PROGRAMMING MANUAL

for

MAZATROL FUSION 640M
MAZATROL FUSION 640M 5X
MAZATROL FUSION 640M NEXUS
MAZATROL PROGRAMMING

MANUAL No. : H735PG0019E

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

03. 2004

SAFETY PRECAUTIONS

Preface

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

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

Rule

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

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

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

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

Basics

WARNING

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

Remarks on the cutting conditions recommended by the NC

WARNING

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

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

Programming

WARNING

- Fully check that the settings of the coordinate systems are correct. Even if the designated program data is correct, errors in the system settings may cause the machine to operate in unexpected places and the workpiece to abruptly move out from the machine in the event of contact with the tool.
- During surface velocity hold control, as the current workpiece coordinates of the surface velocity hold control axes approach zeroes, the spindle speed increases significantly. For the lathe, the workpiece may even come off if the chucking force decreases. Safety speed limits must therefore be observed when designating spindle speeds.
- Even after inch/metric system selection, the units of the programs, tool information, or parameters that have been registered until that time are not converted. Fully check these data units before operating the machine. If the machine is operated without checks being performed, even existing correct programs may cause the machine to operate differently from the way it did before.
- If a program is executed that includes the absolute data commands and relative data commands taken in the reverse of their original meaning, totally unexpected operation of the machine will result. Recheck the command scheme before executing programs.
- If an incorrect plane selection command is issued for a machine action such as arc interpolation or fixed-cycle machining, the tool may collide with the workpiece or part of the machine since the motions of the control axes assumed and those of actual ones will be interchanged. (This precaution applies only to NC units provided with EIA functions.)
- The mirror image, if made valid, changes subsequent machine actions significantly. Use the mirror image function only after thoroughly understanding the above. (This precaution applies only to NC units provided with EIA functions.)
- If machine coordinate system commands or reference position returning commands are issued with a correction function remaining made valid, correction may become invalid temporarily. If this is not thoroughly understood, the machine may appear as if it would operate against the expectations of the operator. Execute the above commands only after making the corresponding correction function invalid. (This precaution applies only to NC units provided with EIA functions.)
- The barrier function performs interference checks based on designated tool data. Enter the tool information that matches the tools to be actually used. Otherwise, the barrier function will not work correctly. (This precaution applies only to the M640MT/MT 5X/T/T NEXUS/TN and M640M Pro/MT Pro.)
- The system of G-code and M-code commands differs between the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060) and the machines equipped with M640MT/MT 5X/T/T NEXUS/TN/MT Pro (such as the INTGEREX non e-Series, the SQT Series, the MPX Series and the QTN Series).
  Issuance of the wrong G-code or M-code command results in totally non-intended machine operation. Thoroughly understand the system of G-code and M-code commands before using this system.

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

- For the machines equipped with M640M Pro (e-Series such as the INTGEREX e-410, e-650 and e-1060), programmed coordinates can be rotated using an index unit of the MAZATROL program and a G68 command (coordinate rotate command) of the EIA program. However, for example, when the B-axis is rotated through 180 degrees around the Y-axis to implement machining with the turning spindle No. 2, the plus side of the X-axis in the programmed coordinate system faces downward and if the program is created ignoring this fact, the resulting movement of the tool to unexpected positions may incite collisions.
  To create the program with the plus side of the X-axis oriented in an upward direction, use the mirror function of the WPC shift unit or the mirror imaging function of G-code command (G50.1, G51.1).
- After modifying the tool data specified in the program, be sure to perform the tool path check function, the solid check function, and other functions, and confirm that the program operates properly. The modification of tool data may cause even a field-proven machining program to change in operational status.
  If the user operates the machine without being aware of any changes in program status, interference with the workpiece could arise from unexpected operation.
  For example, if the cutting edge of the tool during the start of automatic operation is present inside the clearance-including blank (unmachined workpiece) specified in the common unit of the MAZATROL program, care is required since the tool will directly move from that position to the approach point because of no obstructions being judged to be present on this path.
  For this reason, before starting automatic operation, make sure that the cutting edge of the tool during the start of automatic operation is present outside the clearance-including workpiece specified in the common unit of the MAZATROL program.

CAUTION

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

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

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

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

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

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

Operations

WARNING

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

CAUTION

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

OPERATIONAL WARRANTY FOR THE NC UNIT

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

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

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

Operating Environment

1. Ambient temperature

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

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

2. Relative humidity

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

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

- NOTE -

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 WINDOW FUNCTIONS
Chapter 4 PROGRAM CREATION
Chapter 5 PRIORITY FUNCTION FOR THE SAME TOOL
Chapter 6 COORDINATES MEASUREMENT FUNCTION
Chapter 7 TPC DATA CREATION
Chapter 8 BACKGROUND PROGRAMMING
Chapter 9 CASE OF APPEARANCE OF ALARM
Chapter 10 THREE-DIGIT G-FORMAT

Appendix

Appendix 1 LIST OF M CODES
Appendix 2 PROGRAM EXAMPLES
Appendix 3 WHAT TO DO IN SUCH A CASE?

- NOTE -

Introduction

This manual describes only programming based on the MAZATROL language of the MAZATROL FUSION 640M 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 system and use its capabilities to their maximum.

NOTE:

The MAZATROL FUSION 640M controls the machining center 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 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 M-1, M-2, M-32 or MAZATROL FUSION 640M system will be able to understand the functions and usage of the M640M 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 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		An M-code list, 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 bya 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.

H735P10017E

CONTENTS

Page

1 MAZATROL PROGRAM 1-1

2 SYSTEM OF COORDINATES 2-1

2-1 Machine Coordinates System 2-1

2-2 Workpiece Coordinates System 2-2

2-3 Machine Coordinates System and Workpiece Coordinates System 2-3

2-4 Basic Coordinates 2-3

2-5 Auxiliary Coordinates 2-4

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 4-3

4-3 Registration of Tools 4-5

4-3-1 Calling up the TOOL FILE display 4-5

4-3-2 Registration of tools 4-6

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 4-11

4-4-4 Creating a program 4-13

4-4-5 Creation of common unit 4-13

4-4-6 Creation of basic coordinates system unit 4-15

4-4-7 Face machining unit 4-17
4-4-8 Line machining unit 4-27
4-4-9 Point machining unit (1) 4-37
4-4-10 Point machining unit (2) 4-46
4-4-11 End unit 4-53
4-4-12 End of the program 4-54
4-5 Registration of Tools in a Tool Data File 4-56
4-5-1 Pocket numbers and tool numbers 4-56
4-5-2 TOOL DATA display 4-57
4-5-3 TOOL LAYOUT display 4-59
4-5-4 Registration of tools 4-62
4-6 Check of the Machining Path 4-65

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 three types:

Point machining unit

- Drilling
- RGH CBOR machining
- RGH BCB machining
- Reaming
- Tapping
- Boring
  - Boring of through hole
  - Boring of stepped through hole
  - Boring of non-through hole
  - Boring of stepped non-through hole
- Back boring
- Circular milling
- Counterbore-tapping

Line machining unit

- Central linear machining
- Right-hand linear machining
- Left-hand linear machining
- Outside linear machining
- Inside linear machining
- Right-hand chamfering
- Left-hand chamfering
- Outside chamfering
- Inside chamfering

Face machining unit

- Face milling
- End milling-top
- End milling-step
- Pocket milling
- Pocket milling-mountain
- Pocket milling-valley
- End milling-slot
- 3-D (option)

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. Auxiliary coordinates system unit

Use to specify the auxiliary coordinates system (OFFSET).

6. Special mode unit

There are the following special mode units. It is possible that these units marked with an asterisk (*) can not be used or executed in certain machine models.

M code ..... Output of M code

Sub-program..... Calling up a sub-program

Pallet changing*..... Changing of pallet

Indexing*..... Use to specify the angle of the indexing table.

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

7. 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.

8. MMS unit

Automatic measurement of a basic coordinates system (WPC). MMS unit cannot be used for certain machines or will not be performed even if programmed.

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 of the machining center 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

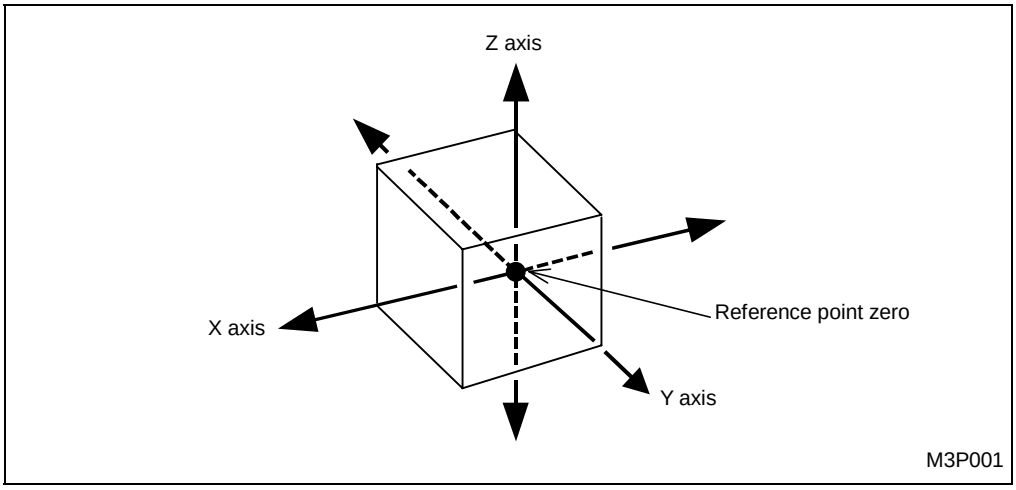


Fig. 2-1 Coordinates system

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.

Note: The following figure represents the case of vertical machining center.

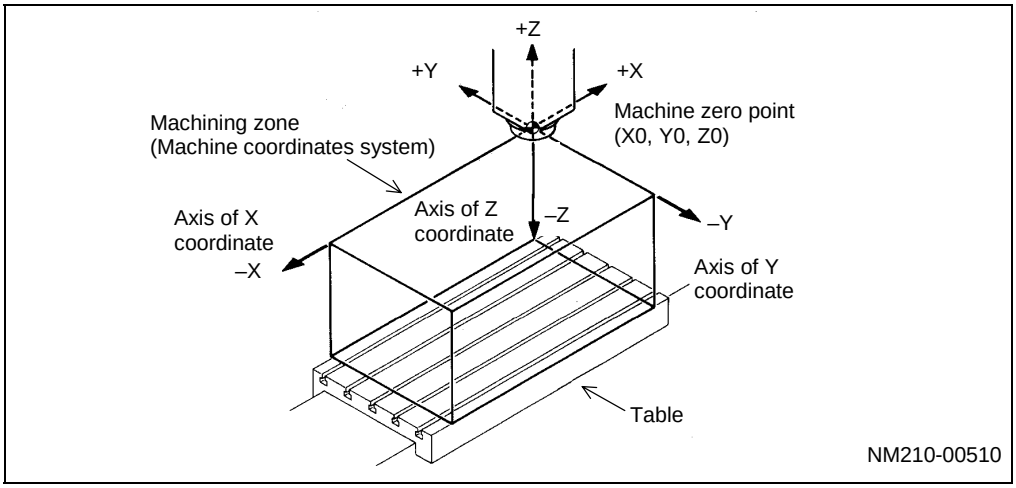


Fig. 2-2 Machine coordinates system

2-2 Workpiece Coordinates System

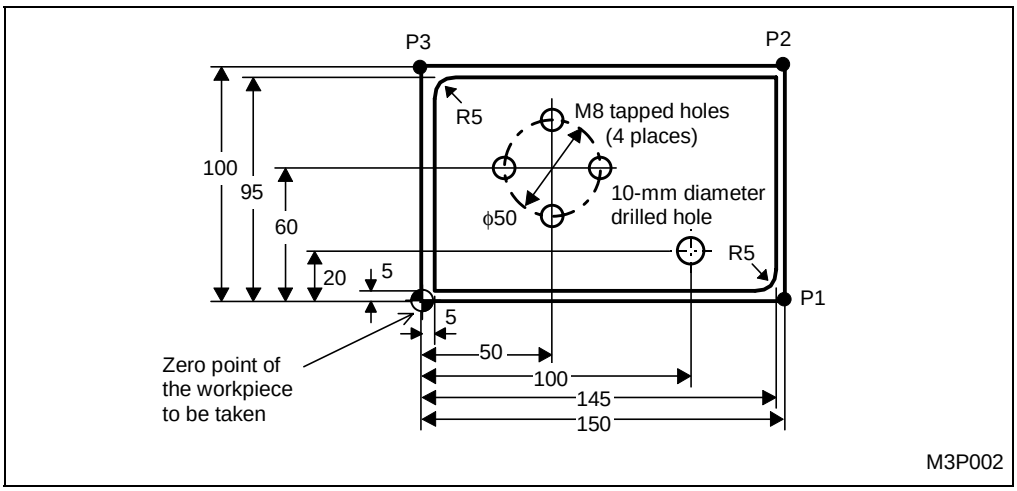
If the program is prepared on the basis of the machine coordinates system, the entering of the machining position and of the form of machining is very complex, tedious and inflexible.

Consequently, a temporary reference point is taken in the machine coodinates 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



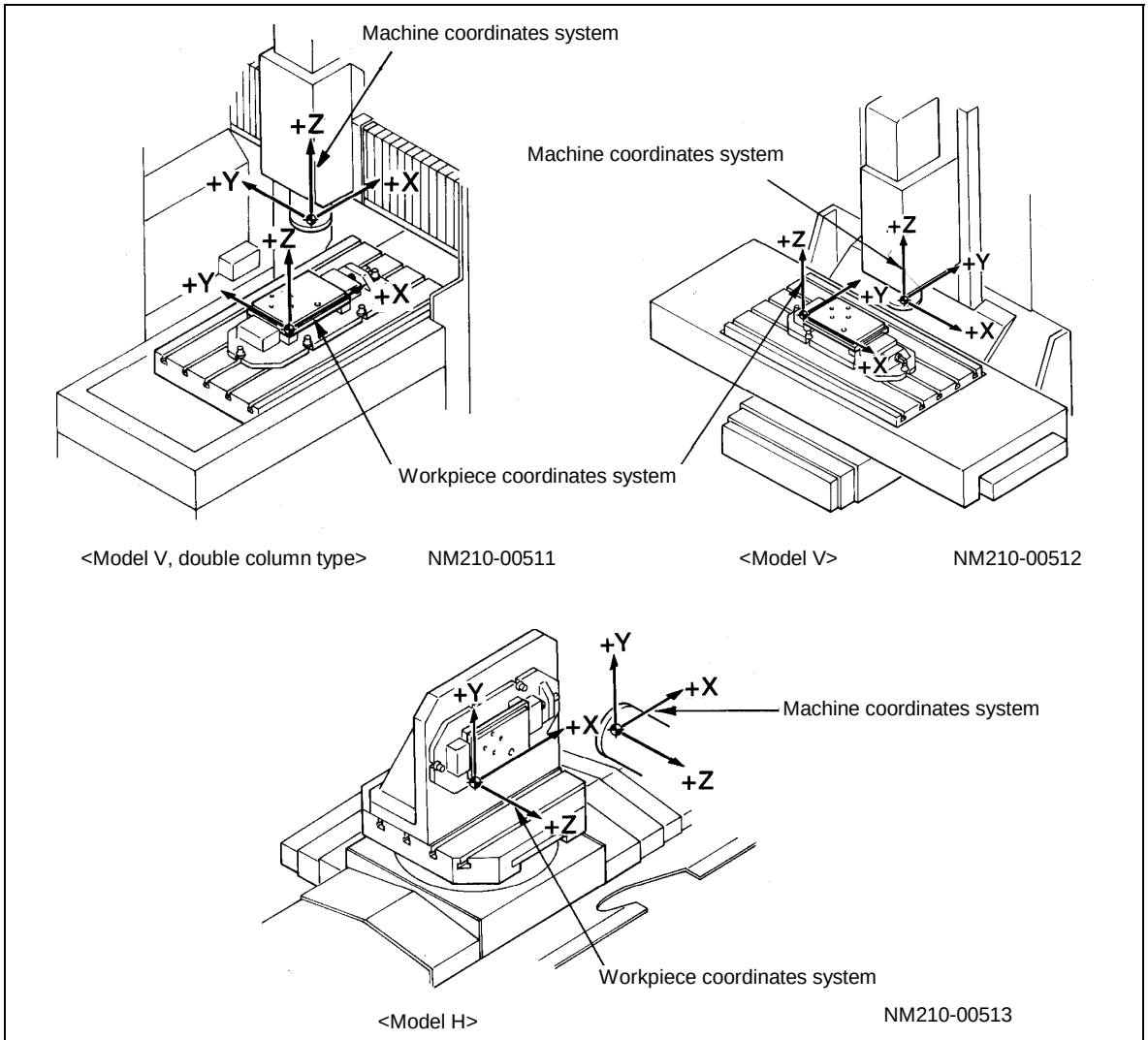
- When the dimensions are entered of the configuration on the basis of the above plan of the workpiece, the bottom left hand corner is taken as the workpiece zero point.
- In this case, the value of the coordinates of the configuration is the following:

Workpiece zero point = (0, 0, 0)
P1 = (150, 0, 0)
P2 = (150, 100, 0)
P3 = (0, 100, 0)

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.



Note: The above relationship may slightly differ according to the type of machine being used.

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

2-4 Basic 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 basic coordinates unit.

