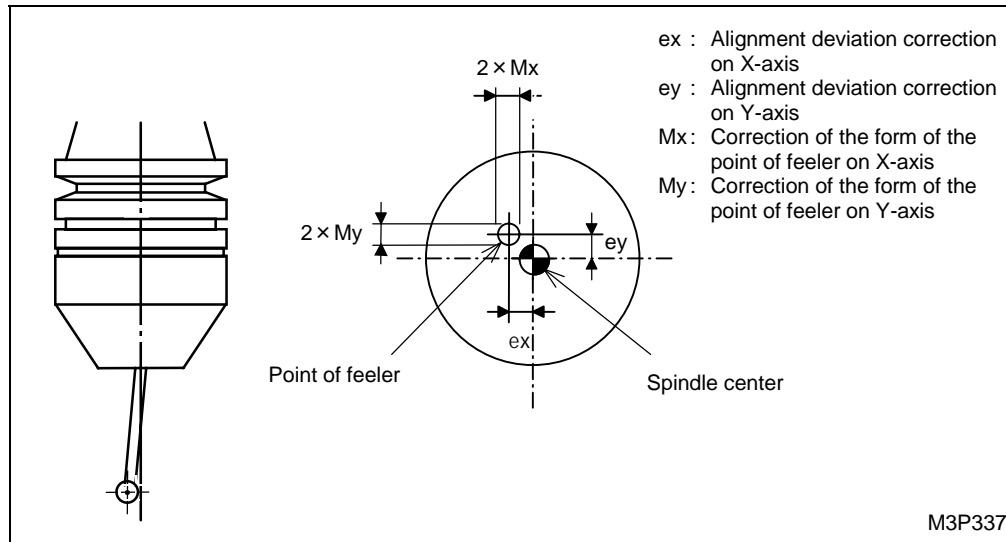
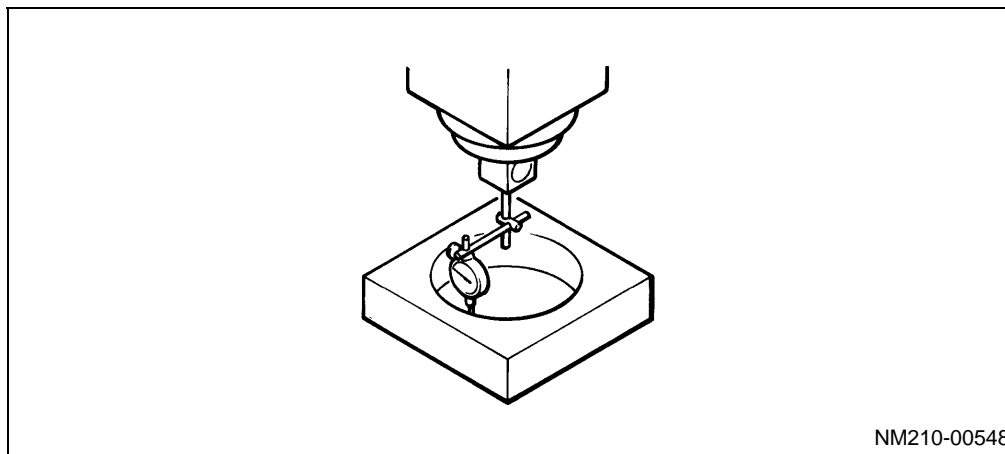


Fig. 4-25 Correction of calibration measurement

- (1) Mount a reference block having a reference hole ($\phi 50$ to $\phi 100$).
- (2) Measure the inside diameter of the reference hole.

Note: It must be correctly measured to within 1/1000 by means of a dial type comparator for measuring cylindricity.

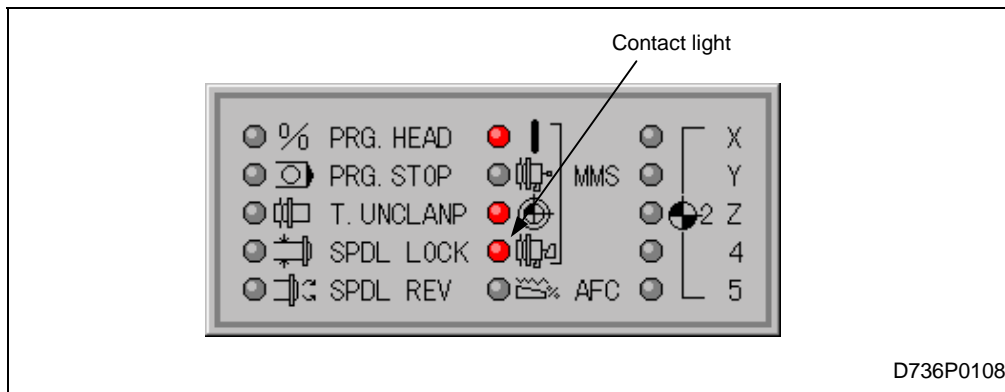


- (3) Move the spindle axis to the center of the reference hole. (Perform the alignment operating. The pivoting of the lever type tester should be less than 4μ).

Note: The precision of all measurements depends on the precision of this alignment.

- (4) Prepare the calibration measurement program.
 - The same as for the general measurement program, press the menu key **CALIBR.**, then prepare the program.
- (5) Enter the X, Y coordinates of the spindle axis in the X and Y items of the WPC unit by using the menu keys **WPC SEARCH** and **TEACH** on the **PROGRAM** display. For details, refer to Chapter 6, "COORDINATES MEASUREMENT FUNCTION."

- (6) Mount the feeler on the spindle.
- (7) In manual mode, move the axes in order to place the point of the feeler in contact with top of the workpiece having the reference hole.
 - Move the Z-axis slowly until the contact light (MMS) lights.



- (8) Enter the Z coordinate of the spindle in the Z column of the WPC unit using the menu keys **WPC SEARCH** and **TEACH** on the **PROGRAM** display.
 - For details, refer to Chapter 6, "COORDINATES MEASUREMENT FUNCTION."
- (9) Select the automatic operation mode and start the calibration measurement program, which causes the measurement → ATC (return to magazine) of the feeler.
 - Thus, the correction data is automatically entered in the parameters (**L1** to **L4**).

Example of calibration measurement program

UNo.	UNIT	ADD.WPC	X	Y	th	Z	C		
1	WPC-0		[1]	[1]	0.	[2]	0.		
UNo.	UNIT	TOOL	NOM-φ	No.	U.SKIP				
2	MMS	TOL SENS	5						
SNo.	PTN	X	Y	Z	C	R	D/L	K	
1	CAL	[3]	[3]	[4]	0	◆	[5]	[6]	
UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE
3	END	0	0						

No.	Item	Description
[1]	X, Y	Specify the X and Y coordinates of the workpiece zero-point (fundamental coordinates) in the machine coordinate system. - Coordinates of the center of the reference hole entered by operation (3).
[2]	Z	Specify the Z coordinates of the workpiece zero-point (fundamental coordinates) in the machine coordinate system. - Coordinates of the top of the reference hole entered by operation (7).
[3]	X, Y	Specify the coordinates of the center of the hole based on the workpiece zero-point (fundamental coordinates). - In this example, 0 because the workpiece zero-point equals to the center of the reference hole.
[4]	Z	Specify the depth to which the point of feeler is inserted in the reference hole in order to make the measurement.
[5]	D/L	Specify the measurement value of the inside diameter of the reference hole. - Inside diameter of the reference hole, measured by operation (2).
[6]	K	Feed distance at skip speed (K = 2 to 5)

Note 1: These corrections are essential data for ensuring the deviation of the system by the feeler. The precision of other measurements are influenced by the precision of the positioning of the machine itself and of this calibration measurement.

Note 2: The calibration measurement is not necessary for each deviation of the coordinates system, but it must obligatorily be made when the feeler is used for the first time or when it is replaced by a new one.

4-22-6 Type of measurement

Select the type of measurement for the deviation of the coordinates system. The six types of measurement available are described in the following descriptions 1 to 6:

- Measurement of the reference surfaceX FACE, Y FACE, Z FACE, C FACE
- Measurement of groove center.....X GROOVE, Y GROOVE, Z GROOVE, C GROOVE
- Measurement of center of projection width ..X STEP, Y STEP, Z STEP, C STEP
- Measurement of boring center.....X-Y BORE, Y-Z BORE
- Measurement of boss centerX-Y BOSS, Y-Z BOSS
- Measurement of workpiece inclinationX-Y-th. CNR

1. Measurement of the reference surface

The measurement of the reference surface is of 4 types: X, Y, Z, and C.

- X reference face (X FACE)
- Y reference face (Y FACE)
- Z reference face (Z FACE)
- C reference face (C FACE)

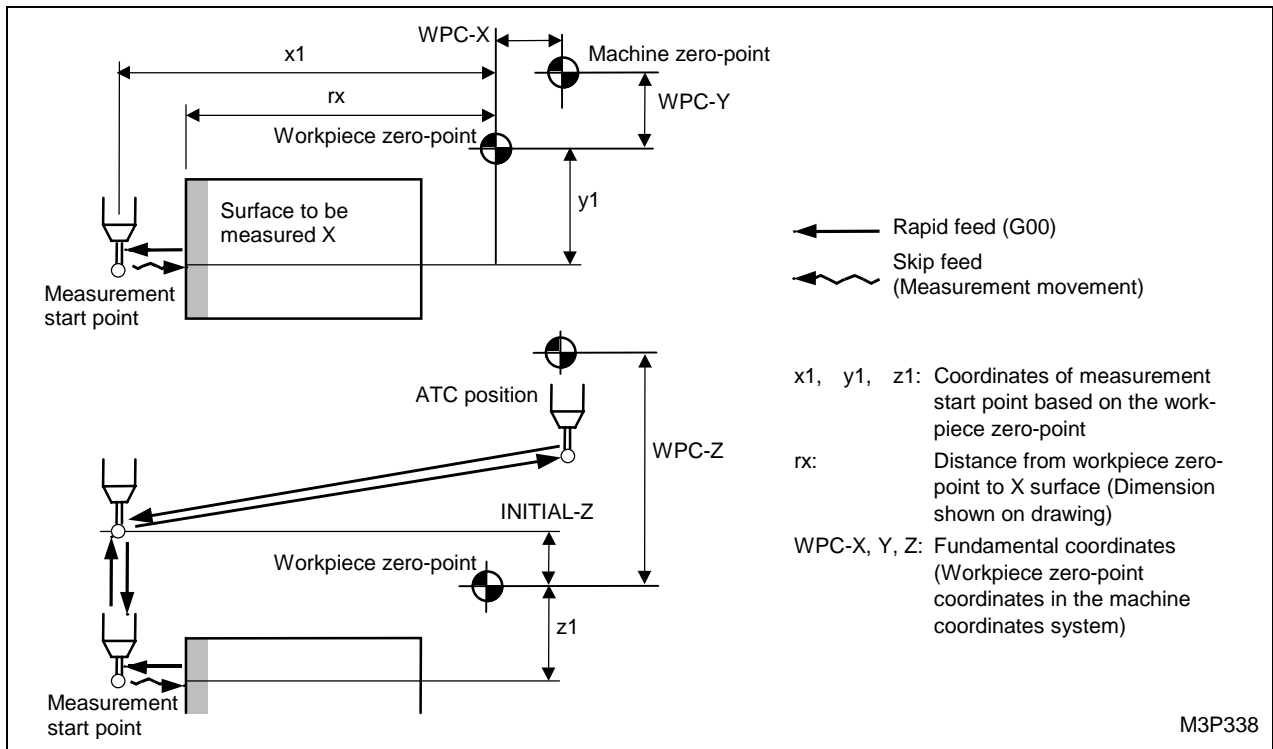
A. X reference face

The X-axial fundamental coordinate is corrected by entering the position of the X reference face in the workpiece coordinates system.

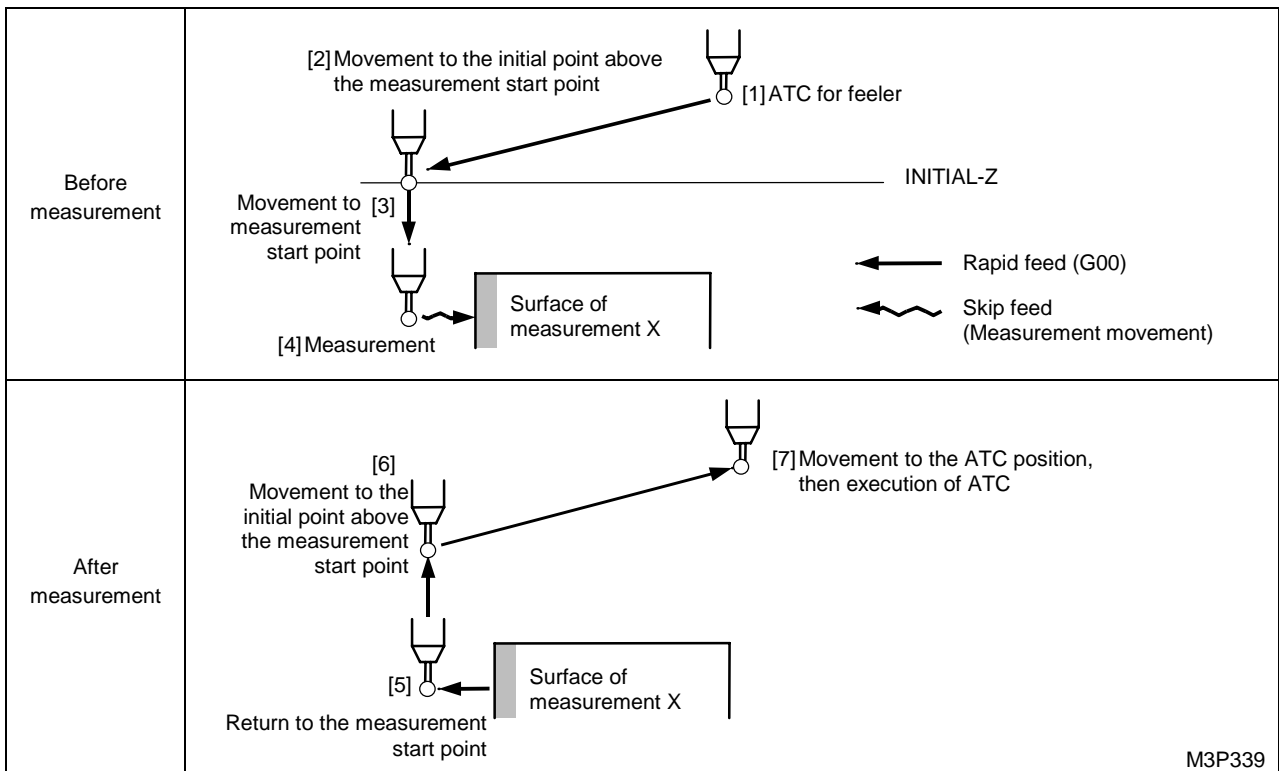
SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	X-FACE	x1	y1	z1	c1	0	rx	◆	◆	←X

◆: Data are not necessary to set here.

Note: Instead of “5”, the axis name is indicated on the NC display. This also applies to the other measurement patterns.

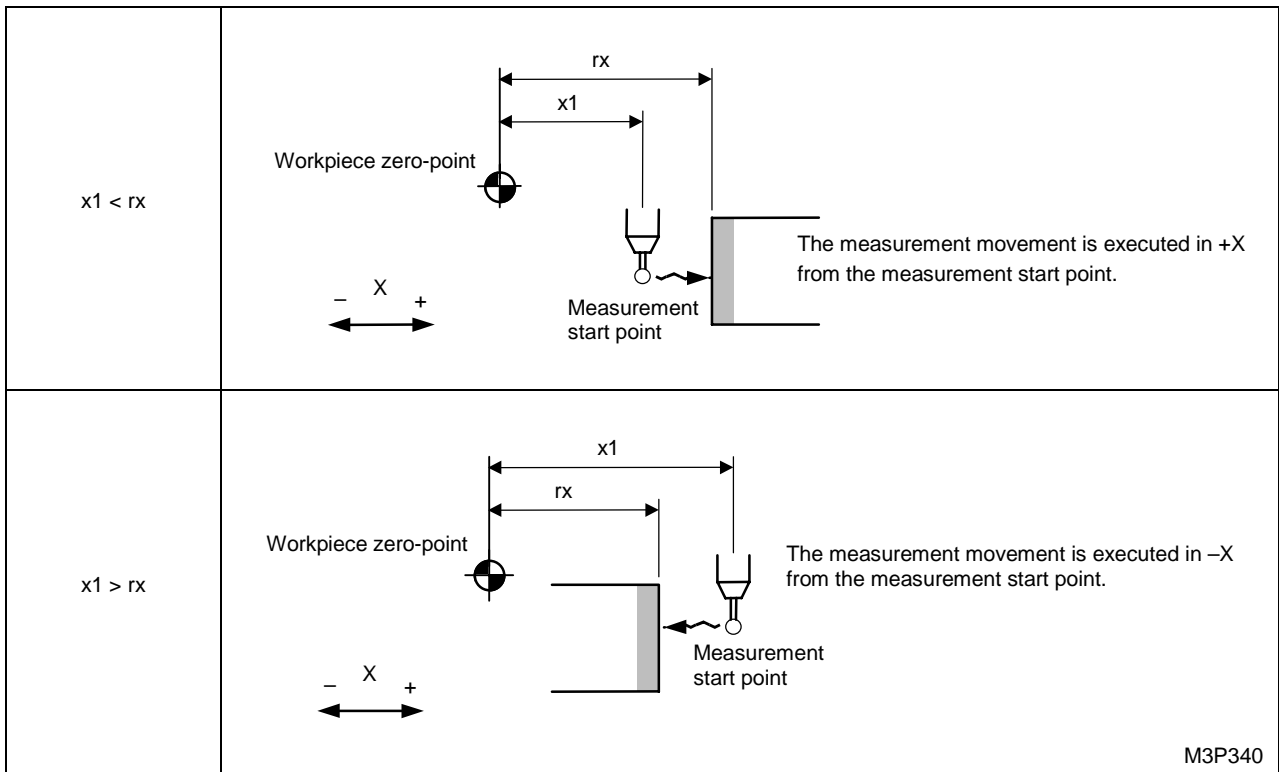


[Measurement movement]



Note 1: The X fundamental coordinate is corrected so that the X coordinate of the measurement surface is equal to the coordinate entered in rx when the feeler is moved to the initial point [6].

Note 2: The direction in which the measurement movement is made from the measurement start point depends on x_1 and rx .



B. Y reference face

The Y-axial fundamental coordinate is corrected by entering the position of the Y reference face in the workpiece coordinates system.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Y-FACE	x1	y1	z1	c1	0	ry	◆	◆	←X

◆: Data are not necessary to set here.

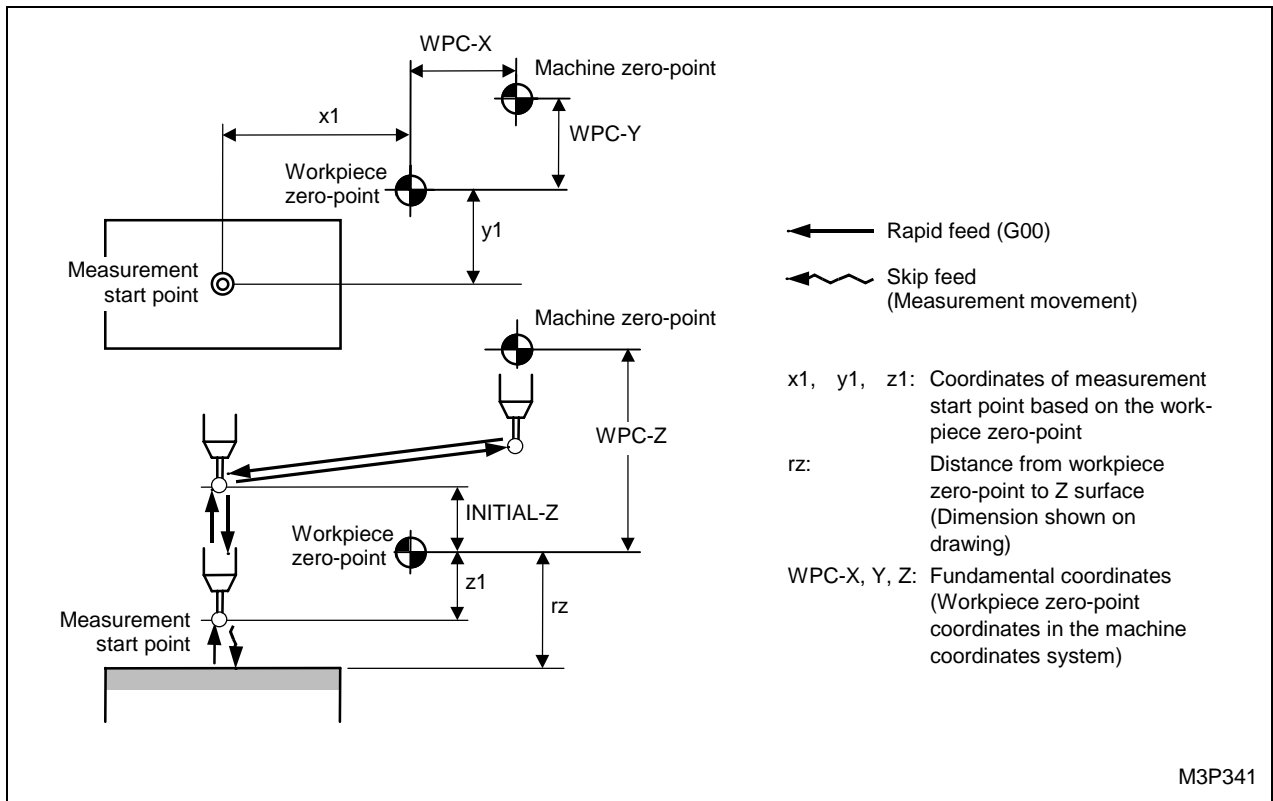
The movement is, in principle, identical to that for the X reference face. ry signifies the distance from the workpiece zero-point to the Y surface (dimension shown on the drawing).

C. Z reference face

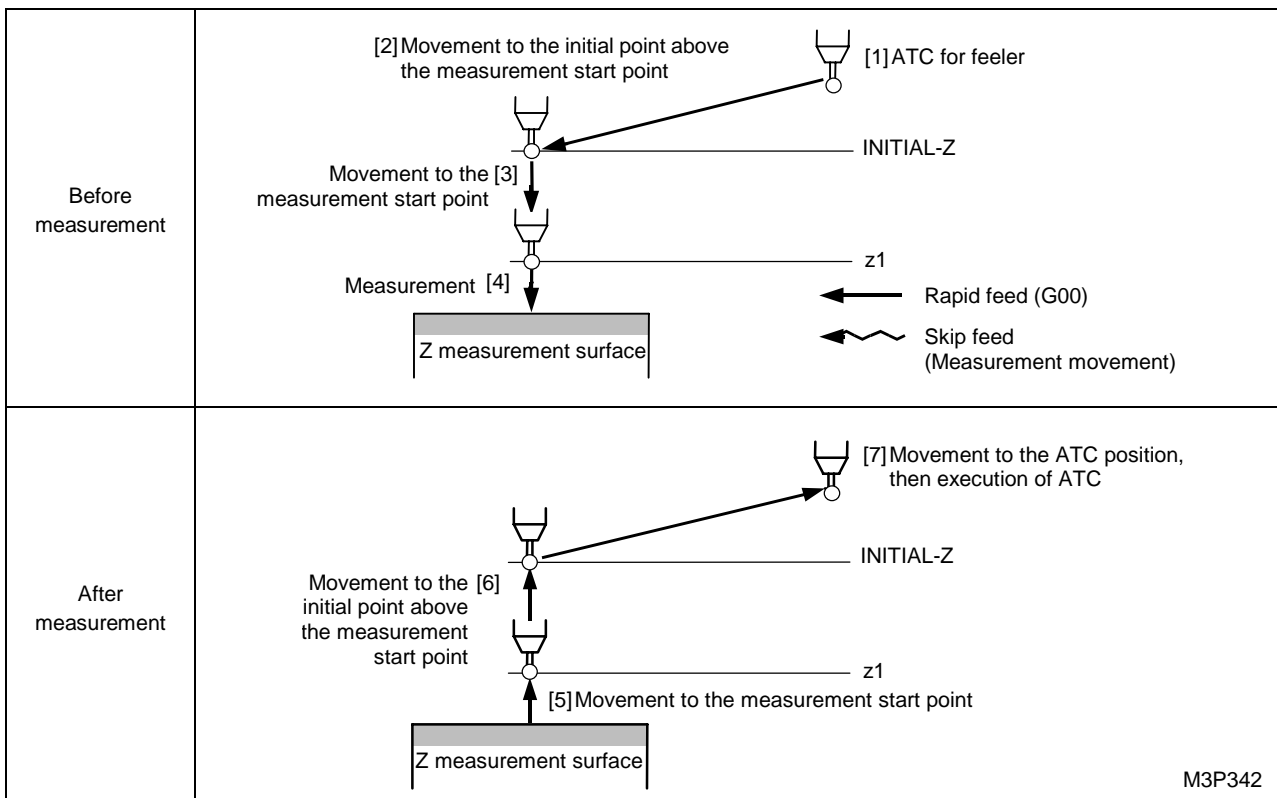
The Z-axial fundamental coordinate is corrected by entering the position of the Z reference face in the workpiece coordinates system.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Z-FACE	x1	y1	z1	c1	0	rz	◆	◆	←X

◆: Data are not necessary to set here.



[Measurement movement]



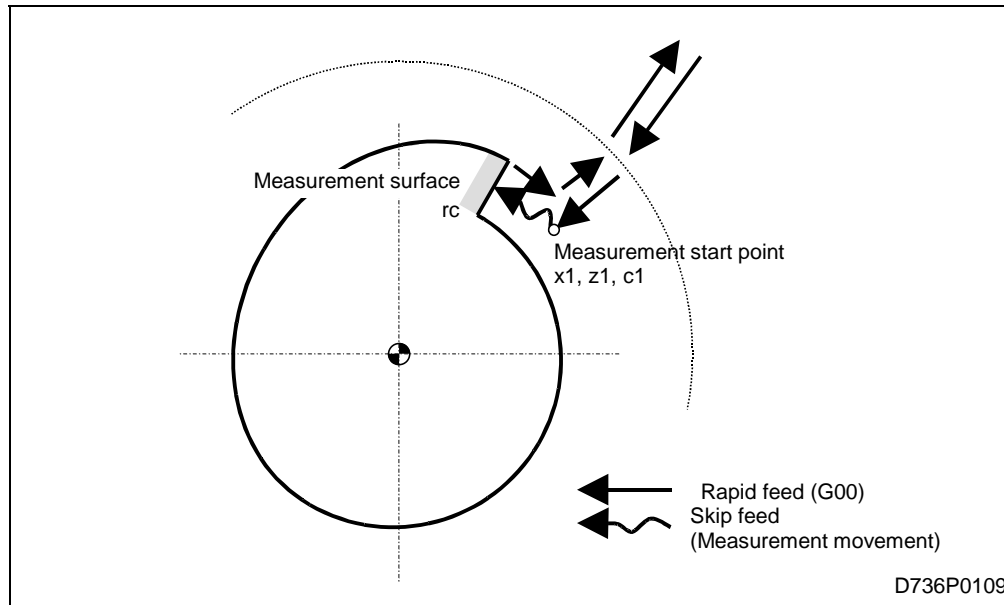
Note: The Z fundamental coordinate is corrected so that the Z coordinate of the measurement surface is equal to the coordinate entered in r_z , when the feeler is moved to the initial point [6].

D. C reference face

The C-axis fundamental coordinate is corrected by entering the position of the C reference face in the workpiece coordinates system.

SNo.	PTN	X	Y	Z	C	S	R	D/L	K	DIR.
1	C-FACE	x1	◆	z1	C1	0	rc	◆	◆	←X

◆: Data are not necessary to set here.



2. Measurement of groove center

The measurement of groove center is available in four types according to whether the direction of the groove width is on the X-, Y-, Z- or the C-axis.

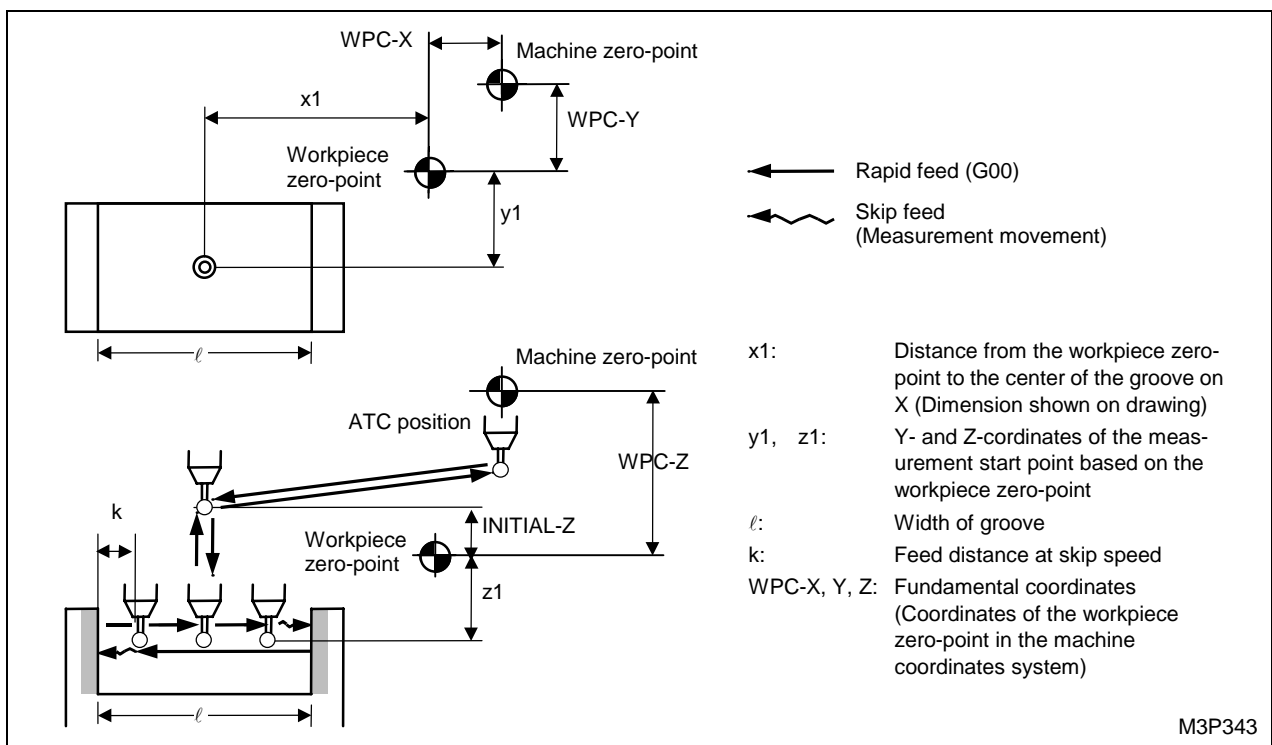
- Groove center-X (X GROOVE)
- Groove center-Y (Y GROOVE)
- Groove center-Z (Z GROOVE)
- Groove center-C (C GROOVE)

A. Groove center-X

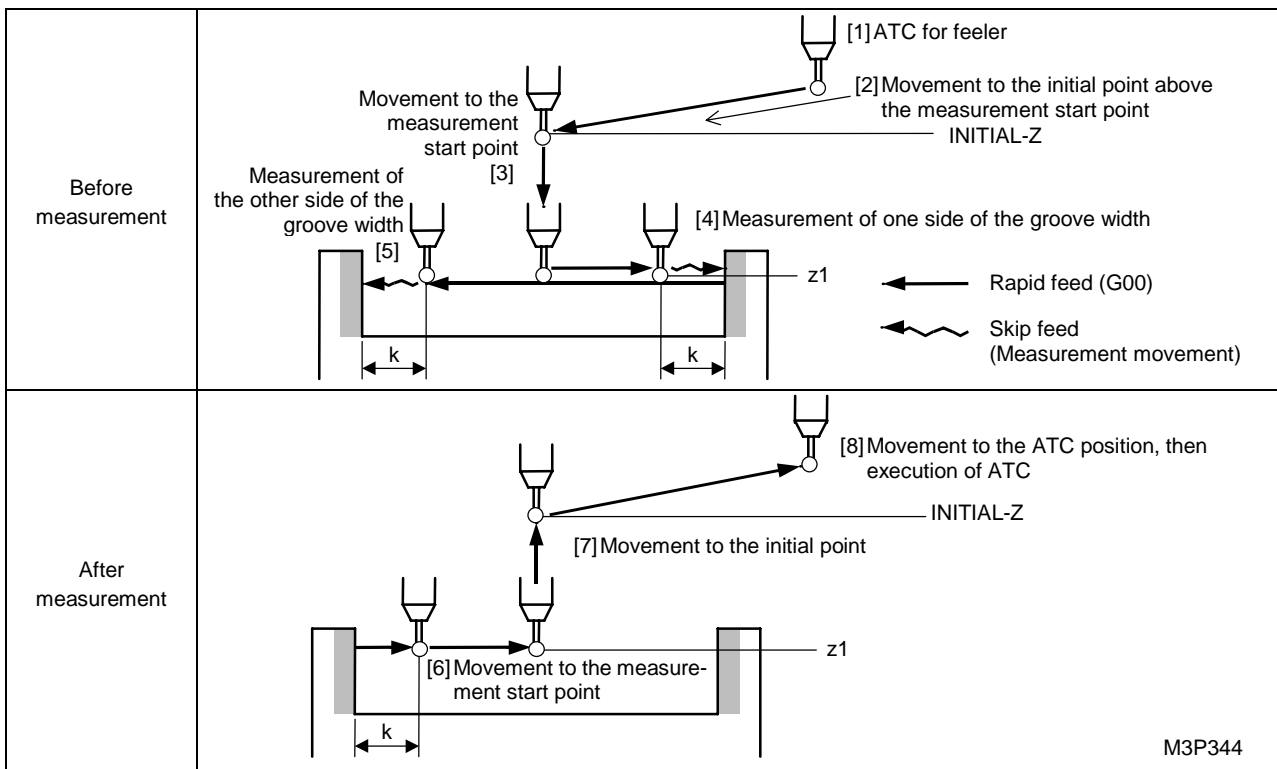
The X-axial fundamental coordinate is corrected by entering the position of the groove center in the workpiece coordinates system and the groove width.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	X-GROOVE	x1	y1	z1	c1	0	◆	ℓ	k	←Z

◆: Data are not necessary to set here.



[Measurement movement]



Note: The X fundamental coordinate is corrected so that the coordinate of the center of the groove to be measured is equal to the coordinate entered in x1 when the feeler is moved to the initial point [7].

B. Groove center-Y

The Y-axial fundamental coordinate is corrected by entering the position of the groove center in the workpiece coordinates system and the groove width.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Y-GROOVE	x1	y1	z1	c1	0	◆	ℓ	k	←Z

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the center of groove-X.

y1 signifies the distance on Y from the workpiece zero-point to the center of the groove width (dimension marked on the drawing).

C. Groove center-Z

The Z-axial fundamental coordinate is corrected by entering the position of the groove center in the workpiece coordinates system and the groove width.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Z-GROOVE	x1	y1	z1	c1	0	◆	ℓ	k	←Z

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the center of groove-X.

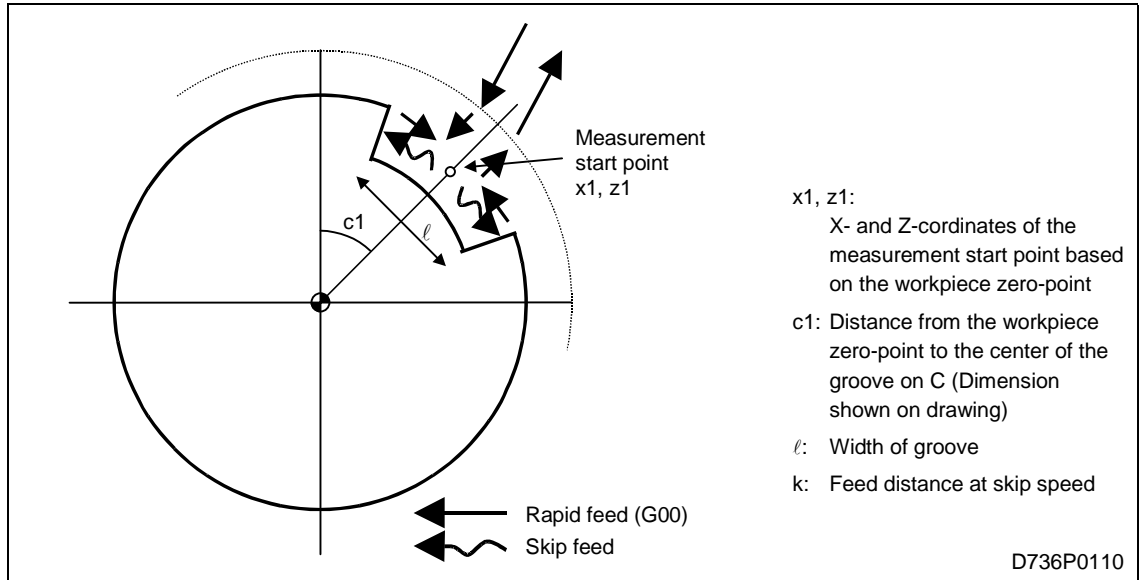
z1 signifies the distance on Z from the workpiece zero-point to the center of the groove width (dimension marked on the drawing).

D. Groove center-C

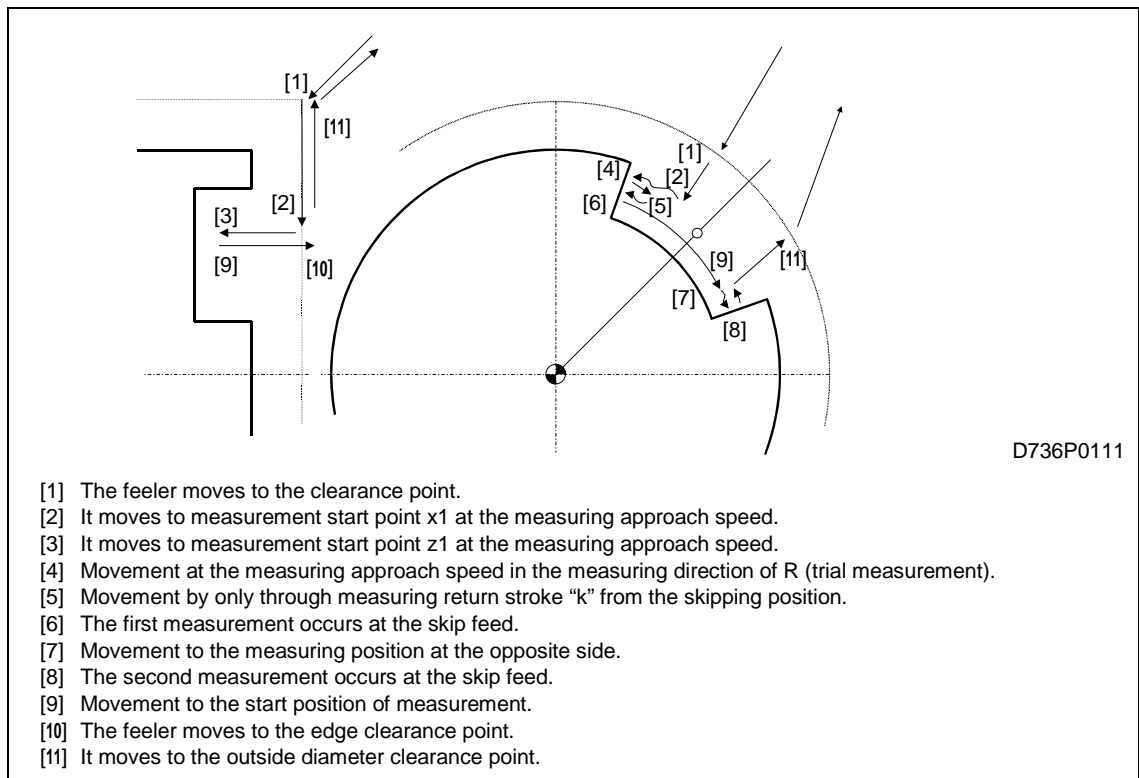
The C-axis fundamental coordinate is corrected by entering the position of the groove center in the workpiece coordinates system and the groove width.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	C-GROOVE	x1	◆	z1	c1	0	CCW	ℓ	k	←Z

◆: Data are not necessary to set here.



[Measurement movement]



Note: If the sensor operates during movement at the measuring approach speed in steps [2] and [3], these steps will be repeated again (this is referred to as the retry function). The retry function is described later in this manual.

3. Measurement of center of projection width

The measurement of the center of the projection width is available in four types according to whether the direction of the width of the projection is on the X-, Y-, Z-, and C-axis.

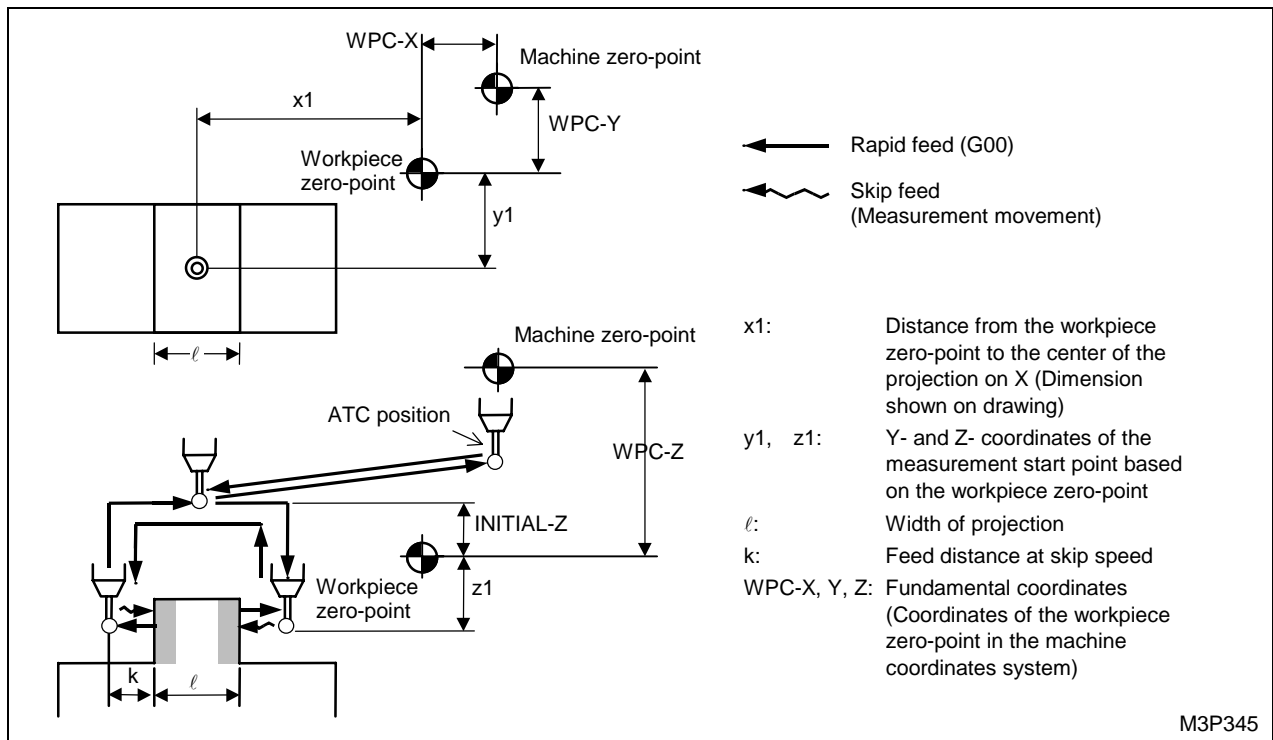
- Projection width on X-axis (X STEP)
- Projection width on Y-axis (Y STEP)
- Projection width on Z-axis (Z STEP)
- Projection width on C-axis (C STEP)

A. Projection width on X-axis

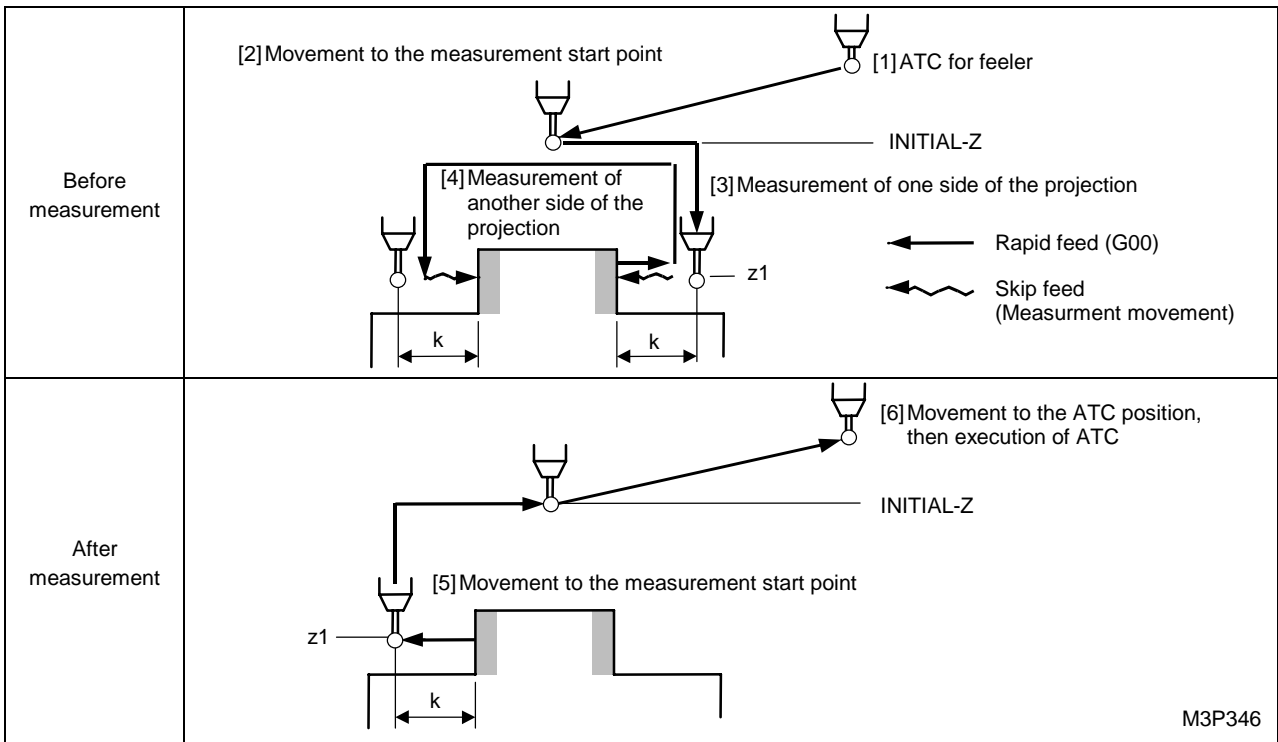
The X-axial fundamental coordinate is corrected by entering the position of the projection center in the workpiece coordinates system and the projection width.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	X-STEP	x1	y1	z1	c1	0	◆	ℓ	k	←Z

◆: Data are not necessary to set here.



[Measurement movement]



Note: The X fundamental coordinate is corrected so that the coordinate of the center of the projection width to be measured is equal to the coordinate entered in x1 when the feeler is moved to the initial point [5].

B. Projection width on Y-axis

The Z-axial fundamental coordinate is corrected by entering the position of the projection center in the workpiece coordinates system and the projection width.

SN _o .	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Y-STEP	x1	y1	z1	c1	0	◆	ℓ	k	←Z

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the projection width on the X-axis.

y1 signifies the distance on Y from the workpiece zero-point to the center of the projection width (dimension shown on the drawing).

C. Projection width on Z-axis

The Y-axial fundamental coordinate is corrected by entering the position of the projection center in the workpiece coordinates system and the projection width.

SN _o .	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	Z-STEP	x1	y1	z1	c1	0	◆	ℓ	k	↓X

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the projection width on the X-axis.

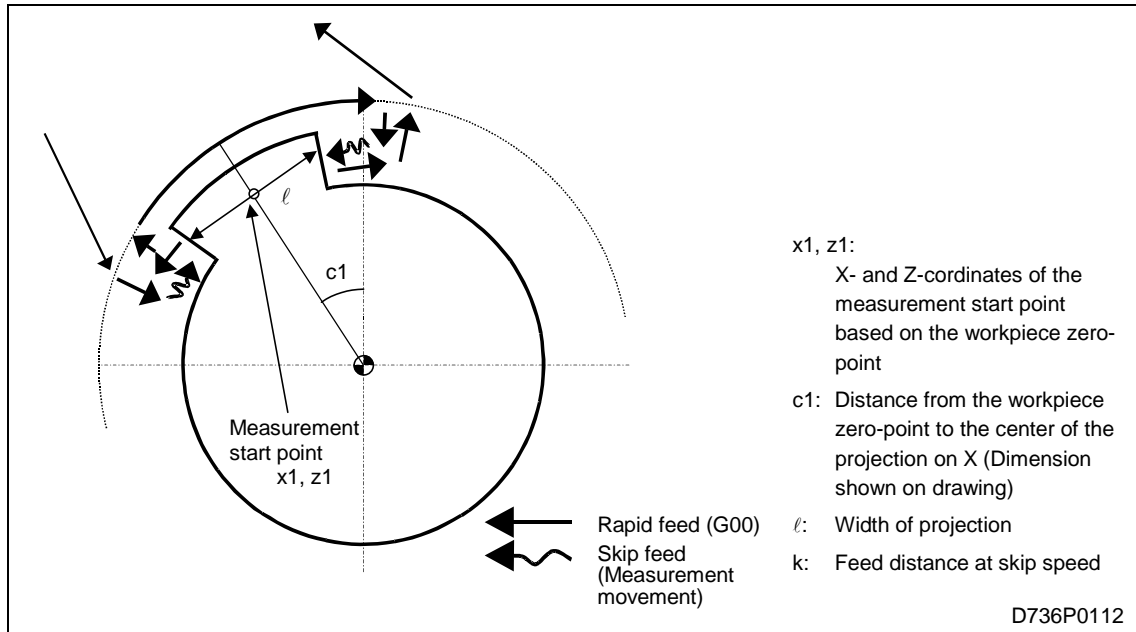
z1 signifies the distance on Z from the workpiece zero-point to the center of the projection width (dimension shown on the drawing).

D. Projection width on C-axis

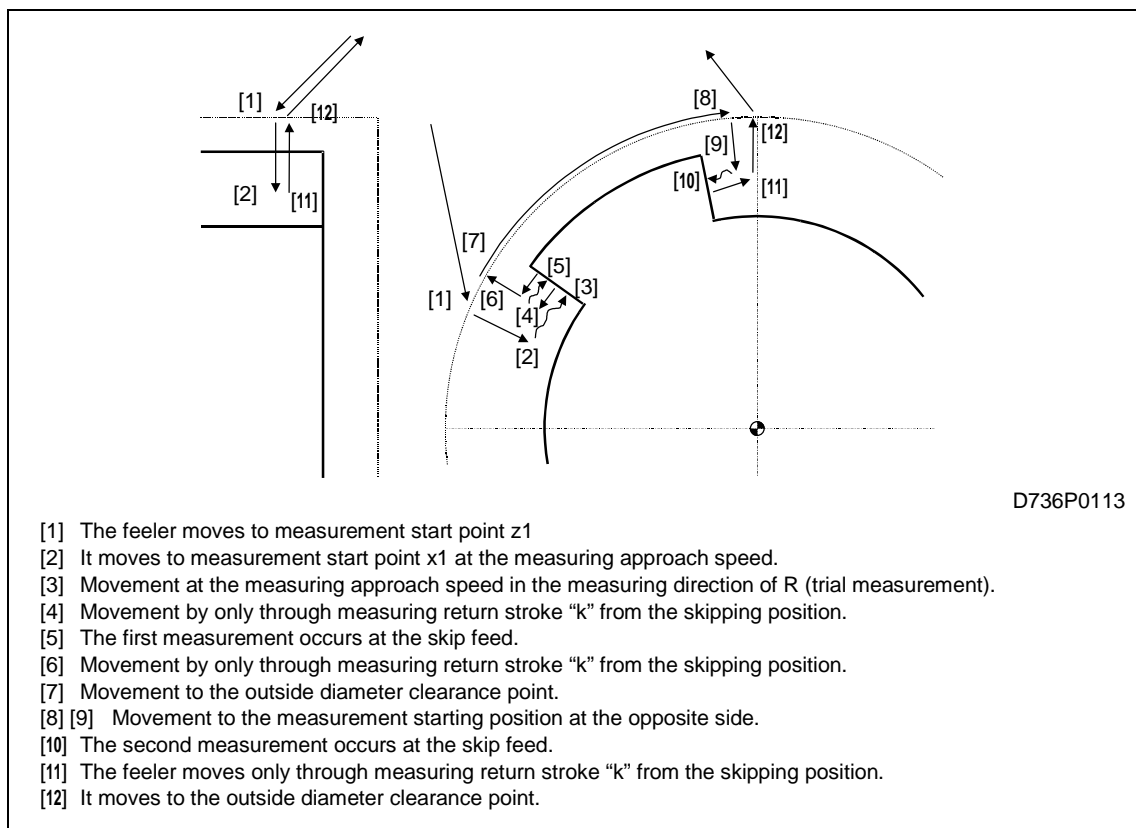
The C-axis fundamental coordinate is corrected by entering the position of the projection center on C-axis in the workpiece coordinates system and the projection width.

SNo.	PTN	X	Y	Z	C	S	R	D/L	K	DIR.
1	C-STEP	x1	◆	z1	c1	0	cw	ℓ	k	←Z

◆: Data are not necessary to set here.



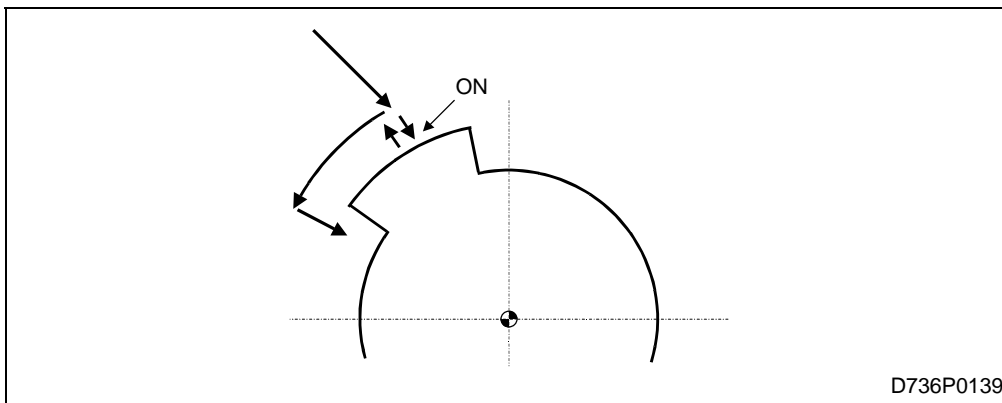
[Measurement movement]



Note: 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.

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.



4. Measurement of boring center

The measurement of boring center is available in two types according to whether the direction of the boring center is on the X-Y or Y-Z-axis.

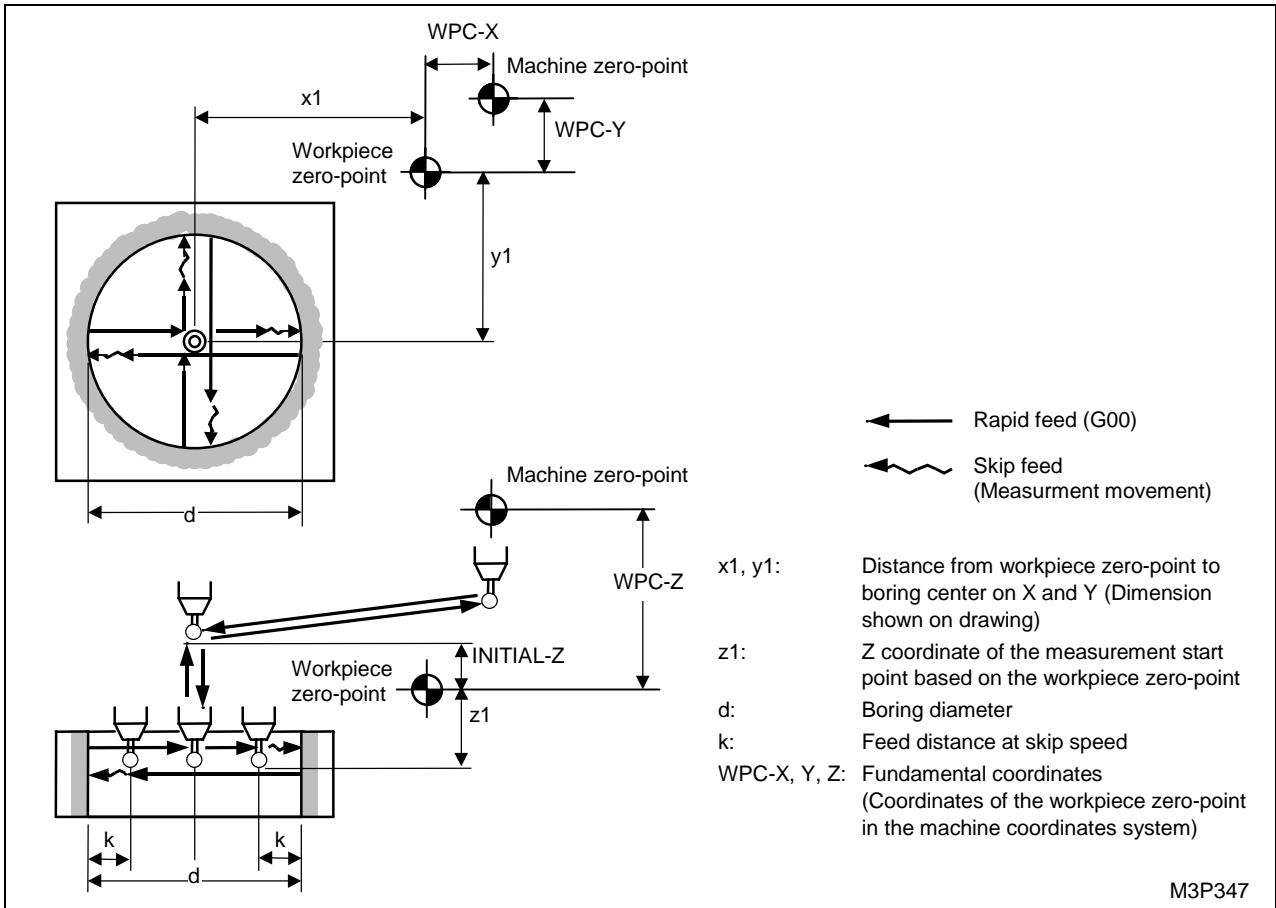
- Boring center on X-Y axis (X-Y BORE)
- Boring center on Y-Z axis (Y-Z BORE)

A. Boring center (X-Y BORE)

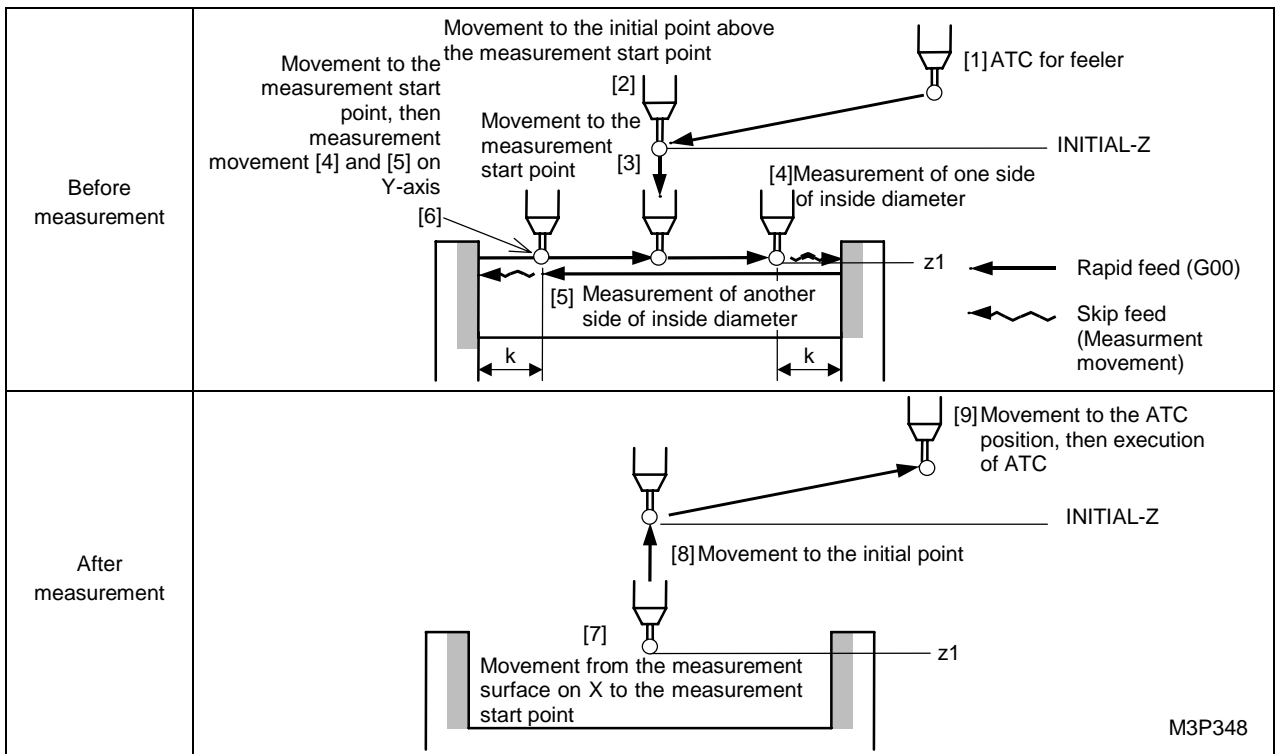
The X- and Y-axial fundamental coordinate are corrected by entering the position of the boring center in the workpiece coordinates system and also the boring diameter.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	BORE X-Y	x1	y1	z1	c1	0	◆	d	k	←Z

◆: Data are not necessary to set here.



[Measurement movement]



Note: The X and Y fundamental coordinates are corrected so that the X and Y coordinates of the boring center to be measured are equal to the coordinates x_1 and y_1 respectively when the feeler is moved to the initial point [8].

B. Boring center (Y-Z BORE)

The Y- and Z-axial fundamental coordinates are corrected by entering the position of the boring center in the workpiece coordinates system and also the boring diameter.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	BORE Y-Z	x1	y1	z1	c1	0	◆	d	k	↓X

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the boring center on the X-Y axis.

y1 and z1 signify the distance on Y and Z from the workpiece zero-point to the center of the boring (dimension shown on the drawing).

5. Measurement of boss center

The measurement of boss center is available in two types according to whether the direction of the boss is on the X-Y or Y-Z axis.

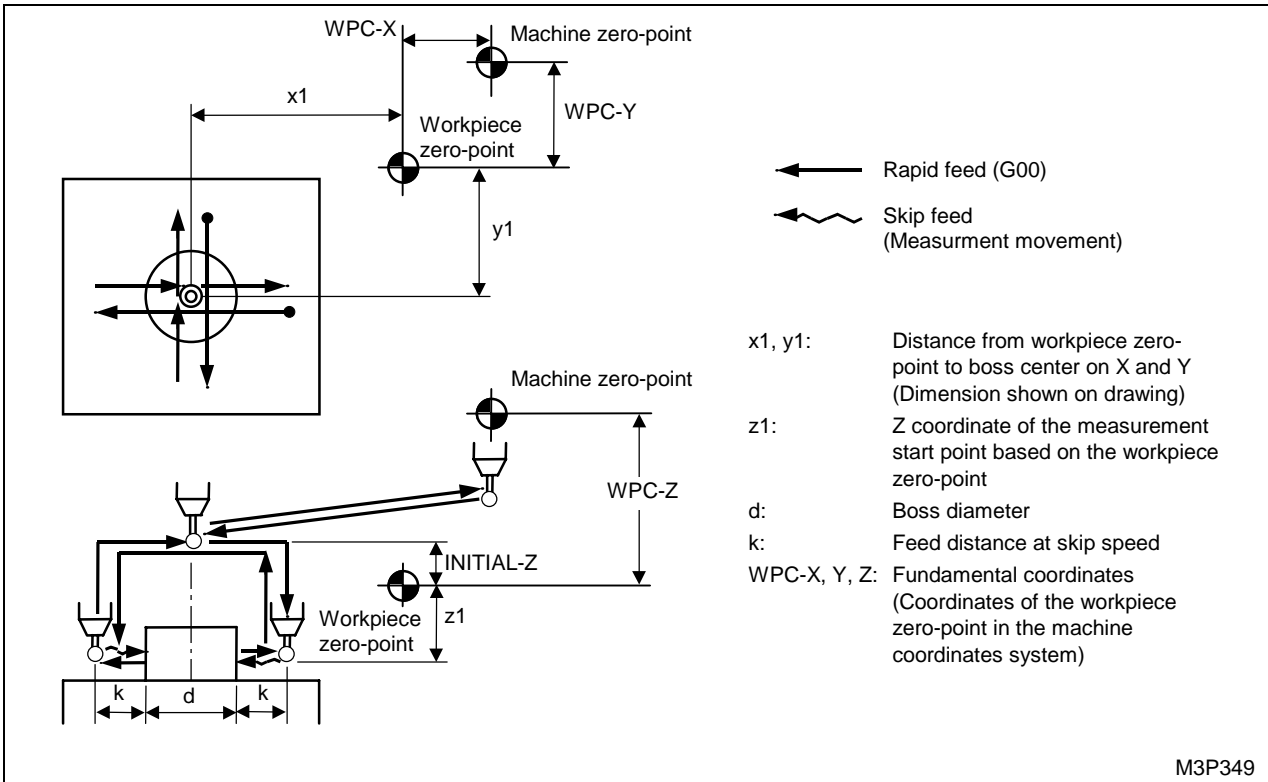
- Boss center on X-Y axis (X-Y BOSS)
- Boss center on Y-Z axis (Y-Z BOSS)

A. Boss center (X-Y BORE)

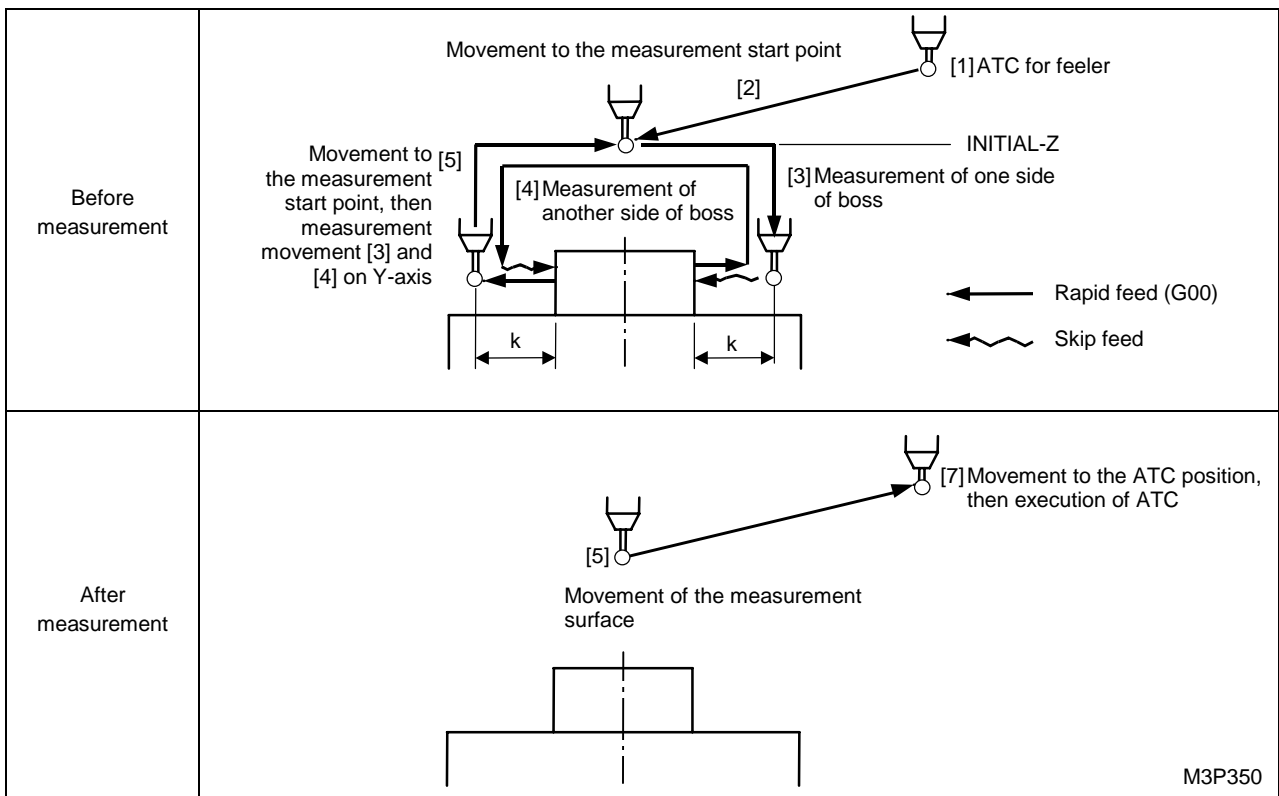
The X- and Y-axial fundamental coordinates are corrected by entering the position of the boss center in the workpiece coordinates system and also the diameter of the boss.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	BOSS X-Y	x1	y1	z1	c1	0	◆	d	k	←Z

◆: Data are not necessary to set here.



[Measurement movement]



Note: The X and Y fundamental coordinates are corrected so that the X and Y coordinates of the boss center to be measured are equal to the coordinates x_1 and y_1 respectively at the time of the movement of the feeler to the initial point [6].

B. Boss center (Y-Z BOSS)

The Y- and Z-axial fundamental coordinates are corrected by entering the position of the boss center in the workpiece coordinates system and also the diameter of the boss.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	BOSS Y-Z	x_1	y_1	z_1	c_1	0	◆	d	k	↓X

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the boring center on the X-Y axis.

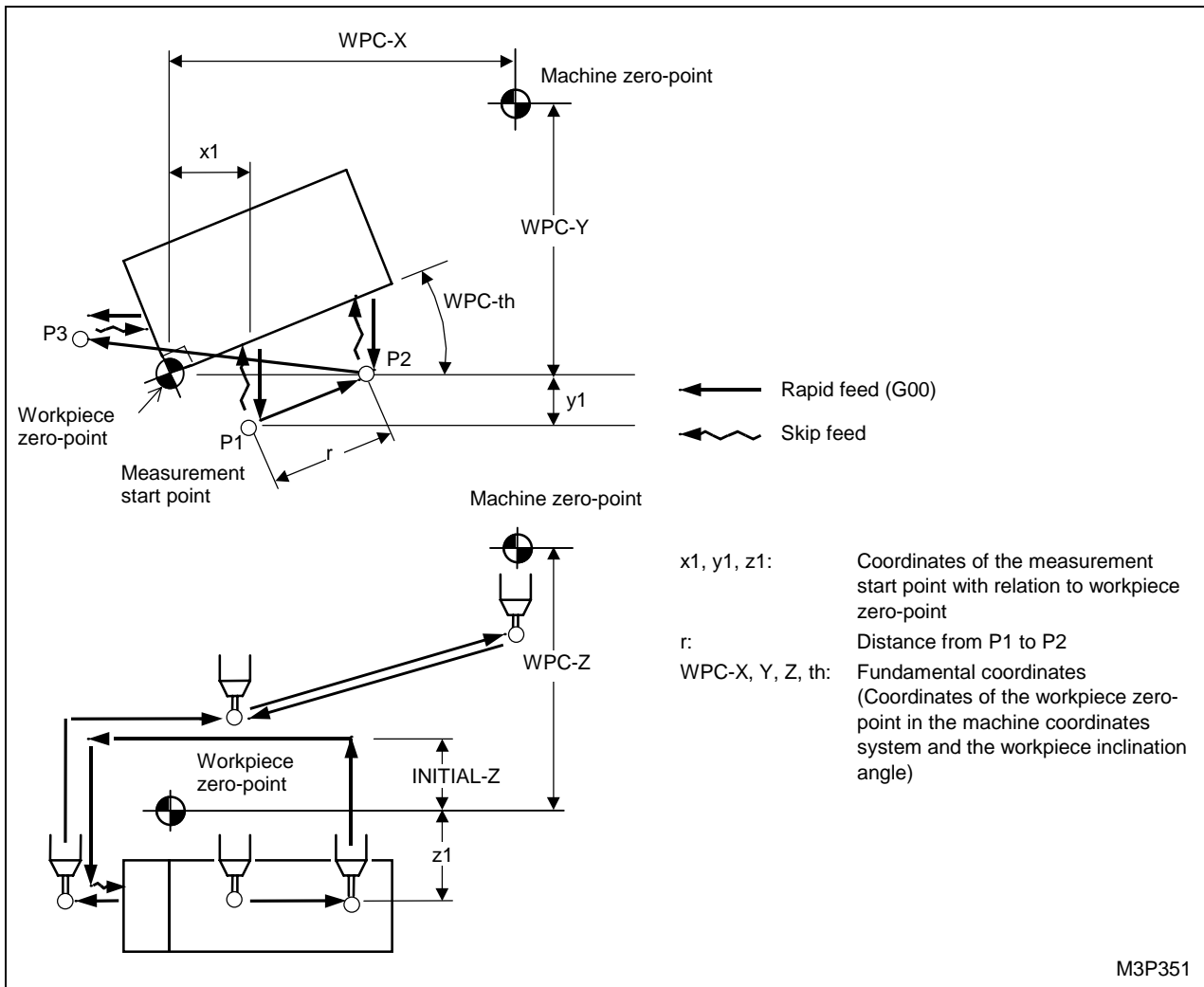
y_1 and z_1 signify the distance on Y and Z from the workpiece zero-point to the center of the boss (dimension shown on the drawing).

6. Measurement of workpiece inclination (X-Y-th. CNR)

The X- and Y-axial fundamental coordinate and the inclination of the workpiece coordinates system are corrected by specifying the workpiece zero-point at the corner of the workpiece to be measured.