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

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

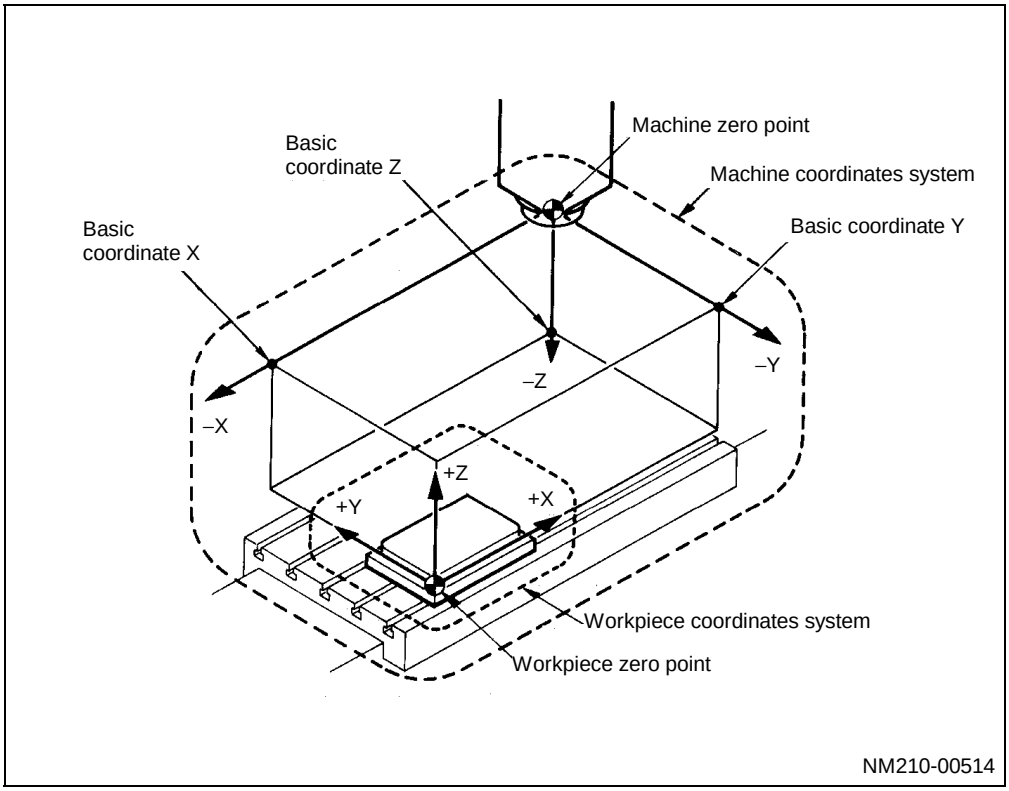
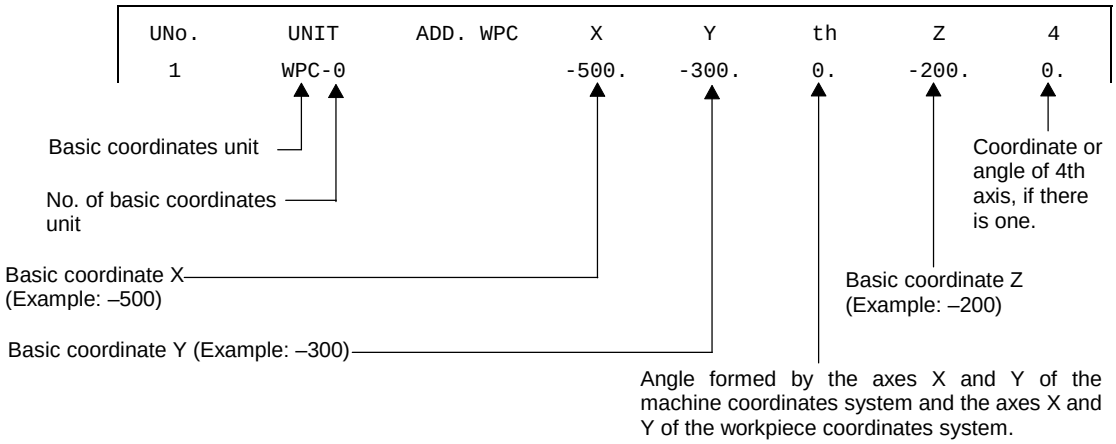


Fig. 2-4 Basic coordinates

Example of entering of the basic coordinates unit:

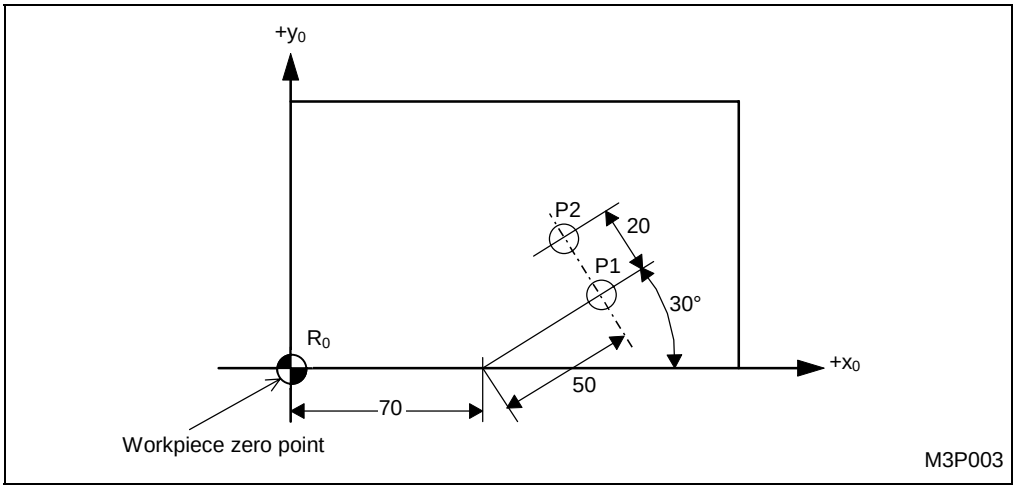


2-5 Auxiliary Coordinates

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.

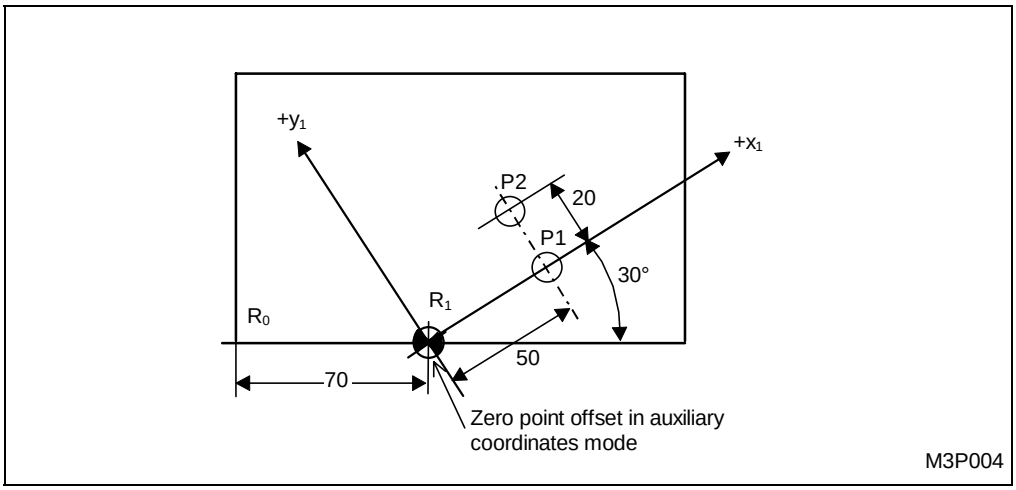
1. Example of entering of auxiliary coordinates



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

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

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:

UNo.	UNIT	U (X)	V (Y)	D (th)	W (Z)
2	OFFSET	70.	0.	30.	0.

Auxiliary coordinates unit

Offsetting on X axis

Offsetting on Y axis

Angle with relation to workpiece coordinates system

Offsetting on Z axis

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 basic coordinates is introduced. In this case, the state without a system of auxiliary coordinates is assumed. (When the system of basic 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.)

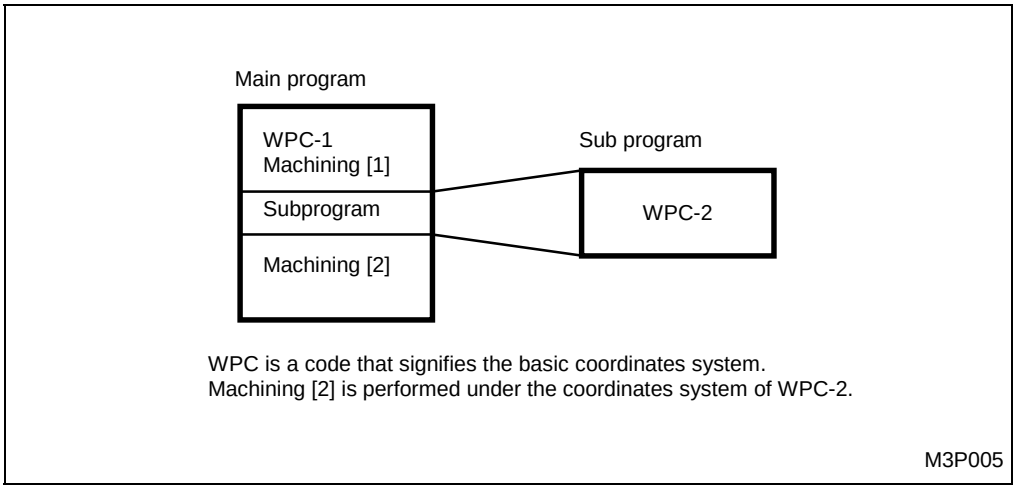


Fig. 2-5 Basic coordinates system after execution of subprogram

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 in Fig. 3-1 and Fig. 3-2.

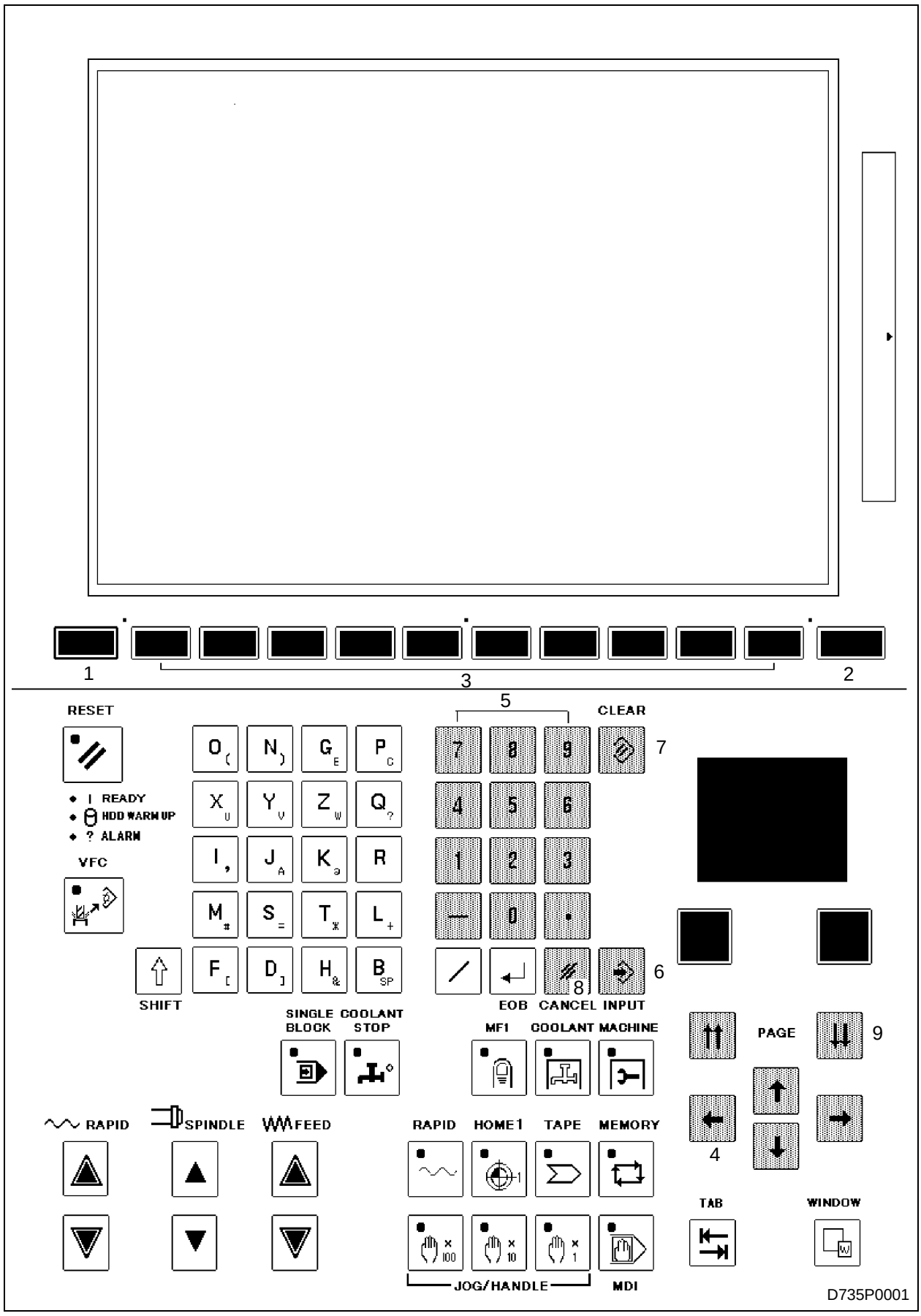


Fig. 3-1 Keys and switches used for creating a MAZATROL program <M640M, M640M-5X> (portion)

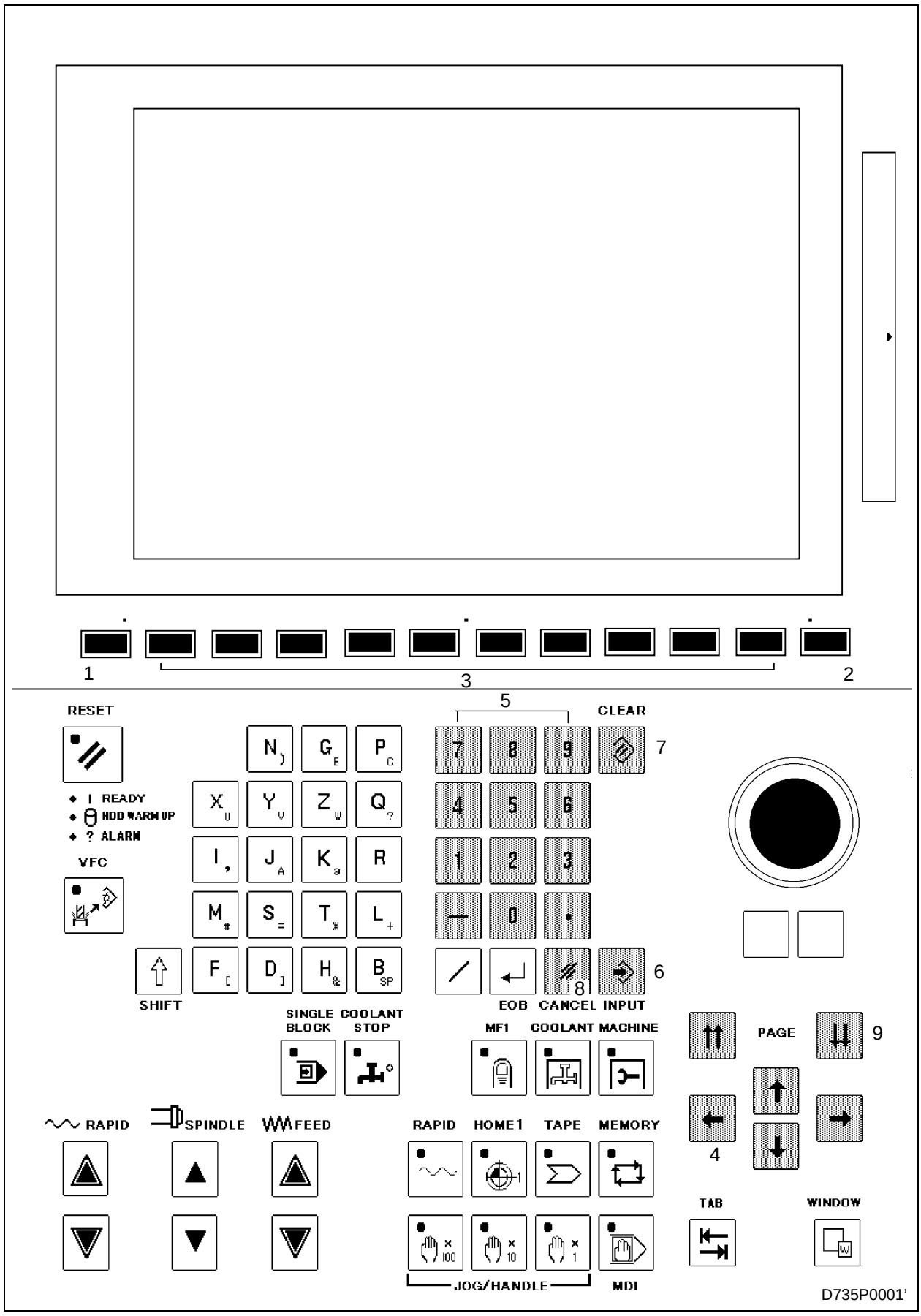


Fig. 3-2 Keys and switches used for creating a MAZATROL program <M640M NEXUS> (portion)

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 apropriate 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.

Note: The numbers in the No. column corresponds to those in Fig. 3-1 and 3-2.

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.

Note 1: It is possible to monitor the path of the tool even if the tool is not recorded in the tool data. (The path of the tool is laid out by taking the diameter of the tool = the nominal diameter and the length of the tool = 0).

Note 2: Automatic operation can not be performed if the tool used is not recorded in the tool data.

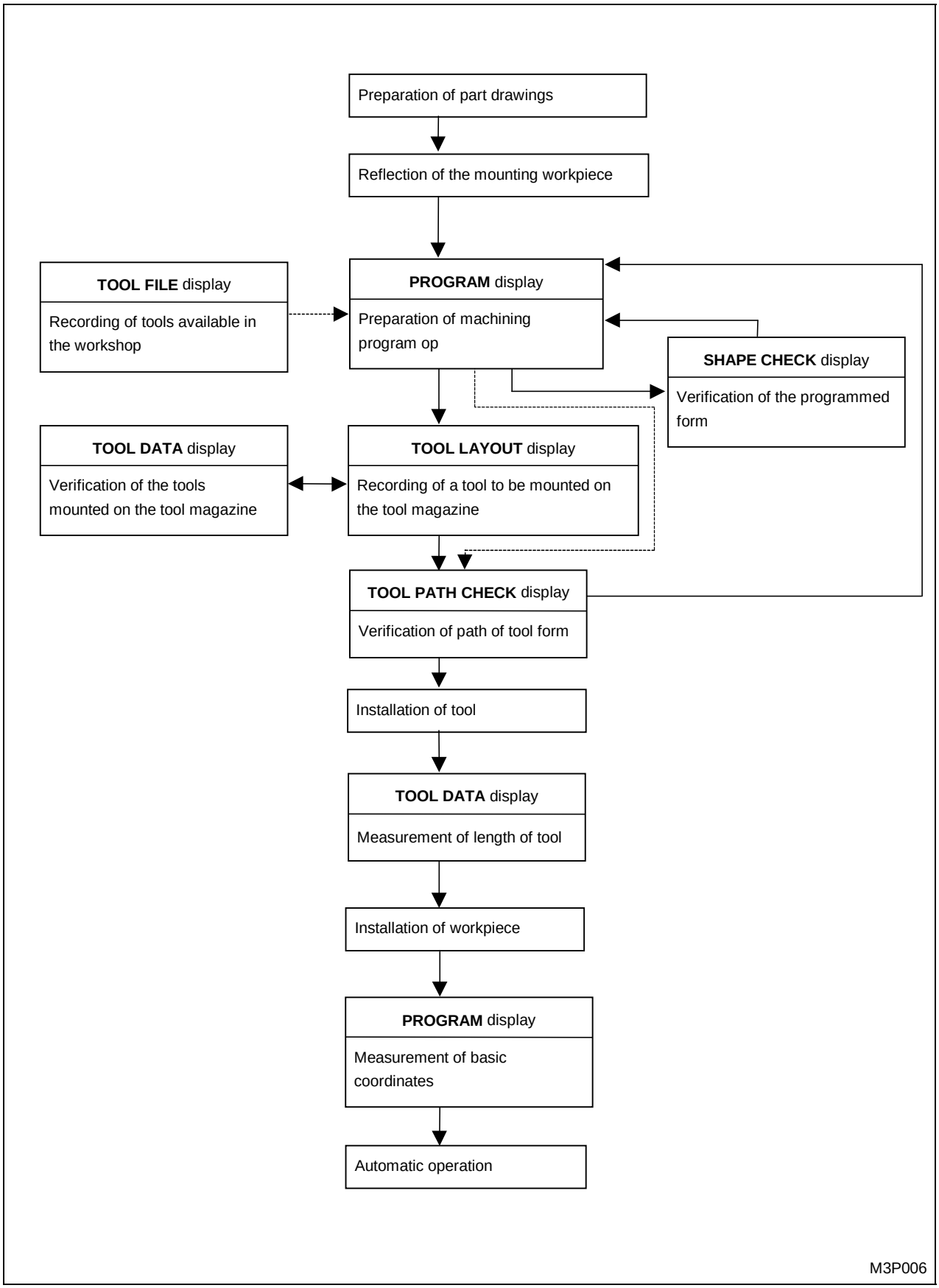


Fig. 4-1 Procedure before performing the machining

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.