SNo.	PTN	X	Y	Z	C	5	R	D/L	K	DIR.
1	XYthCNR	x1	y1	z1	c1	0	r	◆	◆	←Z

◆: Data are not necessary to set here.



A. Direction of movement of the feeler probe

The axial movement direction for measurement of inclination of the workpiece is of four types which are determined automatically according to preset data.

- Direction of movement of the probe on the basis of the position of the workpiece zero-point and also the content of the data.

Article	X	Y	Z	4	R	X	Y	Z	4	R
Data	$x1 > 0$	$y1 > 0$	—	—	$r > 0$	$x1 < 0$	$y1 > 0$	—	—	$r < 0$
[1]						[2]				
[3]						[4]				
<p>Corner where a workpiece zero point is located</p>						M3P352				
Article	X	Y	Z	4	R	X	Y	Z	4	R
Data	$x1 > 0$	$y1 < 0$	—	—	$r > 0$	$x1 < 0$	$y1 < 0$	—	—	$r < 0$

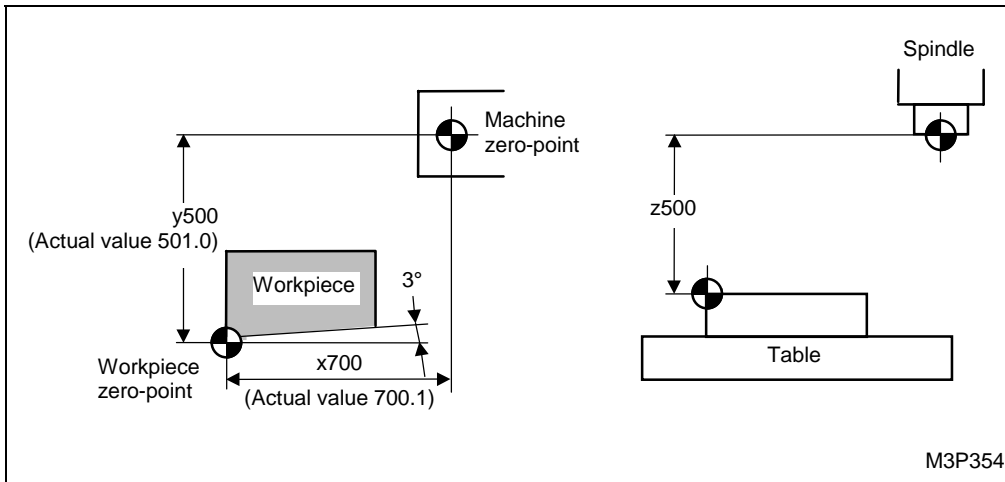
Note: In types [1] to [4] above, the positions related to the points P1 and P2 depend on the sign of r.

B. Measurement movement according to type [3]

Measurement at P1, P2	Measurement at P3	After measurement
M3P353		

Note: The X, Y and th fundamental coordinates are corrected so that the coordinates X and Y of the corner obtained by measurement serve as workpiece zero-point when the feeler is moved to the initial point [9].

Example: Correction of the workpiece zero-point for the workpiece placed as shown below.



- This is a part of program corresponding to MMS.

UNo.	UNIT	ADD.	WPC	X	Y	th	Z	C
1	WPC-0			-700.	-500.	0.	-500.	0.
UNo.	UNIT		TOOL	NOM-φ	No	U.SKIP	\$	
2	MMS		TOL SENS	5.		0.	◆	
SNo.	PTN		X	Y	Z	4	R	D/L
1	XYthCNR		10.	-10.	-10.	0	100	◆ ◆



- The execution of the MMS unit has the effect of correcting the fundamental coordinates as follows.

UNo.	UNIT	ADD.	WPC	X	Y	th	Z	C
1	WPC-0			-700.1	-501.	3.	-500.	0.

Thus the workpiece zero-point is located at the real corner.

C. Positioning of other measuring points P2, P3 other than the measurement start point P1

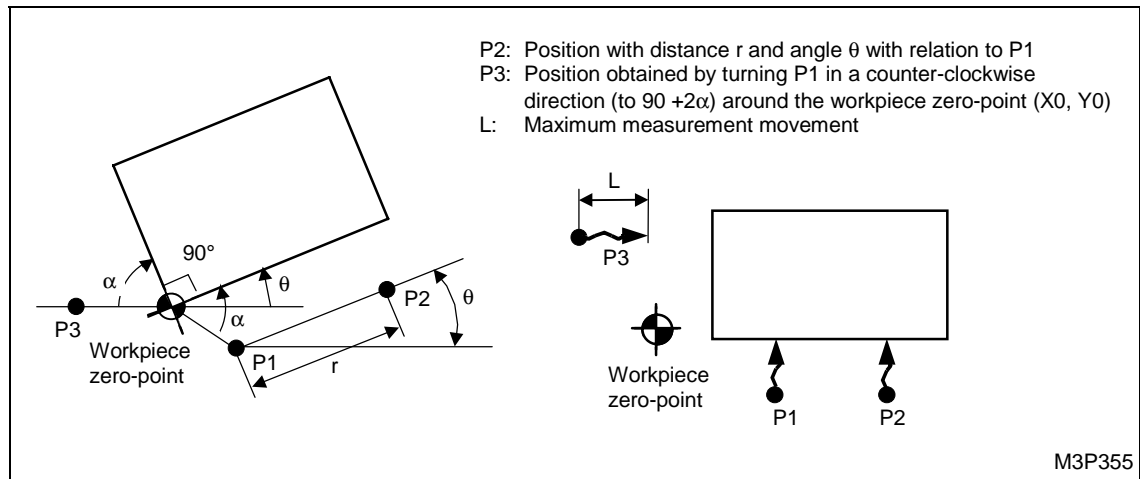
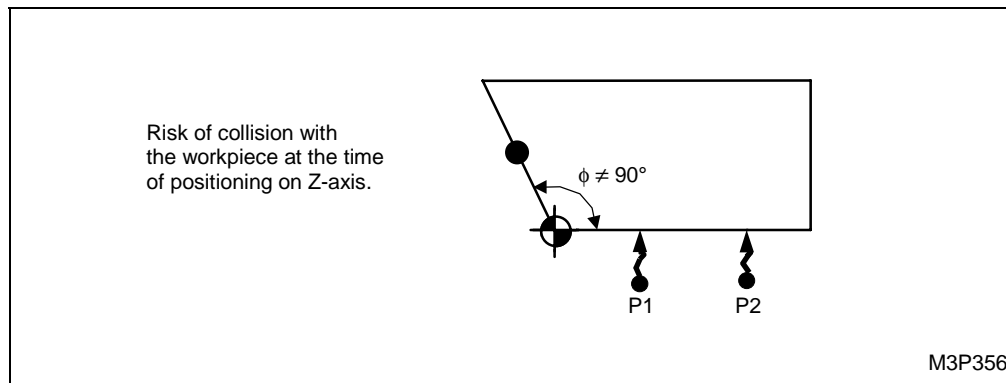


Fig. 4-26 Measurement start point P1 and positions of other measuring points P2 and P3

D. Precautions

The measurement of the inclination of the workpiece requires that the workpiece zero-point to be located at the corner of the workpiece and that the angle of the corner be 90° . If these conditions are not fulfilled, the followings (1), (2) can occur.

- (1) When the workpiece zero-point has been specified at a point distant from the corner of the workpiece, it is possible that the feeler will not come into contact with the workpiece or that the feeler probe will be damaged.
- (2) If the angle of the corner to be measured is not 90° , it is possible that the feeler probe will be damaged. Moreover, the correction of the coordinates is not made correctly for the measured corner.



4-23 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.

Note: 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.

4-23-1 Procedure for selecting workpiece measuring unit

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

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

(2) Press the menu key >>>. The following menu will be displayed.

C-POINT	C-LINE	INDEX	M CODE	SUB	MMS	WORKPICE	TOOL	WORKPICE	>>>
MACH-ING	MACH-ING			PROGRAM		MEASURE	MEASURE	SHAPE	

(3) Press the menu key **WORKPICE MEASURE**.

4-23-2 Setting the unit data in workpiece measuring unit

UNO.	UNIT	PART	COMPENSATE	OFS-TOOL	COMP.DATA	SNS-TOOL	No.	INTERVAL	OUTPUT
	WORK MES					TOL SENS			

Cursor position	Description										
COMPENSATE	Select from the menu whether the measurement results are to be used to correct tool data. <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 10%;">YES</td> <td style="width: 10%;">NO</td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> <td style="width: 10%;"></td> </tr> </table>	YES	NO								
YES	NO										
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.										
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. <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 15%;">DIAMETER</td> <td style="width: 15%;">LENGTH</td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> <td style="width: 15%;"></td> </tr> </table> 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.	DIAMETER	LENGTH								
DIAMETER	LENGTH										
SNS-TOOL	Enter the nominal diameter and identification code of the touch sensor.										
No.	Enter the machining priority number.										
INTERVAL	Specify the intervals at which the workpiece measuring unit is to be executed.										
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.										

4-23-3 Setting the sequence data in workpiece measuring unit

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1											

Cursor position	Description																																	
PTN	<p>Select a workpiece measuring pattern from the menu.</p> <p>A press of the >>> menu key displays menus in the order of [1]→[2] →[3] →[1].</p> <table border="1"> <tr> <td>OUTER X DIA</td> <td>OUTER Y DIA</td> <td>INNER X DIA</td> <td>INTER Y DIA</td> <td></td> <td>X GROOVE</td> <td>Y GROOVE</td> <td>Z GROOVE</td> <td></td> <td>>>></td> <td>[1]</td> </tr> <tr> <td>X WIDTH</td> <td>Y WIDTH</td> <td>Z WIDTH</td> <td>+X STEP</td> <td>-X STEP</td> <td>+Y STEP</td> <td>-Y STEP</td> <td>+Z STEP</td> <td>-Z STEP</td> <td>>>></td> <td>[2]</td> </tr> <tr> <td>INNER GROOVE</td> <td>INNER WIDTH</td> <td>EXT MILLING</td> <td>EXT TURNING</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>>>></td> <td>[3]</td> </tr> </table>	OUTER X DIA	OUTER Y DIA	INNER X DIA	INTER Y DIA		X GROOVE	Y GROOVE	Z GROOVE		>>>	[1]	X WIDTH	Y WIDTH	Z WIDTH	+X STEP	-X STEP	+Y STEP	-Y STEP	+Z STEP	-Z STEP	>>>	[2]	INNER GROOVE	INNER WIDTH	EXT MILLING	EXT TURNING						>>>	[3]
OUTER X DIA	OUTER Y DIA	INNER X DIA	INTER Y DIA		X GROOVE	Y GROOVE	Z GROOVE		>>>	[1]																								
X WIDTH	Y WIDTH	Z WIDTH	+X STEP	-X STEP	+Y STEP	-Y STEP	+Z STEP	-Z STEP	>>>	[2]																								
INNER GROOVE	INNER WIDTH	EXT MILLING	EXT TURNING						>>>	[3]																								
SPT-X SPT-Y SPT-Z	<p>Specify the starting position of measurement.</p> <p>Setup data items differ according to the selected measuring pattern.</p>																																	
FPT-X FPT-Y FPT-Z	<p>Specify the ending position of measurement.</p> <p>Setup data items differ according to the selected measuring pattern.</p>																																	
LIM+	Set the upper-limit value of the tolerance.																																	
LIM-	Set the lower-limit value of the tolerance.																																	
BASE	<p>Set the reference position for measurement.</p> <p>0: The starting position is defined as reference.</p> <p>1: The ending position is defined as reference.</p>																																	
DIR.	<p>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.</p> <table border="1"> <tr> <td>←Z</td> <td>←X</td> <td>↓Z</td> <td>↓X</td> <td>→Z</td> <td>→X</td> <td></td> <td></td> <td></td> <td></td> </tr> </table> <p>Selection of the menu key ← X specifies the milling spindle head facing in the direction of the arrow (left) and the approach/escape in X-axial direction during the measurement.</p> <p style="text-align: right;">D736P0107</p>	←Z	←X	↓Z	↓X	→Z	→X																											
←Z	←X	↓Z	↓X	→Z	→X																													

4-23-4 Selection of a measurement type

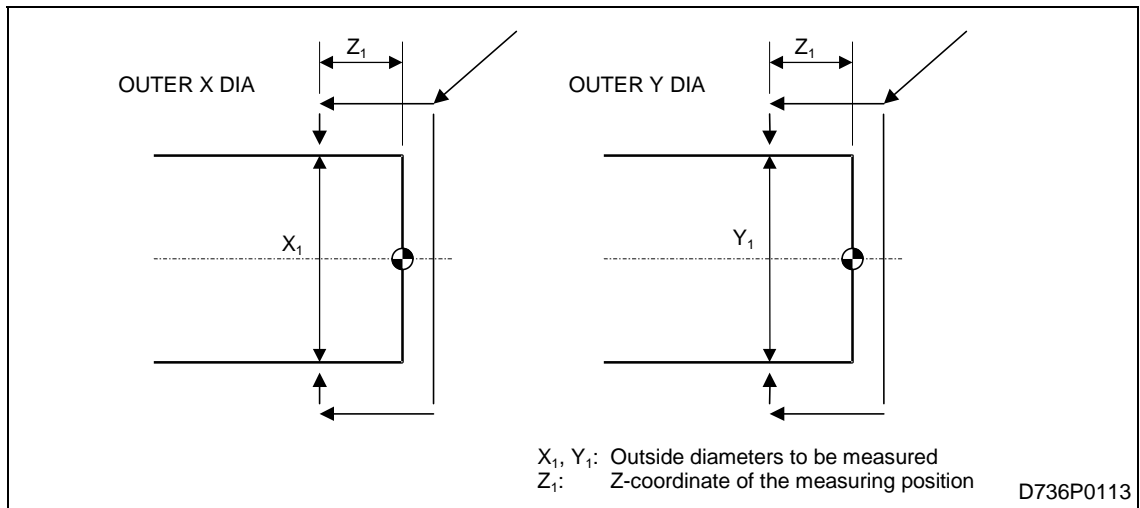
Six types of measurement are provided for the workpiece measurement unit.

- Outside-diameter measurement (OUTER X DIA, OUTER Y DIA)
 To measure the outside-diameter of machined workpiece.
- Inside-diameter measurement (INNER X DIA, INNER Y DIA)
 To measure the inside-diameter of machined workpiece.
- Groove width measurement (X GROOVE, Y GROOVE, Z GROOVE, INNER GROOVE)
 To measure the width of groove or other recesses.
- Protrusion width measurement (X WIDTH, Y WIDTH, Z WIDTH, INNER 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 MILLING, EXT TURNING)
 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.)

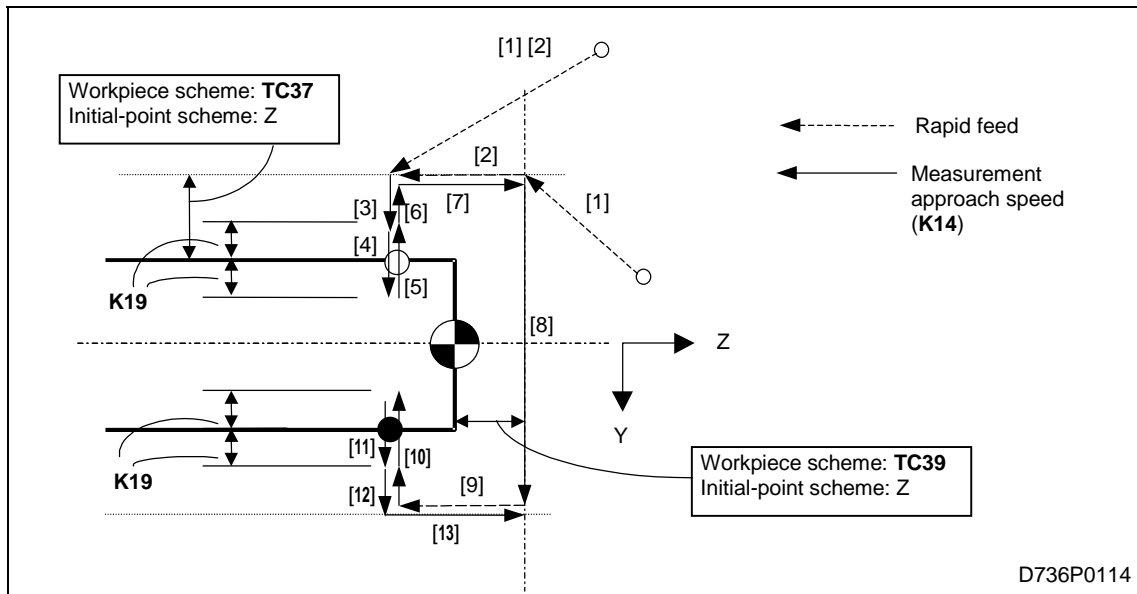
1. Outside-diameter measurement

For **OUTER X DIA**, the X-axial outside diameter of the workpiece is measured, and for **OUTER DIA Y**, the Y-axial outside diameter of the workpiece is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	OUTER X	X ₁	◆	Z ₁	◆	◆	◆	T ₁	T ₂	◆	←Z
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	OUTER Y	◆	Y ₁	Z ₁	◆	◆	◆	T ₁	T ₂	◆	←Z



[Measurement movement]

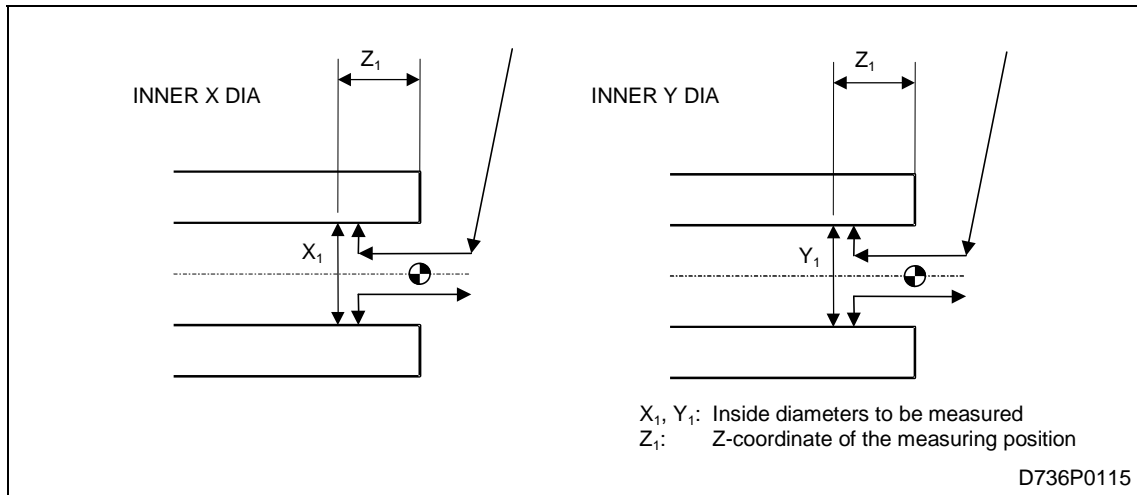


2. Inside-diameter measurement

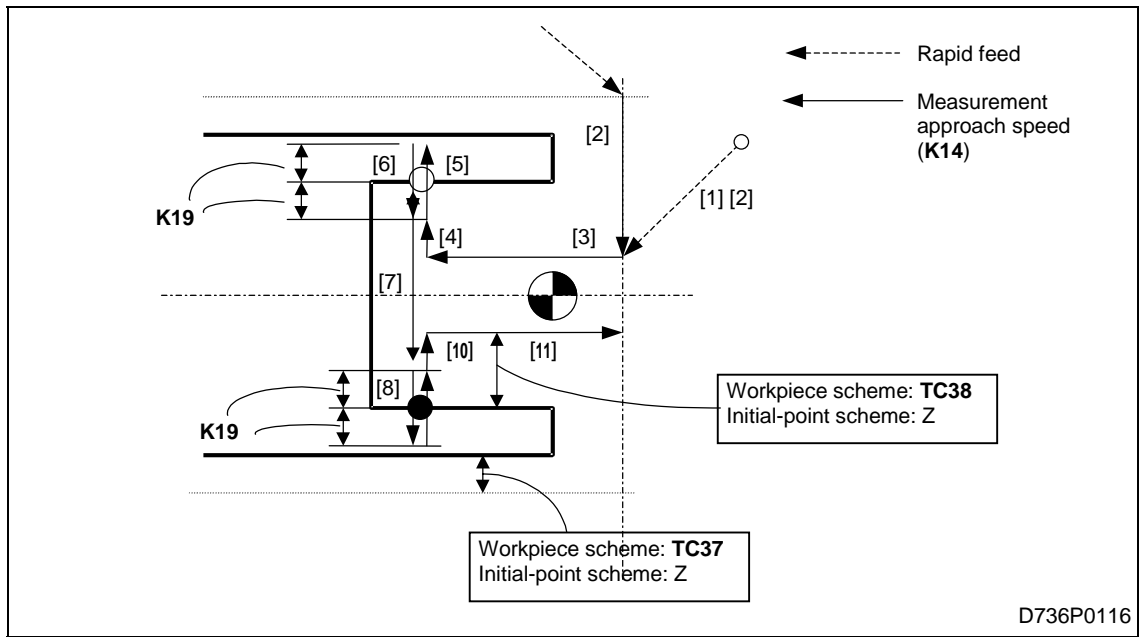
For **INNER X DIA**, the X-axial inside diameter of the workpiece is measured, and for **INNER DIA Y**, the Y-axial inside diameter of the workpiece is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	INNER X	X1	◆	Z ₁	◆	◆	◆	T ₁	T ₂	◆	←Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	INNER Y	◆	Y ₁	Z ₁	◆	◆	◆	T ₁	T ₂	◆	←Z



[Measurement movement]



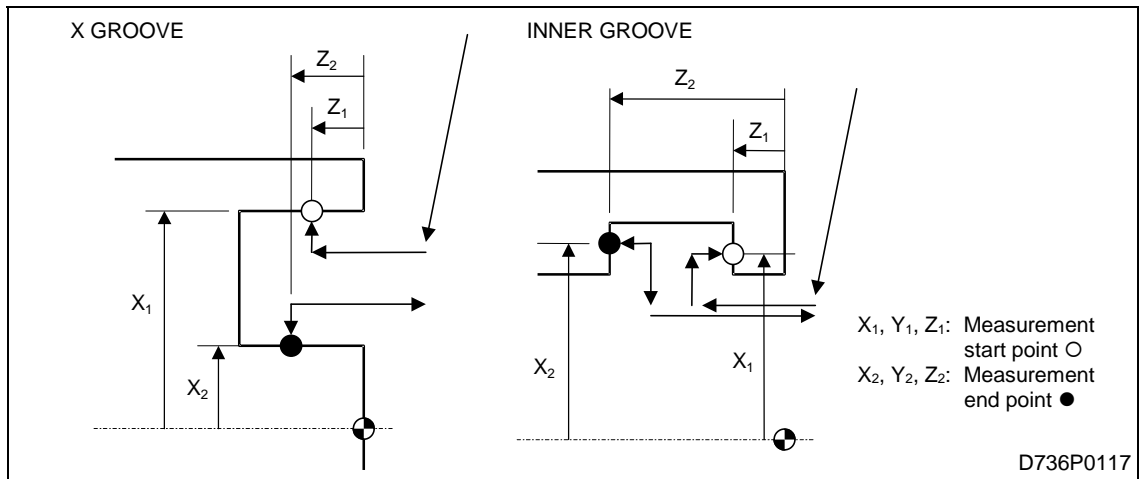
3. Groove width measurement

For **X GROOVE**, **Y GROOVE**, and **Z GROOVE** measurements are performed on X-axial, Y-axial, and Z-axial groove widths, respectively.

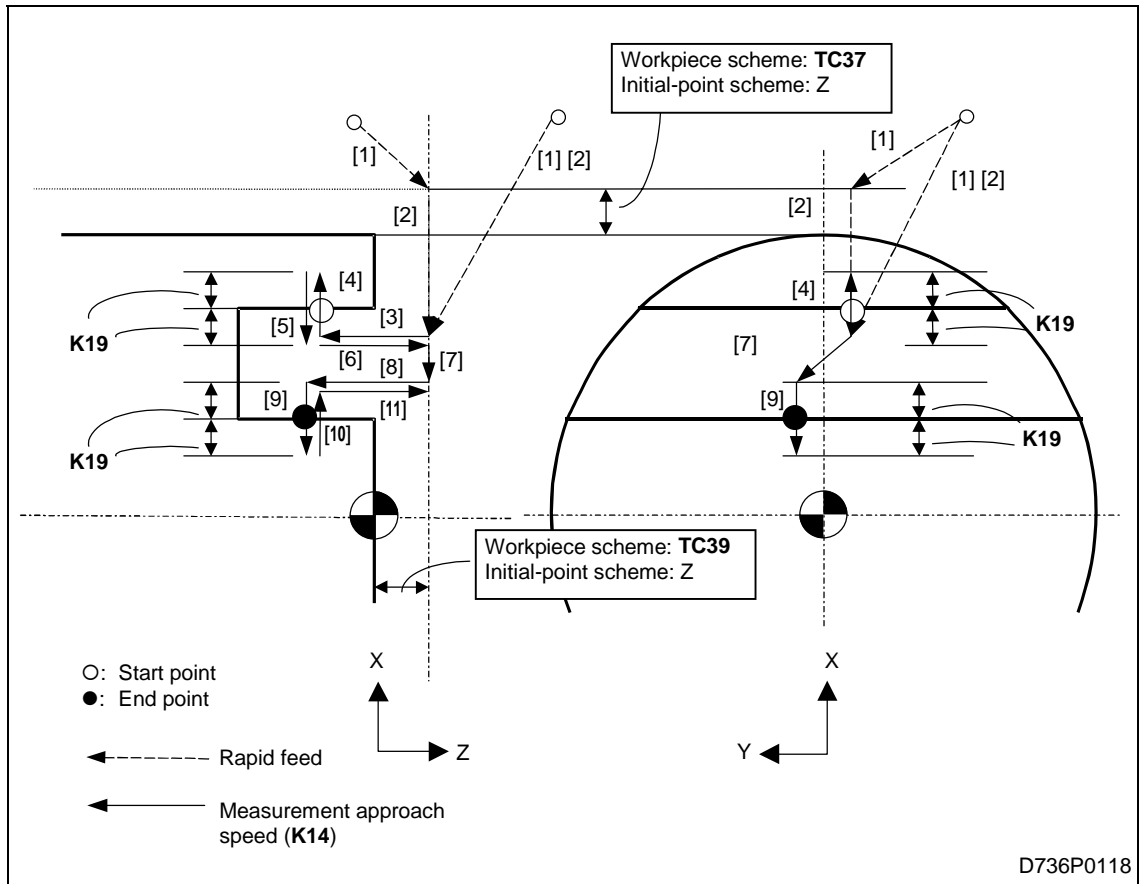
For **INNER GROOVE**, the groove width at the inside diameter side is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	X GROOVE	X ₁	Y ₁	Z ₁	X ₂	Y ₂	Z ₂	T ₁	T ₂	S	←Z

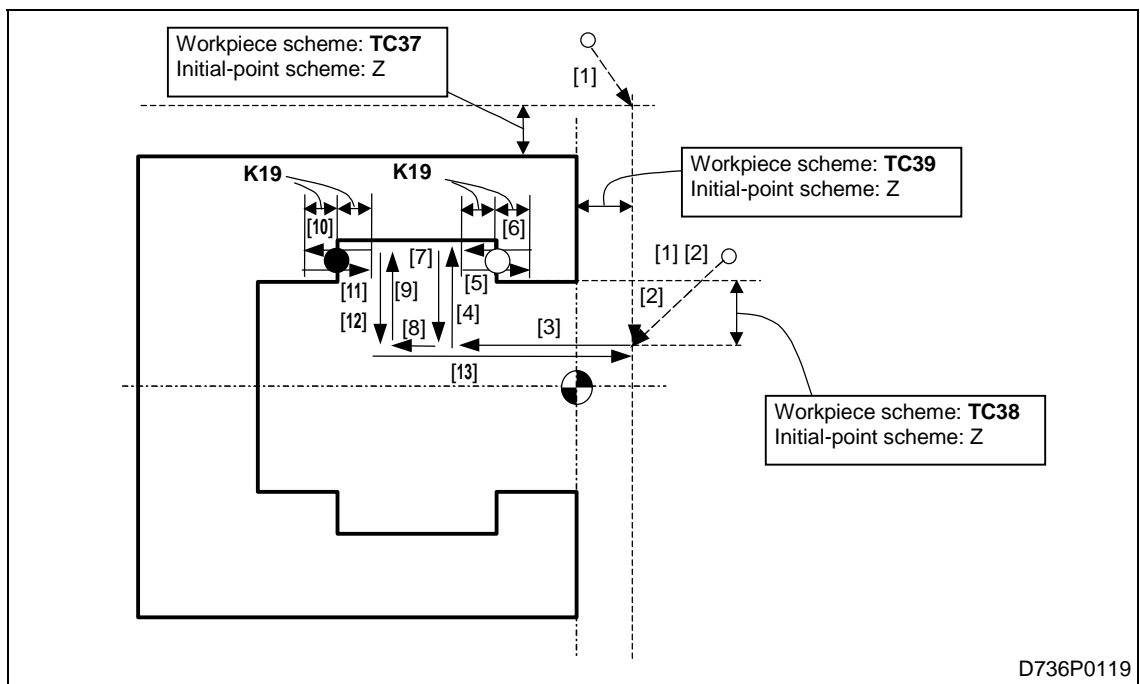
SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	IN GROOVE	X ₁	Y ₁	Z ₁	X ₂	Y ₂	Z ₂	T ₁	T ₂	S	←Z



[Measurement movement (X GROOVE)]



[Measurement movement (INNER GROOVE)]



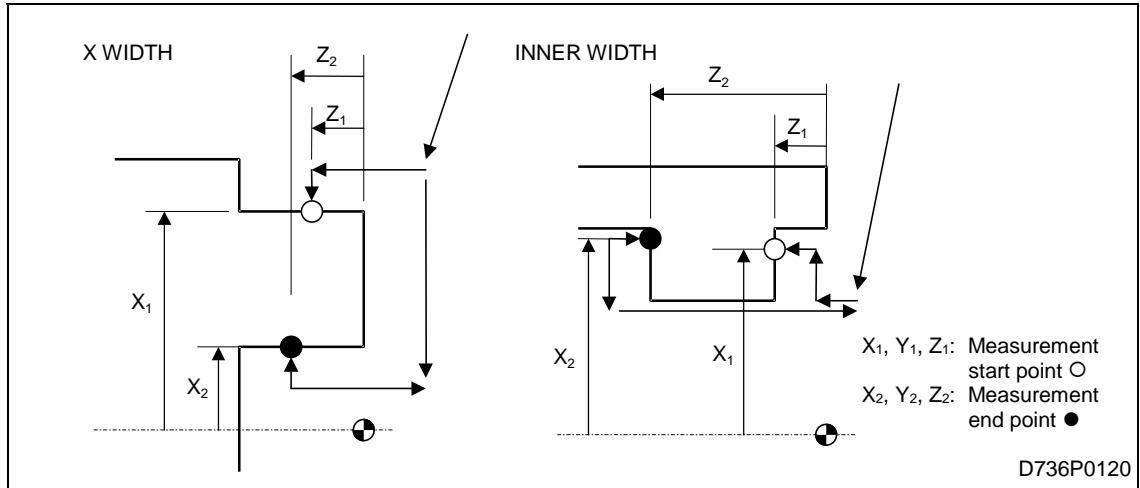
4. Protrusion width measurement

For **X WIDTH**, **Y WIDTH**, and **Z WIDTH** measurements are performed on X-axial, Y-axial, and Z-axial protrusion widths, respectively.

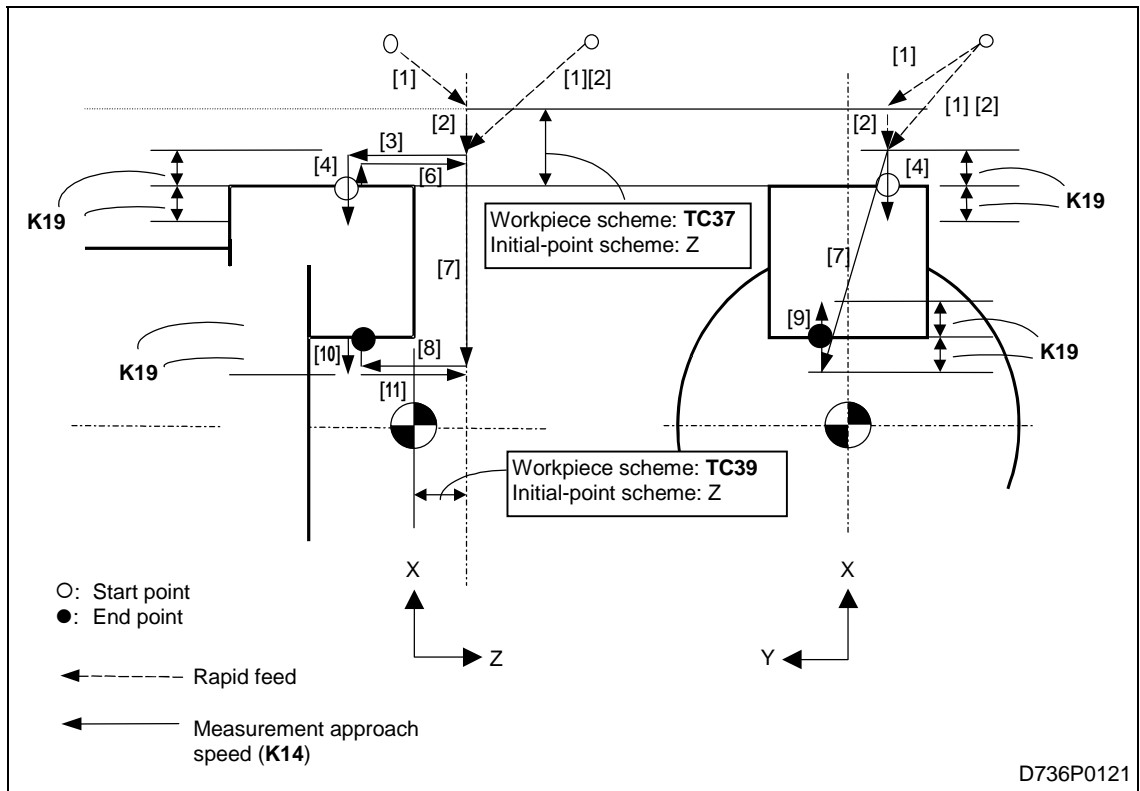
For **INNER WIDTH**, the protrusion width at the inside diameter side is measured.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	X WIDTH	X ₁	Y ₁	Z ₁	X ₂	Y ₂	Z ₂	T ₁	T ₂	S	←Z

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	IN WIDTH	X ₁	Y ₁	Z ₁	X ₂	Y ₂	Z ₂	T ₁	T ₂	S	←Z

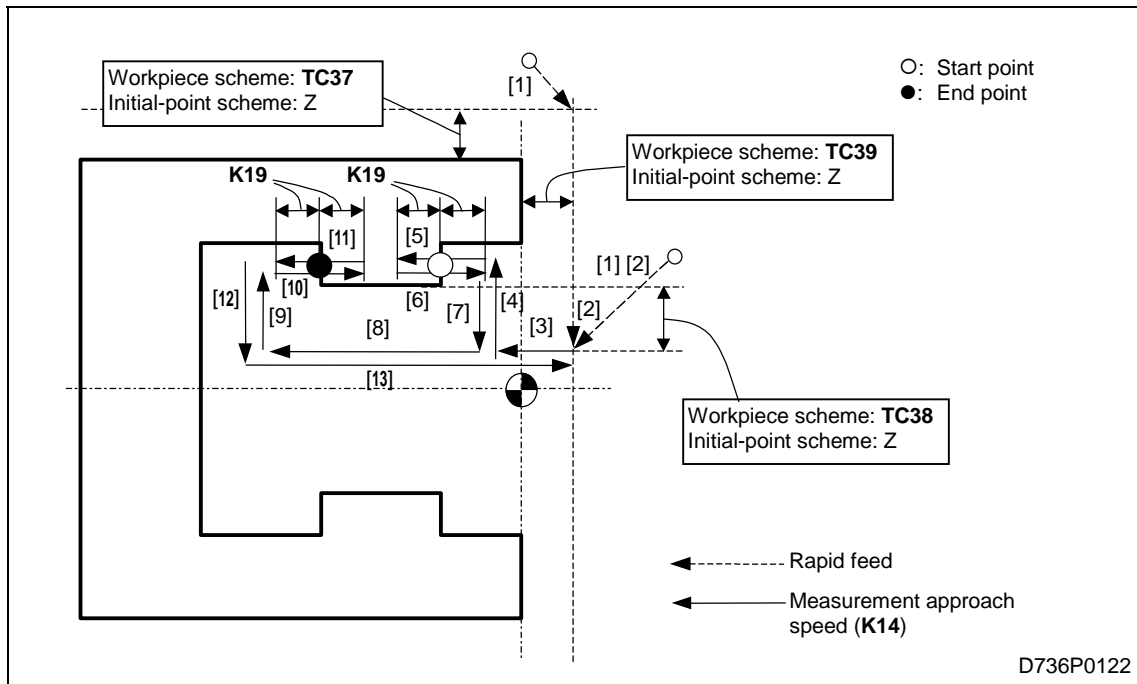


[Measurement movement (X WIDTH)]



Note: The safety profile clearance back (**TC40**) is used for measurement at the No. 2 spindle side.

[Measurement movement (INNER WIDTH)]

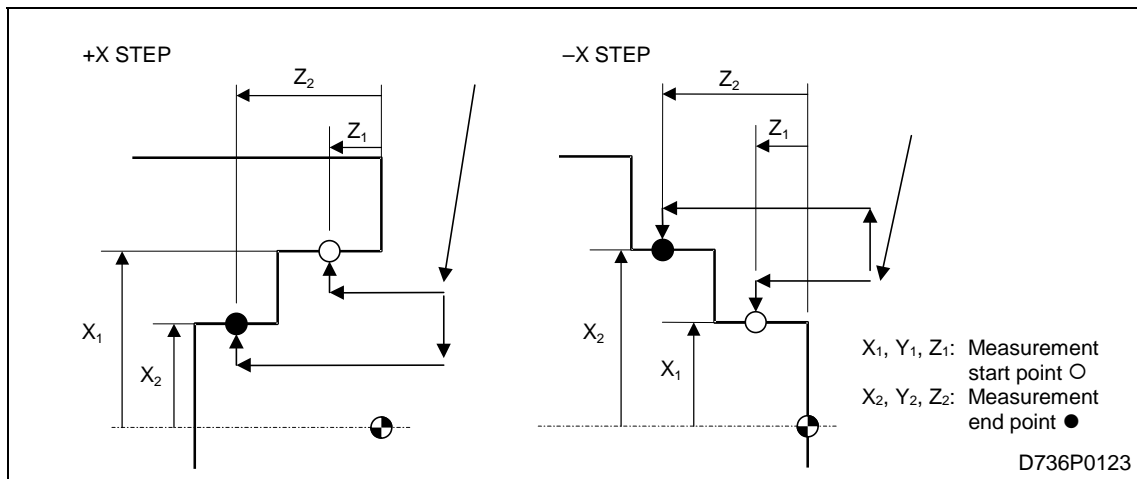


5. Step distance measurement

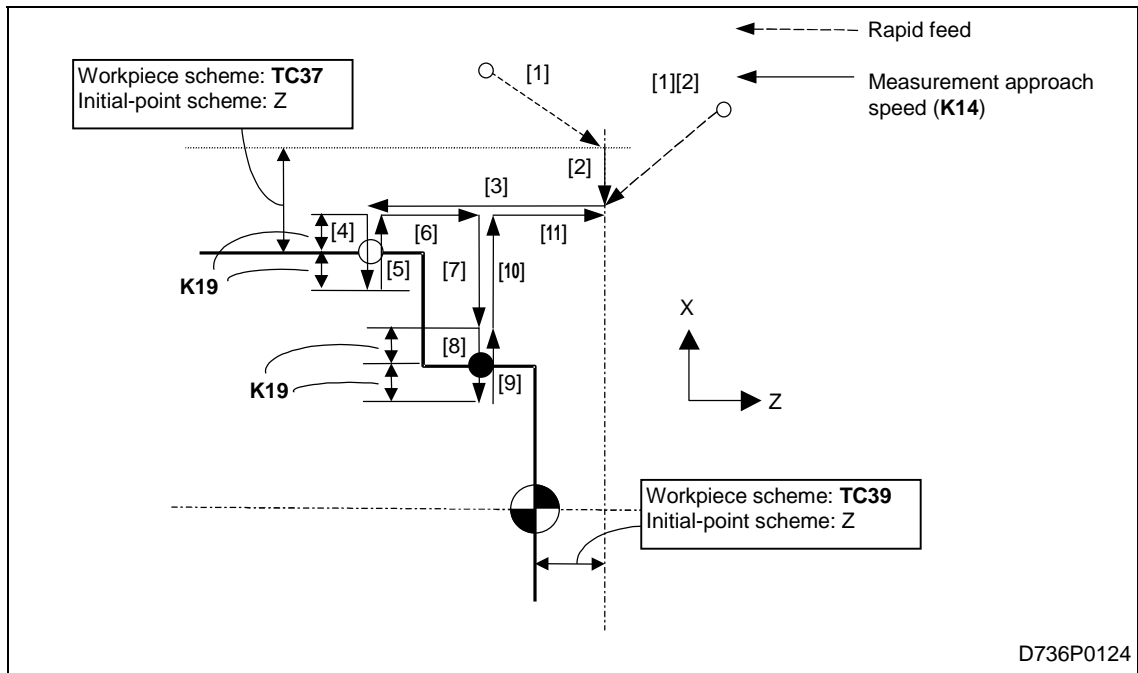
For **+X STEP**, **+Y STEP**, and **+Z STEP**, surface level differences in the plus directions of the X-axis, Y-axis, and Z-axis, respectively, are measured.

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.

SN0.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	+X STEP	X ₁	Y ₁	Z ₁	X ₂	Y ₂	Z ₂	T ₁	T ₂	S	← Z



[Measurement movement (-X STEP)]



6. External measurement

For **EXT MILLING**, data that has been measured using an external measuring unit is read and the measured data is incorporated into milling tool data.

For **EXT TURNING**, measured data is incorporated into turning tool data.

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	EXT MIL	◆	N	Z ₁	◆	◆	◆	T ₁	T ₂	◆	◆

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT- X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	DIR.
1	EXT TURN	P	N	Z ₁	◆	◆	◆	T ₁	T ₂	◆	◆

P: Select a Remote turning (**EXT TURNING**) correction target from the following menu:

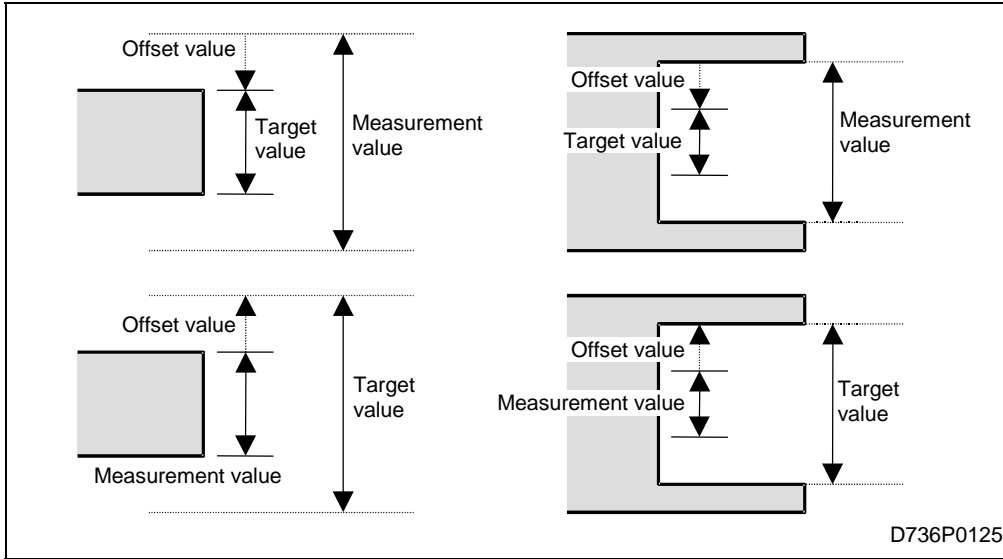
WEAR	WEAR								
X	Z								

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.

4-23-5 Offset value and the direction of offset

1. Outside and inside diameter measurement



$$\text{Target value X (Y)} = | \text{Starting position X (Y)} |$$

$$\text{Measurement value X (Y)}$$

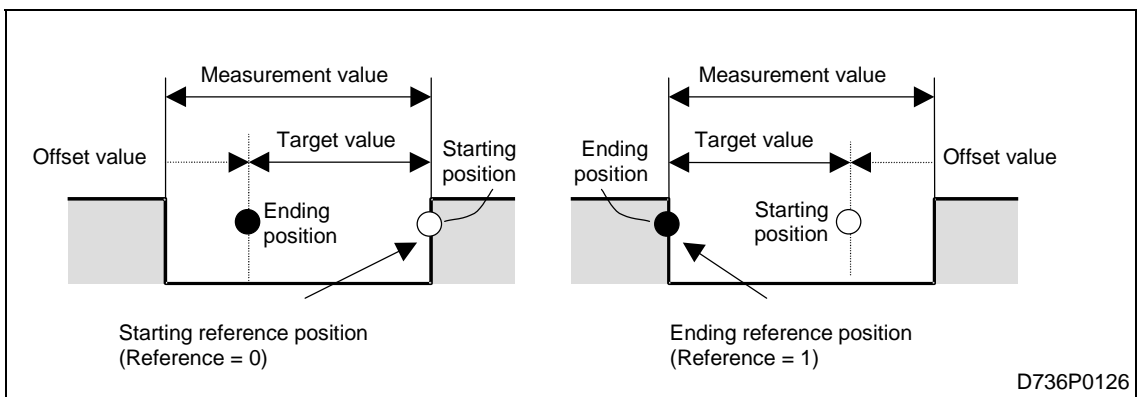
$$= | \text{Measured value \#1} - \text{Measured value \#2} | - 2 \times \text{Touch sensor stylus ball radius X (Y)}$$

$$\text{Offset value X (Y)} = \text{Target value X (Y)} - \text{Measurement value X (Y)}$$

[Offset for outside- and inside diameter measurement]

	Measuring direction	Offset
Turning tool	X	Wear offset value X = Wear offset value X + Offset value X
	Y	Wear offset value X = Wear offset value X + Offset value Y

2. Groove width and inner groove width measurements



$$\text{Target value X (YZ)} = | \text{Starting position X(YZ)} - \text{Ending position X(YZ)} |$$

$$\text{Measurement value X (YZ)}$$

$$= | \text{Measured value \#1} - \text{Measured value \#2} | + 2 \times \text{Touch sensor stylus ball radius X(Y)}$$

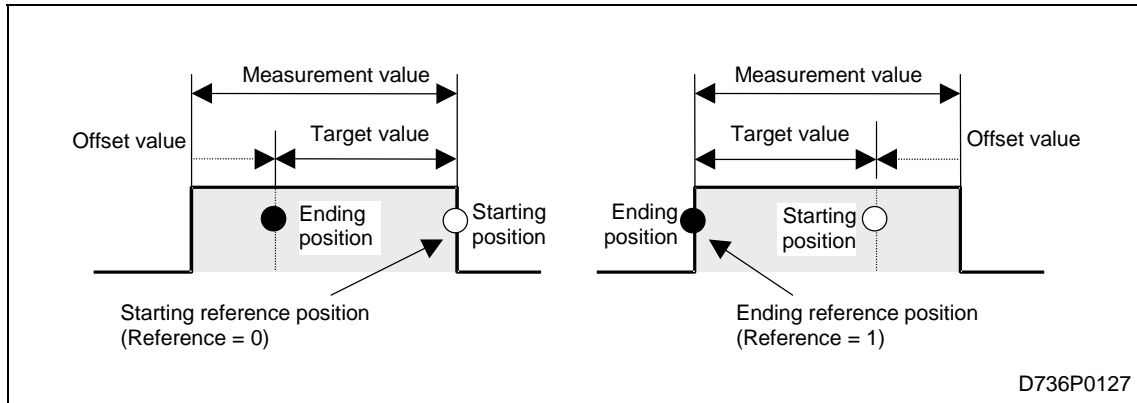
$$\text{Offset value X(YZ)} = \text{Target value X(YZ)} - \text{Measurement value X(YZ)}$$

The value "X(L3)" is used as the Z-axial radius of the touch sensor stylus ball.

[Offset for groove width measurement]

	Measuring direction	Offset	Starting reference position (Reference = 0)	Ending reference position (Reference = 1)
Turning tool	X	Wear offset value X =	Wear offset value X - Offset value X	Wear offset value X + Offset value X
	Y	Wear offset value X =	Wear offset value X - Offset value Y	Wear offset value X + Offset value Y
	Z	Wear offset value Z =	Wear offset value Z - Offset value Z	Wear offset value Z + Offset value Z
Milling tool		Tool diameter =	Tool diameter + Offset value / 2	

3. Protrusion width and inner protrusion width measurements



Target value X (YZ) = | Starting position X (YZ) – Ending position X (YZ) |

Measurement value X (YZ)

= | Measured value #1 – Measured value #2 | – 2 × Touch sensor stylus ball radius X (Y)

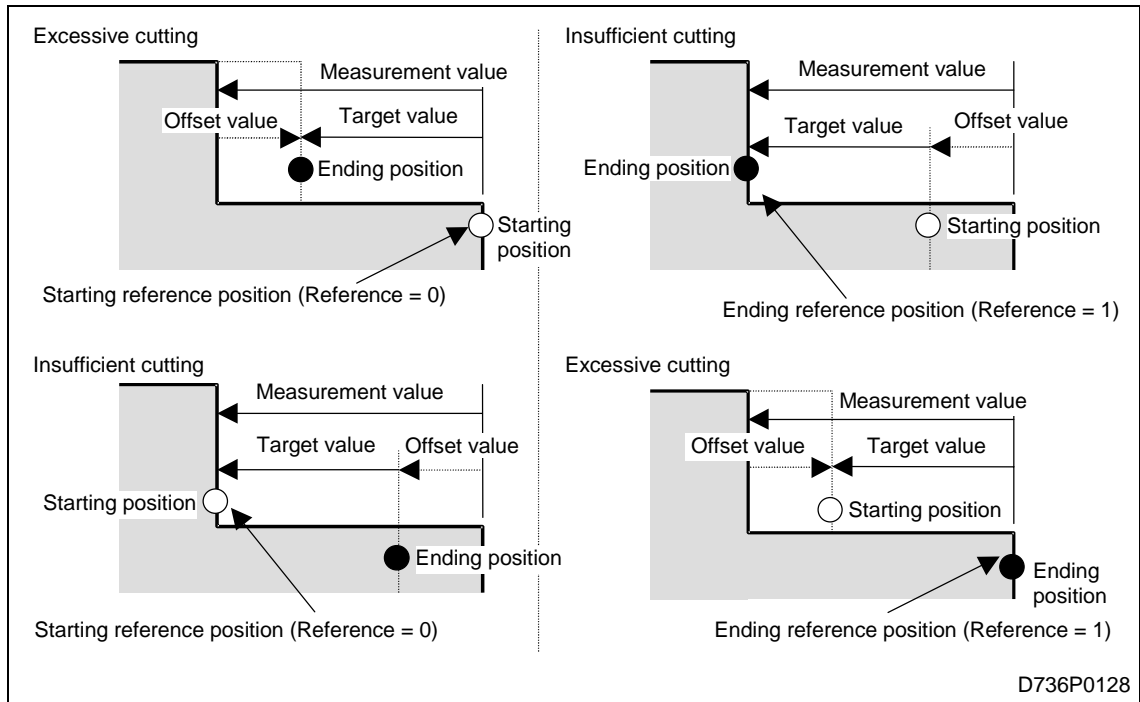
Offset value X (YZ) = Target value X (YZ) – Measurement value X (YZ)

The value “X (L3)” is used as the Z-axial radius of the touch sensor stylus ball.

[Offset for protrusion width measurement]

	Measuring direction	Offset	Starting reference position (Reference = 0)	Ending reference position (Reference = 1)
Turning tool	X	Wear offset value X =	Wear offset value X - Offset value X	Wear offset value X + Offset value X
	Y	Wear offset value X =	Wear offset value X - Offset value Y	Wear offset value X + Offset value Y
	Z	Wear offset value Z =	Wear offset value Z - Offset value Z	Wear offset value Z + Offset value Z
Milling tool		Tool diameter =	Tool diameter – Offset value / 2	

4. Step distance measurement



$$\text{Target value X (YZ)} = | \text{Starting position X (YZ)} - \text{Ending position X (YZ)} |$$

$$\text{Measurement value X (YZ)} = | \text{Measured value \#1} - \text{Measured value \#2} |$$

$$\text{Offset value X (YZ)} = \text{Target value X (YZ)} - \text{Measurement value X (YZ)}$$

[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 value X =	Wear offset value X - value X	Wear offset value X + value X
	Y	Wear offset value X =	Wear offset value X - value Y	Wear offset value X + value Y
	Z	Wear offset value Z =	Wear offset value Z - value Z	Wear offset value Z + value Z
Milling tool		Tool length =	Tool length - Offset value	Tool length + Offset value
		Tool diameter =	Tool diameter + Offset value / 2	

5. External measurement

Target value = Value specified in the workpiece measuring sequence

$$\text{Offset value X} = \text{Target value} - \text{Measurement value X}$$

$$\text{Offset value Y} = \text{Target value} - \text{Measurement value Y}$$

$$\text{Offset value Z} = \text{Target value} - \text{Measurement value Z}$$

$$\text{Tool length offset value} = \text{Target value} - \text{Measured tool length}$$

$$\text{Tool diameter offset value} = \text{Target value} - \text{Measured tool diameter}$$

[Offset for remote measurement]

	Object of offset	Offset
Turning tool	Wear offset value X =	Wear offset value X + Offset value X
	Wear offset value Y =	Wear offset value Y + Offset value Y
	Wear offset value Z =	Wear offset value Z + Offset value Z
Milling tool	Tool length =	Tool length + Offset value
	Tool diameter =	Tool diameter + Offset value / 2

4-23-6 Correction judgment

Correction data that has been obtained from measurements is handled as follows:

[Correction judgment conditions]

Condition	Result
Correction value > $\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18}$	An alarm is displayed.
$\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18} \geq$ Correction value $\geq \frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17}$	The correction is performed.
$\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17} >$ Correction value > $-\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17}$	The correction is not performed.
$-\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K17} \geq$ Correction value $\geq -\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18}$	The correction is performed.
$-\frac{\text{Tolerance upper} - \text{Tolerance lower}}{100} \times \mathbf{K18} >$ Correction 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 value, and tolerance upper/tolerance lower.

4-23-7 Retry function and an alarm

If the correction value enters an alarm region, measurement will be repeated the number of times specified in the parameter. If the situation does not improve even so, the following alarm will be displayed:

157 MEASURED RESULT MALFUNCTION

The retry function is executed the number of times set as "Retry count" in parameter **K23**. If the setting of the parameter is 0, the retry function is not executed and the above alarm is immediately displayed.

4-24 Tool Measuring Unit (TOOL MES)

The tool measuring unit sets tool lengths or diameters automatically.

4-24-1 Procedure for selecting tool measuring unit

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

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

- (2) Press the menu key >>>. The following menu will be displayed.

C-POINT	C-LINE	INDEX	M CODE	SUB	MMS	WORKPIECE	TOOL	WORKPIECE	>>>
MACH-ING	MACH-ING			PROGRAM		MEASURE	MEASURE	SHAPE	

- (3) Press the menu key **TOOL MEASURE**.

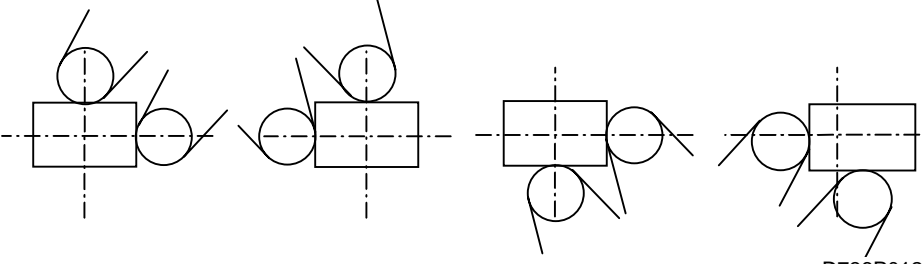
4-24-2 Setting the unit data in tool measuring unit

UNo.	UNIT	COMPENSATE	SNS-TOOL	No.	INTERVAL	OUTPUT
	TOOL MES					

Cursor position	Description										
COMPENSATE	Select from the menu whether the measurement results are to be used to correct tool data. <table border="1" style="width: 100%; text-align: center;"> <tr> <td>YES</td> <td>NO</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> </table>	YES	NO								
YES	NO										
SNS-TOOL	Enter the nominal diameter (nominal size) and identification code of the touch sensor.										
No.	Enter the machining priority number.										
INTERVAL	Specify the intervals at which the workpiece measuring unit is to be executed.										
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 .										

4-24-3 Setting the sequence data in tool measuring unit

SNo.	PTN	TOL.LENGTH/X	TOL.DIA./Z	UNIT	DIR.
1					

Cursor position	Description									
PTN	<p>Select a tool measuring pattern from the menu.</p> <table border="1"> <tr> <td>LASER MEASURE</td> <td>TOOL EYE #1</td> <td>TOOL EYE #2</td> <td>TOOL EYE #3</td> <td>TOOL EYE #4</td> <td></td> <td></td> <td></td> <td></td> </tr> </table> <p>Specify the following measuring directions for the TOOL EYE: #1. O. D., Forward #2. O. D., Reverse #3. I. D., Forward #4. I. D., Reverse</p>  <p style="text-align: right;">D736P0129</p>	LASER MEASURE	TOOL EYE #1	TOOL EYE #2	TOOL EYE #3	TOOL EYE #4				
LASER MEASURE	TOOL EYE #1	TOOL EYE #2	TOOL EYE #3	TOOL EYE #4						
TOL. LENGTH/X	<p>Enter the tool length or the X-axial tolerance upper-limit value. If the tool length or the X-axial tolerance is not entered, tool length or X-axial measurement will not occur.</p>									
TOL. DIA./Z	<p>Enter the tool diameter or the Z-axial tolerance upper-limit value. If the tool diameter or the Z-axial tolerance is not entered, tool diameter or Z-axial measurement will not occur.</p>									
UNIT	<p>Enter 0 to retract the tool nose measuring table after execution of the measuring unit, or enter 1 if the measuring table is not to be retracted.</p> <p>When the noses of multiple tools are to be measured in succession, the measuring time can be minimized by entering 1, since the measuring table does not need to be extended or retracted each time.</p> <p>However, enter 0 for the last tool whose nose is to be measured using the nose measuring unit. If 1 remains entered, the program will be completed with the measuring table extended.</p>									
DIR.	<p>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.</p> <table border="1"> <tr> <td>←</td> <td>↓</td> <td>→</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> </table> <p>Selection of the menu item ← specifies the approche/escape in left direction during the measurement.</p>	←	↓	→						
←	↓	→								

4-24-4 Measuring patterns

The following measuring patterns are available for tool measurement:

- Laser measurement

The tool lengths or diameters of milling tools are measured.

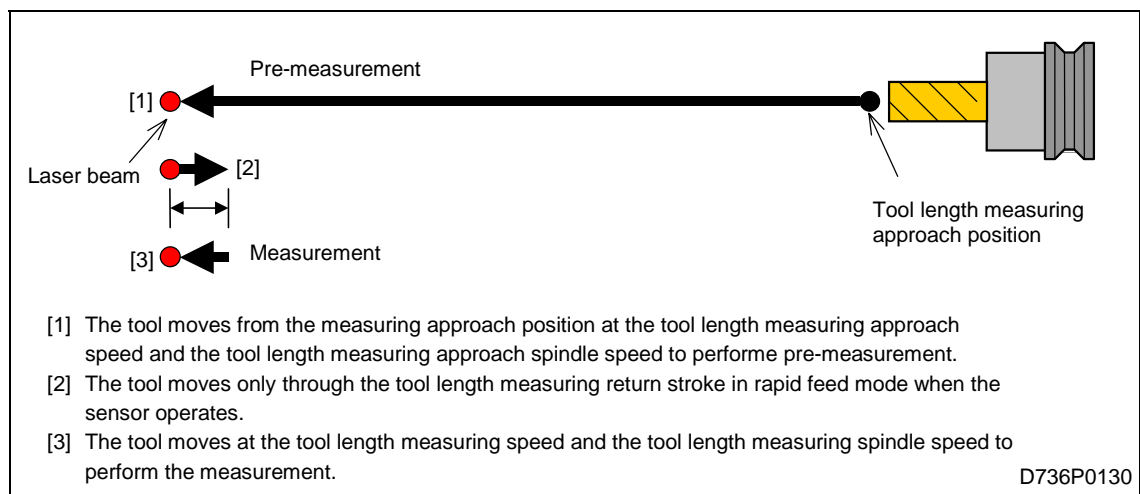
The types of tools whose diameters can be measured are limited to end-milling tools, face-milling tools, special tools, and ball end-milling tools.

- TOOL EYE measurement

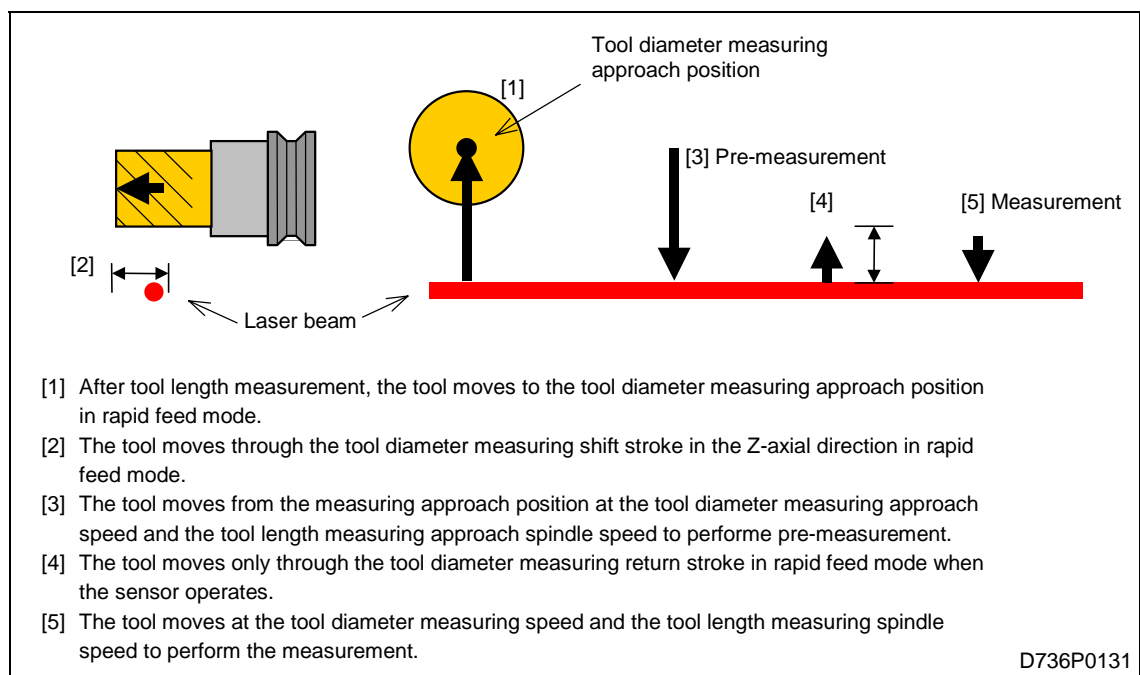
Tool lengths A or diameters B of turning tools are measured.

1. Tool paths during laser measurement

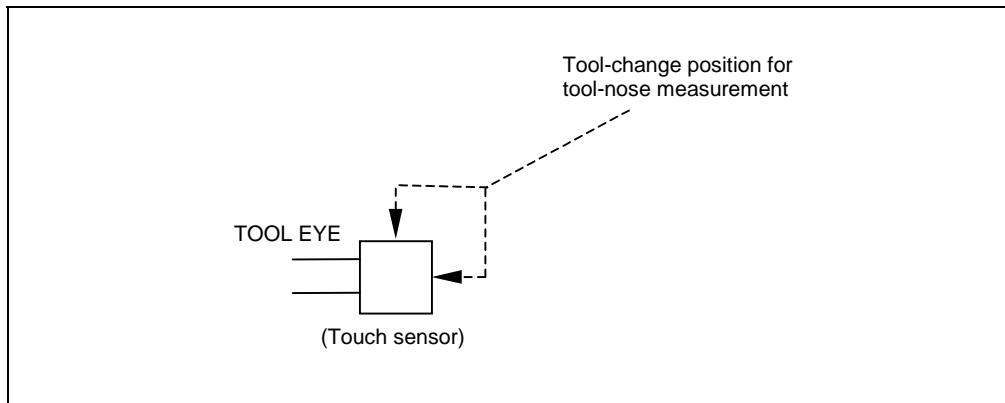
A. Tool length measurement



B. Tool diameter measurement



C. Tool path during TOOL EYE measurement



Note: The moment that the tool-change position for tool-nose measurement is reached, the TOOL EYE begins to advance. This must be considered when setting the parameter for tool-nose measurement tool-change position to prevent collision between TOOL EYE and tool.

5 PRIORITY FUNCTION FOR THE SAME TOOL

The program is executed by numeric order from its head. Consequently, the ATC is repeated for each of the tools specified in the tool sequence. This priority function for the same tool is intended to reduce the number of ATCs 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 sequence that the priority number can be specified.

1. Tool sequence of machining unit
2. Manual program unit (In the case of absence of a tool, the priority number can not be specified.)
3. MMS unit
4. M-code unit

5-1 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 face mill, 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).

5 PRIORITY FUNCTION FOR THE SAME TOOL

Program without priority number

UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE				
0	CBN STL	ONLY MILL.	50.	1	OFF				
UNo.	UNIT	ADD. WPC	X	Y	th				
1	WPC-1		-300.	-300.	0.				
UNo.	UNIT	DEPTH	SRV-Z	BTM					
2	FCE MILL	0.	5.	1					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD		
1	FCE MILL	100.	A	?	?	XBI			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y				
1	SQR	0.	0.	100.	100.				
UNo.	UNIT	DIA	DEPTH	CHMF					
3	DRILLING	15.	15.	5.					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA			
1	CTR-DR	20.		10.	◆	◆			
2	DRILL	15.		15.	15.	◆			
3	CHAMFER	20.		999.	0.	15.			
FIG	PTN	Z	X	Y	AN1	AN2			
1	PT	0.	20.	20.					
UNo.	UNIT	DIA	DEPTH	CHMF					
4	DRILLING	20.	20.	5.					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA			
1	CTR DR	20.		10.	◆	◆			
2	DRILL	20.		20.	20.	◆			
3	CHAMFER	20.		999.	0.	20.			
FIG	PTN	Z	X	Y	AN1	AN2			
1	PT	0.	60.	60.	◆	◆			
UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE
5	END	0	0						

Program with priority number

UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE				
0	CBN STL	ONLY MILL.	50.	1	OFF				
UNo.	UNIT	ADD. WPC	X	Y	th				
1	WPC-1		-300.	-300.	0.				
UNo.	UNIT	DEPTH	SRV-Z	BTM					
2	FCE MILL	0.	5.	1					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD		
1	FCE MILL	100.	1	?	?	XBI			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y				
1	SQR	0.	0.	100.	100.				
UNo.	UNIT	DIA	DEPTH	CHMF					
3	DRILLING	15.	15.	5.					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA			
1	CTR-DR	20.	2	10.	◆	◆			
2	DRILL	15.	1	15.	15.	◆			
3	CHAMFER	20.		999.	0.	15.			
FIG	PTN	Z	X	Y	AN1	AN2			
1	PT	0.	20.	20.					
UNo.	UNIT	DIA	DEPTH	CHMF					
4	DRILLING	20.	20.	5.					
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA			
1	CTR DR	20.	2	10.	◆	◆			
2	DRILL	20.	1	20.	20.	◆			
3	CHAMFER	20.		999.	0.	20.			
FIG	PTN	Z	X	Y	AN1	AN2			
1	PT	0.	60.	60.	◆	◆			
UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE
5	END	0	0						

Prior machining priority number → (points to No. 1 in rows 10, 11, 12)
 Subsequent machining priority number → (points to No. 2 in rows 13, 14, 15)

If one reclassifies these two programs by machining order, the following tables are obtained.

Program without priority number				Program with priority number			
UNo.	SNo.	TOOL	ATC	UNo.	SNo.	TOOL	ATC
2	1	Face mill	} 6 times	2	1	Face mill	} 4 times
3	1	Centering drill		3	1	Centering drill	
3	2	Drill		4	1	Centering drill	
3	3	Chamf. cutter		3	2	Drill	
4	1	Centering drill		4	2	Drill	
4	2	Drill		3	3	Chamf. cutter	
4	3	Chamf. cutter		4	3	Chamf. cutter	

Centering drill —[5]
 Drill —[6]
 Chamf. cutter —[7]
 Centering drill —[2]
 Drill —[3]
 Chamf. cutter —[4]
 Face mill —[1]

Centering drill —[3]
 Drill —[5]
 Chamf. cutter —[7]
 Centering drill —[2]
 Drill —[4]
 Chamf. cutter —[6]
 Face mill —[1]

([1] to [7]: Machining order)

M3P357

Without a specified priority number, the machining is done by the programmed order and the ATCs are executed for each tool. Consequently, in this example, the ATCs are executed 6 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 ATCs to 4.

Note 1: If a different priority number is assigned to the same tool, the machining is done in the order of the priority number.

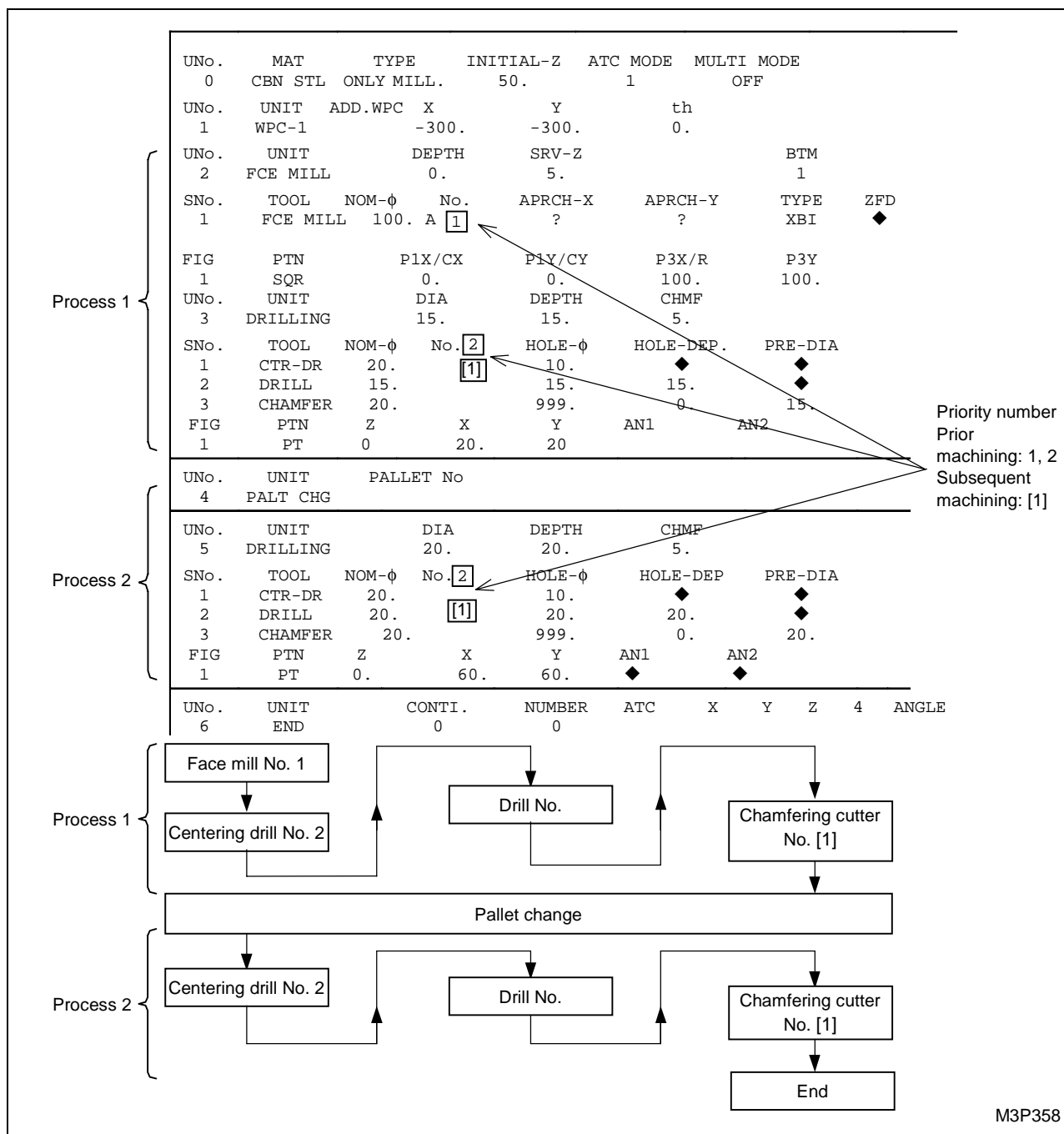
Note 2: When the priority number is assigned to all the tools of the same process, the M-code unit without a priority number is executed once for extra between the prior machining and the subsequent machining.

5-2 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.

- Pallet changing unit
- Process end unit
- Workpiece transfer unit
- Head selection unit

Example: Case where the pallet changing unit has been programmed.
 If the program is executed which contains the priority numbers specified for face mill, centering drill and chamfering cutter, the machining is done in the following order.