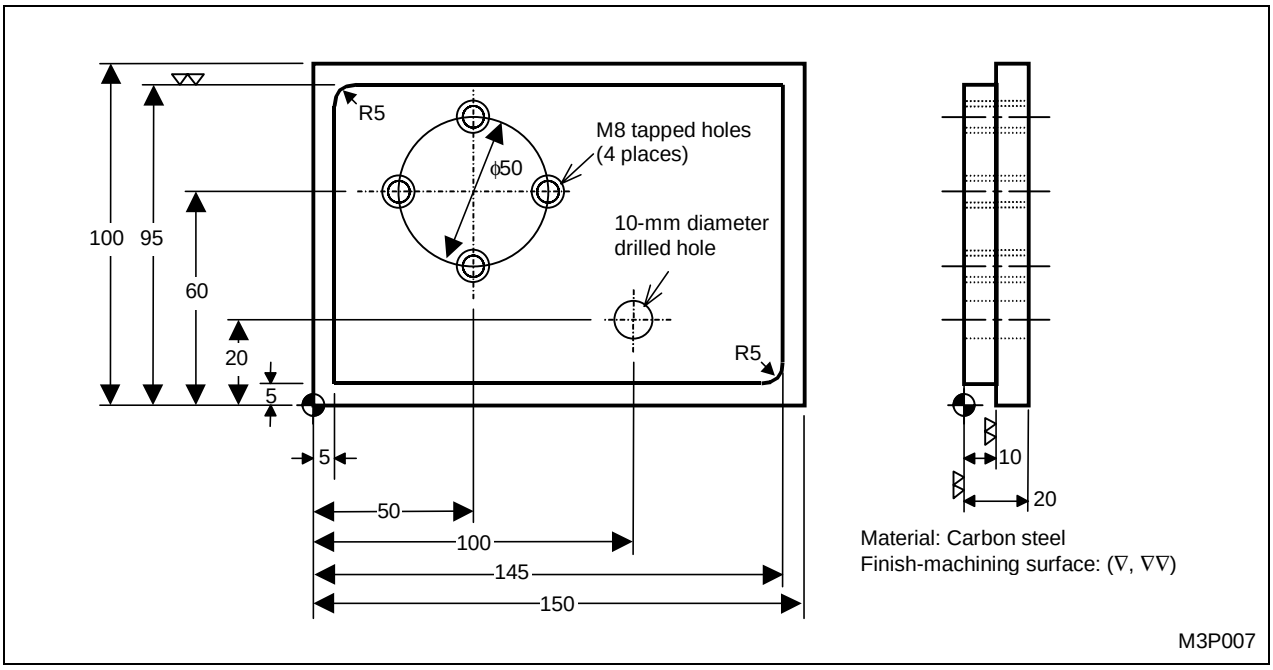
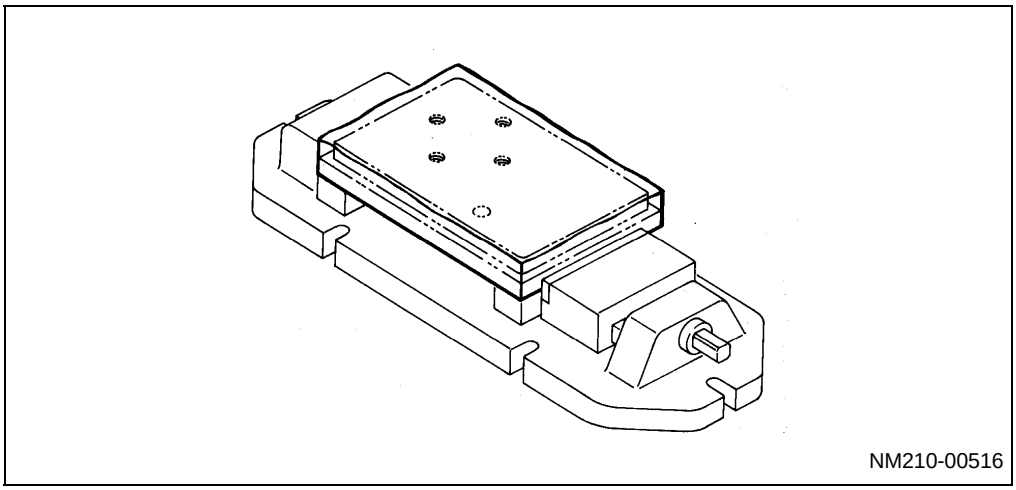
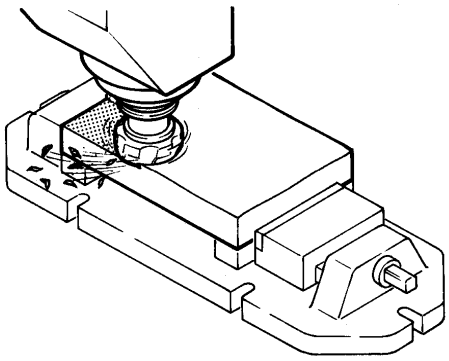
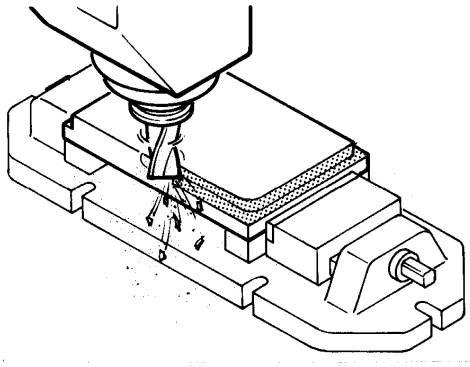
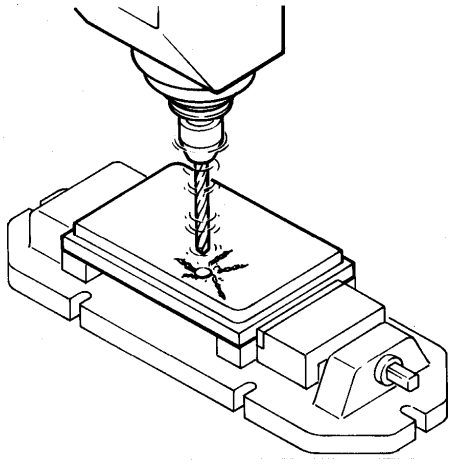
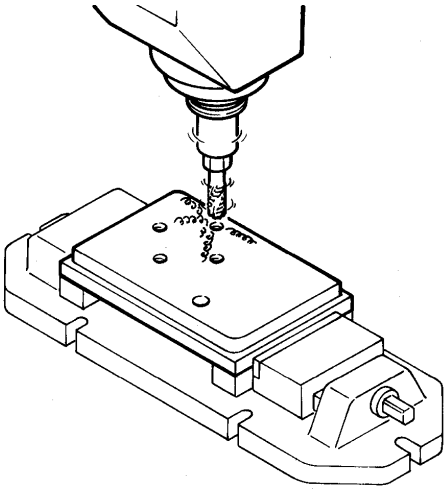


Fig. 4-2 Part drawing example

Held by a vise, the part to be machined is shown below.



Machining of this part consists of the following four processes:

[1] Milling the top face of the workpiece.	[2] End-milling the edges of the workpiece.
 NM210-00517	 NM210-00518
[3] Drilling a 10-mm diameter through-hole.	[4] Drilling four M8 tapped holes.
 NM210-00519	 NM210-00520

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.

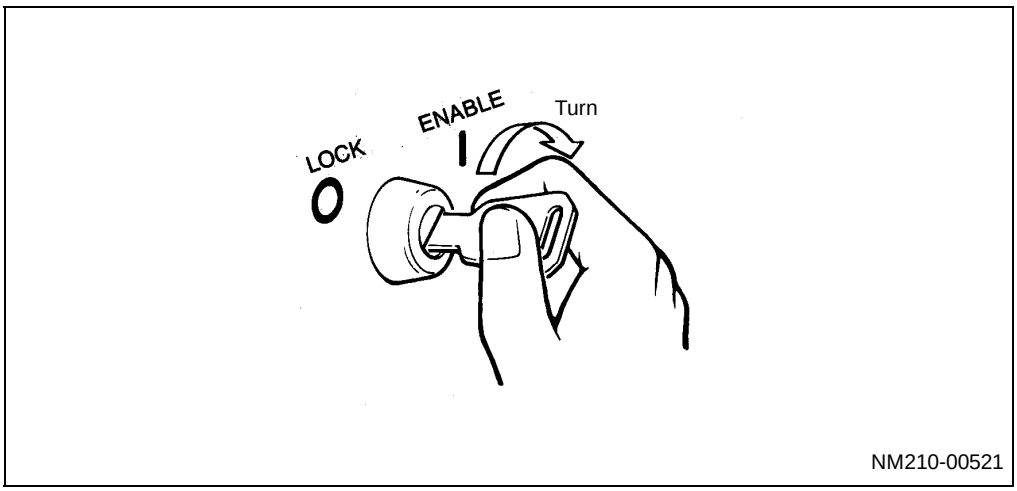
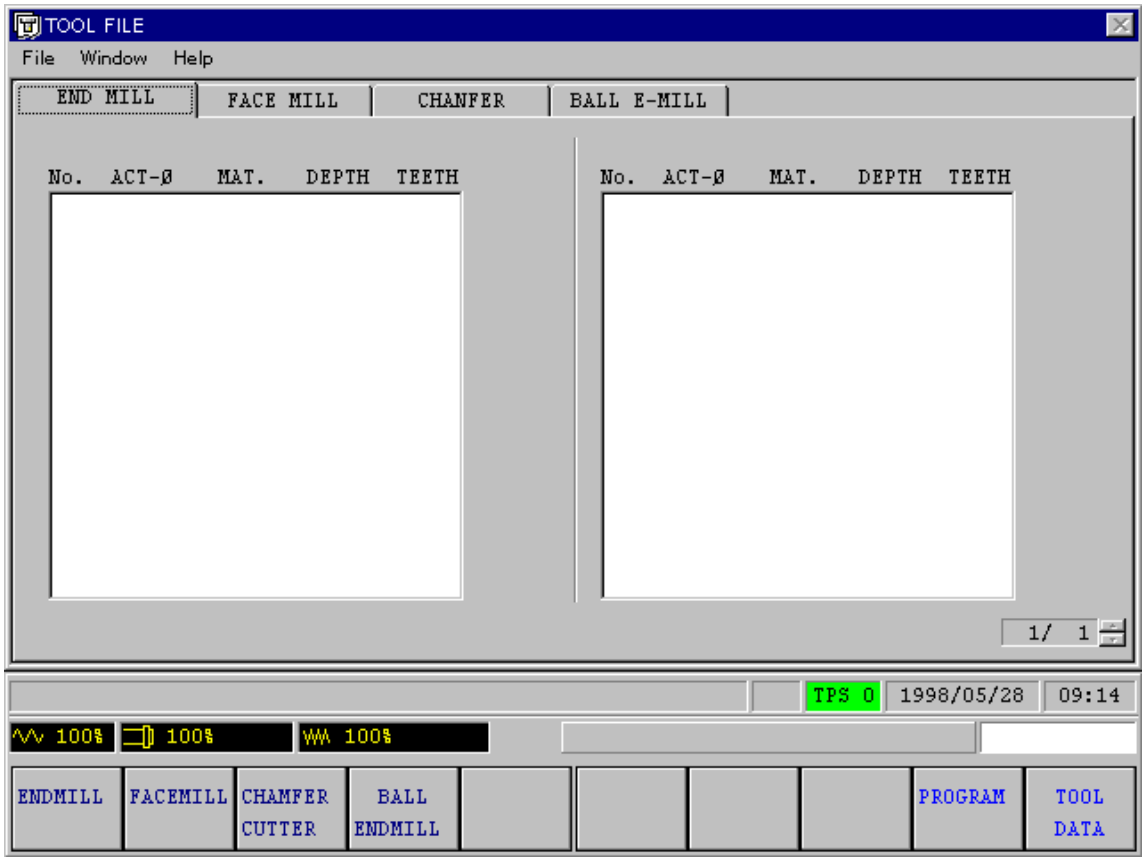


Fig. 4-3 Reprogramming switch

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.

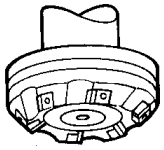
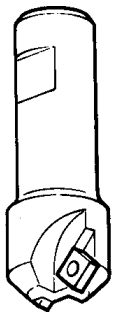
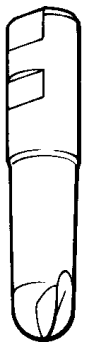
4-3-1 Calling up the TOOL FILE display

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.



D735P0002E

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.

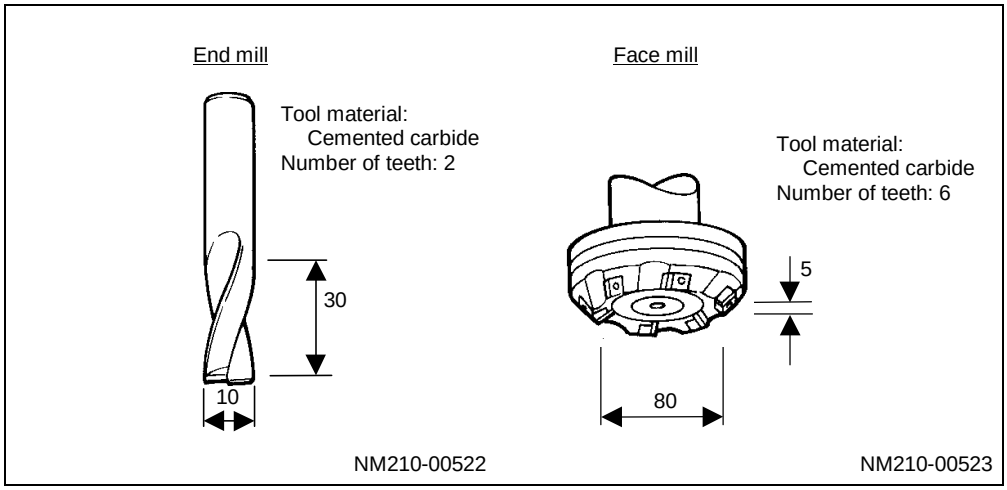


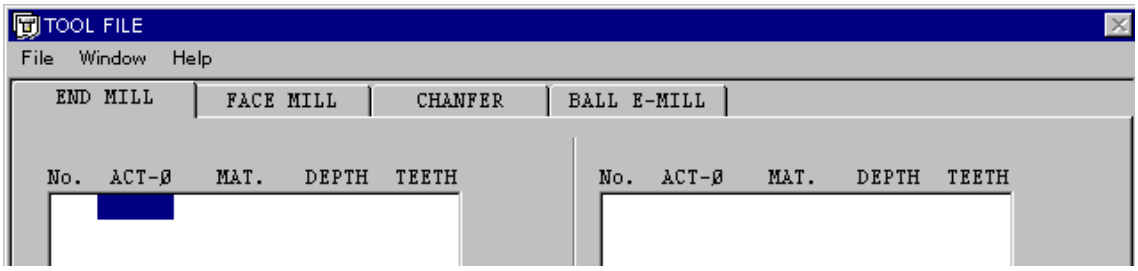
Fig. 4-4 Registered tools example

1. Registering the end mill

Using the page keys, search the display for the tool indicated as E-MILL 10. 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.



D735P0003E

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

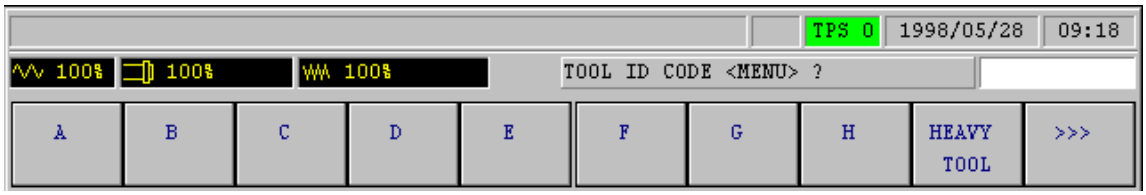
(2) Press the input key [⇨] to register the tool.

➔ The message **NOMINAL DIAMETER?** will be displayed.

(3) Input an approximate tool-diameter value (in this example, input 10 by pressing numeric keys [1] and [0] 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).



D735P0004E

➔ 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**).

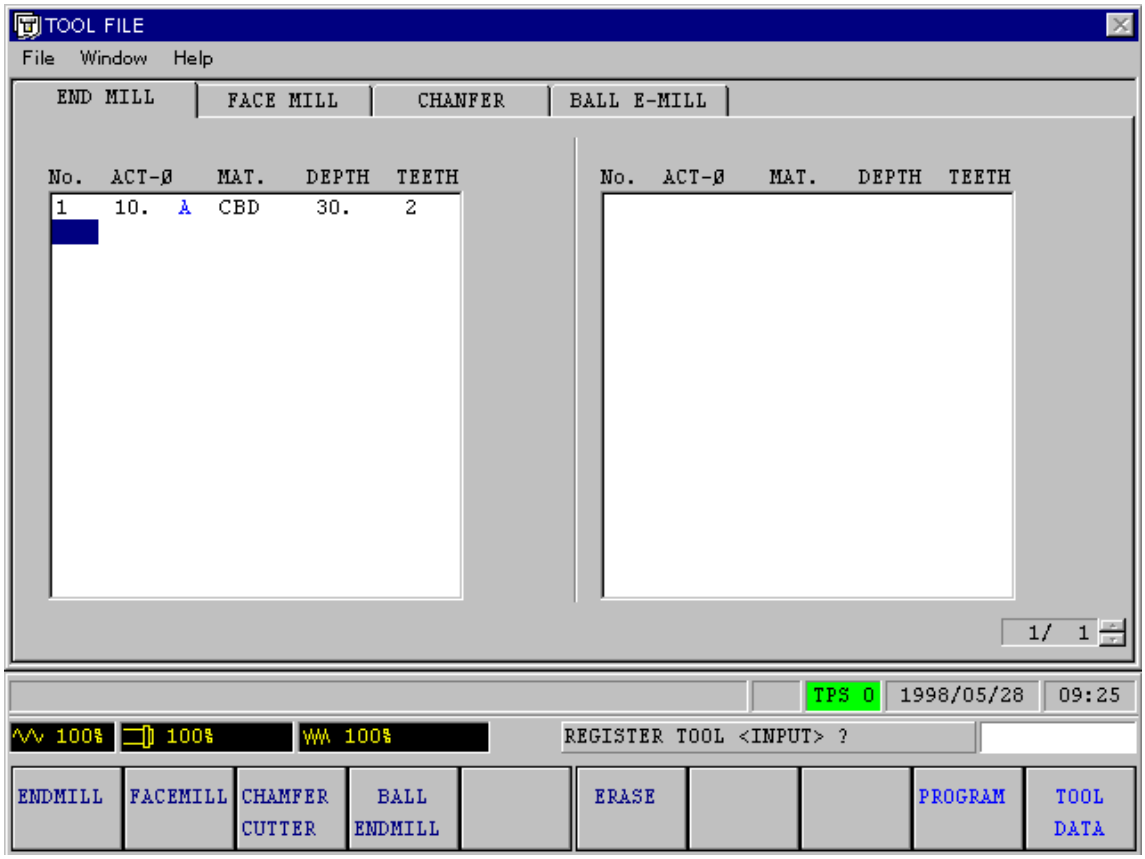
➔ The message **MAX DEPTH OF CUT?** will be displayed.