Priority number
 Prior machining: 1, 2
 Subsequent machining: [1]

M3P358

Note: The combination of the manual program mode unit and the M-code unit permits performing the changing of the pallet. In order to perform this pallet change and the priority function for the same tool in one program, it is necessary to specify the process end unit before and after the unit concerned.

5-3 Editing Function and Input Method of Priority Numbers

5-3-1 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	DEPTH	SRV-Z	
2	FCE MILL	0.	5.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A	█ ← Cursor	

→ The message "MACHINING PRIORITY No.?" is displayed and the menu changes as follows.

MACHINING PRIORITY No.?							
	DELAY PRIORITY		PRI. No. CHANGE	PRI. No. ASSIGN		PRI. No. ALL ERAS	SUB PROG PROC END

(2) Enter the priority number.

There are the three following entry methods:


- Prior machining priority number
To be set by means of numeric keys. The number is displayed in yellow.
- Subsequent machining priority number
To be set by means of numeric keys after having reversed the display of DELAY PRIORITY by selecting its menu key.
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-φ	No.	APRCH-X
1	FCE MILL	100. 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 .

5-3-2 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.

[Menu selection] **PRI. No. ASSIGN** (→ **DELAY PRIORITY**)

- (1) In creating mode, move the cursor to the item “No.”

UNo.	UNIT	DEPTH	SRV-Z	
2	FCE MILL	0.	5.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A		█ ← Cursor

- (2) Press the menu key **PRI. No. ASSIGN**.

→ 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:


- For subsequent machining, enter the intended number after having pressed the menu key **DELAY PRIORITY**.

→ 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-φ	No.	APRCH-X
1	FCE MILL	100. A	2	█
UNo.	UNIT	DEPTH	SRV-Z	
3	FCE MILL	0.	10.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A	2	

Note 1: This function is only useful for a program in the process of editing.

Note 2: Regardless of whether the priority number is entered or not, all the identical tools in the process are marked with the same priority number.

Note 3: If the data cancellation key  is pressed in operation (3) above, all the priority numbers for the identical tools in the process will be erased.

5-3-3 Change of priority numbers

This function is used for changing the priority number entered for all the identical tools in a process.

[Menu selection] **PRI. No. CHANGE** (→ **DELAY PRIORITY**)

- (1) In creating mode, move the cursor to the item “No.”

UNo.	UNIT	DEPTH	SRV-Z	
2	FCE MILL	0.	5.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A		█ ← Cursor

- (2) Press the menu key **PRI. No. CHANGE**.
 - ➔ 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 ↔


- For subsequent machining, press the menu key **DELAY PRIORITY**, and then enter the intended number.

- ➔ 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-φ	No.	APRCH-X
1	FCE MILL	100. A	5	█ ← Cursor
UNo.	UNIT	DEPTH	SRV-Z	
3	FCE MILL	0.	10.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A	5	

Note 1: This function is only useful for a program in the process of editing.

Note 2: Regardless of whether the priority number is entered or not, the same priority number is assigned to all the identical tools in the process.

Note 3: In operation (3), pressing on the data cancellation key  permits the deletion of all the priority numbers for all the identical tools in the process.

5-3-4 Deletion of all the priority numbers

This function is used for deleting all the priority numbers contained in the process or in the program.

[Menu selection] **PRI. No. ALL ERAS**

- (1) In creating mode, move the cursor to the item “No.”

UNo.	UNIT	DEPTH	SRV-Z	
2	FCE MILL	0.	5.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A	█ ← Cursor	

- (2) Press the menu key **PRI No. ALL ERAS**.

➔ 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 keys in the order of **1** .

➔ All the priority numbers in the specified zone are deleted.

SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A		█ ← Cursor
UNo.	UNIT	DEPTH	SRV-Z	
3	FCE MILL	0.	10.	
SNo.	TOOL	NOM-φ	No.	APRCH-X
1	FCE MILL	100. A		

Note: This function is only useful for a program in the process of editing.

5-3-5 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 sub-program.

If in the process constituting the subject of the editing, there is a sub-program containing a process delimitation unit (pallet changing unit or process end unit), press the menu key **SUB PROG PROC END** in order to invert the display, which has the result that the sub-program is treated the same as the process end unit.

Note: The editing function zone can be divided by the sub-program unit. Even if the editing function is executed in delimited zone [1], this has no effect in the zone [2], [3] and [4]. (See the next page.)

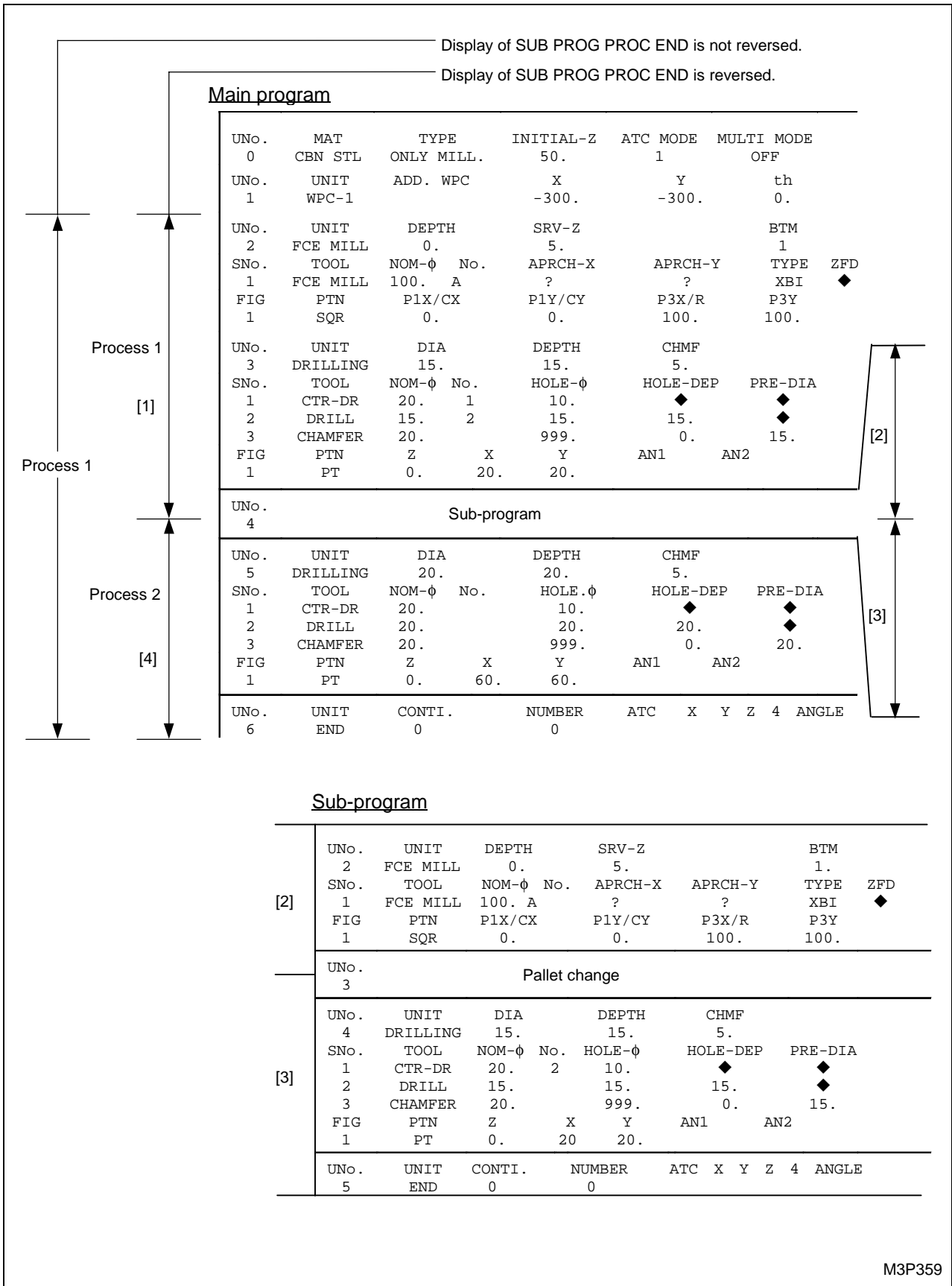


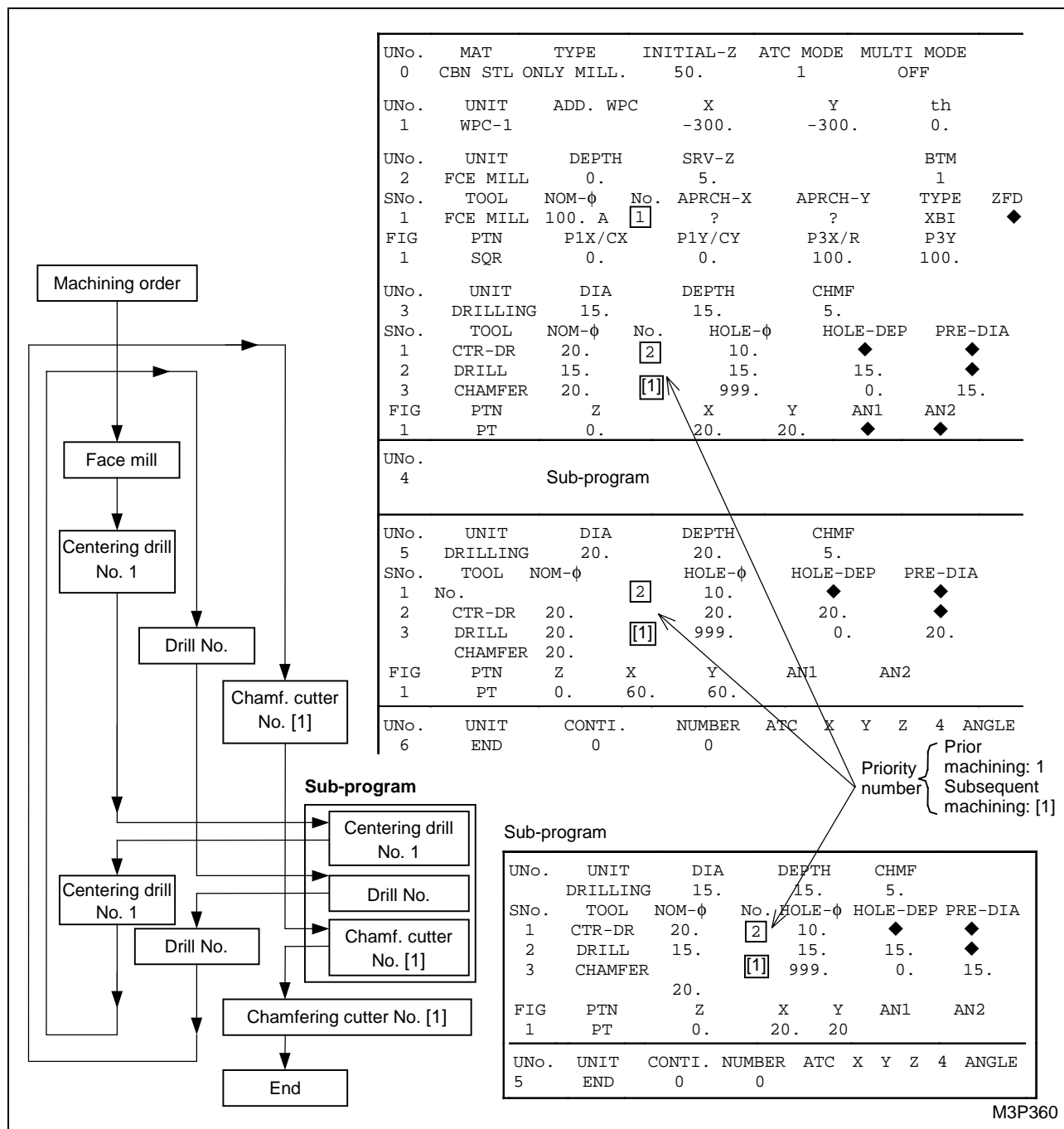
Fig. 5-1 Sub-program unit = process end unit

M3P359

5-4 Relation between the Sub-program Unit and the Priority Machining Function

If one program contains a sub-program unit and the priority function for the same tool, the machining order is as mentioned below.

Example: Entry of priority number for centering drill and chamfering cutter



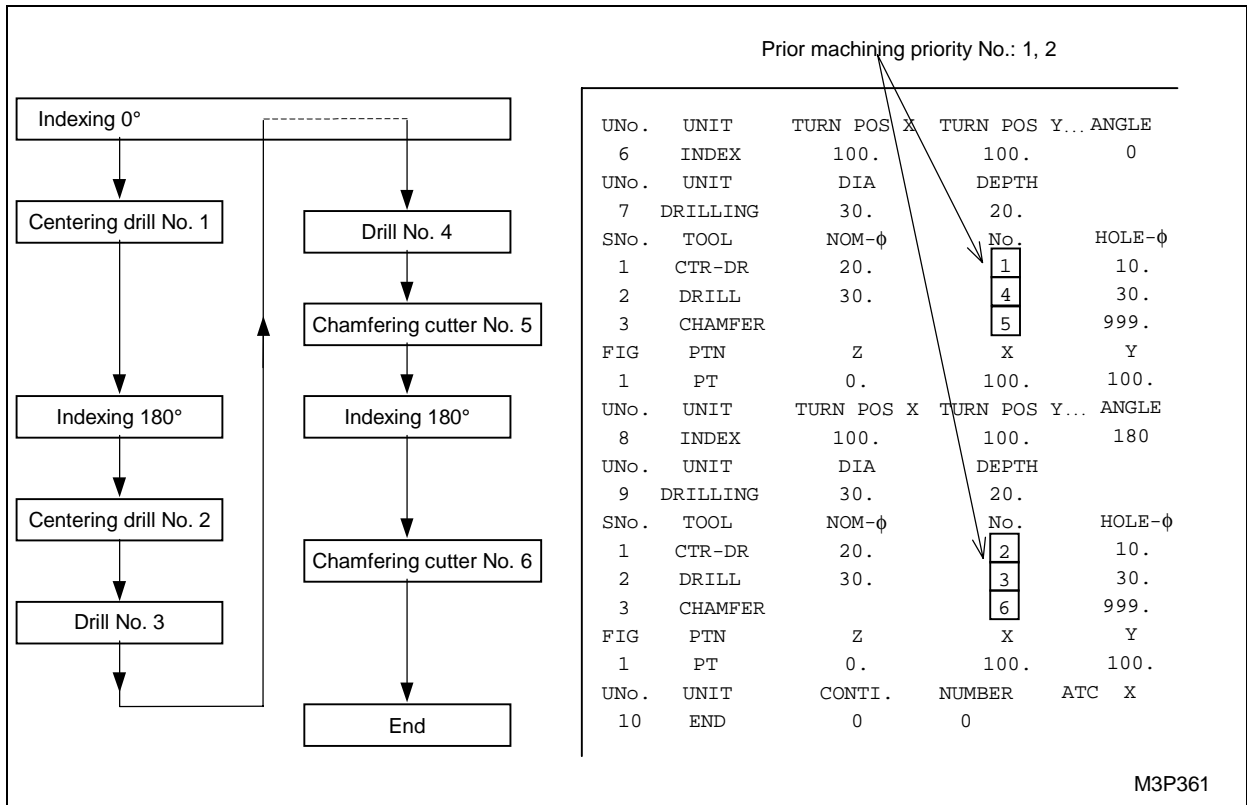
In the process of priority machining search, the sub-program unit is executed as follows:

- In the case where sub-program is a MAZATROL program, the sub-program unit is always executed. (The machinings specified in the sub-program are executed in the numeric order of the priority numbers.)
- In the case where sub-program is an EIA/ISO program, the sub-program unit is only executed once at the time of ordinary machining.

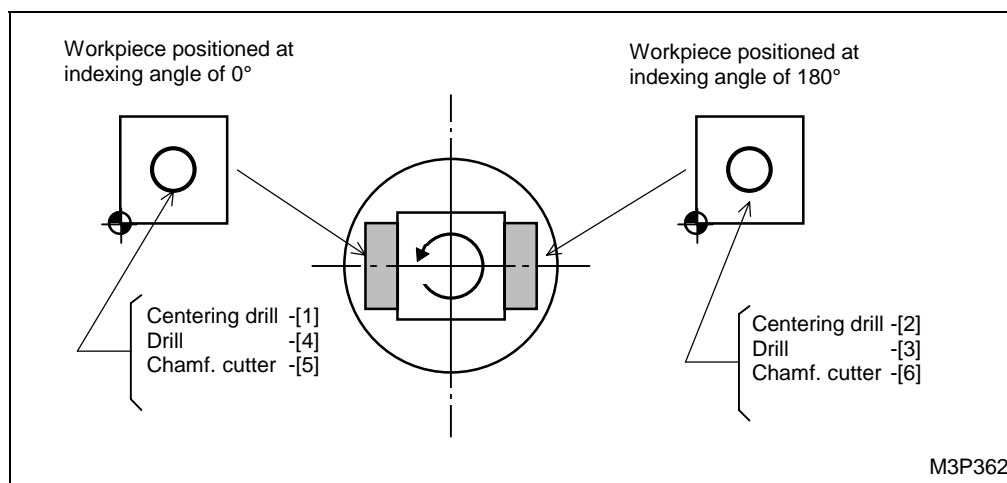
5-5 Relation between the Index Unit and Priority Machining Function

When the machining is performed controlling the machining angle on the indexing table (indexed by B- or M-code), the NC rotating table or the tilting table, it is possible to combine the priority function for the same tool and the index unit, which contributes to the reduction of machining time.

Example: Priority number assigned to centering drill, drill and chamfering cutter



At the time of the execution of this program, the machining order is as follows:



5-6 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.

Without priority No.

With priority No.

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	TYPE	INITIAL-Z	ATC	MODE	MULTI	MODE
0	CBN STL	ONLY MILL.	50.	1		OFF	
UNO.	UNIT	ADD.	WPC	X	Y	th	
1	WPC-1			-300.	-300.	0.	
UNO.	UNIT	DEPTH	SRV-Z			BTM	
2	FCE MILL	0.	5.			1.	
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD
1	FCE MILL	100. A	1	?	?	XBI	◆
FIG	PTN	PLX/CX	PLY/CY		P3X/R	P3Y	
1	SQR	0.	0.		100.	100.	

M-code unit

UNO.	UNIT	DIA	DEPTH	CHMF			
4	DRILLING	15.	15.	5.			
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	
1	CTR-DR	20.	2	10.	◆	◆	
2	DRILL	15.		15.	15.	◆	
3	CHAMFER	20.		999.	0.	15.	
FIG	PTN	Z	X	Y	AN1	AN2	
1	PT	0.	20.	20.			
UNO.	UNIT	DIA	DEPTH	CHMF			
5	DRILLING	20.	20.	5.			
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP.	PRE-DIA	
1	CTR-DR	20.	2	10.	◆	◆	
2	DRILL	20.		20.	20.	◆	
3	CHAMFER	20.		999.	0.	20.	
FIG	PTN	Z	X	Y	AN1	AN2	
1	PT	0.	60.	60.			

UNO.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE
6	END	0	0						

M3P363

5-12

5-7 Relation between Multi-workpiece Machining and the Priority Machining Function

If the multi-workpiece machining process and same-priority-numbered tool data are set both in one program, the order of machining is assigned by the parameter **F71**.

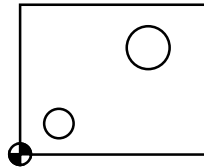
Parameter **F71** = 0: Priority machining is performed workpiece by workpiece.

1: Priority machining is performed on all the workpiece.

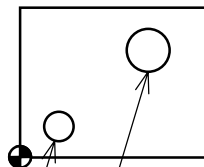
Example: Assigning the priority number to centering drills

UNo.	MAT	TYPE	INITIAL-Z	ATC
0	CST IRN	ONLY MILL.	50.	
OFS	X	Y		
1	0	100		
2	0	200		
UNo.	UNIT	DIA	DEPTH	
1	DRILLING	10.	20.	
SNo.	TOOL	NOM-φ	No.	HOLE-φ
1	CTR-DR	20.	1	
2	DRILL	15.		
FIG	PTN	Z	X	Y
1	PT	0.	15.	15.
UNo.	UNIT	DIA	DEPTH	
2	DRILLING	20.	20.	
SNo.	TOOL	NOM-φ	No.	HOLE-φ
1	CTR-DR	20.	1	
2	DRILL	20.		
FIG	PTN	Z	X	Y
1	PT	0.		
UNo.	UNIT	CONTI.	NUMBER	
3	END	0	0	

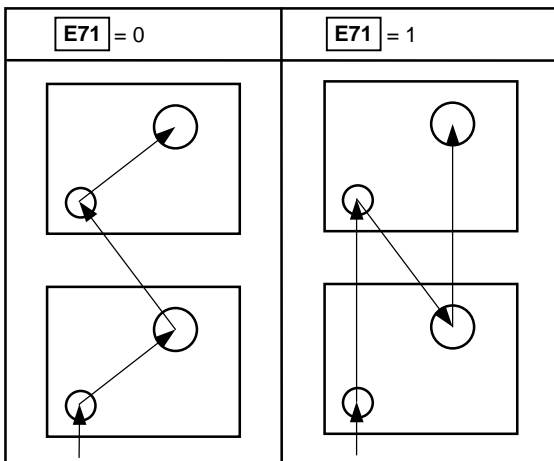
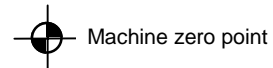
Workpiece of offset No. 2



Workpiece of offset No. 1



Drilling in the unit No. 2
Drilling in the unit No. 1



M3P364

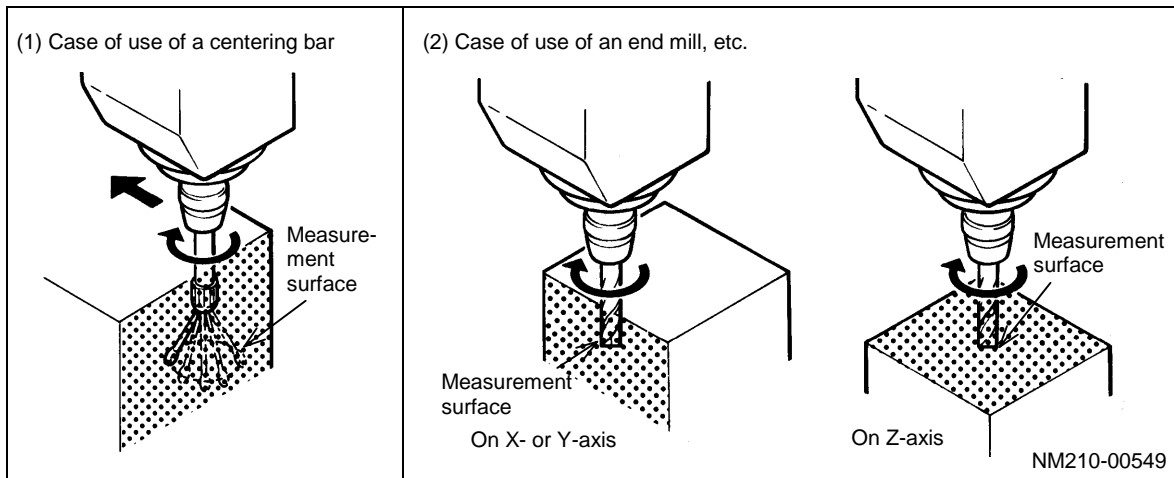
5

PRIORITY FUNCTION FOR THE SAME TOOL

- NOTE -

6 COORDINATES MEASUREMENT FUNCTION

The coordinate measurement function is intended for measuring the fundamental coordinates by using a measuring device (feeler or centering bar) or a tool like an end mill.



6-1 Method of Measurement of Coordinates by TEACH Function

On the basis of the position of the tool edge or the edge of a measuring device in contact with the workpiece to be machined, the fundamental coordinates can be calculated and recorded in the fundamental coordinates unit.

- (1) In ATC mode, mount the measuring device or the tool on the spindle.
- (2) Press the display selector key (key located to the left of the menu keys) to display the menu related to the display.
- (3) Pressing the menu key **PROGRAM**.
- (4) Press the menu key **WPC MSR**.
 → The following menu will be displayed.

WPC		TEACH	+Z	+X	-X	+Y	-Y	-Z
SEARCH			SENSOR	SENSOR	SENSOR	SENSOR	SENSOR	SENSOR

- (5) Pressing the menu key **WPC SEARCH** selects the WPC unit (fundamental coordinates system unit) in which the measured fundamental coordinates are recorded.
- (6) Press the input key.
 → The cursor is moved to the article X of the WPC unit.

UNo.	UNIT	ADD. WPC	X	Y
2	WPC-0		█	

Note: If a plane definition unit exists, the cursor will move to the article X of the plane definition unit, instead X of the WPC unit.

SNo.	WORK No.	ADD. WPC	X	Y
1	100		█	

- (7) Make the spindle turn.

(8) Move the spindle.

Go to manual mode and put the tool in contact with the surface of the workpiece to be measured by the manual pulse handle.

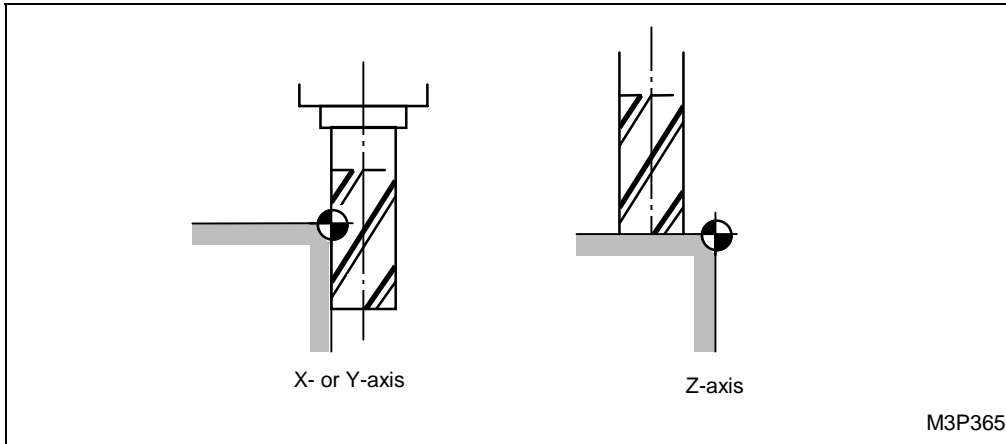


Fig. 6-1 Tool in contact with the surface

(9) Press the menu key **TEACH**.

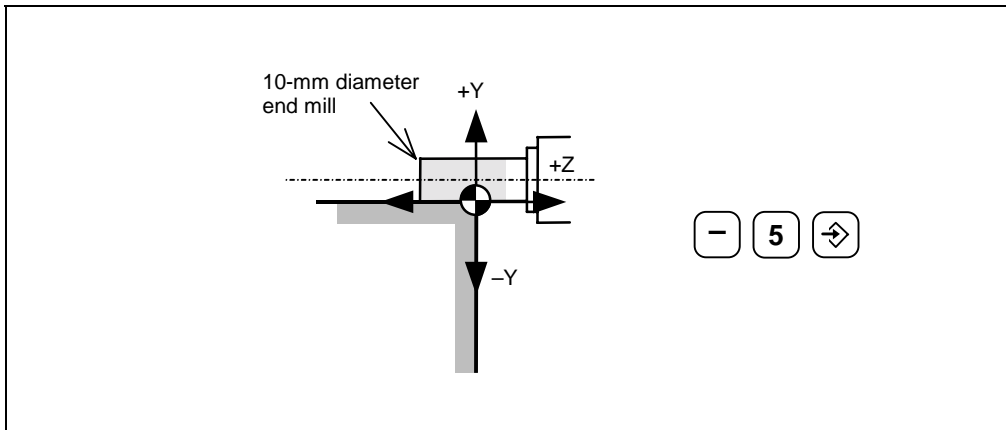
WPC		TEACH	+Z	+X	-X	+Y	-Y	-Z
SEARCH			SENSOR	SENSOR	SENSOR	SENSOR	SENSOR	SENSOR

➔ The display of **TEACH** is reversed.

(10) Enter the coordinate of the workpiece zero-point viewed from the center of the tool by means of numeric keys.

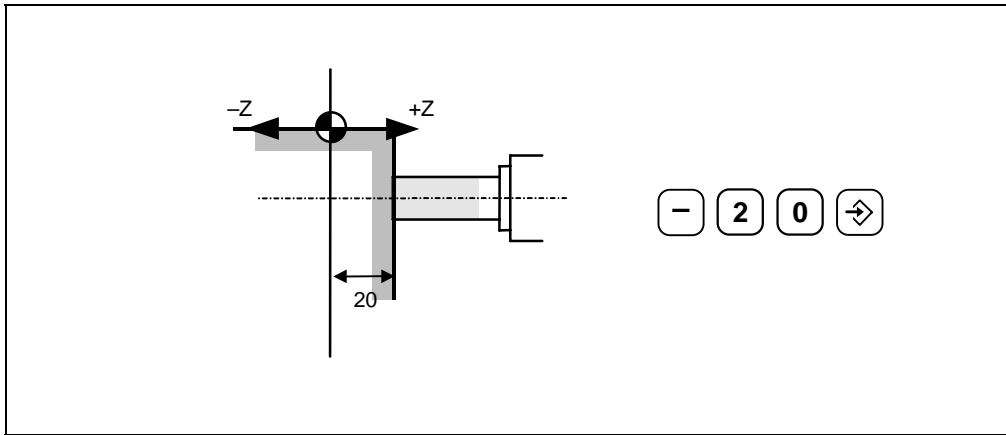
<For a radial direction>

Set the radial coordinate of the workpiece origin when seen from the tool center.



<For an axial direction>

Set the axial coordinate of the workpiece zero-point when seen from the tool nose. The system, after making reference to the tool length data included in the tool data, automatically calculates the fundamental coordinates.



(11) The fundamental coordinate is thus automatically calculated and recorded in the WPC unit.

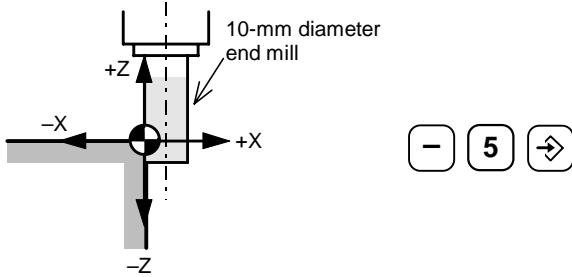
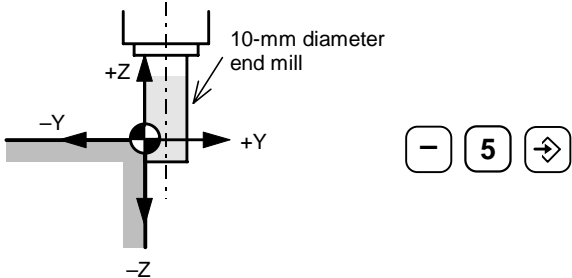
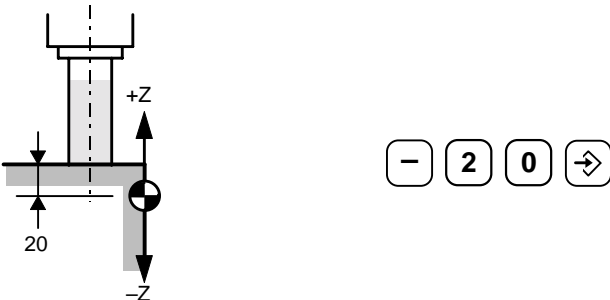
UNo.	UNIT	ADD. WPC	X	Y
2	WPC-0		-210.	█

(12) Repeat the above-indicated operations to record the fundamental coordinates on the other axes.

Note: During execution of WPC units or plane definition sequences, the system searches only for the data present at after the cursor position. Also, the alarm message “407 DESIGNATED DATA NOT FOUND” will be displayed if the cursor position is not followed by a WPC unit.

SNo.	WORK No.	ADD. WPC	X	Y
1	100			█

Setting tool nose position strage data for each of the planes to be machined (in the table below, axis names are for machine coordinates)


<p>X (WPC-X)</p>	 <p>10-mm diameter end mill</p> <p>-X +X</p> <p>+Z</p> <p>-Z</p> <p>- 5 ↗</p>
<p>Y (WPC-Y)</p>	 <p>10-mm diameter end mill</p> <p>-Y +Y</p> <p>+Z</p> <p>-Z</p> <p>- 5 ↗</p>
<p>Z (WPC-Z)</p>	 <p>+Z</p> <p>-Z</p> <p>20</p> <p>- 20 ↗</p>

6-2 Method of Measurement of Coordinates by MDI-MMS

The use of the feeler permits automatic calculation of the fundamental coordinates and recording thereof in the WPC unit.


- (1) In ATC mode, mount the feeler on the spindle.
- (2) Press the display selector key (key located to left of the menu keys) to display the menu related to the display.
- (3) Press the menu key **PROGRAM**.
- (4) Pressing the menu key **WPC MSR** displays the coordinates measurement menu.

WPC SEARCH		TEACH	+Z SENSOR	+X SENSOR	-X SENSOR	+Y SENSOR	-Y SENSOR	-Z SENSOR
---------------	--	-------	--------------	--------------	--------------	--------------	--------------	--------------

- (5) Pressing the menu key **WPC SEARCH** selects the WPC unit in which are recorded the measured fundamental coordinates.
- (6) Press the input key .
 - ➔ The cursor is moved to the item X of the WPC unit.

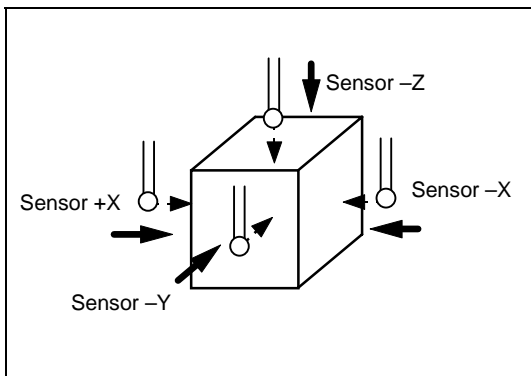
UNo.	UNIT	ADD.	WPC	X	Y	th	Z
2	WPC-0			█			

- (7) Move the spindle.
 - Go to manual mode and by using the manual pulse handle, bring the feeler point to a point close to the surface to be measured.
 - Note:** Do not put the feeler point in contact with the surface to be measured.
 - The position in which the feeler point is stopped is the measurement start point.

- (8) Press the MDI run key  to go to MDI mode.
- (9) Press the menu selector key (key located to the right of the menu keys) to display the menu containing **WPC SEARCH**.

WPC SEARCH		TEACH	+Z SENSOR	+X SENSOR	-X SENSOR	+Y SENSOR	-Y SENSOR	-Z SENSOR
---------------	--	-------	--------------	--------------	--------------	--------------	--------------	--------------

- (10) Select the measurement direction by means of the corresponding menu key.



Coordinate measurement with MDI-MMS is available for both the side and the top.

(11) Enter the coordinate of the face with which the feeler point comes into contact (measurement surface) in the workpiece coordinates system.

Example: Entry of 20

Press the keys **2**, **0**, and **↔** in this order.

(12) Press the start button.

➔ The feeler moves slowly at skip speed and after its point comes into contact with the surface to be measured, the feeler returns to the measurement start point at rapid feed.

- Thus, the fundamental coordinate is automatically calculated and displayed.

UNo.	UNIT	ADD. WPC	X	Y	th	Z	4
2	WPC-0		220.	█			

Note: The fundamental coordinates will not be accurately measured if MDI-MMS is performed with the NC unit not containing the correction data based on tool length measurement with a touch sensor and on calibration measurement with a feeler.

7 BARRIER INFORMATION DATA CREATION

7-1 BARRIER INFORMATION Display

On the **BARRIER INFORMATION** display, set the data related to chuck barriers, jaw and tailstock barriers. Setting the data validates the barriers to prevent tools from interfering with the chucks, jaw and tailstock. The data can also be used for tool path displaying on the **TOOL PATH CHECK** display.

- (1) Set the cursor on the common unit data.

UNo.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
0	**	***	**	***	***	***	***	***

}

UNo.	UNIT	CB-DIA	CB-DEP	CHMP	BTM	DIA	DEPTH
2	RGH CBOR	***	***	**	**	**	***

SNo.	TOOL	NOM-φ No.	HOLE-φ	HOLE-DEP	PRE-DIA	PPR-DEP	RGH	DEPTH	C-SP	FR	M	M
1	**	**	**	**	**	**	**	**	**	**	**	**

Place the cursor at this line.

- (2) Press the menu selector key, and then press the menu key **BARRIER INFORM**.

Pressing the menu key **BARRIER INFORM** makes the **BARRIER INFORMATION** display of the unit which was designated at step (1) above.

Display items differ according to the particular specifications of the machine.

For a machine having a tailstock

← Common data line

Barrier information data

D736P0132E

For a machine having a No. 2 turning spindle

← Common data line

Barrier information data

D736P0133E

Only main turning spindle data is displayed for a machine not having a tailstock or a No. 2 turning spindle.