(6) Input the maximum depth to which the workpiece can be cut in the axial direction by one cutting operation (in this example, input 30 by pressing numeric keys [3] and [0] 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 10-mm diameter end mill will be completed as shown below.



D735P0005E

2. Registering the face mill

The operations to be carried out are similar to those required for registering the 10-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 80. 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?** ..... **8** **0** ⇨

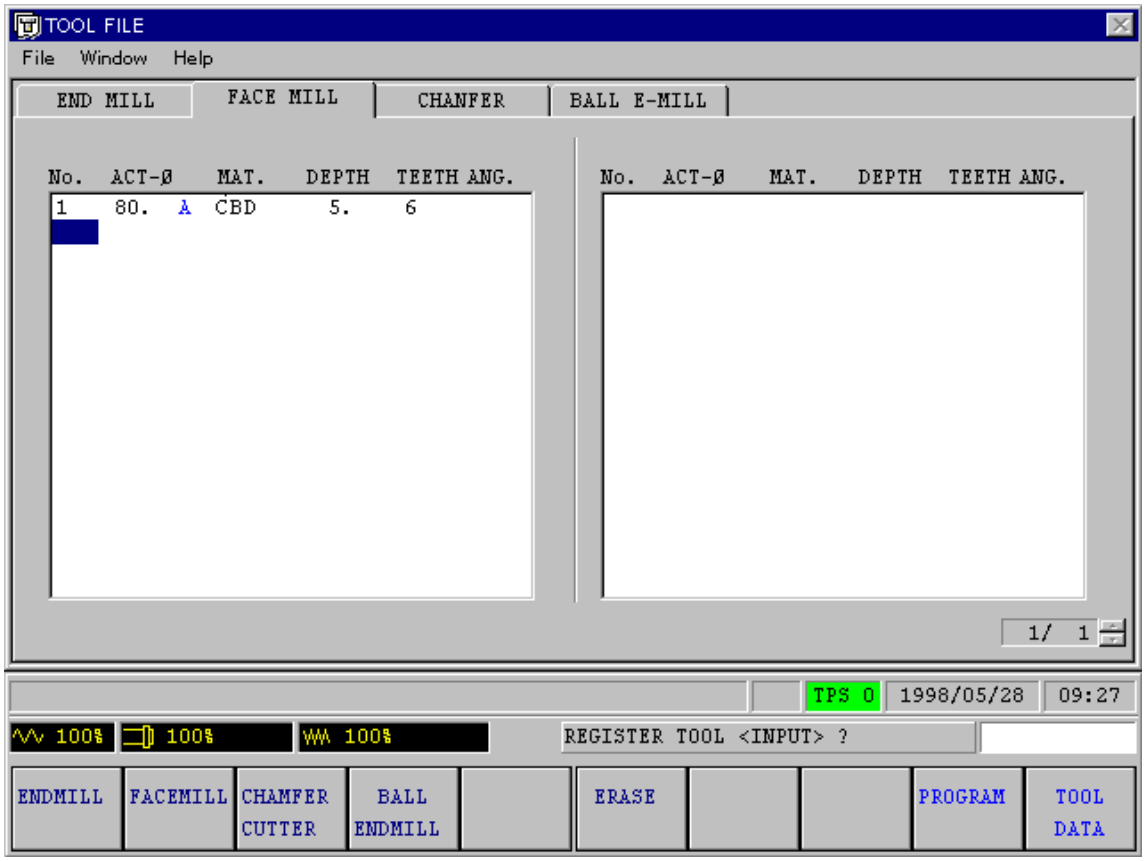
(5) **TOOL ID CODE <MENU>**? ..... **A**

(6) **TOOL MATERIAL <MENU>**? ..... **CARBIDE**

(7) **MAX DEPTH OF CUT?** ..... **5** ⇨

(8) **NUMBER OF TEETH?** 6

➔ This completes registration of the 80-mm diameter face mill as shown below.

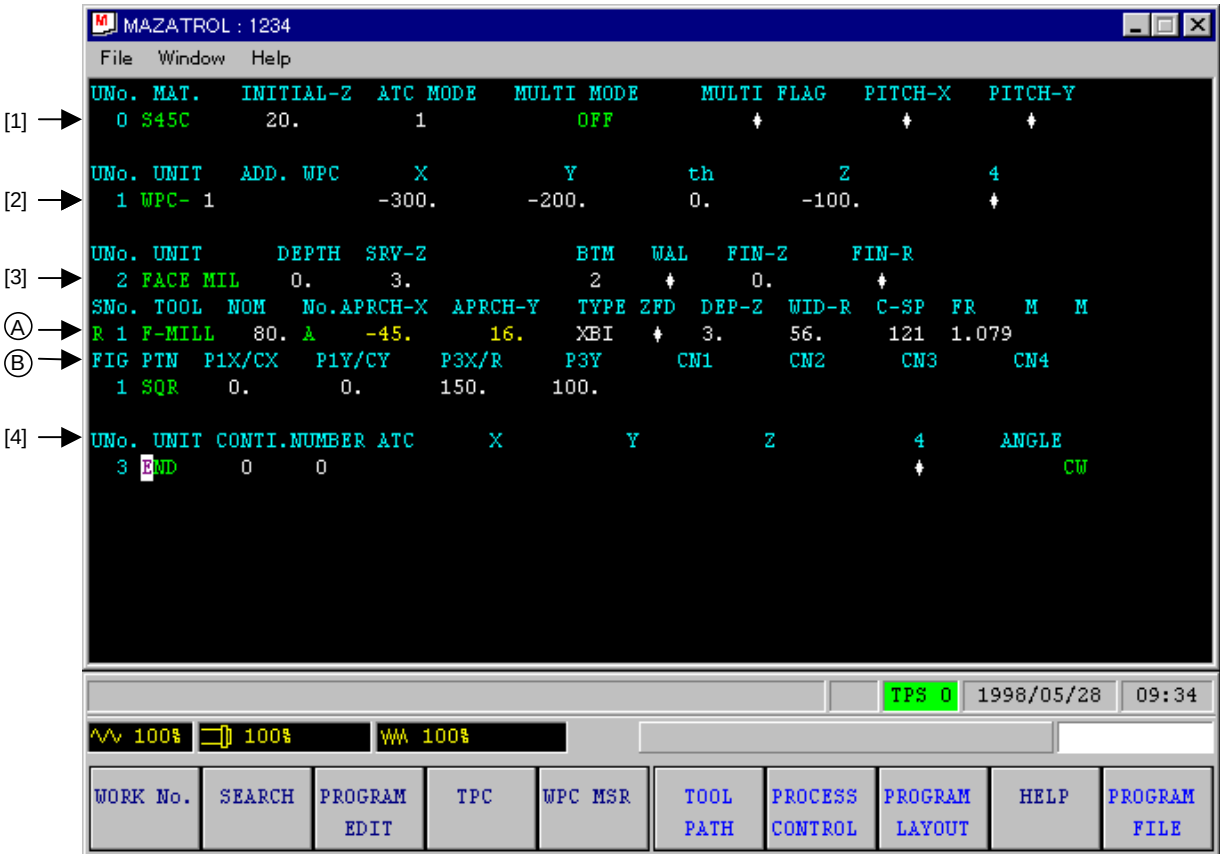


D735P0006E

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.



D735P0007E

[1] Common unit
Basic data related to the entire program, such as the material of the workpiece, is to be input to this unit.

[2] Basic coordinates system unit
The coordinate values of the workpiece zero point in the machine coordinates system are to be input to this unit.

[3] 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.

[4] End unit
Denotes the end of a program (i.e., the completion of machining).

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 presented then as shown below.

D735P0008E

Note: 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

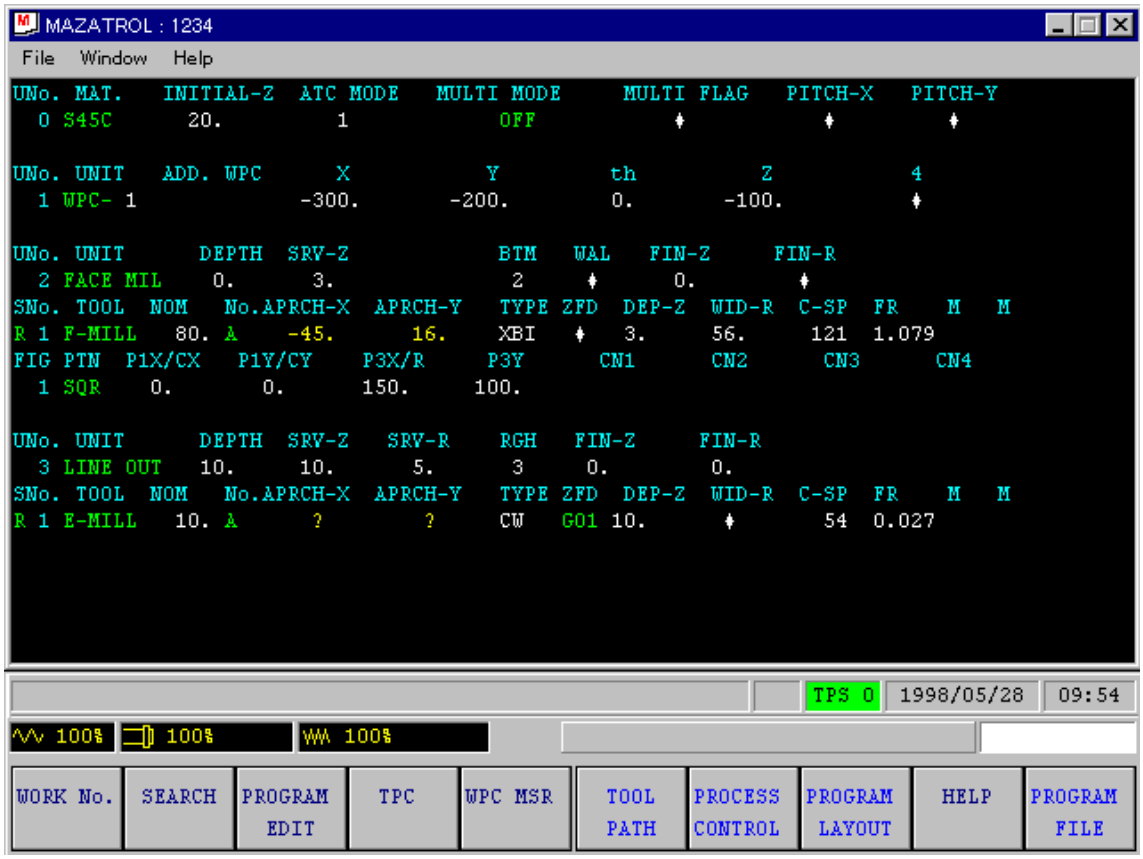
Number the individual programs to be created. These numbers identify the individual programs just as part number in a part drawing identifies the individual parts. These numbers are referred to as workpiece numbers, and the desired number from 1 to 99999999 can be selected for each program. Here, workpiece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

(1) Press the menu key **WORK No.**

(2) **WORK No.**..... 1 2 3 4 →

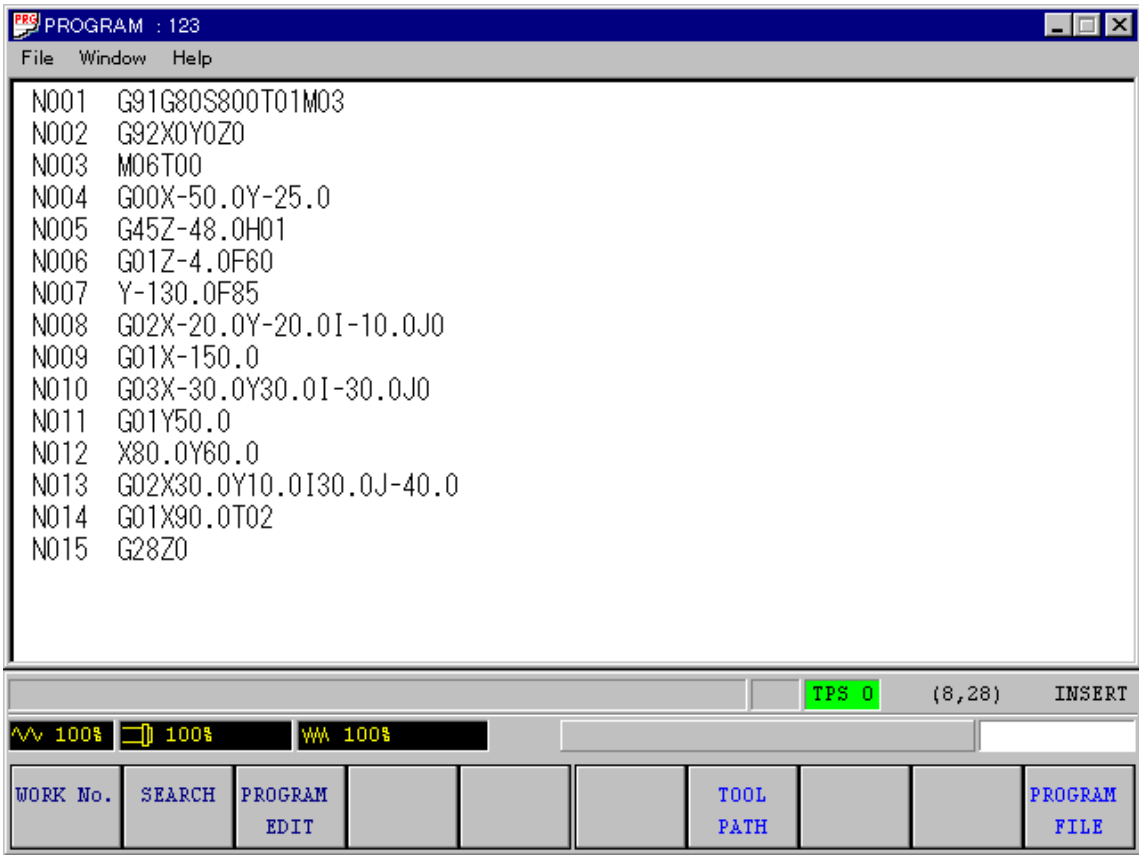
➔ The message **NEW PROGRAM <MENU>?** will be displayed.
If either one of the two displays [1] and [2] shown below is displayed instead of this message, then this indicates that workpiece No. 1234 has already been used. In that case, input a different workpiece number.

[1] MAZATROL program



D735P0009E

[2] EIA/ISO program



D735P0010E

Note: In M640M, 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

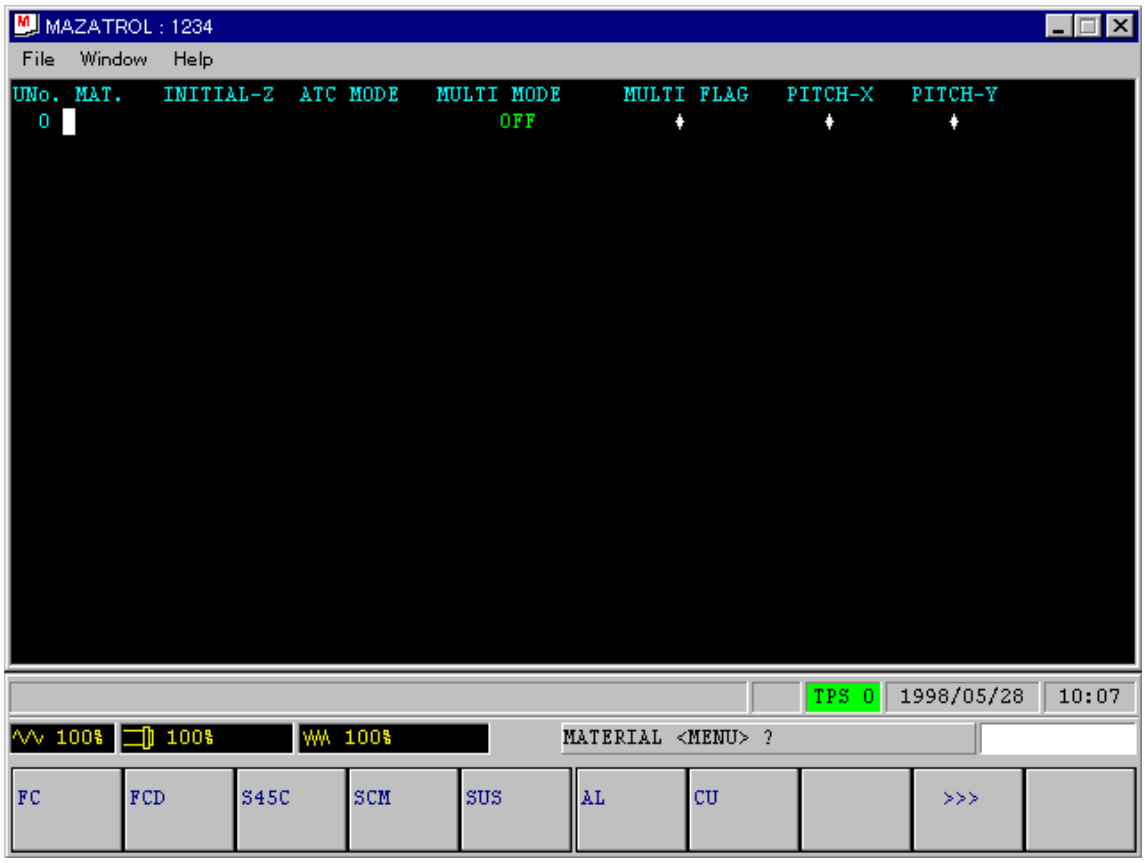
Let us create a program.

(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.



D735P0011E

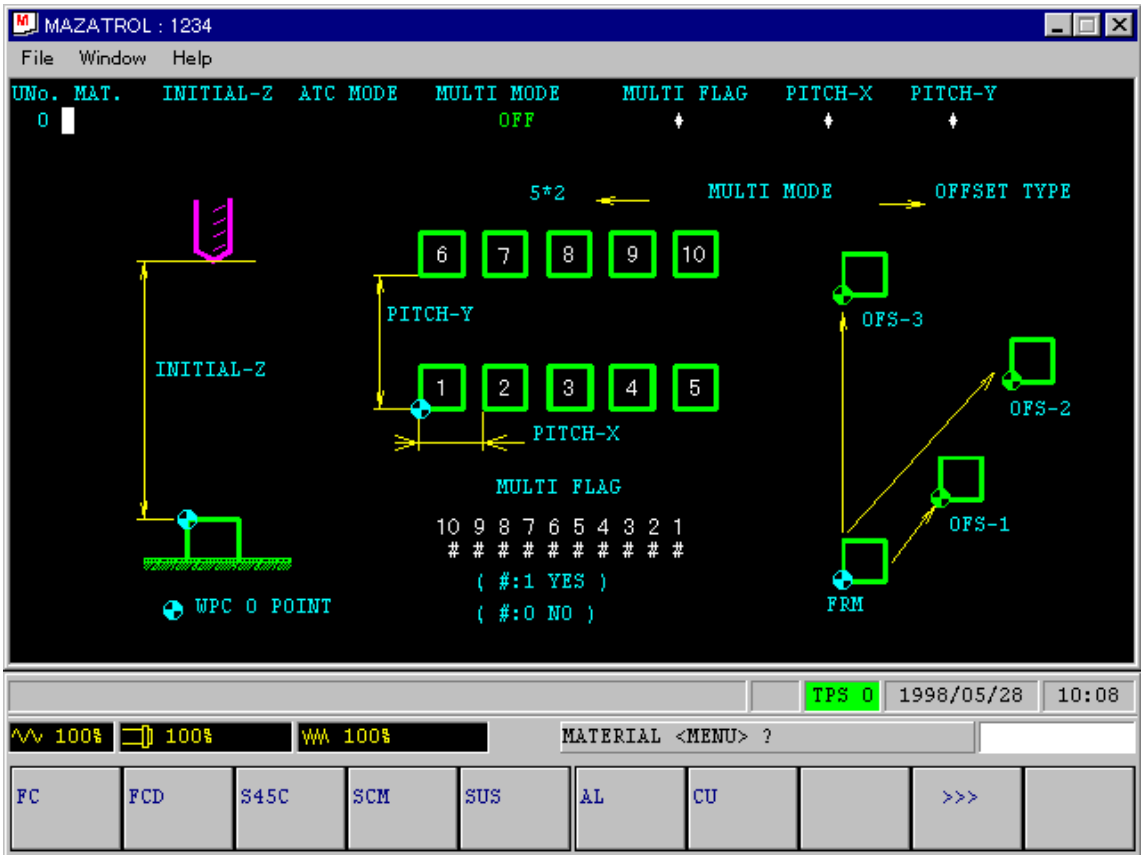
Fig. 4-6 PROGRAM display (edit mode)

4-4-5 Creation of common unit

The common unit is the program unit that must be created at the head position of a program. Basic data on the entire program is to be input to this unit.

The help window is described here before the creation of common unit. Prior to input of data to the unit, full details of the data to input can be displayed on the screen for ease of programming. This display is referred to as the help window display. Carry out the following operation to display the help window display:

(1) Press the menu key **HELP**.
The following help window display will be presented.



D735P0012E

Now, start creating the common unit. The data for the article MAT, INITIAL-Z and ATC MODE will be specified here.

MAT — Specify the material of the workpiece.

INITIAL-Z — Specify the initial height of the tool where it comes close to the workpiece for machining purposes (see figure below).

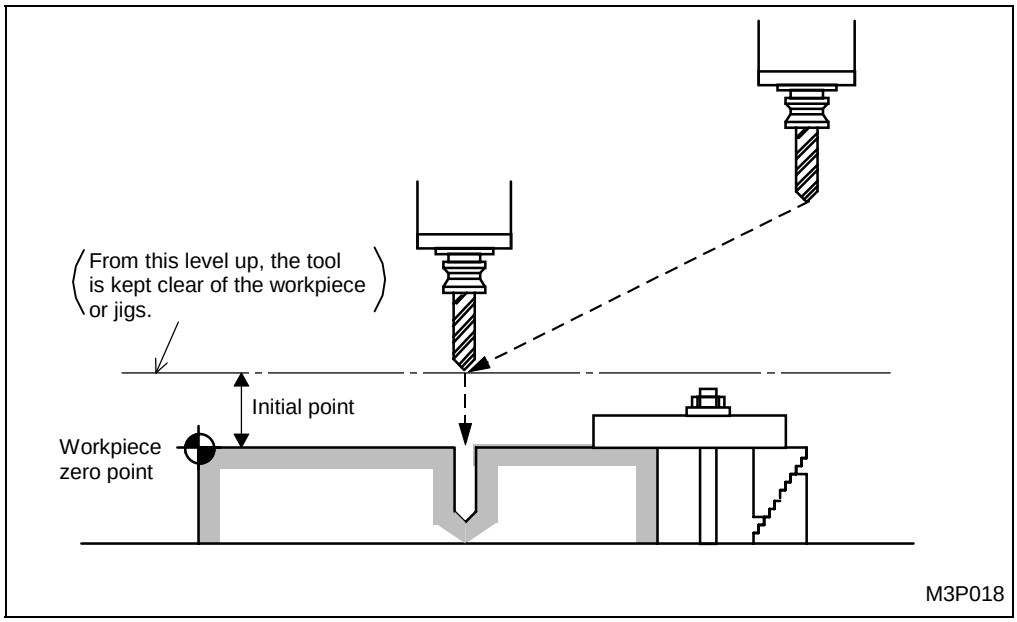


Fig. 4-7 Tool path

ATC MODE Specify the manner in which the tool is to be returned to the ATC position for tool change.

MULTI MODE Use this function when more than one workpiece of the same type are to be arranged on the table for successive machining. Do not use this function here.

(2) **MATEIRAL <MENU>?** **S45C**

(3) **INITIAL POINT Z (CLEARANCE)?** ..... 2 0 ⏎

(4) **ZERO RETURN <Z. X+Y: 0, X+Y+Z: 1>?** ..... 1 ⏎

(5) **MULTI MODE <MENU>?** ..... **MULTI OFF**

The common unit will be created as shown below.

When the unit is created, the help window will be closed and the program will await data input to the next unit.

D735P0013E

4-4-6 Creation of basic coordinates system unit

Let us set the basic coordinates (the coordinate values of the workpiece zero point in the machine coordinate system). This data is to be input using the coordinates measuring funciton (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.

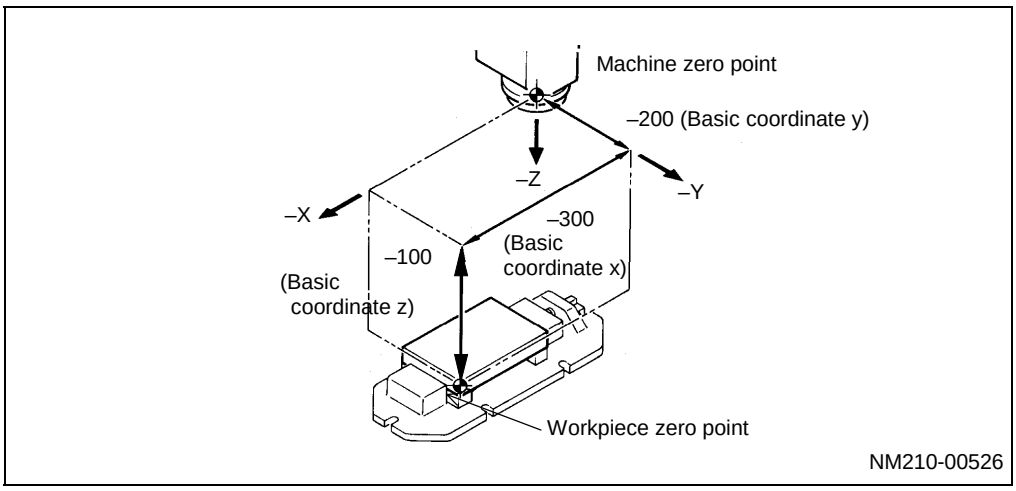
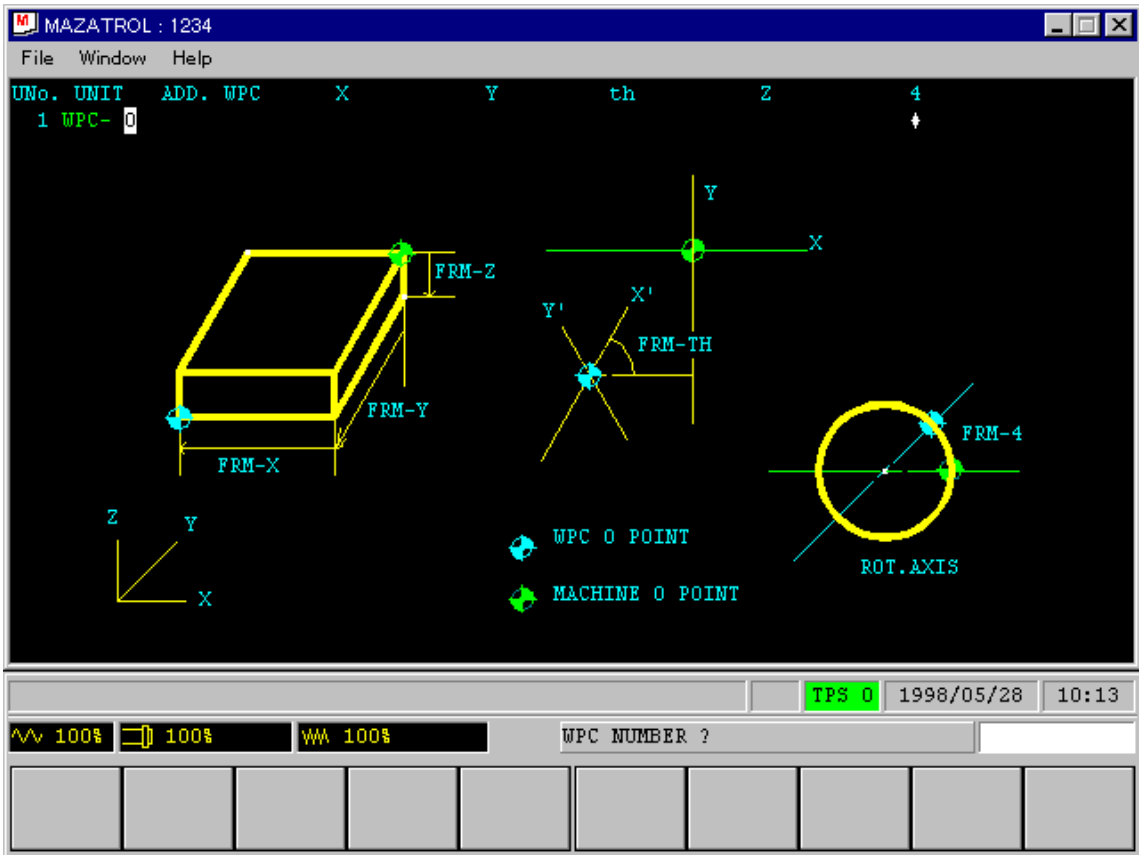


Fig. 4-8 Workpiece mounting position

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

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

➔ The following help window will be presented. “WPC” is the abbreviation of the workpiece coordinates.



D735P0014E

(3) **WPC NUMBER?**..... 1 ⏎

(4) **WORKPIECE COORDINATE, WPC-X?** ..... − 3 0 0 ⏎

(5) **WORKPIECE COORDINATE, WPC-Y?** ..... − 2 0 0 ⏎

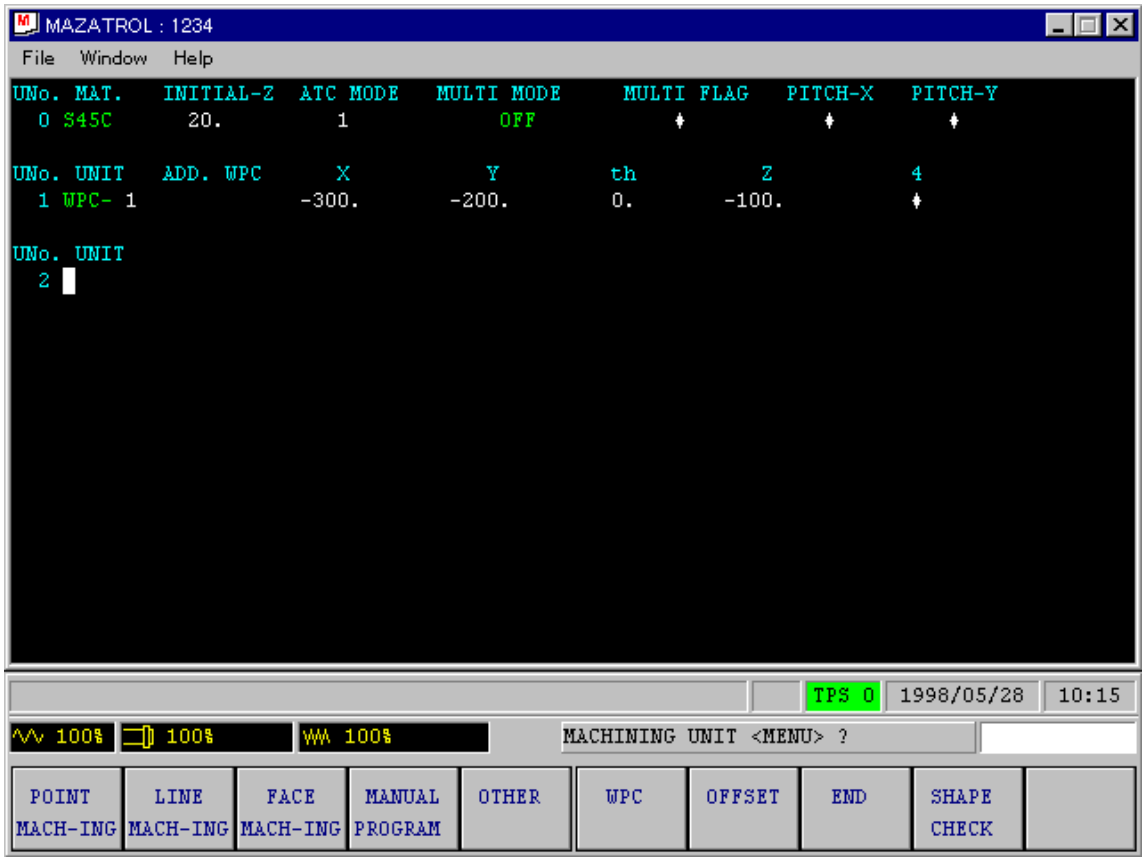
(6) **WORKPIECE COORDINATE, WPC-th?** ..... 0 ⏎

(7) **WORKPIECE COORDINATE, WPC-Z?** ..... [−] [1] [0] [0] [→]

(8) **WORKPIECE COORDINATE, WPC-4?**..... [0] [→]

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

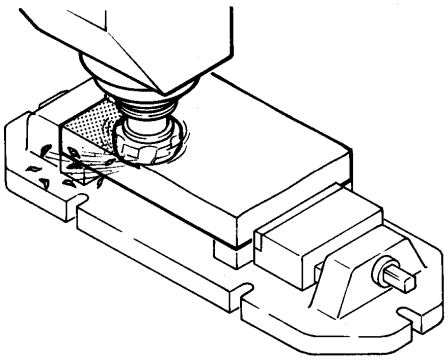
When the unit is created, the help window will be closed and the program will await data input to the next unit.



D735P0015E

4-4-7 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.



NM210-00517

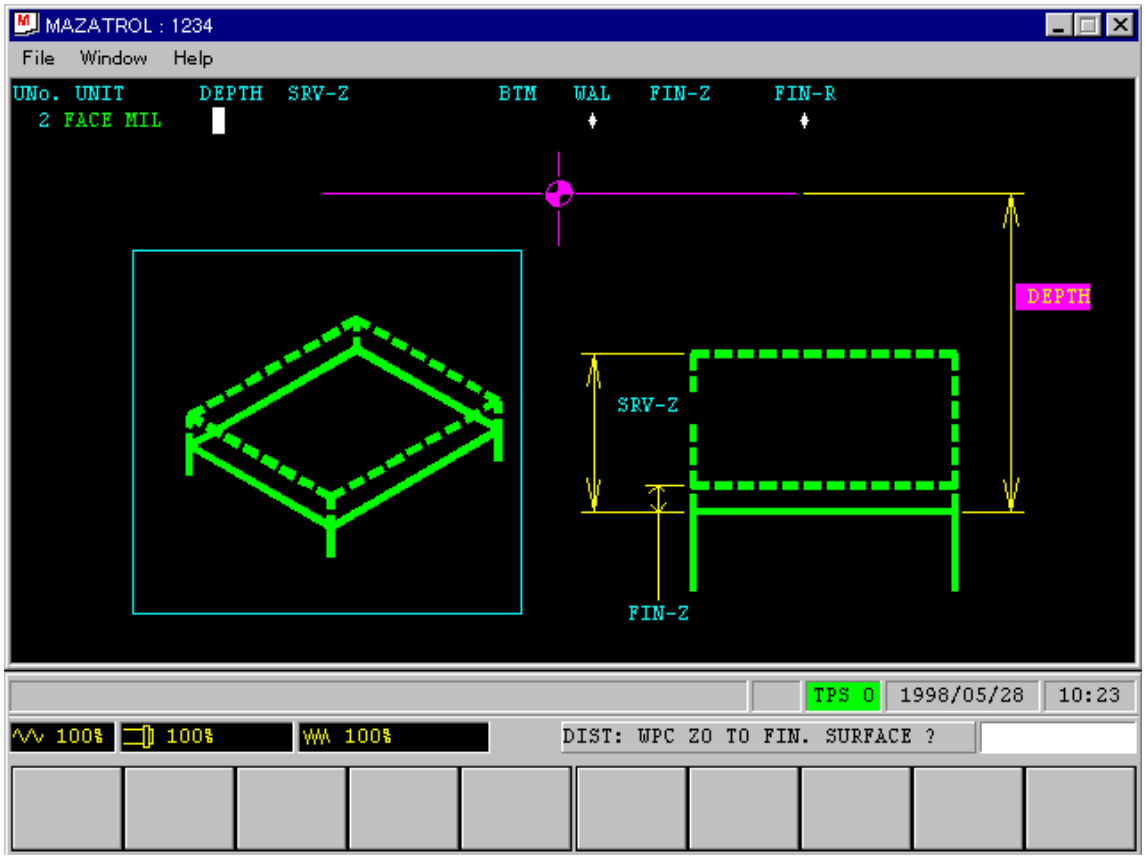
1. Creation of face machining unit

(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.



D735P0016E

Now, start creating the face machining unit. The data for the article DEPTH, SRV-Z, BTM, and FIN-Z will be specified here.

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.

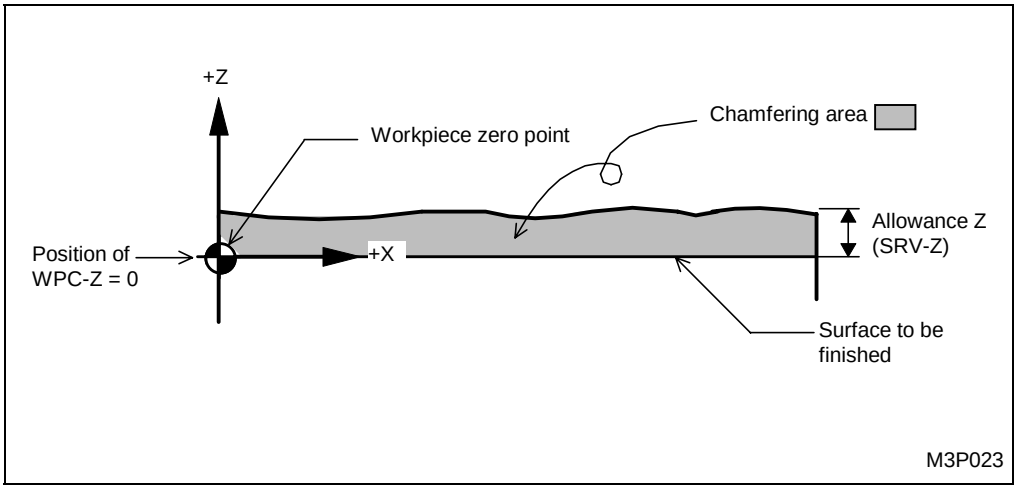


Fig. 4-9 DEPTH and SRV-Z

SRV-Z .....Input the chamfering thickness of the workpiece. An approximate value may be input because the workpiece surface is not plane.