<Display data description>

No.	Item	Unit	Data description
[1]	CHUCK	—	Identification of whether or not the chuck barrier is effective
[2]	JAW No.	—	Number of the jaw to be used
[3]	GRIP DIA	mm (inch)	Diameter of that part of the workpiece that is to be chucked
[4]	TAIL BARRIER	—	Identification of whether or not the tailstock barrier is effective
[5]	TAIL STOCK	—	Identification of whether or not the tailstock is to be used
[6]	REF.TO T/S	mm (inch)	Distance from the reference workpiece zero-point to the end of the tailstock

* These items are valid only for MAZATROL programs.
Refer to the description of parameters **BA4**, **BA16** to **BA18**, **BA39** to **BA41** and **BA125** given in the Parameter List for the barrier function applied to EIA/ISO programs.

(3) Set the data in indicated item or change the data displayed in indicated item.

- On common unit line above, the data cannot be changed on the **BARRIER INFORMATION** display.

Note 1: The following menu is displayed while the **BARRIER INFORMATION** display remains on the screen:

			BAR. INF. END	BAR. INF. CANCEL				
--	--	--	------------------	---------------------	--	--	--	--

Pressing the menu key **BAR. INF. END** calls up the **PROGRAM** display anew.

Note 2: Setting or changing barrier information data displays “+” mark on the left side of the common unit number on the **PROGRAM** display.

For units whose barrier information data has been set or changed, one more blocks of program memory will be used.

UNo.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
+ 0	CST IRN	MILL&TURN.	100.	0.	120.	1.	1	3000

Note 3: Carry out the following procedure to cancel the entire barrier information data that has been set (or changed).

1) Press the menu key **BAR. INF. CANCEL**.

2) Set “-9999.”

The entire current barrier information data is cancelled and initial barrier information data is displayed on the **BARRIER INFORMATION** display. Also, the + mark on the **PROGRAM** display is deleted.

7-2 Barrier Information Data Registration

1. Data registration

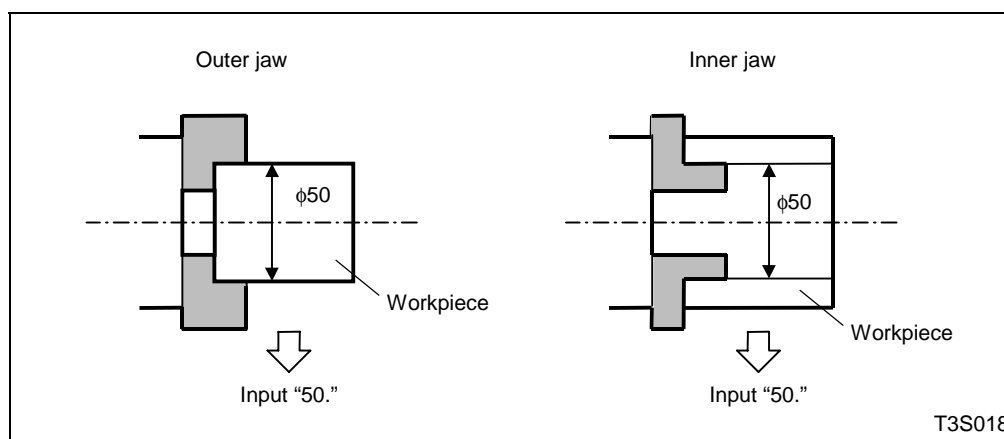
The following data can be registered on the **BARRIER INFORMATION** display:

For the machine with the No. 2 spindle, set the data of the No. 1 spindle or the No. 2 spindle in the SPDL 1 or SPDL 2 group, respectively.

- (1) Place the cursor on the data item CHUCK.
 - Press the cursor key to call the cursor on the display. Then, move the cursor to the position CHUCK.
- (2) Specify whether or not the chuck barriers are to be made effective.
 - Pressing the menu key **VALID** validates the chuck barriers.
 - Pressing the menu key **INVALID** invalidates the chuck barriers.
 - After a menu key has pressed, the cursor will move to the data item JAW No.
- (3) Set the number of the jaw to be used.
 - To check jaw numbers, press the menu key **CHUCK JAW DATA** to display the **CHUCK JAW DATA** display.
 - Set 0 if the collet chuck is to be used.
 - After data has been set, the cursor will move to the data item GRIP DIA.

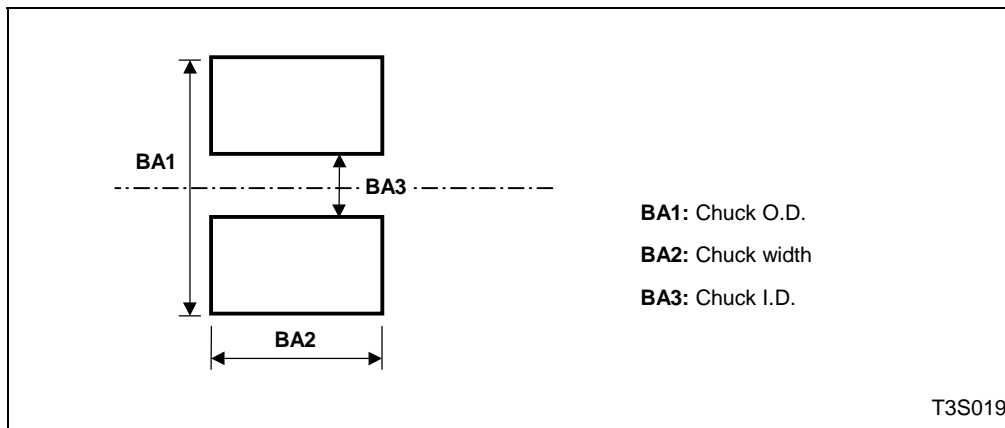
Note: Data can be entered with a jaw name by setting up "1" at bit 2 of parameter **TC142**.

- (4) Set the diameter of that section of the workpiece that is to be chucked.

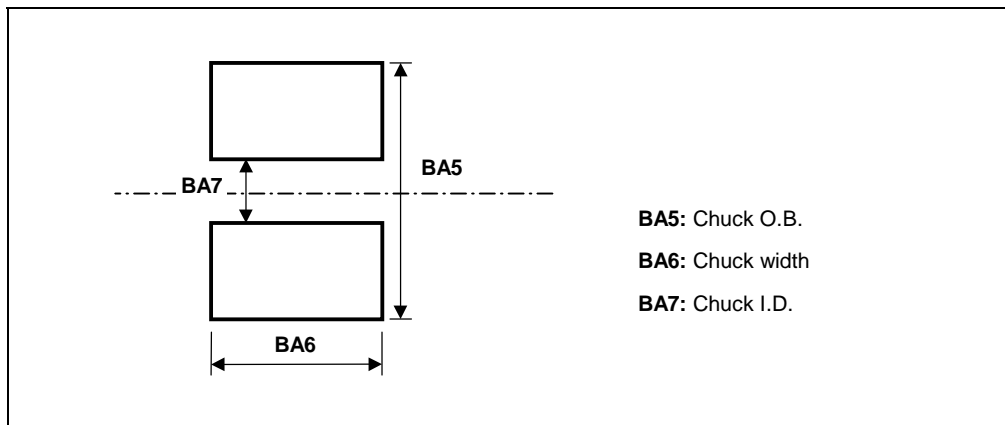


- Input 0 if the collet chuck is to be used.

Note: Chuck shape data and chuck barriers:
The shape data of the chuck is registered in parameters **BA1** through **BA3**.



The shape data of chuck of the No. 2 spindle side is registered in parameters **BA5** through **BA7**.

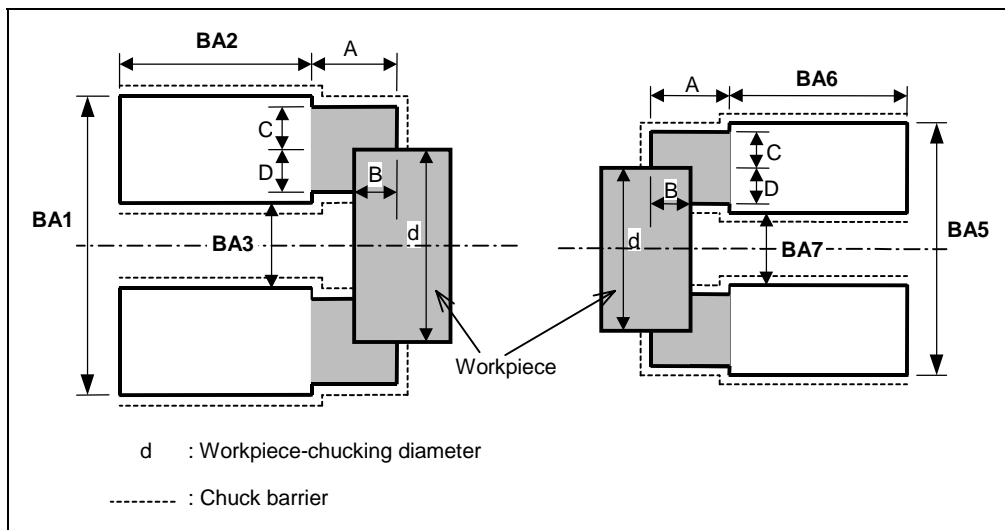


For three-jaw chucks, the chuck barriers are automatically arranged slightly outside the chucks or jaws. In that case, they are arranged according to the following three factors:

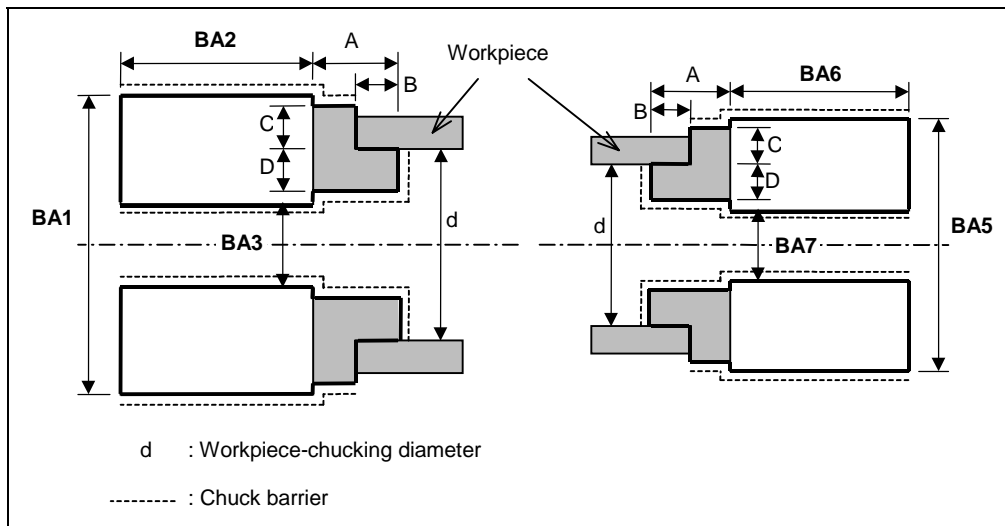
- Chuck shape data within the parameters mentioned above
- Jaw shape data registered in the **CHUCK JAW DATA** display (Dimensions of the section A through D)
- Workpiece-chucking diameter data set on the **BARRIER INFORMATION** display.

The position where the tool stops in barrier differs according to the selected feed rate or other factors. In general, the time where the tool stops in barrier area delays as the feed rate increases.

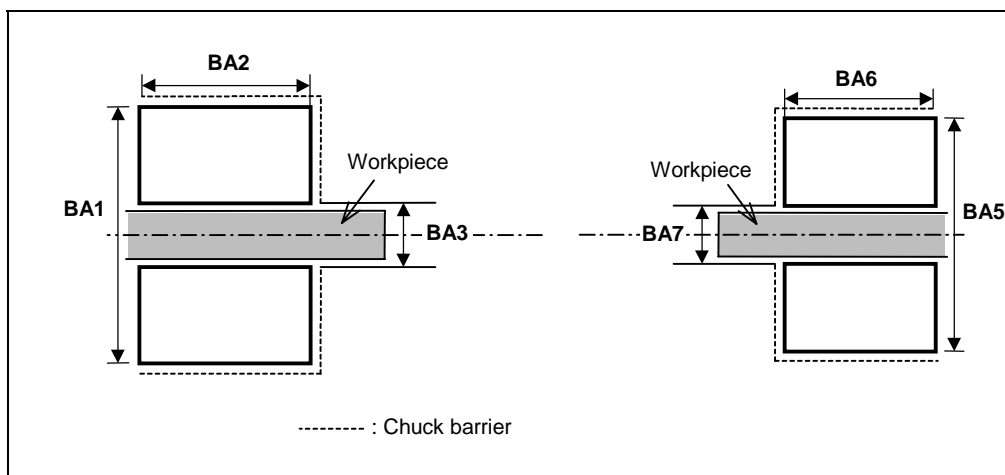
[When using the outer jaw]



[When using the inner jaw]



For the collet chuck, the chuck barriers are only arranged according to the chuck shape data registered in the above-mentioned parameters.



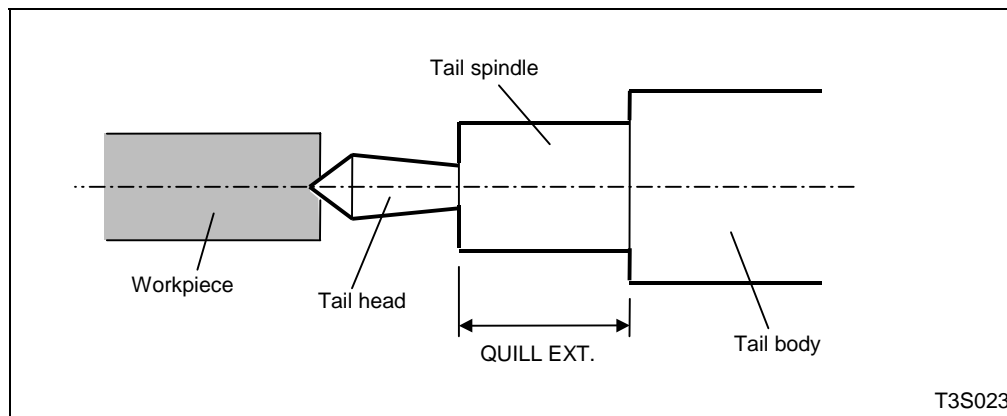
2. Setting data related to tailstock barriers

- (1) Place the cursor on the data item TAIL BARRIER.
- Press the cursor key to call the cursor on the display. Then, move the cursor to the position shown below:

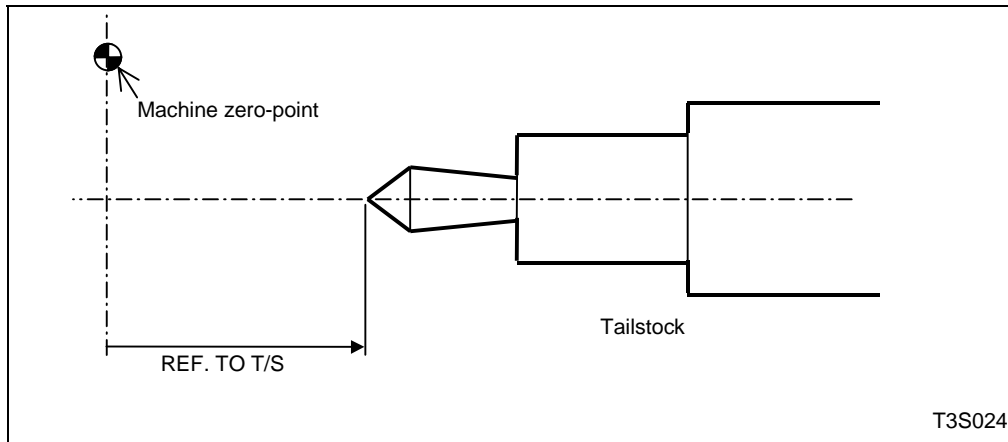
```

TAIL
TAIL BARRIER [INVALID ]
TAIL STOCK   [UNUSED   ]
REF.TO T/S   [         ]
  
```

- (2) Specify whether or not the tailstock barriers are to be validated.
- Pressing the menu key **VALID** validates the tailstock barriers.
 - Pressing the menu key **INVALID** invalidates the tailstock barriers.
 - ➔ After data has been set, the cursor will move to the data item TAIL STOCK.
(In the data item TAIL BARRIER, INVALID is preset as its initial value. To validate the tailstock barriers, therefore, call the cursor on the display and move it to the data item TAIL STOCK.)
- (3) Specify whether or not the tailstock is to be used.
- Press the menu key **USE** if the tailstock is to be used when executing the program.
 - Press the menu key **UNUSE** if the tailstock is not to be used when executing the program.
 - ➔ If **USE** has been set, the cursor will move to the data item QUILL EXT.
 - ➔ If **UNUSE** has been set, the cursor will move to the data item REF. TO T/S.
- (4) Set the projection length of the tail spindle.
- Using the data keys, specify the projection length shown in the figure below. Then, press the input key.

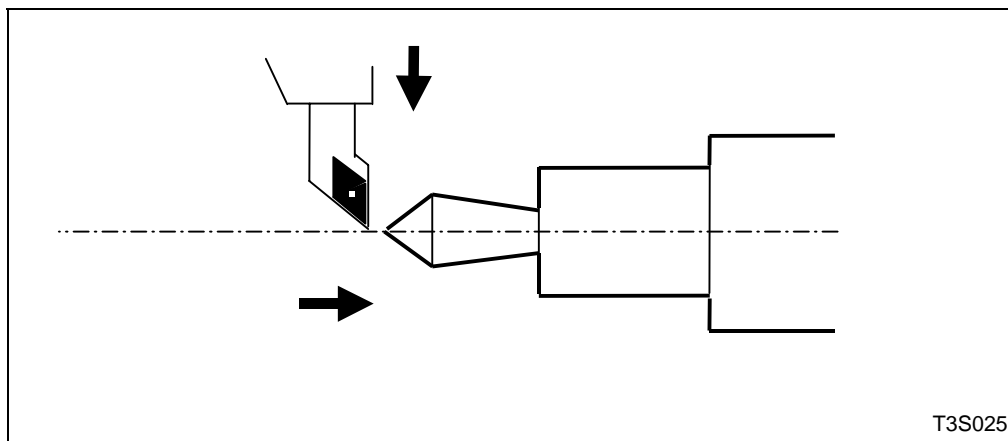


- (5) Set the Z-coordinate of the tailstock end with respect to the reference workpiece zero-point.
 - Set the data using the TEACH function.
 - For details for the data setting procedure, see <Setting the end Z-coordinate> below.



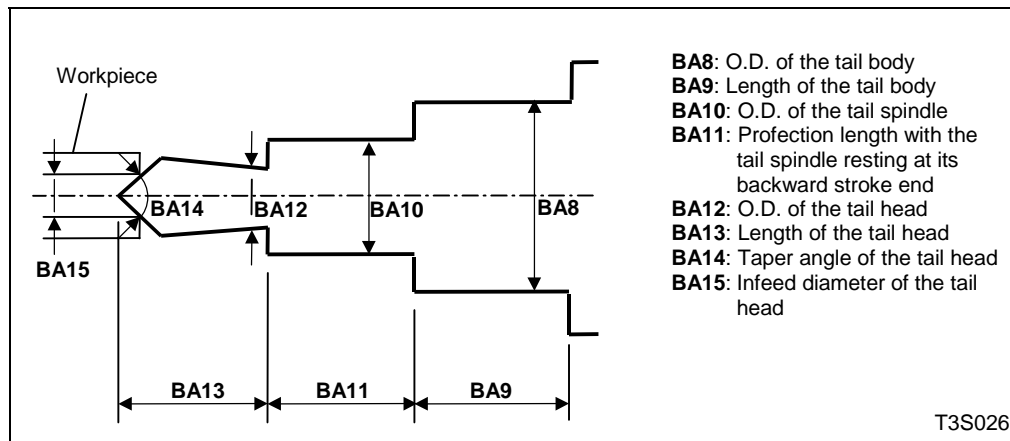
<Setting the end Z-coordinate>

- (1) Index a tool which has already been set.
- (2) Bring the nose of the tool into contact with the end of the tailstock.
 - Move the X-axis and the Z-axis under manual operation until the tool nose has touched the end of the tailstock.



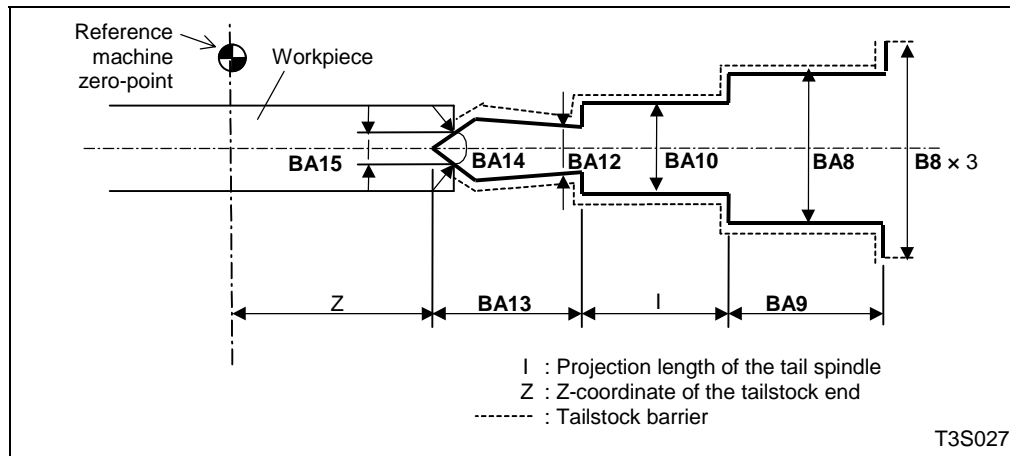
- (3) Place the cursor on the data item REF. TO T/S.
 - ➔ The message "T/S BARRIER SET <TEACH>?" will be displayed.
- (4) Press the menu key **TEACH**.
 - ➔ The **TEACH** will assume a reverse display status.
- (5) Press the input key.
 - ➔ The data will be registered in the data item REF. TO T/S automatically and the reverse display status of the **TEACH** will be cleared.

Note: Tailstock shape data and tailstock barriers:
Tailstock shape data is pre-registered in parameter **BA8** through **BA15**.



Tailstock barriers will be arranged slightly outside the tailstock according to the tailstock shape data pre-registered in the above parameters (**BA11** excluded), and the data set in the data item QUILL EXT. and REF. TO T/S on the **BARRIER INFORMATION** display.

Example: Arrangement of the tailstock barriers



- NOTE -

8 TPC DATA CREATION

8-1 Setting Tool Path Control (TPC) Data

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
- Fundamental coordinates system unit (WPC)
- Fundamental coordinate shifting unit (WPC SHIFT)
- Auxiliary coordinates system unit (OFFSET)
- End unit
- Sub-program unit
- Pallet changing unit
- Index unit
- Process end unit
- Three-dimensional surface machining unit (3-D)
- Materials shape unit
- Head selection unit

(1) Set the cursor on the unit data where the TPC data is to be set.

Example: If the TPC data is to be set for the RGH CBOR unit

UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	PITCH-X	PITCH-Y
0	*	***	**	***	***	***	***

}

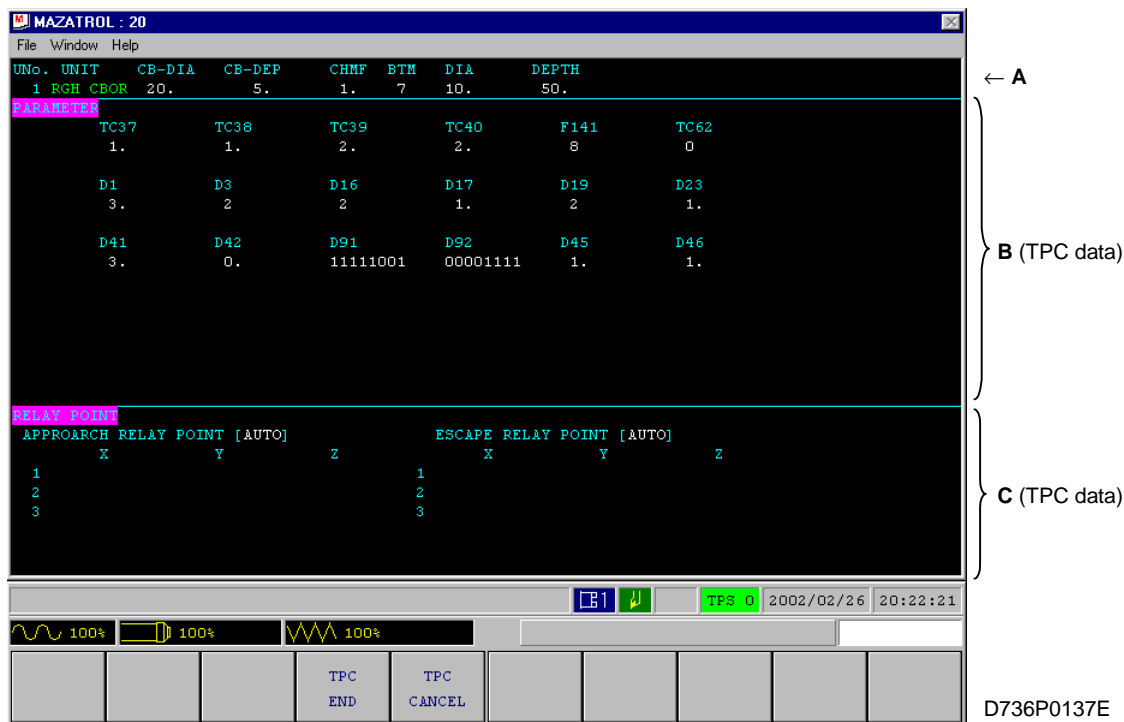
UNo.	UNIT	CB-DIA	CB-DEP	CHMP	BTM	DIA	DEPTH
2	RGH CBOR	***	***	**	**	**	***

SNo.	TOOL	NOM-φ No.	HOLE-φ	HOLE-DEP	PRE-DIA	PPR-DEP	RGH	DEPTH	C-SP	FR	M	M
1	**	**	**	**	**	**	**	**	**	**	**	**

Place the cursor at this line.

- (2) Press the menu selector key, and then press the menu key **TPC**.

Pressing the menu key **TPC** makes the **TPC** display of the unit which was designated at step (1) above.



- (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 in the area **B** shown above. Parameters denoting the distance are usually preset in 0.1 mm (0.01 inch) steps, but on the **TPC** display, they will be displayed in 1 mm (1 inch) steps.

Example: Data preset in parameter **D42:** 50 (in 0.1 mm steps)
 ↓
D42 data displayed on the **TPC** display: 5.0 (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 overridden with the new data.

The parameter settings will not change even if the displayed data is changed on the **TPC** display.

- The area C shown above 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 menu key **MANUAL**. 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.

Note 1: The following menu is displayed while the **TPC** display remains on the screen:

			TPC END	TPC CANCEL					
--	--	--	------------	---------------	--	--	--	--	--

Pressing the menu key **TPC END** 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, maximum three more blocks of program memory will be used.

If TPC data has not ben set:

UNo.	MAT	TYPE		INITIAL-Z	
0	**	ONLY MILL.		***	
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	
1	RGH CBOR	***	***	**	
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP
1	**	**	.	**	**
FIG	PTN	Z	X	Y	
1	**	***	***	**	
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	
2	RGH CBOR	***	***	**	
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP
1	**	**	.	**	**
FIG	PTN	Z	X	Y	
1	**	***	***	**	

If TPC data has been set for unit No. 2:

UNo.	MAT	TYPE		INITIAL-Z	
0	**	ONLY MILL.		***	
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	
1	RGH CBOR	***	***	**	
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP
1	**	**	.	**	**
FIG	PTN	Z	X	Y	
1	**	***	***	**	
UNo.	UNIT	CB-DIA	CB-DEP	CHMF	
+ 2	RGH CBOR	***	***	**	
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP
1	**	**	.	**	**
FIG	PTN	Z	X	Y	
1	**	***	***	**	



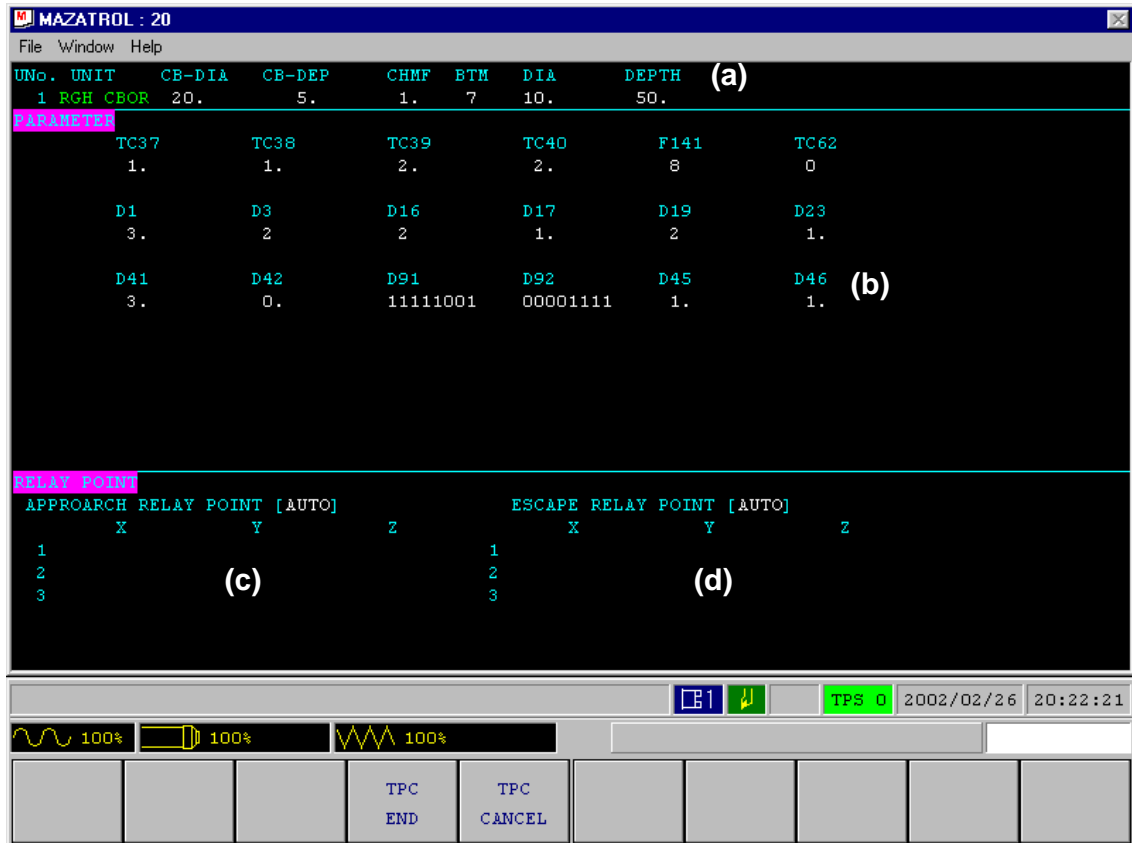
Note 3: Carry out the following procedure to cancel the entire TPC data that has been set (or changed).

- 1) Press the menu key **TPC CANCEL**.
- 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.

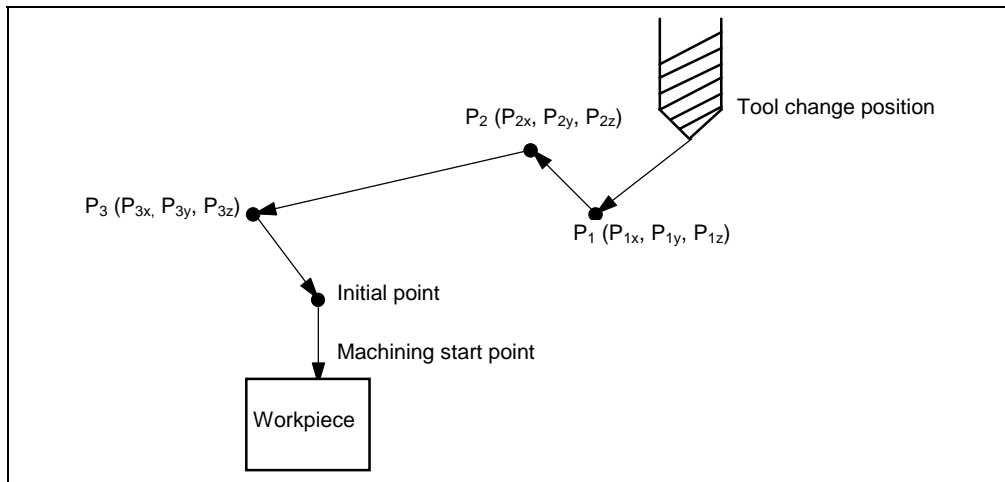
8-2 Description of Each TPC Data Item



D936P0140E

- (a) Unit data for which the **TPC** display is called up. The data cannot be changed with the TPC data displayed on the screen.
- (b) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display. Refer to the separate Parameter List for details of parameter data. Parameters denoting the distance are usually set in 0.1 mm (or 0.01 in.) steps, but they will be displayed here in 1 mm (or 1 in.) steps.

- (c) Use this section to modify the approach path so that interference does not occur. To modify the path, first set the cursor at [AUTO] of the required section and then press the menu key **MANUAL**, to display [MANU]. Finally enter the coordinates of the relay points on three lines (1, 2 and 3) in the desired order of relaying.



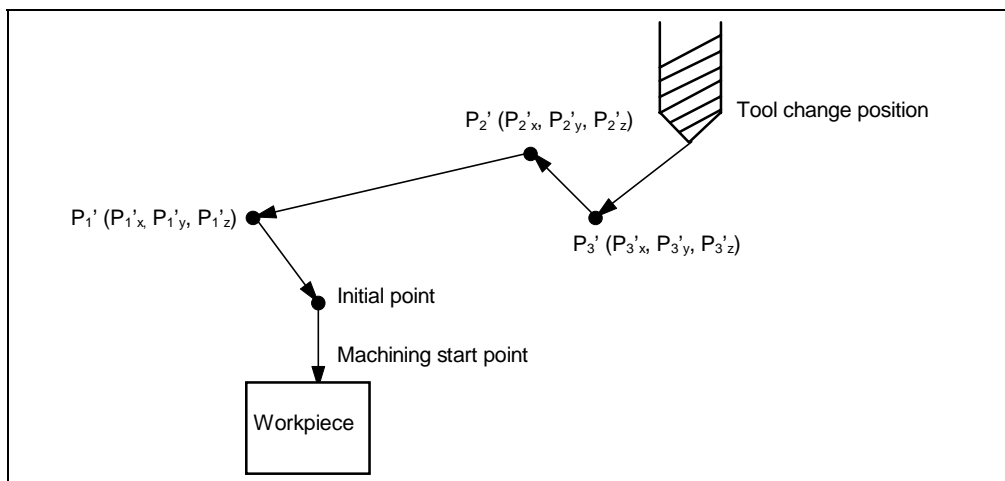
For approaching path from the tool change position through relay points P₁, P₂ and P₃ to the machining start point, as shown above, set data as follows:

APPROACH RELAY POINT [MANU]

	X	Y	Z	
1	P _{1x}	P _{1y}	P _{1z}	Path through P ₁
2	P _{2x}	P _{2y}	P _{2z}	Path through P ₂
3	P _{3x}	P _{3y}	P _{3z}	Path through P ₃

Specify the position using the program coordinates system.

- (d) Use this section to modify the escape path so that interference does not occur. Refer to the description in (c) for details on data setting.



ESCAPE RELAY POINT [MANU]

	X	Y	Z	
1	P _{1'x}	P _{1'y}	P _{1'z}	Path through P ₁ '
2	P _{2'x}	P _{2'y}	P _{2'z}	Path through P ₂ '
3	P _{3'x}	P _{3'y}	P _{3'z}	Path through P ₃ '

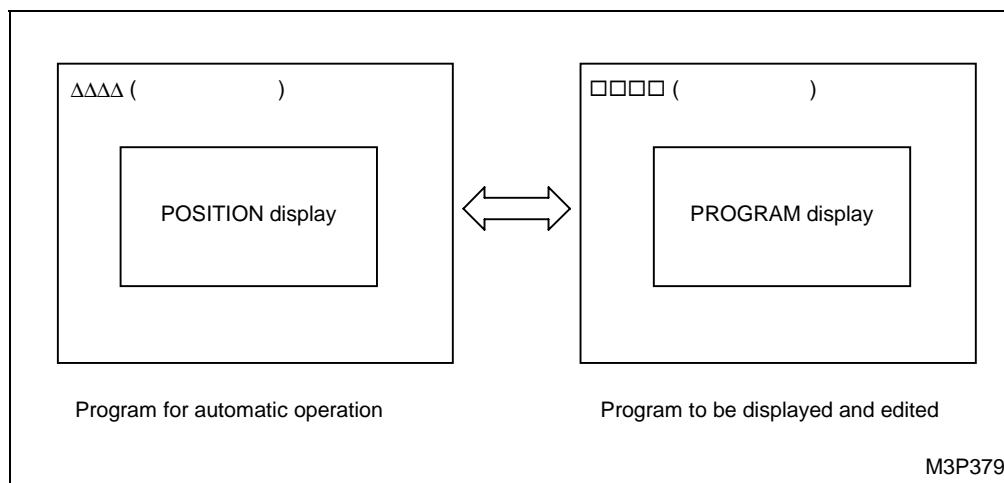
Specify the position using the program coordinates system.

- NOTE -

9 BACKGROUND PROGRAMMING

For the purpose of more efficient programming, M640M Pro permits preparing a program even during automatic operation. This function is called Background programming.