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

FIN-Z.....Input the finish milling thickness. 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 →

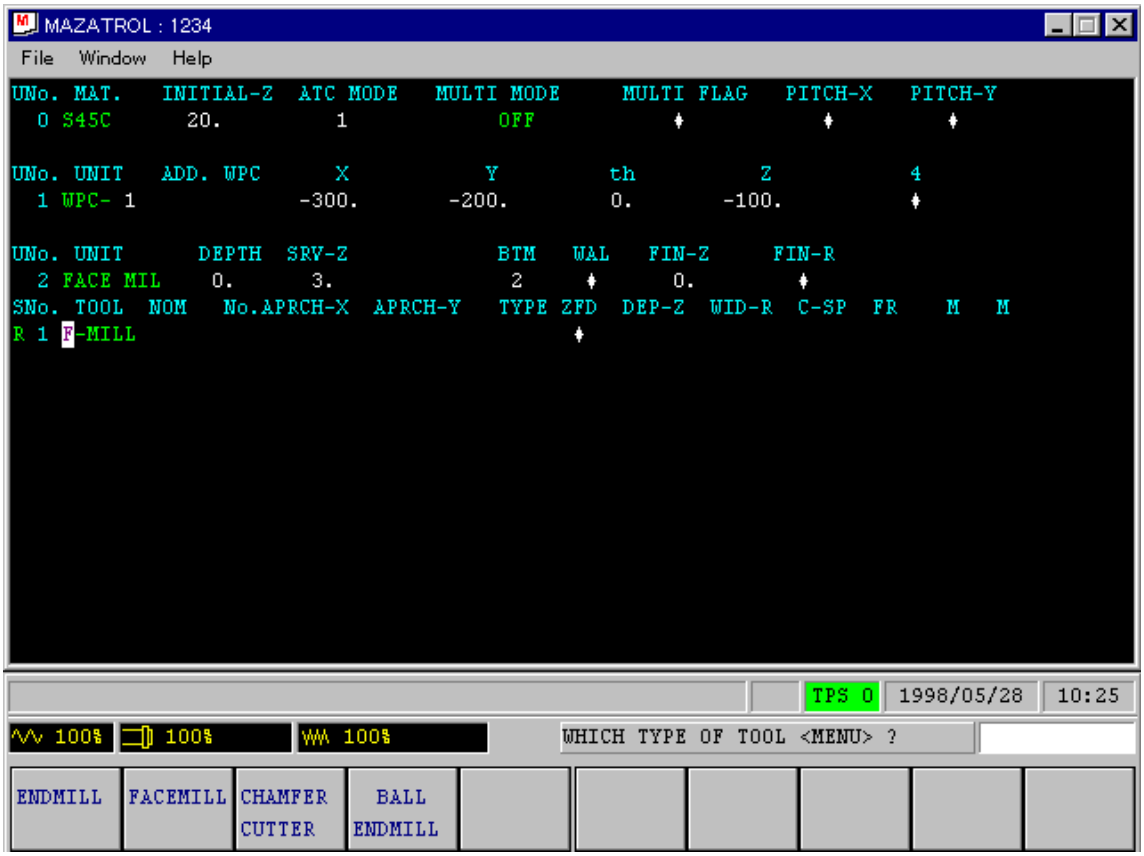
(5) **Z AXIS STOCK REMOVAL?** ..... 3 →

(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.)

A 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 machining unit. Here, a face mill has been selected.



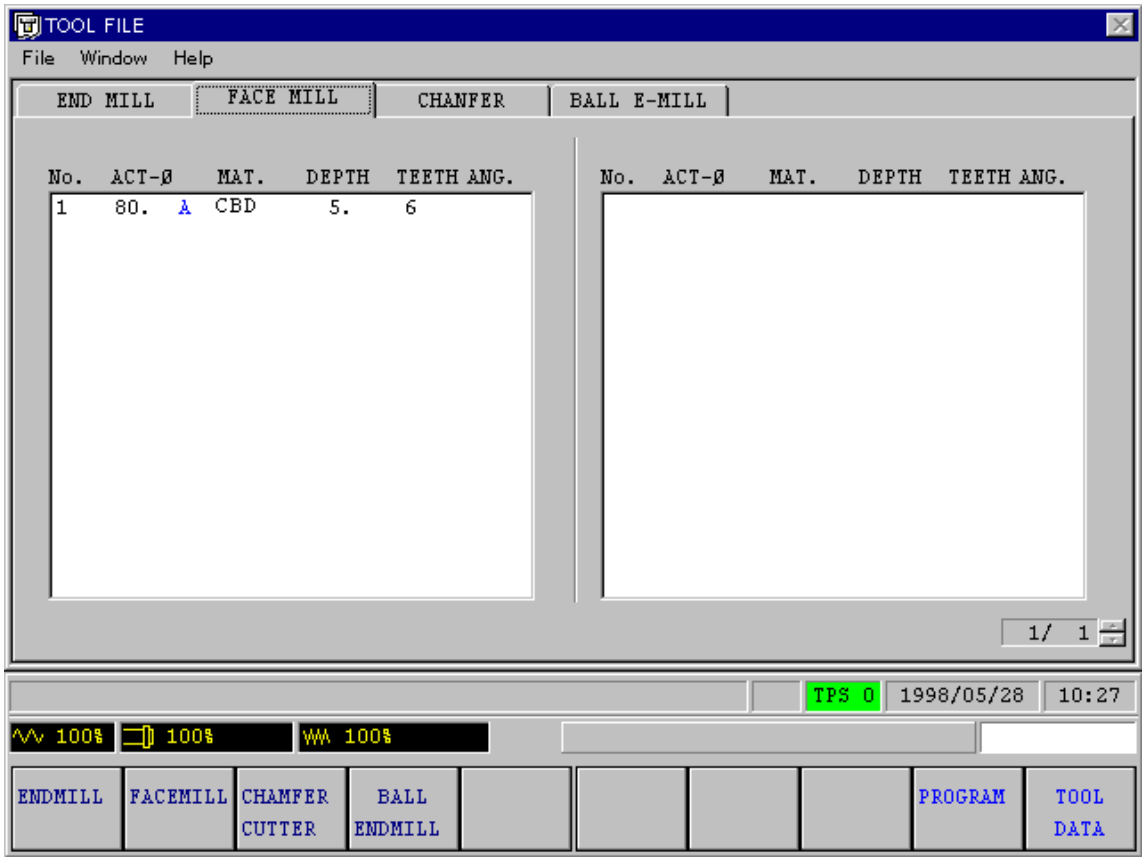
D735P0017E

2. Creation of tool sequence

(1) While the message **WHICH TYPE OF TOOL <MENU>?** is displayed, press the cursor key (→).

(2) Press the menu key **TOOL FILE** to call the **TOOL FILE** display on the screen, and check that the previously registered tool is included in the display.

➔ The following **TOOL FILE (FACEMILL)** display shows that the 80 mm diameter face mill is registered.



D735P0018E

(3) Press the menu key **PROGRAM** to return to the **PROGRAM** display and continue programming.

(4) **NOMINAL DIAMETER?** ..... 8 0 →

(5) **TOOL FILE CODE**?..... **A**

(6) **MACHINING PRIORITY No.?** ..... →

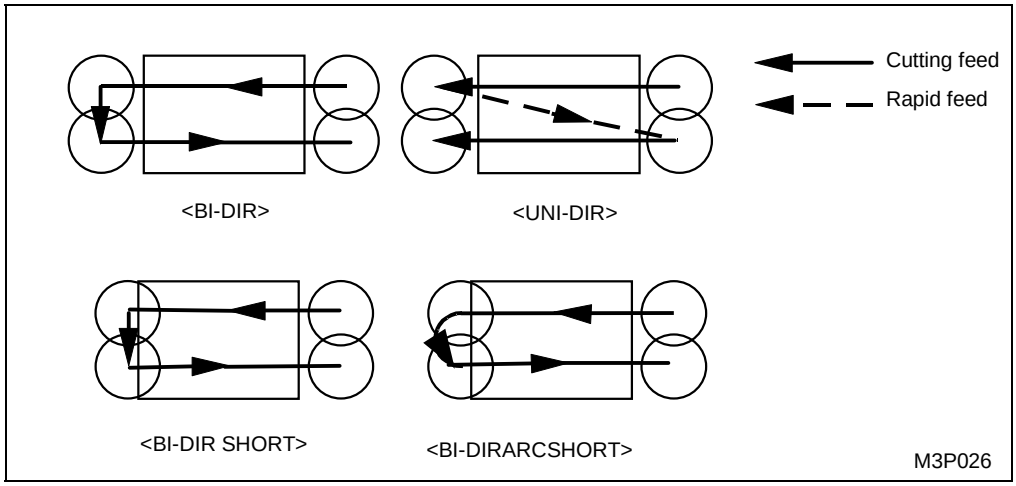
(Skip to the next item since the identical-tool priority function is not used here. Refer to Part 2, Chapter 5.)

(7) **APPROACH POINT X, AUTO → <MENU>?** ..... **AUTO SET**

(8) **APPROACH POINT Y, AUTO → <MENU>?** ..... **AUTO SET**

(9) **CUTTING DIRECTION <MENU>?** ..... **X BI-DIR**

The possible directions of cutting are shown below.



(10) **DEPTH OF CUT?** ..... **AUTO SET**

(11) **WIDTH OF CUT?** ..... **AUTO SET**

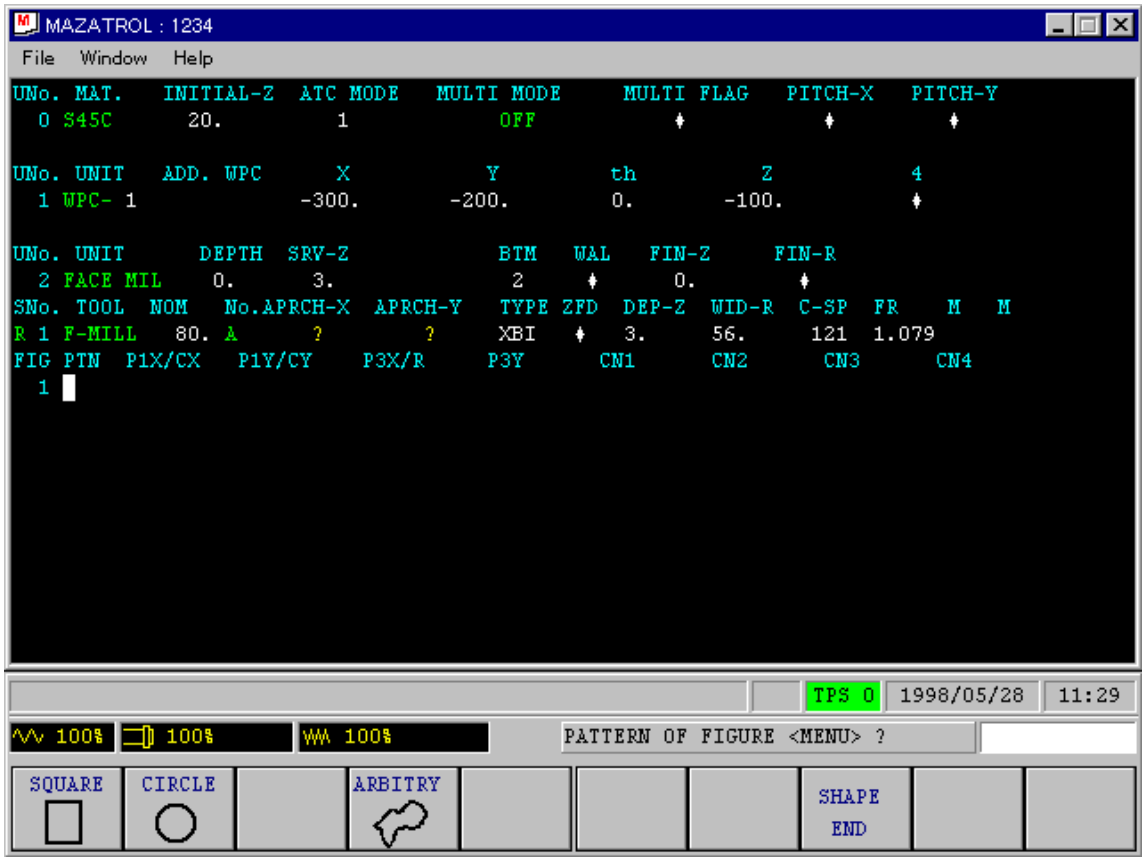
(12) **CUTTING SPEED, AUTO → <MENU>?** ..... **AUTO SET**

(13) **FEEDRATE, AUTO → <MENU>?** ..... **AUTO SET**

(14) **M CODE?** ..... [→]

(15) **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 article APRCH-X, APRCH-Y, however, maintains ? since the necessary data is automatically set upon completion of the tool path check.



D735P0019E

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.

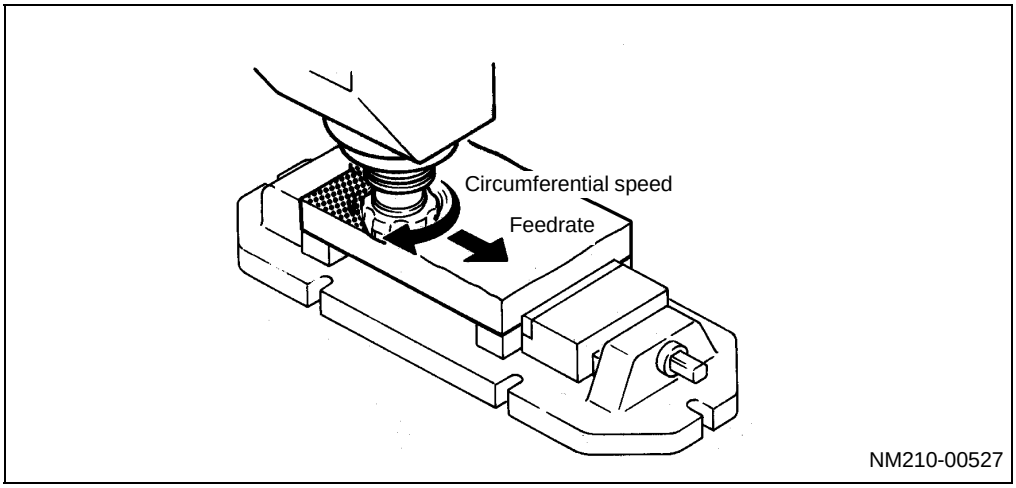
Circumferential speed (C-SP):

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.

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

Feedrate (FR):

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

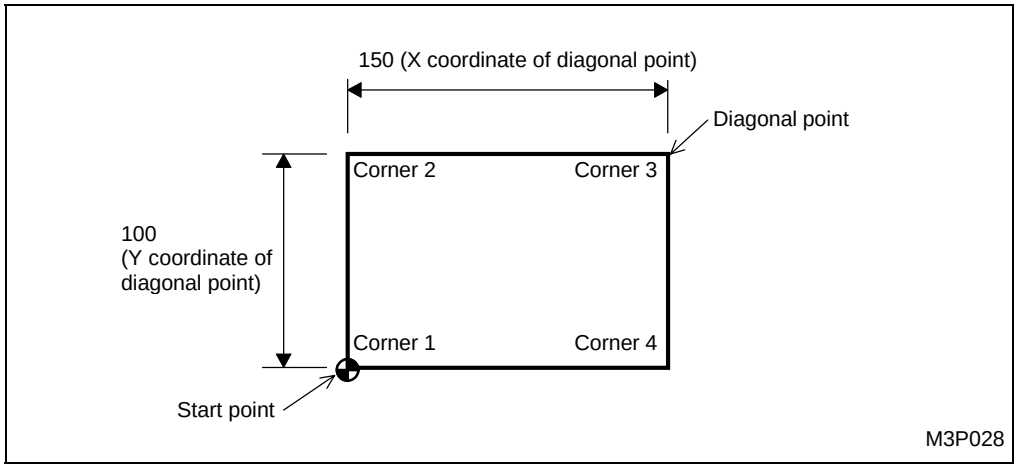


4. Creation of shape sequence

Next, let us program a shape sequence.

Read the machining area from the drawing, and input the coordinate values necessary for the machining shape. For face milling, the top face of workpiece is to be machined.

Thus, input the dimensions of a shape such as that shown below.



Note: The X- and Y-coordinates of the start point are those of the workpiece zero point.

(1) **PATTERN OF FIGURE <MENU>?** ..... **SQUARE**

(2) **CORNER 1 COORDINATE X?** ..... 0 →

(3) **CORNER 1 COORDINATE Y?** ..... 0 →

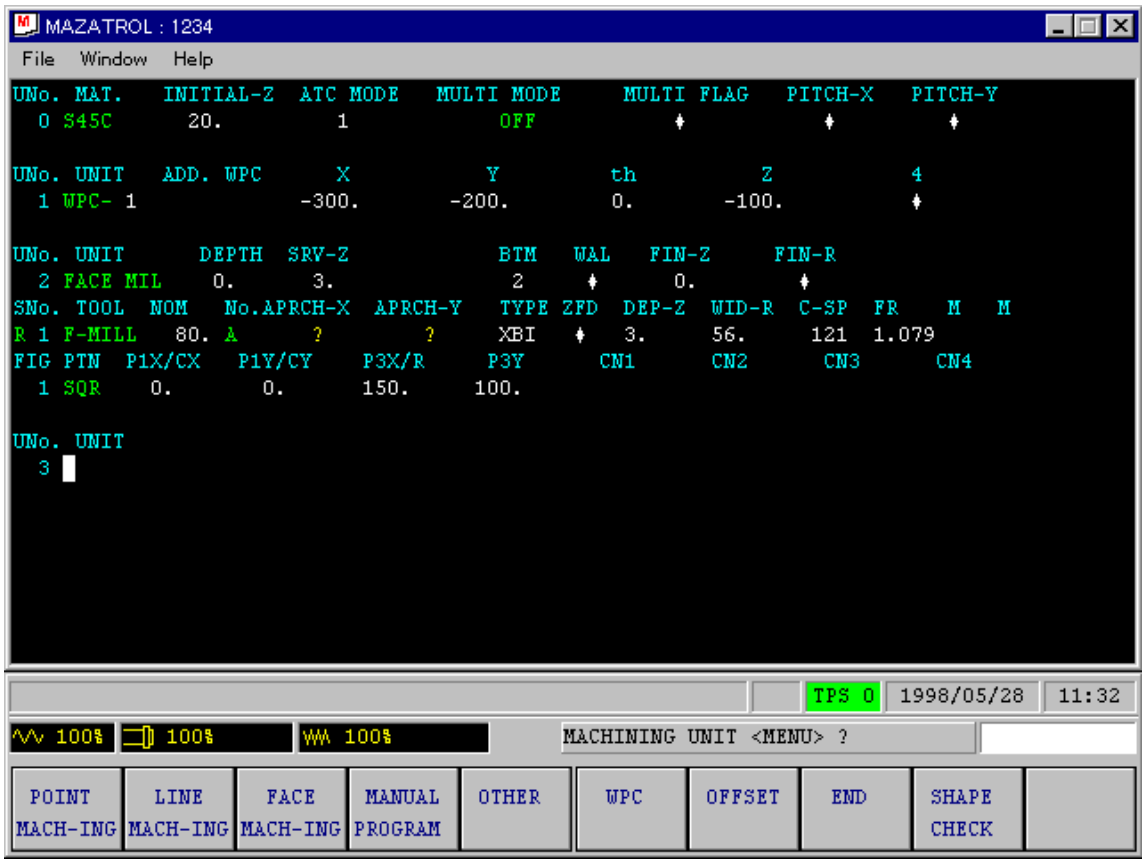
(4) **CORNER 3 COORDINATE X?** ..... 1 5 0 →

(5) **CORNER 3 COORDINATE Y?** ..... 1 0 0 →

(6) **CORNER 1 CHAMFER?** ..... ↓

(7) **PATTERN OF FIGURE <MENU>?** ..... **SHAPE END**

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



D735P0020E

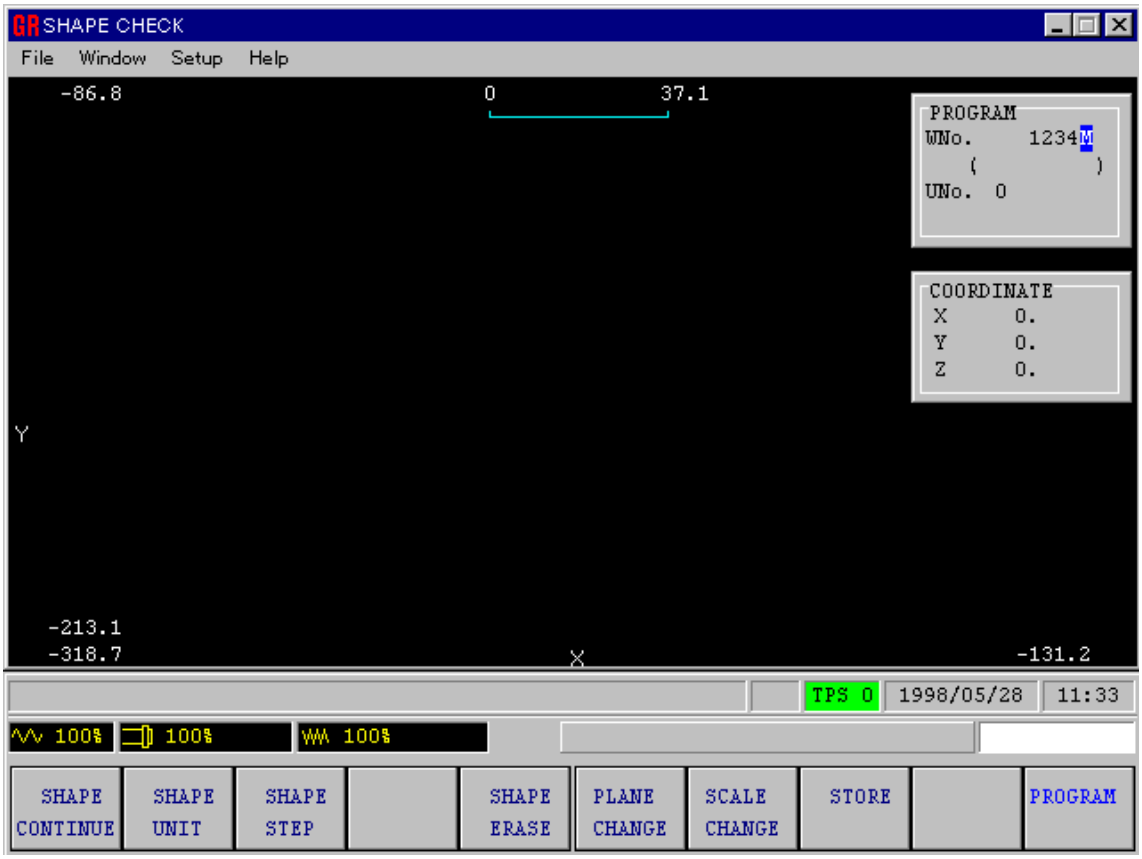
5. Shape check

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

Of the varioius 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**.

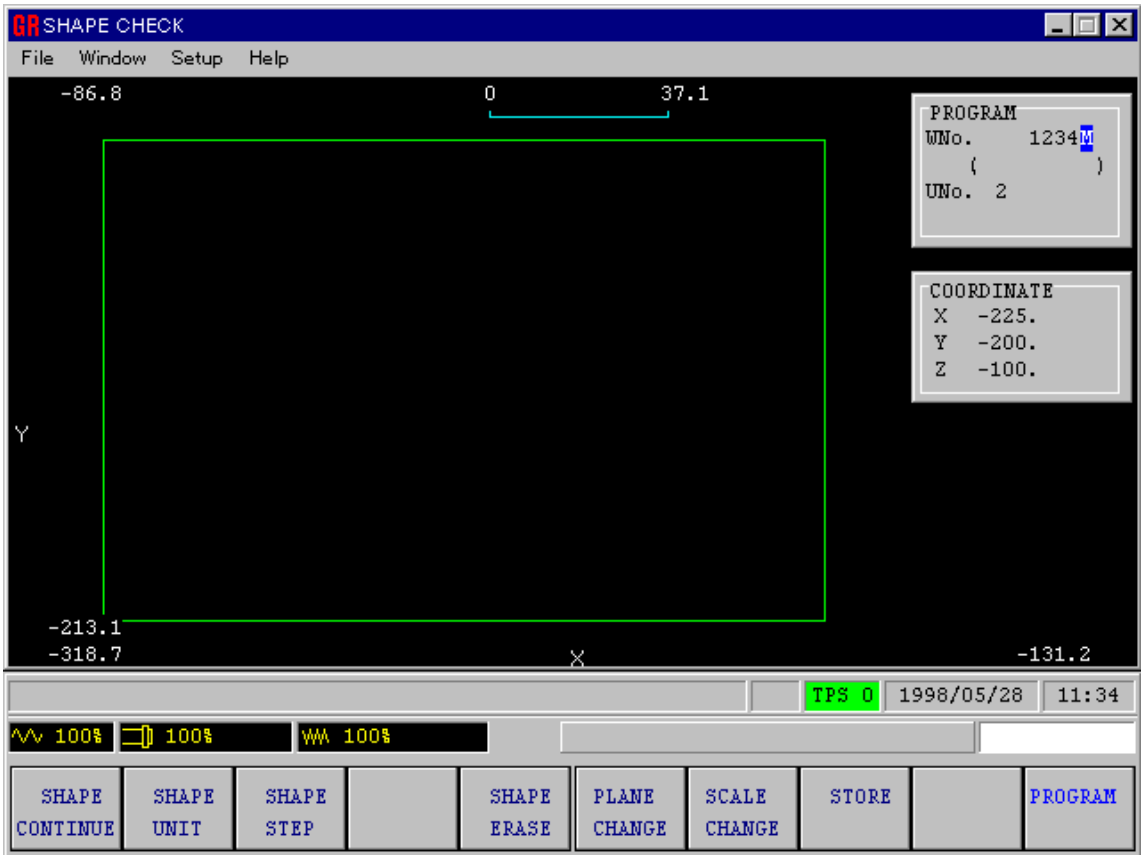
➔ The following **SHAPE CHECK** display will be displayed.



D735P0021E

(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.



D735P0022E

(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.

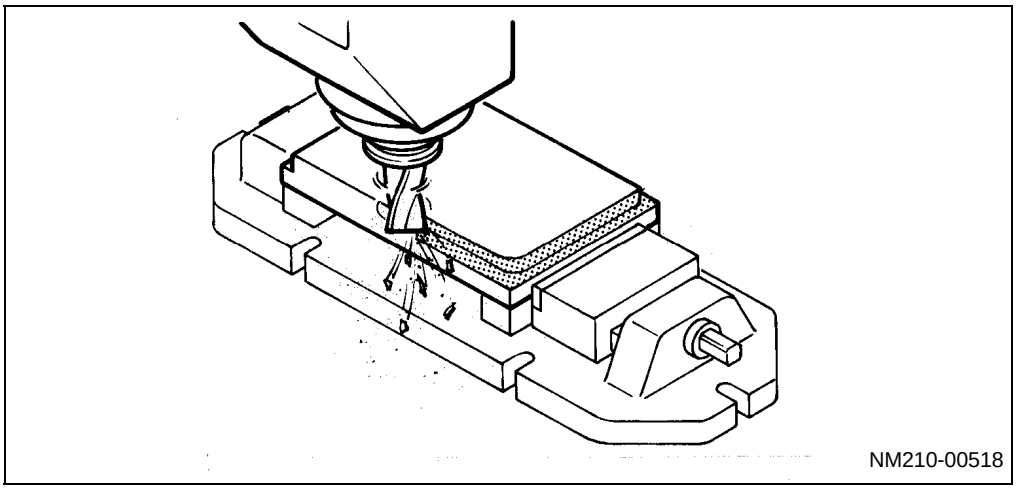
D735P0023E

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

This completes a face milling unit for use in face machining.

4-4-8 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.



NM210-00518

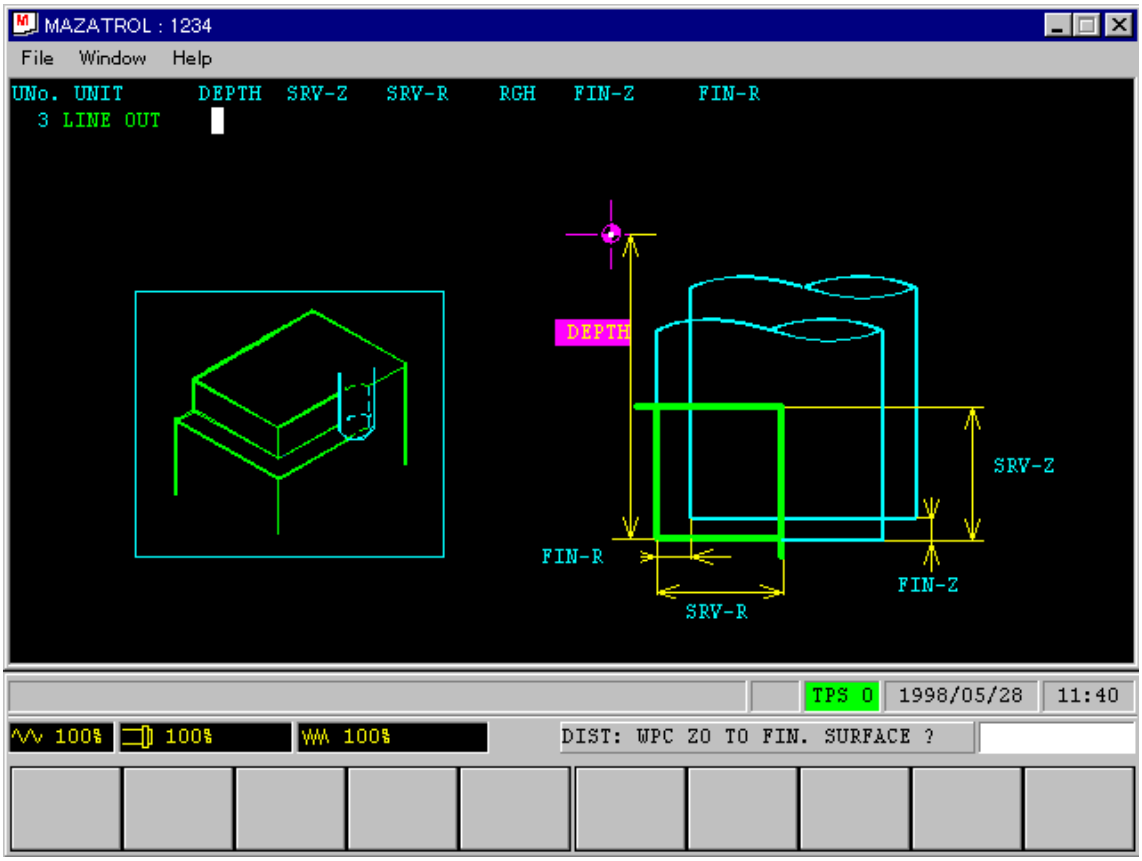
1. Creation of line machining unit

(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 help window will be presented.



D735P0024E

Now, start creating the line machining unit. The data for the article DEPTH, SRV-Z, SRV-R and RGH will be specified here.

DEPTH..... Input a depth value of 10.

SRV-Z ..... Input 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..... Input the wall thickness of the workpiece through which it is to be chamfered in the radial direction (see figure below).

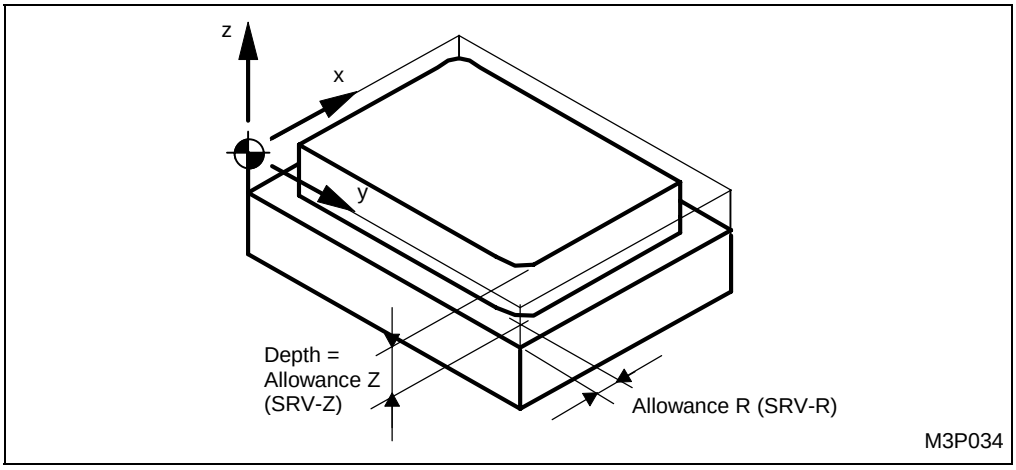


Fig. 4-10 SRV-Z and SRV-R

(4) **DIST: WPC Z0 TO FIN. SURFACE?** ..... 1 0 ⏎

(5) **Z AXIS STOCK REMOVAL?** ..... 1 0 ⏎

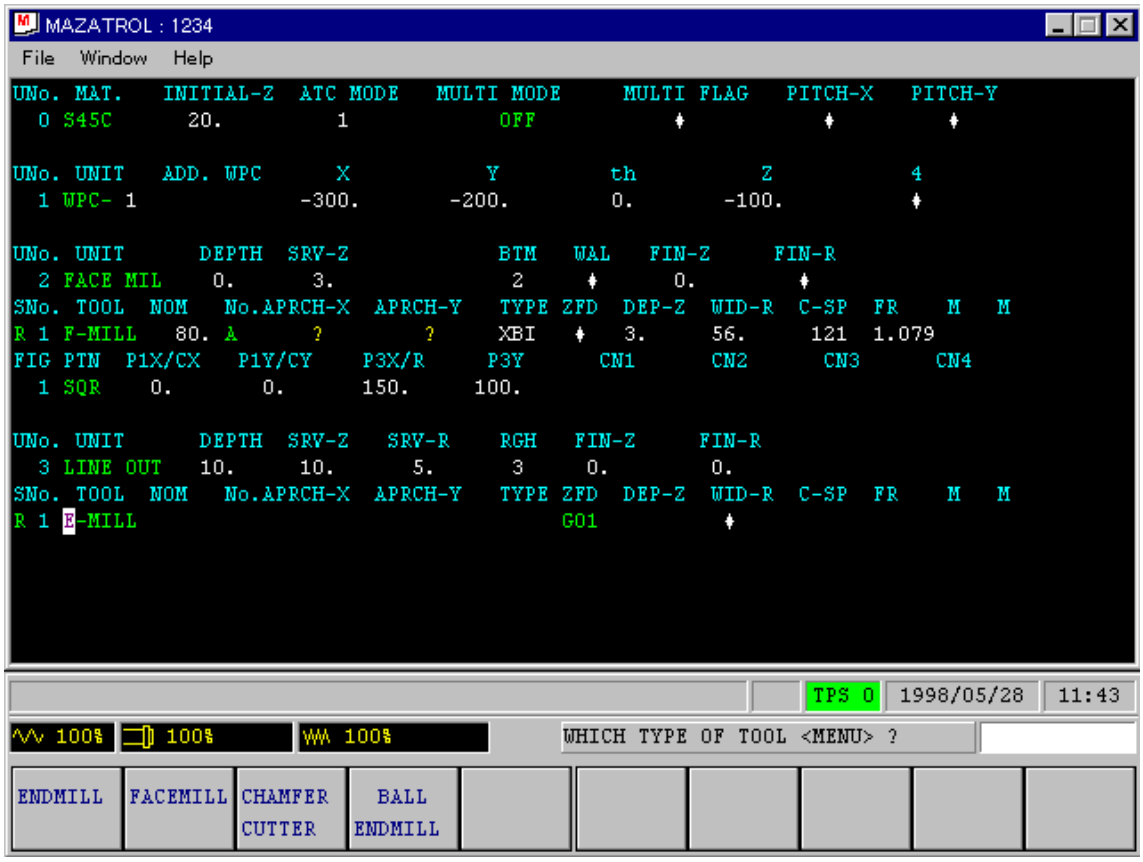
(6) **X/Y AXIS STOCK REMOVAL?** ..... 5 ⏎

(7) **SURFACE ROUGHNESS <MENU>?** 3

(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.



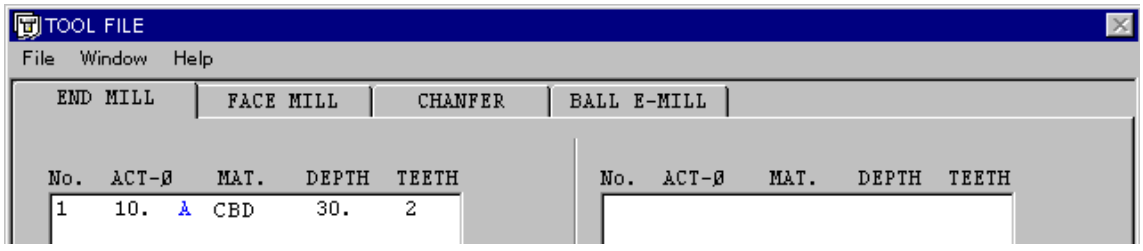
D735P0025E

2. Creation of tool sequence

(1) While the message **WHICH TYPE OF TOOL <MENU>?** is displayed, press the cursor key [→].

(2) Press the menu key **TOOL FILE** 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 (ENDMILL)** display shows that the 10-mm diameter end mill is registered.



D735P0026E

(3) Press the menu key **PROGRAM** to return to the **PROGRAM** display and continue programming.

(4) **NOMINAL DIAMETER?** ..... [1] [0] [→]

(5) **TOOL FILE CODE <MENU>?** ..... **A**

(6) **MACHINING PRIORITY No.?** ..... [→]

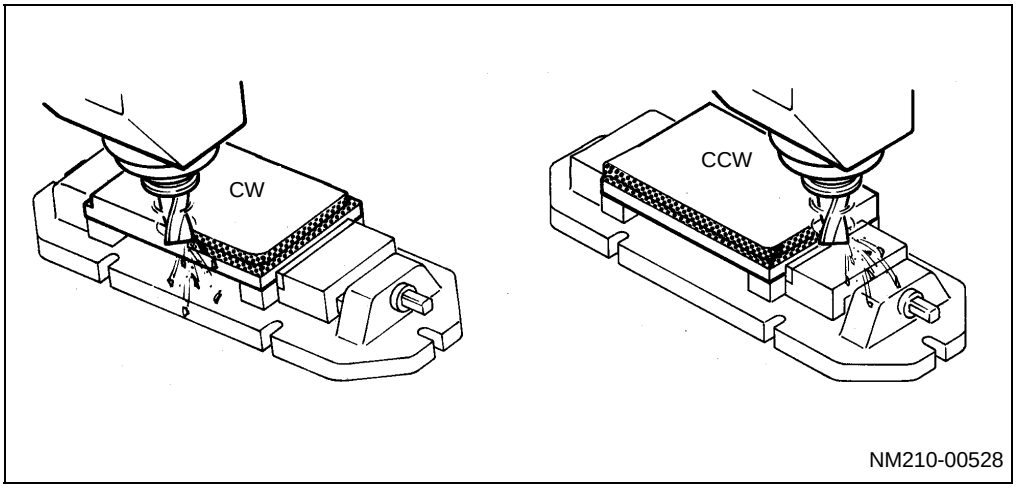
(Skip to the next article since the priority function for the same tool is not to be used here. See Part 2, Chapter 5.)