The workpiece number of the program to be executed is entered in the **POSITION** display while the workpiece No. of the program to be created or to be edited is entered in the **PROGRAM** display.



Note 1: It is possible that the displayed workpiece number on the **POSITION** display does not coincide with that on the **PROGRAM** display.

Therefore, perform the operation after confirmation of the workpiece number presented actually on the display.

Note 2: The program in the process of automatic operation and its sub-program can not be edited.

- NOTE -

10 CASE OF APPEARANCE OF ALARM

1. Alarm display

Alarm is displayed either in red or blue.



RedLights up in the event of machine failure.

Blue.....Lights up if erroneous operations are carried out during creating or editing a program.

2. Clearing alarm

When the alarm is displayed during creating and editing the program, clear it by the following procedure.

- (1) Check the alarm number and message in the alarm display section of the screen.
- (2) Refer to the Alarm List for M640M Pro or Alarm list in the Maintenance Manual of the machine and eliminate the cause of the alarm by referring to the alarm No., alarm message, cause of the alarm and action to be taken to eliminate the cause.

- (3) Clear the alarm display with either the clear key  or the reset key .

RedClear the display with the reset key.

BlueClear the display with the clear key.

Note: While background programming is done in the automatic operation mode, pressing the reset key in order to cancel an alarm also has the effect of returning the automatic operation to the initial status.

- NOTE -

11 THREE-DIGIT G-FORMAT

11-1 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.

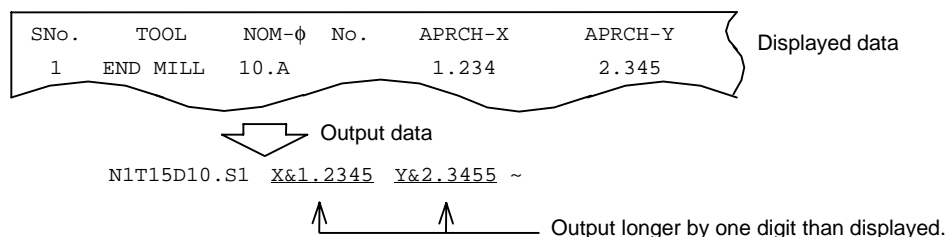
Data that have been output to external units in the three-digit G-format can be edited using a personal computer, and if the edited data are restored into the NC unit, the corresponding original data within the NC unit will be auto-modified according to the required edition.

11-2 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 including the barrier information, 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.
- 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:



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.

11-3 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	<i>Program No.</i> O99999999	(M	G	3	-	2	4	1)	EOB	<i>Description in 3-digit G-format</i>	EOB	EOR %

(With program name)

EOB	<i>Program No.</i> O99999999	(M	G	3	-	2	4	1	:	<i>Program name</i>)	EOB	<i>Description in 3-digit G-format</i>	EOB	EOR %

- Program number
The program number is assigned following "O."
Program numbers are output in eight digits (normal setting; Nos. 1 to 99999999) or four digits (Nos. 1 to 9999) according to the setting of the related parameter. The distinction in the number of digits must be observed even in the designation of the program to be output. An error will result if an illegal program number is designated.
- Identifier
The code "(MG3-241)" succeeding the program number identifies a MAZATROL program described in the three-digit G-format of the M640M Pro.
- Program name
The program name is assigned in the parentheses with the identifier separated by a colon.
The maximum available number of characters is usually 48 for naming a program stored in the NC memory.
An excess in characters will be given away.

2. Unit

- Common unit (ONLY MILL.) G300

UNo.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
U	()	D	Z	A	B	C	X	Y
		0: ONLY MILL.			1: OFF			
					2: 5*2			
					3: OFFSET TYPE			

- Common unit (MILL & TURN.) G300

UNo..	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM
U	()	D	X	I	Z	C	A	S
		1: MILL&TURN.						

- Fundamental coordinates system unit G379

UNo.	WPC No.	Turning spindle rotation center flag	ADD.WPC	X	Y	th	Z
U	A	C	B	X	Y	E	Z
		0: Not turning center		1 - 6: G54 - G59			
		1: Turning center		7 - 16: A - K			
				17 - 64: G54.1P1 - G54.1P48			
				4 (Rotational axis) /4 (Linear axis)		5	6
				J/L		F	K

- Auxiliary coordinates system unit G380

UNo.	U(X)	V(Y)	D(th)	W(Z)
U	X	Y	D	Z

- End unit G301

UNo.	CONTI.	NUMBER	ATC	X	Y	Z	4(Rotational axis) /4(Linear axis)	5	6	ANGLE	DIR.
U	B	C	A	X	Y	Z	J/L	F	K	D	E
											0: CW
											1: CCW
											2: NEAR DIR.

- Sub-program unit G303

UNo.	WORK No.	\$ REPEAT	Measurement flag	\$ V:
U	W	V L	F	0 - 9 Unit skip (0 - 9)
			0: Sub program contains no measurement	-1 - -4 Multi-workpiece machining (A - D)
			1: Sub program contains measurement	

- Manual program mode unit G305

UNo.	TOOL	Tool shape	NOM-φ/(Nominal Diameter/Size)	ID code	Priority No.
U	T	&T	D	S	P

* For nominal diameter of taps and taps for turning, refer to "Tapping unit."

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 (SPECIAL)
43	SPECIAL

&T:	Tool shape
1	OUT OUTER DIAMETER
2	IN INNER DIAMETER
3	EDG EDGE
4	IN INNER (BAK)
5	EDG EDGE (BAK)
17	001
18	002
19	003
20	004
21	005
22	006
23	007
24	008
25	009

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:	Priority No.
0	No priority No.
1 - 99	Priority No. for prior machining
-1 - -99	Priority No. for subsequent machining

- M-code unit G302

UNo.	Priority No.	M1	M2	M3	M4	M5	M6	M7	M8	M9	M10	M11	M12
U	P	MA	MB	MC	MD	ME	MF	MG	MH	MI	MJ	MK	ML

→ Refer to "Manual program mode unit."

- Coordinates measuring unit G304

UNo.	TOOL	NOM-φ	ID code	Priority No.	U. SKIP	\$
U	T	D	S	P	K	V

→ Refer to "Manual program mode unit."

- Pallet changing unit G381

UNo.	PALLET No.	No. of next pallet
U	P	&P

- Process end unit G385

UNo.
U

- Index unit G382

UNo.	TURN POS X	TURN POS Y	TURN POS Z	Angle 4	Angle 5	Angle 6	TURN DIR
U	X	Y	Z	J	D	K	E
							0: CW
							1: CCW
							2: SHORTEST

- Fundamental coordinate shifting unit G306

UNo.	X	Y	Z	4	5	6	θ	MIRROR
U	X	Y	Z	F	J	H	E	M
								0: X-axis mirror image invalid
								1: Valid

- Materials shape unit G307

UNo.	Shape
U	E
	1: OUT
	3: IN

- Workpiece transfer unit G309

UNo.	PAT.	HEAD	SPDL	PUSH
U	P	H	S	J
	0: None	<Case of CHUCK>	0: Spindle stop	0: With pressing
	1: CHUCK	1: 1→2	1: Spindle forward	1: Without pressing
	2: BAR	2: 2→1	2: Spindle reverse	
	3: MOVE	<Case of BAR>	3: Spindle orient	
		1: HEAD1	4: C-axis positioning	
		2: HEAD2	5: Spindle mov. keep	

CHUCK	Z1	Z2	C1	C2	MOVEMENT
K	ZA	ZB	CA	CB	L
0: Chuck open					
1: Chuck close					

- Head selection unit G310

UNo.	TYPE	HEAD
U	P	H
	0: None	0: None
	1: SINGLE	1: HEAD1
	2: SYNCH.	2: HEAD2

- Workpiece measuring unit G308

UNo.	COMPENSATE	OFS-TOOL	Ofs. tool shape	Ofs. tool/ Nom. dia./size	Ofs. tool ID	COMP.DATA	SNS-TOOL
U	H	T	&T	D	I	J	K
	0: YES					0: DIAMETER	
	1: NO					1: LENGTH	

Sensor Nom. dia.	Sensor ID code	No.	INTERVAL	OUTPUT
R	S	P	L	Q
0: No output				
1: Output to RS232C				
2: Output to HDD				

* See the manual program unit for tool shape, ID code and propriety number.

- Tool measuring unit G311

UNo.	COMPENSATE	Tool	Tool shape	Nom. dia./size	ID code	No.	INTERVAL	OUTPUT
U	H	T	&T	D	S	P	L	Q
	0: YES							0: No output
	1: NO							1: Output to RS232C
								2: Output to HDD

* See the manual program unit for tool shape, ID code and propriety number.

- Drilling unit G350

UNo.	DIA	DEPTH	CHMF
U	D	H	C

- RGH CBOR machining unit G351

UNo.	CB-DIA	CB-DEP	CHMF	BTM	DIA	DEPTH
U	&D	&H	C	F	D	H

- RGH BCB machining unit G352

UNo.	CB-DIA	CB-DEP	DIA	DEPTH	CHMF
U	&D	&H	D	H	C

- Reaming unit G353

UNo.	DIA	DEPTH	CHMF	PRE-REAM	CHP
U	D	H	C	A	K
				0: Drilling	
				1: Boring	
				2: End milling	

- Tapping unit G354

UNo.	NOM-	MAJOR- ϕ	PITCH	TAP-DEP	CHMF	CHP	Tap screw type A:	Tap fraction B:
U	*	E	P	H	C	K	1 M	1 1/2
							2 UNn	2 1/4
							3 UN	3 1/8
							4 PT	4 1/16
							5 PF	Nominal diameter D:
							6 PS	Nominal diameter 2 V:
							7 OTHER	

Example:
M10. A1D10.
UNn 1-2 A2D1V2
UN 1H-2 A3D1V2B1
PT 2Q A4D2B2

- Back boring unit G355

UNo.	DIA	DEPTH	BTM	WAL	PRE-DIA	PRE-DEP	CHMF	WAL
U	D	H	I	J	&D	&H	C	&J

- Circular milling unit G356

UNo.	DIA	DEPTH	CHMF	TORNA.	BTM	PRE-DIA	CHMF	PITCH1	PITCH2
U	D	H	C	K	I	&D	&C	E	F
				0: CIRCUL					
				1: TORNADE					
				2: HIGH AC.					

- Counterbore-tapping unit G357

UNo.	NOM-	MAJOR- ϕ	PITCH	TAP-DEP	CHMF	CB-DIA	CB-DEP	CHMF	BTM	CHP
U	*	E	P	H	C	&D	&H	&C	I	K

→ Refer to "Tapping unit."

- Through hole boring unit G358

UNo.	DIA	DEPTH	CHMF	WAL
U	D	H	C	J

- Non-through hole boring unit G359

UNo.	DIA	DEPTH	CHMF	BTM	WAL	PRE-DIA
U	D	H	C	I	J	E

- Stepped through hole boring unit G360

UNo.	CB-DIA	CB-DEP	CHMF	BTM	WAL	DIA	DEPTH	CHMF	WAL
U	&D	&H	&C	&I	&J	D	H	C	J

- Stepped non-through hole boring unit G361

UNo.	CB-DIA	CB-DEP	CHMF	BTM	WAL	PRE-DIA	DIA	DEPTH	CHMF	BTM	WAL
U	&D	&H	&C	&I	&J	E	D	H	C	I	J

- Central linear machining unit G362

UNo.	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z
U	H	Z	R	F	&Z

- Right-hand linear machining unit G363

UNo.	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
U	H	Z	R	F	&Z	&R

- Left-hand linear machining unit G364

UNo.	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
U	H	Z	R	F	&Z	&R

- Outside linear machining unit G365

UNo.	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
U	H	Z	R	F	&Z	&R

- Inside linear machining unit G366

UNo.	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
U	H	Z	R	F	&Z	&R

- Right-hand chamfering unit G367

UNo.	DEPTH	INTER-Z	INTER-R	CHMF
U	H	I	J	C

- Left-hand chamfering unit G368

UNo.	DEPTH	INTER-Z	INTER-R	CHMF
U	H	I	J	C

- Outside chamfering unit G369

UNo.	DEPTH	INTER-Z	INTER-R	CHMF
U	H	I	J	C

- Inside chamfering unit G370

UNo.	DEPTH	INTER-Z	INTER-R	CHMF
U	H	I	J	C

- Face milling unit G371

UNo.	DEPTH	SRV-Z	BTM	FIN-Z
U	H	Z	I	&Z

- End milling-top unit G372

UNo.	DEPTH	SRV-Z	BTM	FIN-Z
U	H	Z	I	&Z

- End milling-step unit G373

UNo.	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R
U	H	Z	I	J	&Z	&R

- Pocket milling unit G374

UNo.	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R	INTER-R	CHMF
U	H	Z	I	J	&Z	&R	K	C

- Pocket milling-mountain unit G375

UNo.	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R
U	H	Z	I	J	&Z	&R

- Pocket milling-valley unit G376

UNo.	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R
U	H	Z	I	J	&Z	&R

- End milling-slot unit G377

UNo.	DEPTH	SRV-Z	SLOT-WID	BTM	WAL	FIN-Z	FIN-R
U	H	Z	D	I	J	&Z	&R

- Inclined-face machining unit (COR. FACE) G378

UNo.	DEPTH	SRV-Z	BTM	FIN-Z
U	H	Z	I	&Z

- C-axis machining units

In the output format of the C-axis machining unit for C-axis drilling, C-axis hole with counter bore (RGH CBOR), etc., data on the face to be machined (M) is sent after the unit number (U) has been sent. Subsequent output formats are the same as those of the normal machining units. See the section on machining units.

Example: C-axis drilling unit G329

UNo.	C-FACE	DIA	DEPTH	CHMF
U	M	D	H	C
	0: None	Same output format as drilling unit		
	1: FACE			
	2: CYLIND			
	3: SLANT (Only point machining)			

C-axis machining unit	3-digit G-code
C-axis DRILLING	G329
C-axis RGH CBOR	G330
C-axis RGH BCB	G331
C-axis REAMING	G332
C-axis TAPPING	G333
C-axis BK CBOR	G334
C-axis CIRC MIL	G335
C-axis CBOR TAP	G336
C-axis BORING T1	G337
C-axis BORING S1	G338
C-axis BORING T2	G339
C-axis BORING S2	G340
C-axis LINE CTR	G341
C-axis LINE RGT	G342
C-axis LINE LFT	G343
C-axis LINE OUT	G344
C-axis LINE IN	G345
C-axis CHMF RGT	G346
C-axis CHMF LFT	G347
C-axis CHMF OUT	G348
C-axis CHMF IN	G349

- Bar-materials machining unit G320

UNo.	PART	CPT-X	CPT-Z	FIN-X	FIN-Z
U	E	X	Z	&X	&Z
	0: None				
	1: OUT (Outside-diameter open type)				
	2: OUT (Outside-diameter middle type)				
	3: IN (Inside-diameter open type)				
	4: IN (Inside-diameter middle type)				
	5: FCE (Front-face open type)				
	6: FCE (Front-face middle type)				
	7: BAK (Back-face open type)				
	8: BAK (Back-face middle type)				

- Copy-machining unit G321

UNo.	PART	CPT-X	CPT-Z	SRV-X	SRV-Z	FIN-X	FIN-Z
U	E	X	Z	I	J	&X	&Z

Refer to "Bar-materials machining unit."

- Corner-machining unit G322

UNo.	PART	FIN-X	FIN-Z
U	E	&X	&Z

Refer to "Bar-materials machining unit."

- Edge-machining unit G323

UNo.	PART	FIN-Z
U	E	&Z

Refer to "Bar-materials machining unit."

- Threading unit G324

UNo.	PART	CHAMF	LEAD	ANG	MULTI	HGT
U	E	C	K	D	R	H

0: None

1: 45 degrees

2: 60 degrees

Refer to "Bar-materials machining unit."

- Grooving unit G325

UNo.	PART	PAT.	No.	PITCH	WIDTH	F.allowance (Grooving pattern #0 - #3) /Overshoot (Grooving pattern #4, #5)
U	E	I	K	F	J	Z

0: #0 (Right-angled or oblique)

1: #1 (Isosceles trapezoidal)

2: #2 (Right-tapered)

3: #3 (Left-tapered)

4: #4 (Right-corner cut-off)

5: #5 (Left-corner cut-off)

Refer to "Bar-materials machining unit."

- Turning drilling unit G326

UNo.	PART	DIA
U	E	D

Refer to "Bar-materials machining unit."

- Turning tapping unit G327

UNo.	PART	NOM-DIA	PITCH
U	E	*	F

Refer to "Bar-materials machining unit."

Refer to "Tapping unit."

- 3-D unit (ROTATE 1) G386

UNo.	DIST/th	MAT-HIGH	FIN	CUT-PROCESS
U	D	H	I	K

CUT-PROCESS	K:
1	R1
2	R1-F2
3	R1-F2-F2
4	F1
5	F1-F2

- 3-D unit (ROTATE 2) G387

UNo.	DIST/th	MAT-HIGH	FIN	CUT-PROCESS
U	D	H	I	K

- 3-D unit (ROTATE 3) G388

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (ROTATE 4) G389

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (PARALL. 1) G390

UNo.	DIST/th	MAT-HIGH	FIN	CUT-PROCESS
U	D	H	I	K

- 3-D unit (PARALL. 2) G391

UNo.	DIST/th	MAT-HIGH	FIN	CUT-PROCESS
U	D	H	I	K

- 3-D unit (PARALL. 3) G392

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (PARALL. 4) G393

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (NORMAL 1) G394

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (NORMAL 2) G395

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- 3-D unit (RULED-S.) G396

UNo.	MAT-HIGH	FIN	CUT-PROCESS
U	H	I	K

- Workpiece measuring sequence

SNo.	PTN	SPT-X	SPT-Y	SPT-Z	FPT-X	FPT-Y	FPT-Z	LIM+	LIM-	BASE	Approach	HEAD
N	A	X	Y	Z	&X	&Y	&Z	V	W	Q	L	R
	1: OUTER X									0: SPT	0: Z dir.	
	2: OUTER Y									1: FPT	1: X dir.	
	3: INNER X											
	4: INNER Y											
	5: X GROOVE											
	6: Y GROOVE											
	7: Z GROOVE											
	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: INNER WIDTH											
	18: INNER GROOVE											

The approach/escape direction of measurement is represented by a combination of L and R.
Ex: Case of INTEGREX e410, e650
L0R0.: ←Z
L1R90.: ↓X
L1R180.: →X

- External measurement

SNo.	PTN	COMP.DATA	MEASURING POINT	TARGET DATA	LIM+	LIM-
N	A	I	J	K	V	W
	19: EXT MIL	0: WEAR X				
	20: EXT TURN	1: WEAR Z				

- Tool measuring sequence

SNo.	PTN	TOL. LENGTH	TOL. DIA.	TOOL EYE	DIR.
N	A	V	W	Q	R
	1: LASER MES.			0: Retract	← : 0
	2: TOOL EYE #1			1: Not retract	↓ : 90
	3: TOOL EYE #2				→ : 180
	4: TOOL EYE #3				
	5: TOOL EYE #4				

- Point machining shape sequence

SNo.	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
N	A	Z	X	Y	DA	DB	TA	TB	F	M	K	P	Q	R
	1: PT													
	2: LIN													
	3: SQR													
	4: GRD													
	5: CIR													
	6: ARC													
	7: CHD													

- C-axis point machining shape sequence (FACE)

SNo.	PTN	Control flag	SPT-Z	SPT-R/X	SPT-th/Y	ANGLE	NUM.	Q	R
N	A	E	Z	X	Y	C	M	Q	R
	1: PT	Bit0=1: SPT-X input							
	5: CIR	Bit0=0: SPT-R input							
	6: ARC	Bit1=1: SPT-Y input							
		Bit1=0: SPT-th input							

- C-axis point machining shape sequence (CYLIND, SLANT)

SNo.	PTN	SQT-X/SHIFT	SPT-R	SPT-th	ANGLE	NUM.	Q	R
N	A	Z	X	Y	C	M	Q	R
	1: PT							
	5: CIR							
	6: ARC							

- Fixed form shape sequence (SQUARE)

SNo.	PTN	P1X	P1Y	P3X	P3Y	CN1	CN2	CN3	CN4	Type of shape sequence
N	A	X	Y	&X	&Y	CA/RA	CB/RB	CC/RC	CD/RD	B
	1: SQR									0: FIX

➤ Set C_ or R_ for corner chamfering or rounding respectively.

- Fixed form shape sequence (CIRCLE)

SNo.	PTN	CX	CY	R	Type of shape sequence
N	A	X	Y	R	B
	2: CIR				0: FIX

- C-axis fixed form shape sequence (FACE, SQUARE)

SNo.	PTN	Control flag	P1Rx/CRx	P1thy/Cthy	P3Rx/R	P3thy	CN1	CN2	CN3	CN4	Type of shape seq.
N	A	E	X	Y	&X	&Y	CA/RA	CB/RB	CC/RC	CD/RD	B
	1: SQUARE	bit0=1: SPT-X input									
		bit0=0: SPT-R input									
		bit1=1: SPT-Y input									
		bit1=0: SPT-th input									
		bit2=1: Diagonal-X input									
		bit2=0: Diagonal-R input									
		bit3=1: Diagonal-Y input									
		bit3=0: Diagonal-th input									
	0: FIX										

Set C_ or R_ for corner chamfering or rounding respectively. ←

- C-axis fixed form shape sequence (FACE, CIRCLE)

SNo.	PTN	Control flag	P1Rx/CRx	P1thy/Cthy	P3Rx/R	Type of shape sequence
N	A	E	X	Y	&X	B
	2: CIR	bit0=1: CENTER-X input				0: FIX
		bit0=0: CENTER-R input				
		bit1=1: CENTER-Y input				
		bit1=0: CENTER-θ input				

- C-axis fixed form shape sequence (CYLID, SQUARE)

SNo.	PTN	P1R /CR	P1th /Cth	P3R/R	P3th	CN1	CN2	CN3	CN4	Type of shape sequence
N	A	X	Y	&X	&Y	CA/RA	CB/RB	CC/RC	CD/RD	B
	1: SQR									0: FIX

➤ Set C_ or R_ for corner chamfering or rounding respectively.

- C-axis fixed form shape sequence (CYLIND, CIRCLE)

SNo.	PTN	P1R/CR	P1th/Cth	P3R/R	Type of shape sequence
N	A	X	Y	&X	B
	2: CIR				0: FIX

- Arbitrary form shape sequence

SNo.	PTN	X	Y	R/th	I	J	P	CNR	RGH/Feedrate set flag	R-FEED	SPT	Type of shape sequence
N	A	X	Y	V	I	J	P	C/R	E	F	S	B
	1: LIN								0: Feedrate		0: No SPT	1: Arbitrary
	2: CW								1: Roughness		1: SPT	
	3: CCW											
	4: FIG-SH											
	5: CW SH											
	6: CCW SH											
	7: REP-EN											

- C-axis arbitrary form shape sequence (FACE)

SNo.	PTN	Control flag	R/x	th/y	RADIUS /th.	I	J	P	CNR	RGH/Feedrate set flag
N	A	D	X	Y	V	I	J	P	C/R	E
		bit0=1: FPT-X input						1: UP		0: Feedrate
		bit0=0: FPT-R input						2: DOWN		1: Roughness
		bit1=1: FPT-Y input						3: LEFT		
		bit1=0: FPT-θ input						4: RIGHT		
		bit3=1: CENTER-X input								
		bit3=0: CENTER-R input								
		bit4=1: CENTER-Y input								
		bit4=0: CENTER-θ input								

R-FEED	SPT	Type of shape sequence
F	S	B
	0: No SPT	1: Arbitrary
	1: SPT	

- C-axis arbitrary form shape sequence (CYLIND)

SNo.	PTN	R	th	RADIU /th.	I	J	P	CNR	RGH/Feedrate set flag	R-FEED	SPT	Type of shape sequence
N	A	X	Y	V	I	J	P	C/R	E	F	S	B
							1: UP		0: Feedrate		0: No SPT	1: Arbitrary
							2: DOWN		1: Roughness		1: SPT	
							3: LEFT					
							4: RIGHT					

- Bar-materials machining/Copy machining sequence

SNo.	PTN	S-CNR	SPT-X	SPT-Z	SPT INTER PT	F-CNR/\$	FIN-X	FIN-Z	FIN INTER PT
N	A	C/R	X	Z	K	&C/&R/Q	&X	&Z	L
	0: None				0: None				0: None
	1: LIN				1: UP				1: UP
	2: TPR				2: DOWN				2: DOWN
	3: □				3: LEFT				3: LEFT
	4: ▢				4: RGT				4: RGT
	5: CTR								

RADIUS/th	FGH/Feedrate set flag	RGH code/ Feedrate	INTER PT flag
I/J	E	F	H
	0: ROUGHNESS		bit0=1: SPT-X?
	1: FEEDRATE/rev		bit1=1: SPT-Z?
			bit2=1: FIN-X?
			bit3=1: FIN-Z?
			bit4=1: SPT-X CONT?
			bit5=1: SPT-Z CONT?
			bit6=1: FIN-X CONT?
			bit7=1: FIN-Z CONT?

- Corner-machining/Edge-machining unit sequence

SNo.	SPT-X	SPT-Z	F-CNR/\$	FIN-X	FIN-Z	FGH/Feedrate set flag	RGH code /Feedrate
N	X	Z	&C/&R/Q	&X	&Z	E	F

- Threading unit sequence

SNo.	SPT-X	SPT-Z	FIN-X	FIN-Z
N	X	Z	&X	&Z

- Grooving unit sequence

SNo.	S-CNR	SPT-X	SPT-Z	F-CNR	FPT-X	FPT-Z	ANGLE	RGH/Feedrate flag	RGH code/ Feedrate	INTER PT flag
N	C/R	X	Z	&C/&R	&X	&Z	J	E	F	H

- Turning drilling and turning tapping sequence

SNo.	SPT-Z	FPT-Z
N	Z	&Z

4. Tool sequence

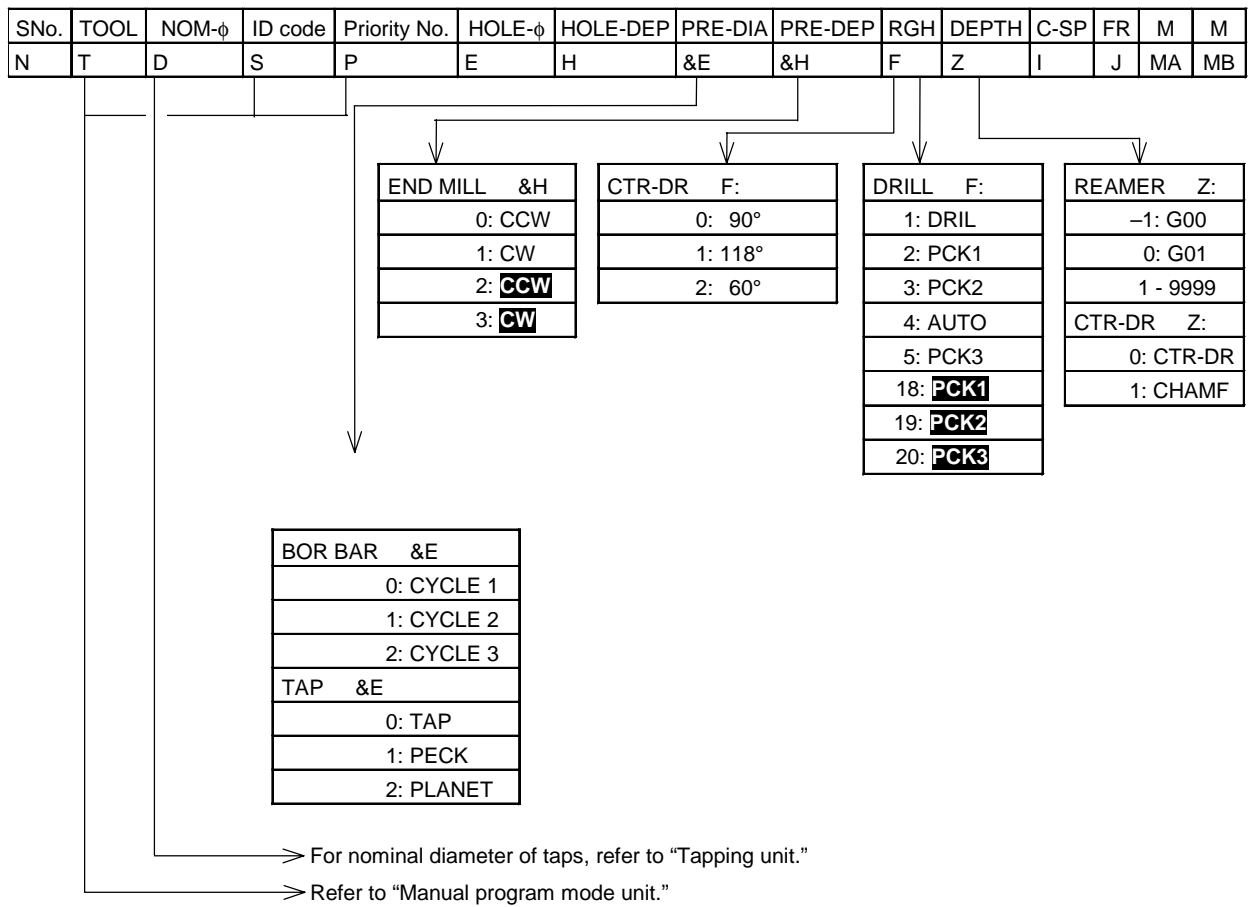
G424U_ Beginning of tool sequence (U: unit No.)

N1
N2
:
:
}

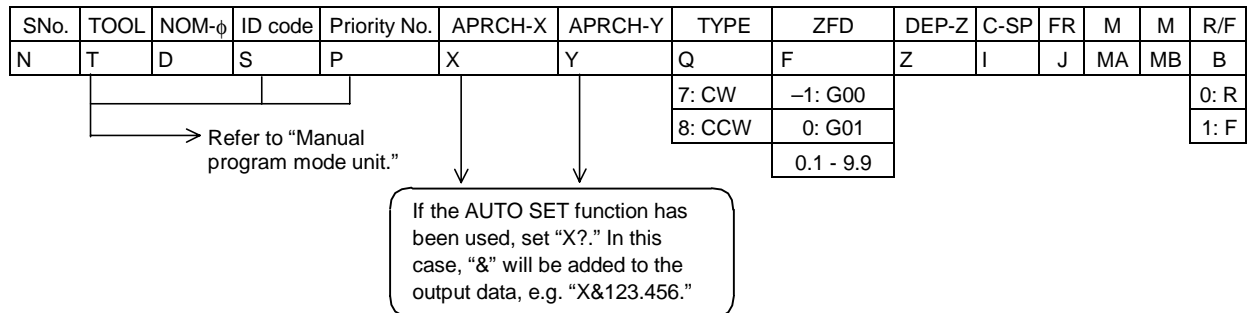
Tool sequence data

G425 End of tool sequence

- Point machining tool sequence



- Line machining tool sequence



- Face machining tool sequence

SNo.	TOOL	NOM	ID code	Priority No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	R/F
N	T	D	S	P	X	Y	Q	F	Z	R	I	J	MA	MB	B
							1: XBI	-1: G00							0: R
							2: YBI	0: G01							1: F
							3: XUN	0.1 - 9.9							
							4: YUN								
							5: XBI-S								
							6: YBI-S								
							7: CW								
							8: CCW								
							9: XB-AS								
							10: YB-AS								

Refer to "Manual program mode unit."

If the AUTO SET function has been used, set "X?". In this case, "&" will be added to the output data, e.g. "X&123.456."

- Turning tool sequence


SNo.	TOOL	Tool shape	Nom. dia./size	ID code	PRI.No.	PAT. (Note)	DEP-1	DEP-2	DEP-3	No of cut passes			
N	T	&T	D	S	P	Q	RA	RB	RC	E			
							FIN-X	FIN-Y	C-SP	FR	M	M	R/F
							X	Y	I	J	MA	MB	F

Refer to "Manual program mode unit".

Note: Machining patterns

Q:	BAR and CORNER units	THREAD unit	T.DRILL unit
0	Perpendicularly upward cutting cycle	#0 STANDARD (Standard pattern)	Drilling cycle (stop hole)
1	High-speed rough-machining cycle	#1 CONST.DEPTH (Constant-threading pattern)	Deep-hole drilling cycle (stop hole)
2	Inside-diameter deep-hole dividing cycle	#2 CONST.AREA (Area-constant pattern)	High-speed deep-hole drilling cycle (stop hole)
3		#0 STANDARD (zigzag threading)	Reaming cycle (stop-hole)
4		#1 CONST.DEPTH (zigzag threading)	Reserve
5		#2 CONST.DEPTH (zigzag threading)	Drilling cycle (through-hole)
6			Deep-hole drilling cycle (through-hole)
7			High-speed deep-hole drilling cycle (through-hole)
8			Reaming cycle (through-hole)

- 3-D machining tool sequence

No.	TOOL	NOM	ID code	Priority No.	APRCH -X	APRCH -Y	TYPE	#T	DEPTH	PITCH	C-SP	FR	M	M	R/F	STATUS
N	T	D	S	P	X	Y	Q	V	Z	W	I	J	MA	MB	B	C
							1: //-1								0: R	
							2:]-1								1: F	
							3: //-2								2: 	
							4:]-2									
							5: XBI									
							6: YBI									
							7: XUN									
							8: YUN									
							9: XBI									
							10: YBI									
							11: XUN									
							12: YUN									

Refer to "Manual program mode unit."

If the AUTO SET function has been used, set "X?". In this case, "&" will be added to the output data, e.g. "X&123.456."

5. TPC

TPC data consists of two types: TPC data (parameter) section and relay-point section. The two types of data are identified as follows:

- The block next to the code G422 (Beginning of TPC data) is always handled as parameter data, and
- Subsequent blocks up to the code G423 (End of TPC data) are handled as relay-point data.

```
G422U_   Beginning of TPC data (U: unit No.)
  :
  :
  :
  :
G423     End of TPC data
```

} TPC data

- The block that succeeds G422U0 (beginning of the TPC data in unit No. 0) is handled as barrier information.

A. TPC data

	A	B	C	D	E	F	H	I	J	K	L	M	P	Q	R	S	Y	Z	&C	&D	&E	&X	&Y	&Z
DRILLING	D1	D3	D16	D17									D41	D42	D91		D45	D46	TC37	TC38	TC39	TC40	F141	TC62
RGH CBOR	D1	D3	D16	D17		D19	D23						D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
RGH BCB	D1	D3	D16	D17	D5	D19		D40					D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
REAMING	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
TAPPING	D1	D3	D16	D17	D22			D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
BK-CBORE	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D33	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
CIRC. MIL	D1		D16	D17		D19	D23						D41	D42	D91	D92			TC37	TC38	TC39	TC40	F141	TC62
CBOR-TAP	D1	D3	D16	D17	D22	D19	D23	D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
BORE T1	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
BORE S1	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
BORE T2	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
BORE S2	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46	TC37	TC38	TC39	TC40	F141	TC62
LINE CTR		E2		E7	E9		E17												TC37	TC38	TC39	TC40	F141	TC62
LINE RGT		E2		E7	E9		E17			E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40	F141	TC62
LINE LFT		E2		E7	E9		E17			E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40	F141	TC62
LINE OUT	E1	E2	E5	E7	E9		E17		E21	E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40	F141	TC62
LINE IN	E1	E2	E5	E7	E9		E17		E21	E22	E23	E24	E25	E95					TC37	TC38	TC39	TC40	F141	TC62
CHMF RGT		E2		E8	E9	E11	E17							E95					TC37	TC38	TC39	TC40	F141	TC62
CHMF LFT		E2		E8	E9	E11	E17							E95					TC37	TC38	TC39	TC40	F141	TC62
CHMF OUT	E1	E2		E8	E9	E11	E17		E21					E95					TC37	TC38	TC39	TC40	F141	TC62
CHMF IN	E1	E2		E8	E9	E11	E17		E21					E95					TC37	TC38	TC39	TC40	F141	TC62
FCE MILL					E9	E12	E15												TC37	TC38	TC39	TC40	F141	TC62
TOP EMIL				E7	E9	E13	E17							E97					TC37	TC38	TC39	TC40	F141	TC62
STEP	E1	E2	E5	E7	E9	E16	E17		E21	E22	E23	E24	E25	E91	E98				TC37	TC38	TC39	TC40	F141	TC62
POCKET	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E92					TC37	TC38	TC39	TC40	F141	TC62
PCKT MT	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E93					TC37	TC38	TC39	TC40	F141	TC62
PCKT VLY	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E94	E98				TC37	TC38	TC39	TC40	F141	TC62
SLOT				E7	E9		E17		E21					E96					TC37	TC38	TC39	TC40	F141	TC62
ANG. FACE					E9	E12																		
MMS	L1	L2	K21	K22															TC37	TC38	TC39	TC40	F141	TC62
WORK MES	K17	K18	K19	K23															TC37	TC38	TC39	TC40	F141	TC62
TOOL MES	K17	K18	K20																TC37	TC38	TC39	TC40	F141	TC62
TRANSFER		TC44	TC57	TC58	TC59																			
BAR	TC67	TC68	TC1	TC5	TC6														TC37	TC38	TC39	TC40	F141	TC62
COPY	TC7																		TC37	TC38	TC39	TC40	F141	TC62
CORNER	TC67	TC68	TC1																TC37	TC38	TC39	TC40	F141	TC62
FACING			TC56	TC76															TC37	TC38	TC39	TC40	F141	TC62
THREAD		TC41	TC77	TC78			TC82												TC37	TC38	TC39	TC40	F141	TC62
T.GROOVE	TC52	TC42	TC43	TC73	TC74		TC75												TC37	TC38	TC39	TC40	F141	TC62
T.DRILL	TC47	TC20	TC11	TC12															TC37	TC38	TC39	TC40	F141	TC62
T.TAP	TC21	TC22																	TC37	TC38	TC39	TC40	F141	TC62

D1, E2 etc. denote parameter type.

The TPC output format of C-axis point-machining and C-axis line-machining is the same as that of normal point-machining and line-machining.

B. Relay point

Relay point type	Relay point setting	X1	Y1	Z1	X2	Y2	Z2	X3	Y3	Z3
A	B	XA	YA	ZA	XB	YB	ZB	XC	YC	ZC
0: APPROACH	0: AUTO									
1: ESCAPE	1: MANU									

C. Barrier information

- Barrier information of turning spindle

Type	JAW No	JAW CODE/JAW NAME	CHUCK	GRIP DIA
A	B	C/()	D	E
0: T.SPIDEL 1			0: INVALID	
1: T.SPIDEL 2			1: VALID	

- Barrier information of tailstock

Type	Use of tailstock	TAIL BARRIER	TAIL STOCK	REF.TO T/S
A	B	C	D	E
1: (Fixed)	0: UNUSED	0: INVALID		
	1: USED	1: VALID		

11-4 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

G10L41T_P_C_D_S_I_E_H_K_R_J_Q_()A_N_W_

TNo.	PKNo.	TOOL	NOM-φ	ID CODE	INTERFER.code	ACT-φ	LENGTH	LENG COMP.	AUXIL	
T	P	C	D	S	I	E	H	K	R/B/Y	
					LIFE TIME	CUT TIME	T.-MAT	BORDER	STATUS1	STATUS2
					J	Q	()	A	N	W

- Turning tool (GENERAL, GROOVE, THREAD, SPECIAL)

G10L41T_P_C_&C_D_S_I_H_&H_E_K_R_A_F_J_Q_Z_()N_W_

TNo.	PKNo.	TOOL	Tool shape	NOM-φ	ID CODE	INTERFER.code	LENGTH A	LENGTH B	WIDTH	CUT DIR.	
T	P	C	&C	D	S	I	H	&H	E	K	
			NOSE-R	CUT ANGLE /GRV DEPTH	EDG-ANG /TIP-WID	LIFE TIME	CUT TIME	LBB No.	MAT	STATUS1	STATUS2
			R	A	F	J	Q	Z	()	N	W

- Turning tool (T.DRILL, T.TAP)

G10L41T_P_C_&C_D_S_I_H_E_K_F_J_Q_(_)N_W_

TNo.	PKNNo.	TOOL	Tool shape	NOM-φ	ID CODE	INTERFER. code	LENGTH	ACT-φ	CUT DIR.
T	P	C	&C	D	S	I	H	E	K

EDG-ANG	LIFE TIME	CUT TIME	MAT	STATUS1	STATUS2
F	J	Q	()	N	W

C:	TOOL
1	CTR-DR
2	DRILL
3	REAMER
4	TAP(M)
5	TAP(UN)
6	TAP(PT)
7	TAP(PF)
8	TAP(PS)
9	TAP(OTHER)
10	BCK FACE
11	BOR BAR
12	B-B BAR
13	CHAMFER
14	FCE MILL
15	END MILL
16	OTHER
17	CHIP VAC
18	TOL SENS
19	BAL EMIL
33	GENERAL
34	GROOVE
35	THREAD
36	T.DRILL
37	T.TAP(M)
38	T.TAP(UN)
39	T.TAP(PT)
40	T.TAP(PF)
41	T.TAP(PS)
42	T.TAP(OTHER)
43	SPECIAL

&C:	TOOL (Section to be machined)
1	OUT OUTER DIAMETER
2	IN INNER DIAMETER
3	EDG EDGE
4	I INNER (BAK)
5	EDG EDGE (BAK)
17	001
18	002
19	003
20	004
21	005
22	006
23	007
24	008
25	009

I:	Interference code
0	Normal diameter
1	MG+
2	MG-
3	Large diameter
4	Small diameter

AUXIL.	
R	End-mill corner R
B	Number of drill cutting edges
Y	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

K:	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

G10L42T_Q_V_P_K_S_R_J_I_D_H_E_N_

TNo.	HOLDER	TIP POS.	THRUST F.	HORSE PW	MAX.ROT.	Tap returning feedrate	GROUP No.	IDNo.
T	Q	V	P	K	S	R	J	I

ACT- ϕ CO.	LENG. CO.	ACT- ϕ CO. No.	LENG. CO. No.
D	H	E	N

- Turning tool

G10L42T_Q_R_V_J_I_E_X_Y_Z_&X_&Y_&Z_

TNo.	HOLDER	HLD. TYPE	INDEX ANG.	GROUP No.	IDNo.	OFFSET No.	WEAR COMP.X	WEAR COMP.Y	WEAR COMP.Z
T	Q	R	V	J	I	E	X	Y	Z

MAX WEAR-X	MAX WEAR-Y	MAX WEAR-Z
&X	\$Y	&Z

2. Tool offset**A. Type A**

G10L10P_R_

B. Type B

Geometric offset for tool length G10L10P_R_

Defacement for tool length G10L11P_R_

Geometric offset for tool radius G10L12P_R_

Defacement for tool radius G10L13P_R_

C. Type C

Geometric offset ZG10L10P_R_

Tool wear compensation ZG10L11P_R_

Geometric offset nose radiusG10L12P_R_

Tool wear compensation nose radius.....G10L13P_R_

Geometric offset X.....G10L14P_R_

Tool wear compensation X.....G10L15P_R_

Geometric offset Y.....G10L16P_R_

Tool wear compensation Y.....G10L17P_R_

Direction.....G10L18P_R_

Offset No.	OFFSET
P	R

3. Tool file

A. End mill, face mill and ball-end mill

G10L49P_C_D_S_()R_H_A_

Tool file No.	TOOL	NOM-φ	ID code	MAT.	DEPTH	No.	ANG
P	C	D	S	()	R	H	A

B. Chamfering cutter

G10L49P_C_D_S_E_()A_

Tool file No.	TOOL	NOM-φ	ID code	MIN-φ	MAT.	ANG
P	C	D	S	E	()	A

➤ Refer to "Tool data."

4. Chuck jaw

G10L69P_M_A_B_C_D_E_N_

JAW No.	CHUCK JAW	Size A	Size B	Size C	Size D	Size E	JAW CODE/JAW NAME
P	M	A	B	C	D	E	N()
	1: OD JAW						
	2: ID JAW						

5. Cutting condition

A. Cutting condition (WORK-MAT)

G10L52P_()

Material No.	WORK-MAT
P	()

B. Milling cutting condition (TOOL-MAT, C-SP, FR)

G10L_P_S_F_()

DRILL	CTR-DR	REAMER	TAP	BOR BAR	B-B BAR	BK FACE	CHAMFER	END MILL	FCE MILL	B-E-MIL	OTHER
G10L53	G10L54	G10L55	G10L56	G10L57	G10L58	G10L59	G10L60	G10L61	G10L62	G10L63	L10L64

No.	C-SP	FR	TOOL-MAT.
P	S	F	()

C. Cutting condition for turning

G10L65P_A_B_C_D_

No.	R-SPD	F-SPD	R-FEED	R-DEPTH
P	A	B	C	D

D. Cutting condition for turning (PERCENTAGE)

G10L66P_A_B_C_D_

No.	R-SPD	F-SPD	R-FEED	R-DEPTH
P	A	B	C	D

E. Cutting condition for turning (PERCENTAGE)

G10L67P_A_B_C_D_()

No.	R-SPD	F-SPD	R-FEED	R-DEPTH	T.-MAT.
P	A	B	C	D	()

6. Workpiece offset**A. Standard**

G10L2P_

P0: Coordinates system shifting P1: G54 P2: G55 P3: G56 P4: G57
P5: G58 P6: G59

B. Additional workpiece coordinate system

G10L20P_

P1: G54.1P1 to P48: G54.1P48

No.	X	Y	Z	4	5	6	7
P	X	Y	Z	A	B	C	D

7. Parameter**A. User parameter**

G10L50 User parameter setting start

D1Z1 1 is set to D1 (example).

D2Z2

⋮

G11 User parameter setting end

D:	E:	F:	I:	TC:	G:	H:
D1 - D108	E1 - E108	F1 - F154	I1 - I224	T1 - T154	Q1 - Q132	H1 - H108

B. Machine parameter

G10L51 Machine parameter setting start

J1Z1 1 is set to J1 (example).

J2Z1

⋮

G11 Machine parameter setting end

J:	K:	L:	M:	N:	S:	SA:	BA:
J1 - J108	K1 - K108	L1 - L108	M1 - M308	N1 - N308	S1 - S308	W1 - W352	X1 - X132

A	B:	C:	O:	SP	SV:	R REGISTER
A1 - A108	B1 - B108	C1 - C108	Y1 - Y108	P1 - P1536	V1 - V1344	R2100 - R2527 R10500 - R11999

C. Pitch error compensation

G10L47 Pitch error compensation setting start
 X1A0B255C0D0E0
 P1Z1 1 is set to #1 (example).
 P2Z1
 ⋮
 X2A0B255C0D0E0
 P1Z1
 ⋮
 G11 Pitch error compensation setting end

AXIS	P1	P2	P3	P4	P5
X	A	B	C	D	E
1: X					
2: Y					
3: Z					
4: 4					
⋮					
14: 14					

Parameter data should not be changed until you have fully understood the meaning of the respective parameter.

8. Additional WPC

G10L43P_X_Y_E_Z_A_B_C_

WPC No.	X	Y	th	Z	A	B	C
P	X	Y	E	Z	A	B	C
1:A							
2:B							
3:C							
4:D							
5:E							
6:F							
7:G							
8:H							
9:J							
10:K							

9. Macro variable

A. In case of foreground

G10L44#100 = 100 100 is set to #100.

B. In case of background

G10L45#100 = 100 100 is set to #100.

10. Pallet management

G10L46P_U_R_W_S_J_N_K_M_A_B_C_D_Q_X_Y_Z_E_

No.	Process No.	PALLET No.	WNo.	STATE	ORDER	NUMBER	UNIT SKIP SWITCH	MULTIPLE SWITCH, MAIN
P	U	R	W	S	J	N	K	M

1: READY

2: CUTTING

3: FINISHED

4: UNFINISH

MULTIPLE SWITCH A	MULTIPLE SWITCH B	MULTIPLE SWITCH C	MULTIPLE SWITCH D	Set up order
A	B	C	D	Q

JIG OFFSET X	JIG OFFSET Y	JIG OFFSET Z	Error No.
X	Y	Z	E

11. Head offset

G10L48P_H_I_F_J_X_Y_Z_S_

OFFSET	HEAD No.	TYPE	ROT. DIR	SPDL. CMD	OFFSET X	OFFSET Y	OFFSET Z	RPM
P	H	I	F	J	X	Y	Z	S

1: V/H→V

1: F

1: 0

2: V/H→H

2: R

2: R

3: V

3: R0

4: H

5: SLN

6: CVR

12. 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	()

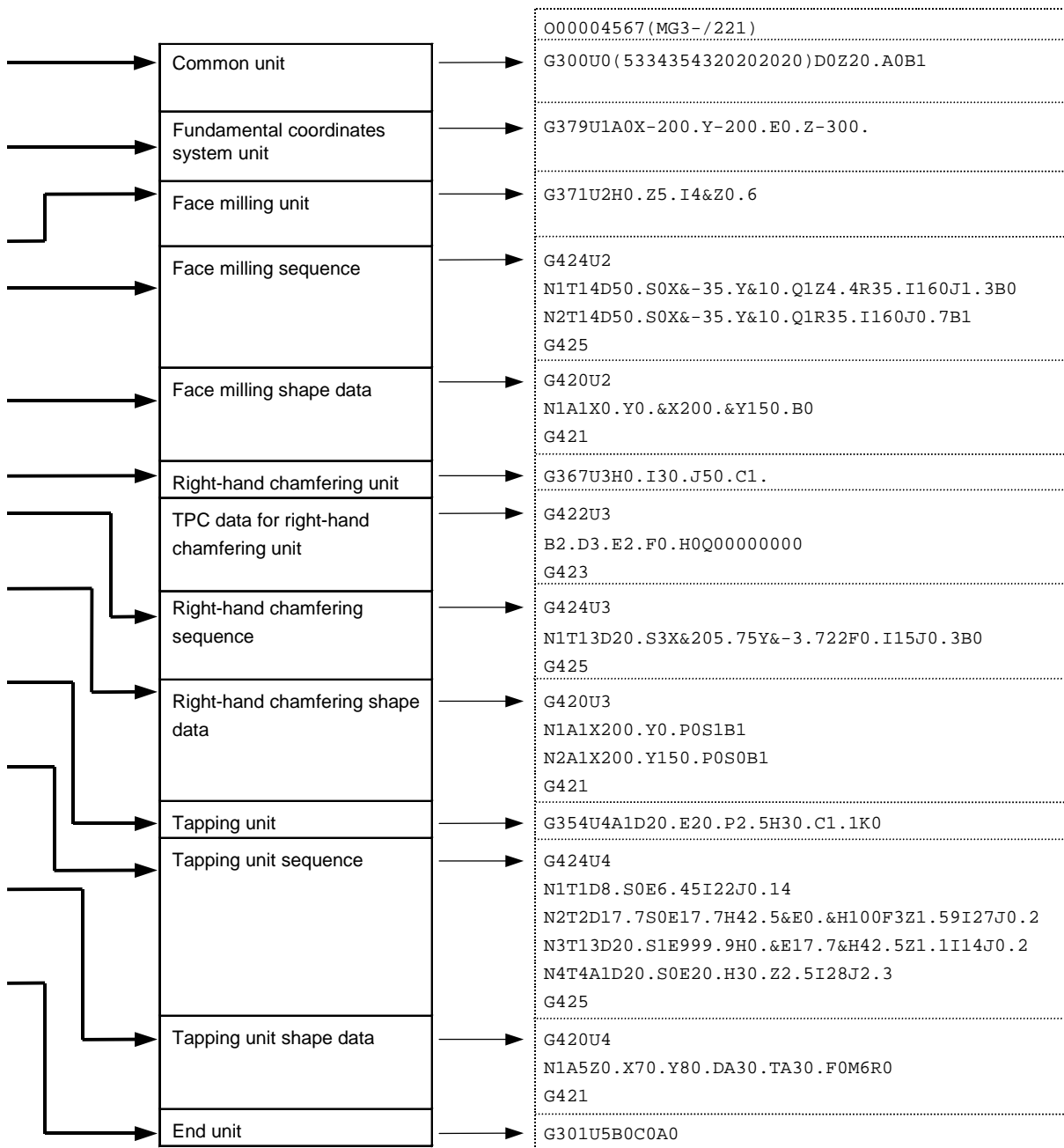
B. Long-term check item

G10L70P_()

Check No.	Check item
P	()

Example of output in use of 3-digit G-format

UNO.	MAT	TYPE	INITIAL-Z	ATC MODE	MULTI MODE	MULTI	FLAG	PITCH-X	PITCH-Y				
0	CBN	STL ONLY MILL.	20.	0	OFF		◆	◆	◆				
UNO.	UNIT	ADD.	WPC	X	Y	th	Z	4					
1	WPC-0			-200.	-200.	0.	-300.	◆					
UNO.	UNIT	DEPTH	SRV-Z		BTM	WAL	FIN-Z	FIN-R					
2	FCE MILL	0.	5.		4	◆	0.6	◆					
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	DEP-R	C-SP	FR	M	M
R1	FCE MILL	50.		-35.	10.	XBI	◆	4.4	35.	160	1.3		
F1	FCE MILL	50.		-35.	10.	XBI	◆	◆	35.	160	0.7		
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4				
1	SQR	0.	0.	200.	150.								
UNO.	UNIT	DEPTH	INTER-Z	INTER-R		CHMF							
3	CHMF RGT	0.	30.	50.		1.							
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
1	CHAMFER	20.C		205.75	-3.722	◆	G01	◆	◆	15	0.3		
FIG	PTN	X	Y	R/th	I	J	P	CNR					
1	LINE	200.	0.										
2	LINE	200.	150.										
UNO.	UNIT	NOM-	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP						
4	TAPPING	M20	20.	2.5	30.	1.1	0						
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	HOLE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	8.		6.45	◆	◆	◆	90°	CTR-D	22	0.14		
2	DRILL	17.7		17.7	42.5	0	100	PCK 1	T1.59	27	0.2		
3	CHAMFER	20.A		999.9	0.	17.7	42.5	◆	C1.1	14	0.2		
4	TAP	M20		20.	30.	◆	◆	FIX	P2.5	28	2.3		
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q R
1	ARC	0.	70.	80.	30.	◆	30.	◆	◆	6	◆	◆	◆ 0
UNO.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE				
5	END	0	0	0				◆	◆				



- NOTE -

APPENDIX

CONTENTS

	Page
APPENDIX-1 PROGRAM EXAMPLES.....	1
APPENDIX-2 WHAT TO DO IN SUCH A CASE?	7

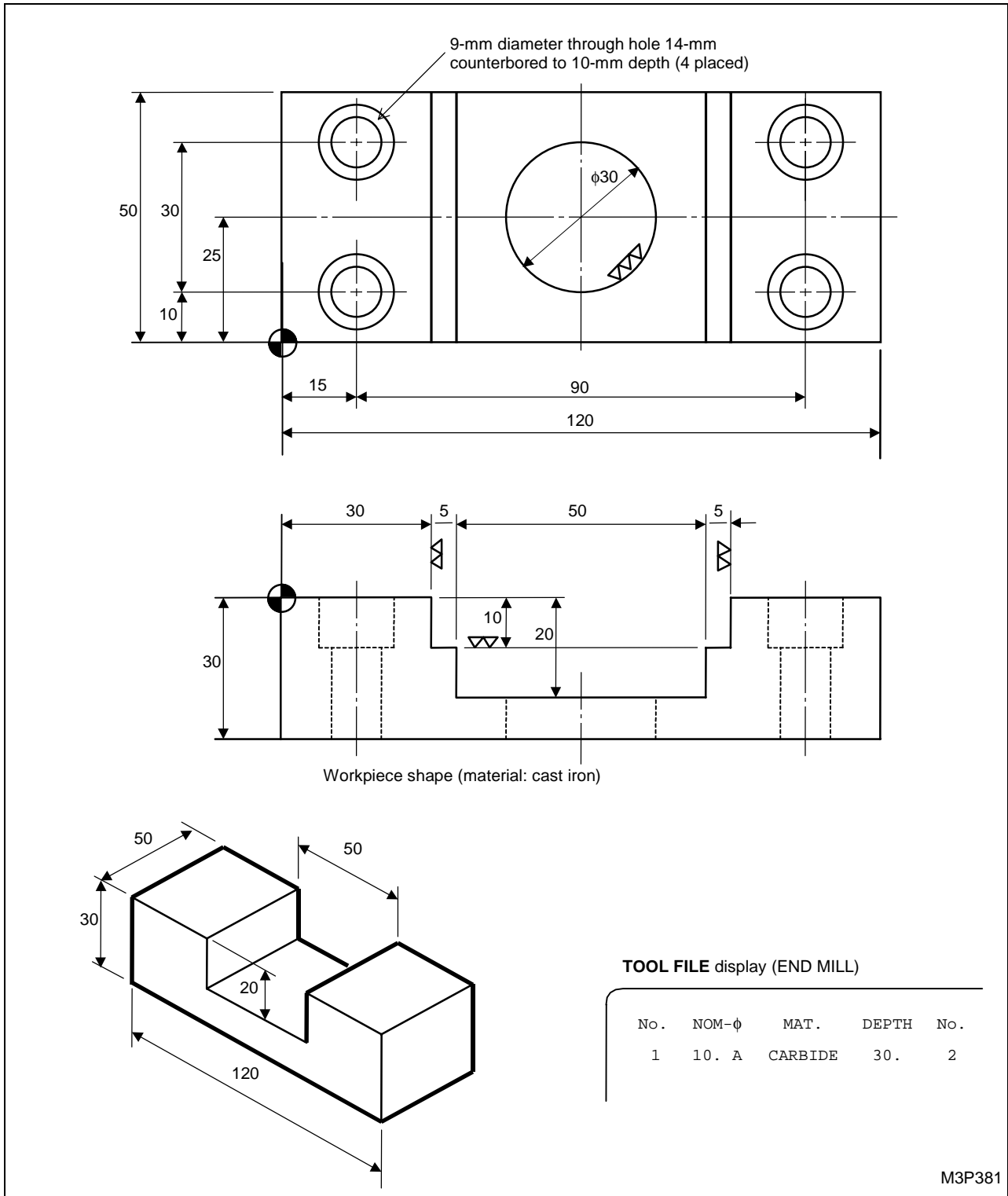
- NOTE -

APPENDIX

APPENDIX-1 PROGRAM EXAMPLES

Program examples are given below.

Example 1:



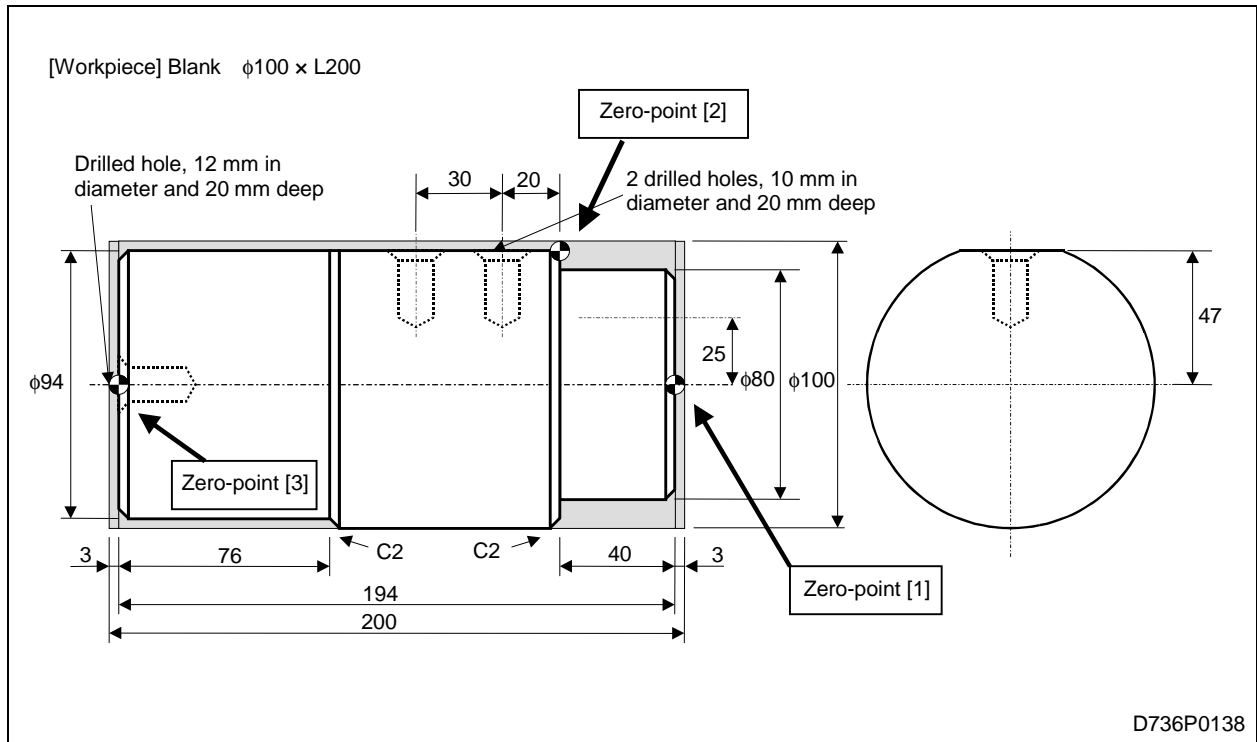
APPENDIX

	MAT	TYPE	INITIAL-Z	ACT	MODE	MULTI	MODE	MULTI	FLAG	PITCH-X	PITCH-Y			
0	CST	IRN ONLY MILL.	20.	1		OFF		◆	◆	◆				
UNO.	UNIT	ADD.	WPC	X	Y	th	Z	4						
1	WPC-1			-100.	-200.	0.	-300.	0.						
UNO.	UNIT	DEPTH	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R						
2	LINE RGT	10.	10.	5.	3		0.	0.						
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
R1	END MILL	10.A		?	?	◆	G00	10.	◆	60	0.039			
FIG	PTN	X	Y	R/th	I	J	P	CNR						
1	LINE	30.	0.											
2	LINE	30.	50.											
UNO.	UNIT	DEPTH	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R						
3	LINE LFT	10.	10.	5.	3		0.	0.						
SNO.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
R1	END MILL	10.A		?	?	◆	G00	10.	◆	60	0.039			
FIG	PTN	X	Y	R/th	I	J	P	CNR						
1	LINE	90.	0.											
2	LINE	90.	50.											
UNO.	UNIT	CB-DIA	CB-DEP	CHMF	BTM	DIA	DEPTH							
4	RGH CBOR	14.	10.	0.	1	9.	30.							
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP.	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20.		9.	◆	◆	◆	90°	◆	20	0.2			
2	DRILL	9.		9.	30.	◆	◆	PCK1	T4.5	22	0.119			
3	END MILL	10.A		14.	10.	9.	◆	1	T6.	51	0.025			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	SQR	0.	15.	10.	0.	90.	90.	30.	0	2	2	0	0	1
UNO.	UNIT	DIA	DEPTH	CHMF	WAL									
5	BORE T1	30.	10.	0.	5									
SNO.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP.	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20.		10.	◆	◆	◆	90°	◆	20	0.2			
2	DRILL	24.		24.	11.	◆	◆	DRILL	T12.	25	0.253			
3	BOR BAR	28.5		28.5	11.	CYCLE 1	0.	0	T 2.25	34	0.126			
4	BOR BAR	29.5		29.5	11.	CYCLE 1	0.	0	T 0.5	38	0.091			
5	BOR BAR	30.		30.	11.	CYCLE 1	0.	0	T 0.25	42	0.066			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	-20.	60.	25.	◆	◆	◆	◆	◆	◆	◆	0	0	0
UNO.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE					
6	END	0	0											

APPENDIX

UNo.	MAT	TYPE	INITIAL-Z	ACT	MODE	MULTI	MODE	MULTI	FLAG	PITCH-X	PITCH-Y			
0	ALUMINUM	ONLY MILL.	20.	1		OFF		◆		◆	◆			
UNo.	UNIT	TYPE	ADD. WPC	X	Y	th	Z	4						
1	WPC-1		-100.	-200.	0.	-300.	0.							
UNo.	UNIT	DEPTH	SRV-Z			BTM	WAL	FIN-Z	FIN-R					
2	STEP	10.	10.			3	3	0.	0.					
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
R1	END MILL	20.A		?	?	CCW	G00	10.	14.	201	0.055			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4					
1	SQR	0.	0.	80.	80.				R 40.					
2	CIR	30.	50.	20.	◆	◆	◆	◆	◆					
UNo.	UNIT	NOM-	MAJOR-φ	PITCH	TAP-DEP	CHMF	CHP							
3	TAPPING	UN1Q-20	6.35	1.27	10.	0.6	0							
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP.	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20.		6.45	◆	◆	◆	90°	◆	48	0.2			
2	DRILL	5.3		5.3	17.35	◆	◆	PCK1	T1.59	54	0.076			
3	TAP	UN1Q-20		6.35	10.	◆	◆	FIX	P1.27	18	1.27			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	ARC	-10.	10.	70.	0.	-90.	60	◆	1	4	◆	◆	0	0
UNo.	UNIT	DEPTH	INTER-Z	INTER-R				CHMF						
4	CHMF OUT	0.	10.	99.				2.						
SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M	
R1	CHAMFER	20.A		?	?	CW	G00	◆	◆	105	0.3			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4					
1	CIR	30.	50.	20.	◆	◆	◆	◆	◆					
UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE					
5	END	0	0											

Example 3: Multi-face machining [The turning, milling, and transfer operations that use INTEGREX H (horizontal type)]



Program

UNO.	MAT	TYPE	OD-MAX	ID-MIN	LENGTH	WORK FACE	ATC MODE	RPM					
0	CBN STL	MILL&TURN.	100.		200.	3.	0						
UNO.	UNIT	TYPE	HEAD										
1	HEAD	SIN	1										
UNO.	UNIT	ADD.WPC	X	Y	th	Z	C						
2	WPC- 0		T.CENTER	T.CENTER	0.	-500.	0.		Zero-point [1]				
UNO.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE B	ANGLE C							
3	INDEX	0.			90.	0.							
UNO.	UNIT	PART	FIN-Z										
4	CORNER	FCE	0.1										
SNo.	TOOL	NOM.	No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1	GENERAL OUT	80.	R	◆	3.	◆	◆	◆	◆	200	0.3		
F2	GENERAL OUT	60.	F	◆	◆	◆	◆	◆	0.	300	0.1		
FIG		SPT-X	SPT-Z	FPT-X	FPT-Z					FGH			
1		100.	3.	0.	0.					▼▼▼5			
UNO.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z							
5	BAR	OUT	100.	0.	0.3	0.1							
SNo.	TOOL	NOM	No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M
R1	GENERAL OUT	80.	A	0	5.	◆	◆	◆	◆	200	0.3		
F2	GENERAL OUT	60.	B	◆	◆	◆	◆	0.	0.	300	0.1		
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/\$	RADIUS/th	RGH				
1	LINE C	2.	◆	◆	80.	40.		◆	▼▼▼5				
2	LINE C	2.	◆	◆	100.	42.		◆	▼▼▼5				
UNO.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z		SHIFT-C	COORD.th						
6	WPCSHIFT	47.	0.	-40.		0.	0.						
UNO.	UNIT	DEPTH	SRV-Z		BTM	FIN-Z							Zero-point [2]
7	FCE MILL	0.	3.		1	0.							

APPENDIX

SNo.	TOOL	NOM-φ	No.	APPCH-X	APPCH-Y	TYPE	ZFD	DEP-Z	DID-R	C-SP	FR	M	M	
R1	FCE MILL	80.	A	?	?	XB X	◆	3.	56.	140	0.2			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y			CN1	CN2	CN3		CN4		
1	SQR	0.	-20.	78.	20.									
UNo.	UNIT	DIA	DEPTH	CHMF										
8	DRILLING	10.	20.	2.										
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20.	A	14.	◆	◆	◆	90°	CTR-D	60	0.1			
2	DRILL	10.	A	10.	20.	0.	100	DRIL T	5.	60	0.1			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	LINE	0.	20.	0.	0.	◆	30.	◆	0	2	◆	◆	0	0
UNo.	UNIT	HEAD	SPDL	PUSH	CHUCK	W1	W2	MOUVEMENT	C1	C2				
9	TRANSFER	1→2	4	1	◆	-600.	0.	600.	0.	0.				
UNo.	UNIT	TYPE	HEAD											
10	HEAD	SIN	2											
UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z				SHIFT-C	COORD.th					
11	WPCSHIFT	0.	0.	0.				0.	0.					
UNo.	UNIT	PART	FIN-Z											
12	CORNER	BAK	0.1											
SNo.	TOOL	NOM.	No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	
R1	GENERAL OUT	80.	C	◆	5.	◆	◆	◆	◆	200	0.3			
F2	GENERAL OUT	60.	D	◆	◆	◆	◆	◆	0.	300	0.1			
FIG		SPT-X	SPT-Z	DEP-X	DEP-Z									FGH
1		100.	-197.	0.	-194.									▼▼▼5
UNo.	UNIT	PART	CPT-X	CPT-Z	FIN-X	FIN-Z								
13	BAR	OUT	100.	194.	0.3	0.1								
SNo.	TOOL	NOM.	No.	PAT.	DEP-1	DEP-2/NUM.	DEP-3	FIN-X	FIN-Z	C-SP	FR	M	M	
R1	GENERAL	OUT	80.	C	0	5.	◆	◆	◆	200	0.3			
F2	GENERAL	OUT	60.	D	◆	◆	◆	0.	0.	300	0.1			
FIG	PTN	S-CNR	SPT-X	SPT-Z	FPT-X	FPT-Z	F-CNR/\$	RADIUS/th	RGH					
1	LINE	C	2.	◆	◆	94.	118.		◆					▼▼▼5
2	LINE	C	2.	◆	◆	100.	116.		◆					▼▼▼5
UNo.	UNIT	TURN	POS X	TURN	POS Y	TURN	POS Z	ANGLE B	ANGLE C					
14	INDEX	0.						180.	0.					
UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z				SHIFT-C	COORD.th					
15	WPCSHIFT	0.	0.	-194.				0.	0.					
UNo.	UNIT	DIA	DEPTH	CHMF										
16	DRILLING	12.	20.	2.										
SNo.	TOOL	NOM-φ	No.	HOLE-φ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	20.	A	16.	◆	◆	◆	90°	CTR-D	60.	0.1			
2	DRILL	12.	A	12.	20.	0.	100	DRIL T	6.	60.	0.1			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0.	0.	0.	◆	◆	◆	◆	◆	◆	◆	0	0	0
UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	B	ANGLE					
17	END													

Zero-point [1]

Zero-point [3]

APPENDIX-2 WHAT TO DO IN SUCH A CASE?

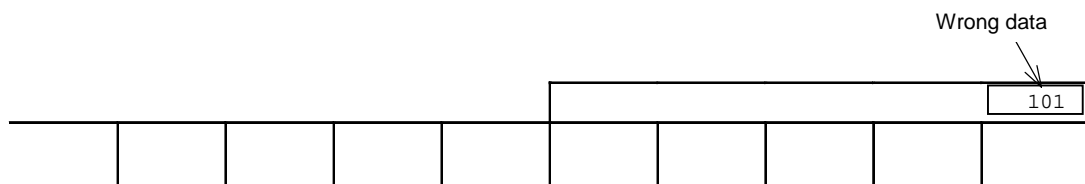
If an erroneous key operation is carried out during creation of a program, you may be placed at a loss what to do. Consult this Appendix in such cases; it offers a description of how to act in case of trouble.


Here, the following states of trouble are covered:

1. A wrong numeric key has been pressed.
2. Wrong data has been input (data change).
3. Wrong data has been input (data erasure).
4. The display selector key or the menu selector key has been pressed inadvertently.
5. How to add a unit(s).
6. How to erase unit(s).
7. How to add a tool sequence(s).
8. How to erase a tool sequence(s).

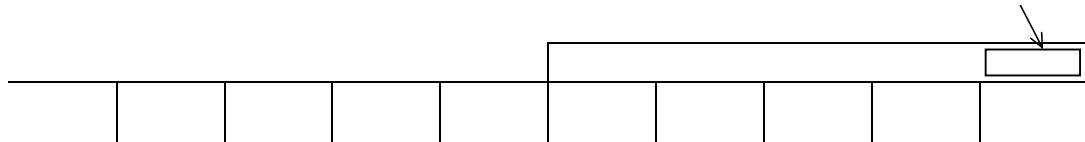
1. Wrong numeric key has been pressed.

⇒ Operate as follows:



- (1) Press the clear key  to erase the data in the data selection area .

The wrong data has been erased.







- (2) Press the correct numeric key(s).

2. Wrong data has been input (data change).





⇒ Operate as follows:

SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y
R1	END MILL	10.A		?	?
FIG	PTN	P1X/CX	PLY/CY	P3X/R	P3Y
1	SQR	5.	5.	146.	95.

This data is to be changed to 145.

- (1) Position the cursor on the wrong data using the cursor keys (   ).

P1Y/CY	P3X/R	P3Y
5.	146	95.

- (2) Input correct data. In this case, press     in this order.

SNo.	TOOL	NOM-φ	No.	APRCH-X	APRCH-Y
R1	END MILL	10.A		?	?
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y
1	SQR	5.	5.	145.	95.





The wrong data has been changed to a correct one.

3. Wrong data has been input (data erasure).


⇒ Operate as follows:

P1X/CX	P1Y/CY	P3X/R	CN1
5.	5.	145.	R5.

This data is to be erased.

- (1) Position the cursor on the wrong data using the cursor keys (   ).

P3Y	CN1
95.	R5

- (2) Press the data cancellation key .

P1X/CX	P1Y/CY	P3X/R	P3Y	CN1
5.	5.	145.	95.	

The wrong data has been erased.

4. The display selector key or the menu selector key has been pressed inadvertently.

⇒ Pressing of the menu selector key causes the original menu to be resumed.

5. How to add a unit(s).

⇒ Refer to Part 2, Subsection 2-2-3, "1. Insertion of a unit."

6. How to erase a unit(s).

⇒ Refer to Part 2, Subsection 2-2-4, "1. Deletion of the unit."

7. How to add a tool sequence(s).

⇒ Refer to Part 2, Subsection 2-2-3, "2. Insertion of a tool sequence."

8. How to erase a tool sequence(s).

⇒ Refer to Part 2, Subsection 2-2-4, "2. Deletion of the tool sequence."