(7) **APPROACH POINT X, AUTO → <MENU>? ……AUTO SET**

(8) **[APPROACH POINT Y, AUTO → <MENU>? ..... AUTO SET**

(9) **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).)



(10) **FEEDRATE Z, <MENU>/DATA <INPUT>?** ..... ➔

(11) **DEPTH OF CUT?** ..... **AUTO SET**

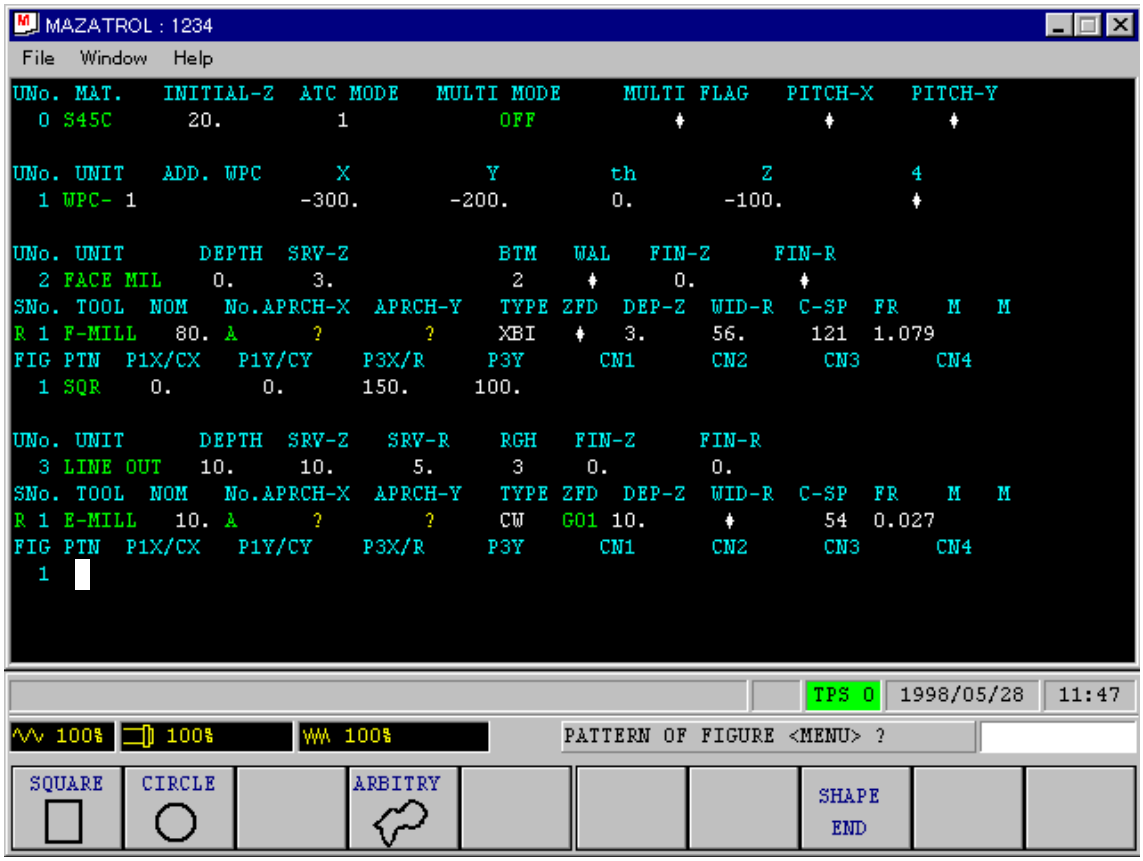
(12) **CUTTING SPEED, AUTO → <MENU>?** ..... **AUTO SET**

(13) **FEED RATE, AUTO → <MENU>?** ..... **AUTO SET**

(14) **M CODE?** ..... ➔

(15) **M CODE?** ..... ➔

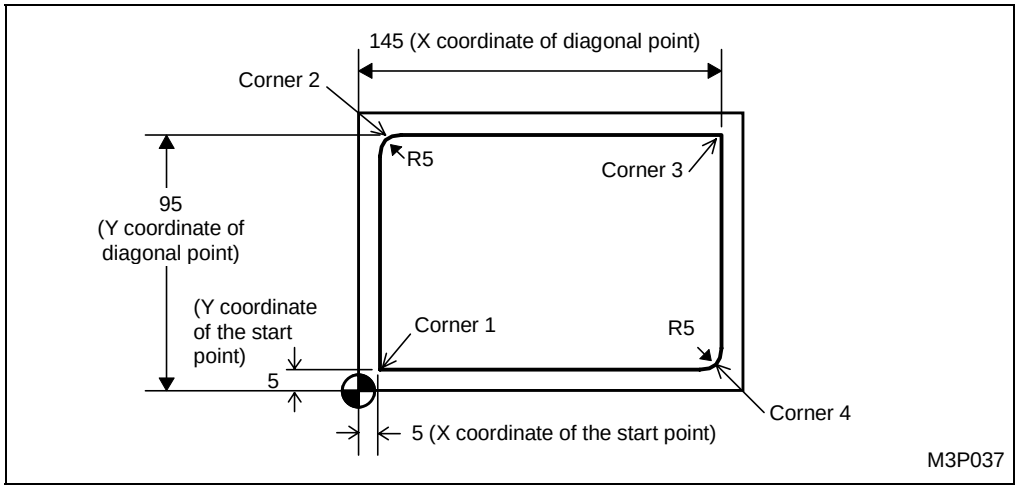
A tool sequence such as that shown below is now completed.



D735P0027E

3. Creation of 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.



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

(2) **CORNER 1 COORDINATE X?** ..... 5 ⏎

(3) **CORNER 1 COORDINATE Y?** ..... 5 ⏎

(4) **CORNER 3 COORDINATE X?** ..... 1 4 5 ⏎

(5) **CORNER 3 COORDINATE Y?** ..... 9 5 ⇨

(6) **CORNER 1 CHAMFER?** ..... ⇨

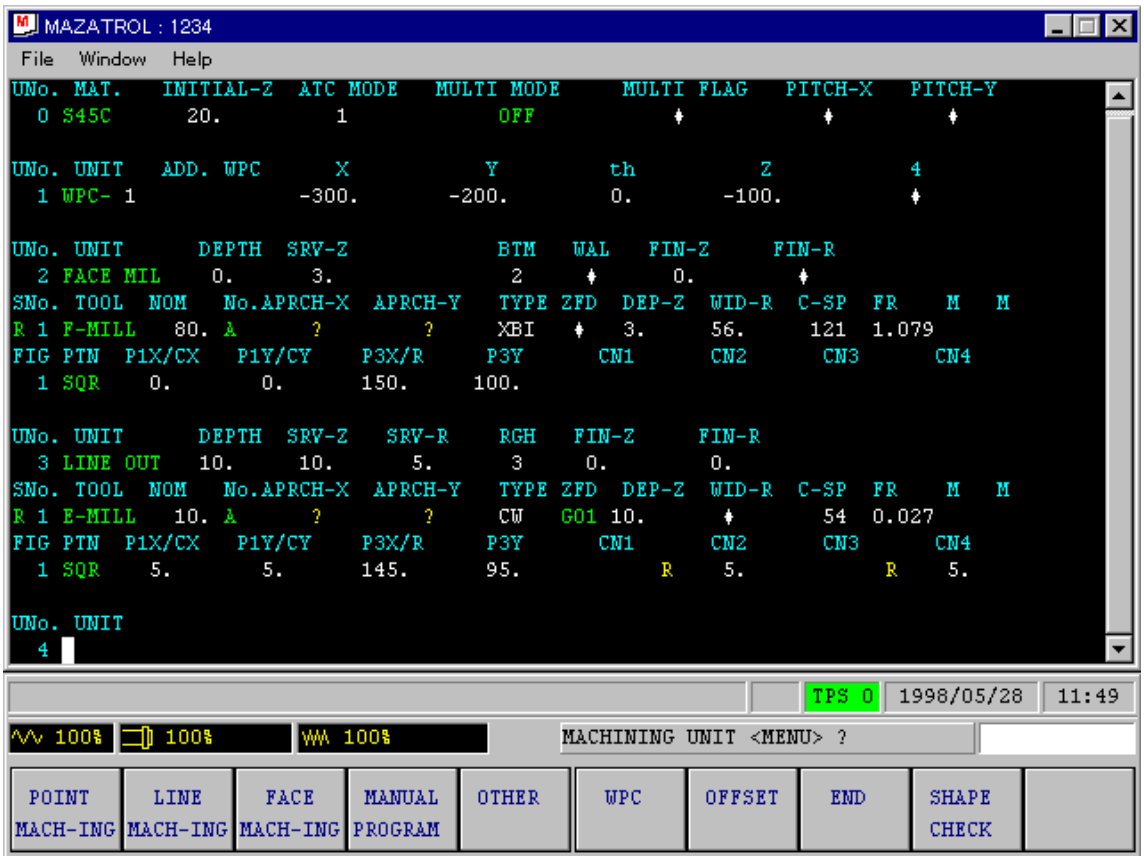
(7) **CORNER 2 CHAMFER?** ..... 5 ⇨

(8) **CORNER 3 CHAMFER?** ..... ⇨

(9) **CORNER 4 CHAMFER?** ..... 5 ⇨

(10) **PATTERN OF FIGURE <MENU>?** ..... **SHAPE END**

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



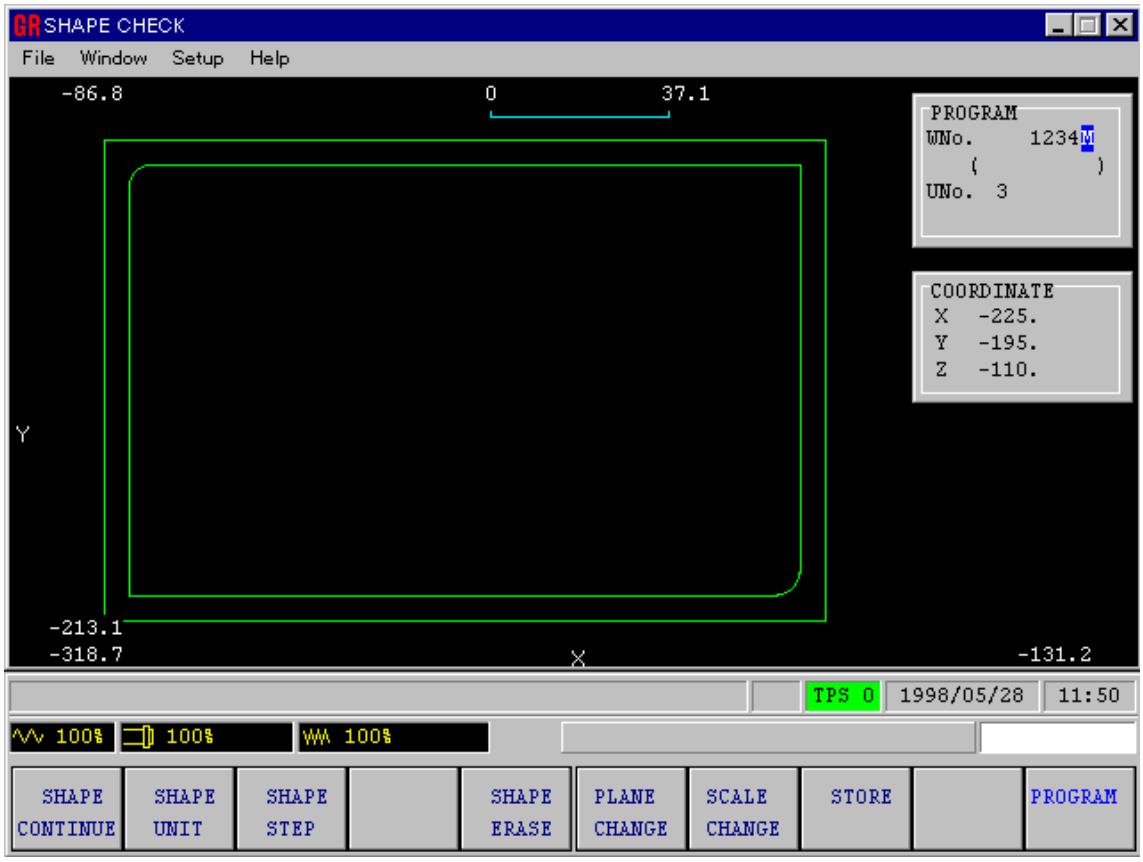
D735P0028E

4. Shape check

Here, let us call the **SHAPE CHECK** display on the screen.

(1) Press the menu key **SHAPE CHECK** to call the **SHAPE CHECK** display.

(2) Press the menu key **SHAPE CONTINUE**.



D735P0029E

The PLANE CHANGE funciton is described here. The shape being displayed in procedural step (2) is that of the workpiece as seen from above. With this function, it becomes possible to display shapes as seen from the side or from an oblique angle.

The selectable plane is one of the four types shown below. It is also possible to display one of two plane pairs (either XY-XZ (XY plane and XZ plane) or XY-YZ (XY plane and YZ plane)) at the same time. See Section 6-3, “SHAPE CHECK Display” of the Operating Manual for the details.

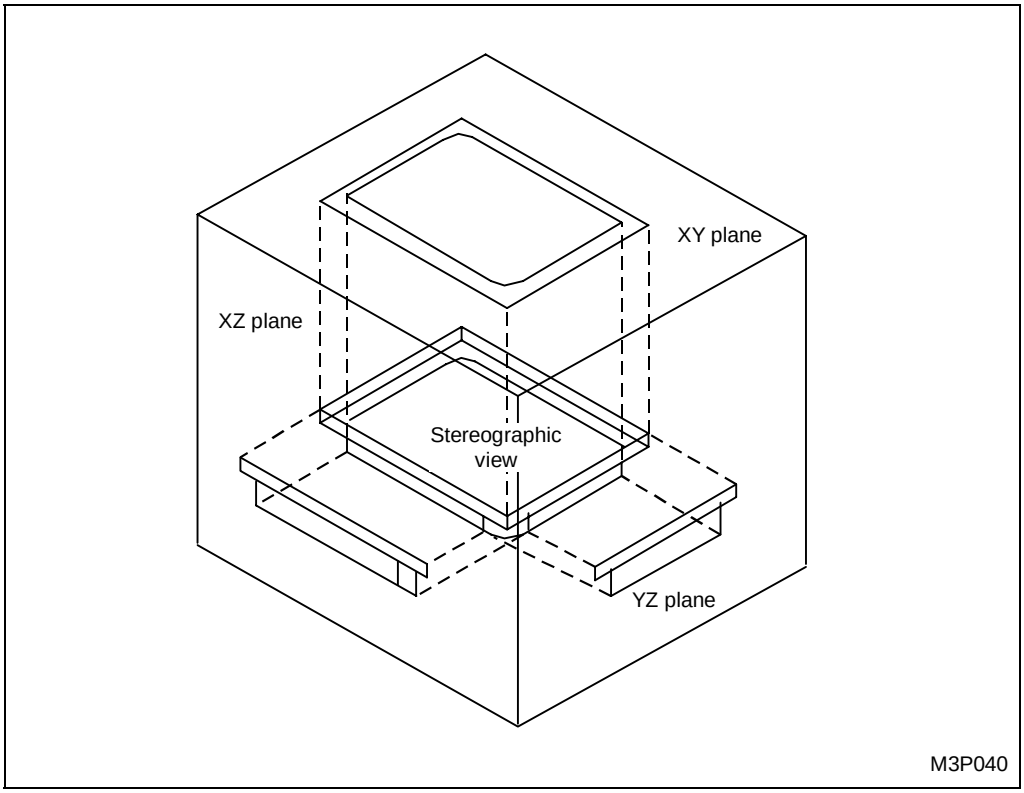


Fig. 4-11 Plane selection

Let us now display shapes of various planes.

XY-XZ plane

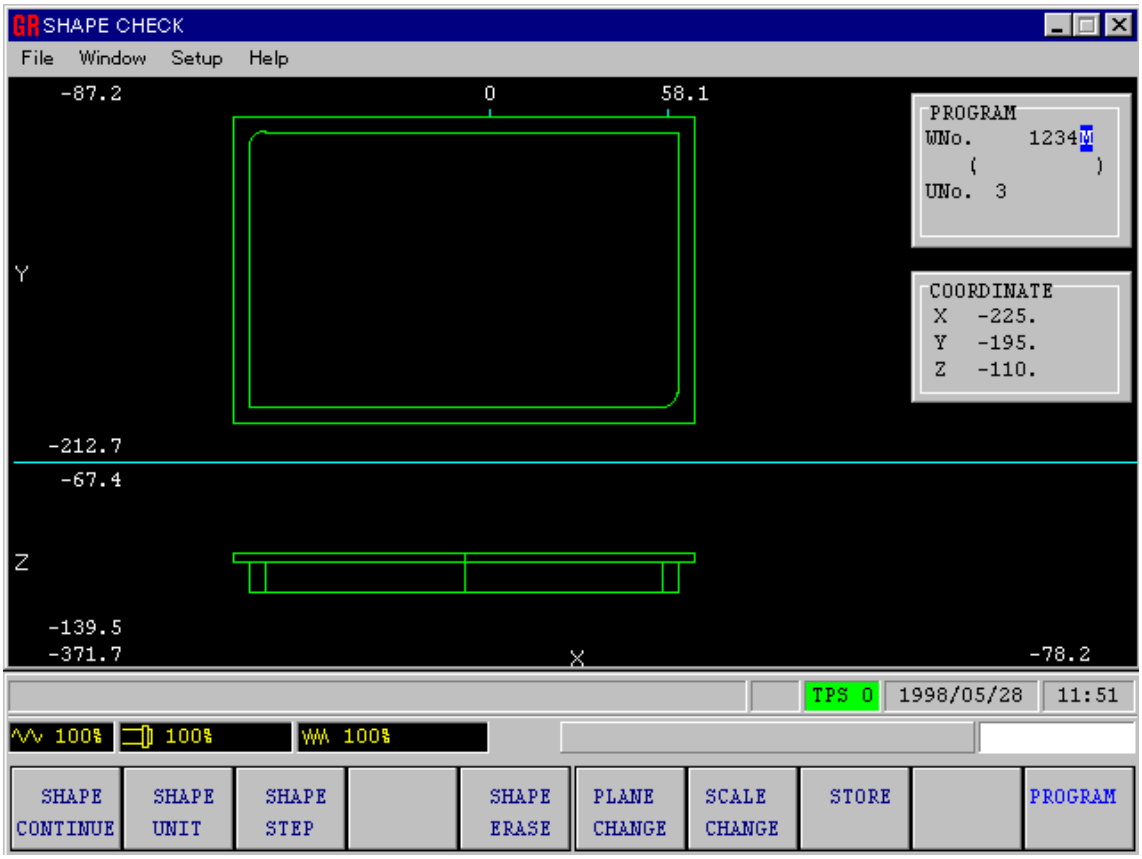
(3) Press the menu key **PLANE CHANGE**.

(4) Press the menu key **XY-XZ**.

➔ Shapes as seen from above and from the side are selected.

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

➔ As shown below, a shape as seen from above (XY plane) and a shape as seen from the side (XZ plane) are displayed at the same time.



D735P0030E

Three-dimensional plan

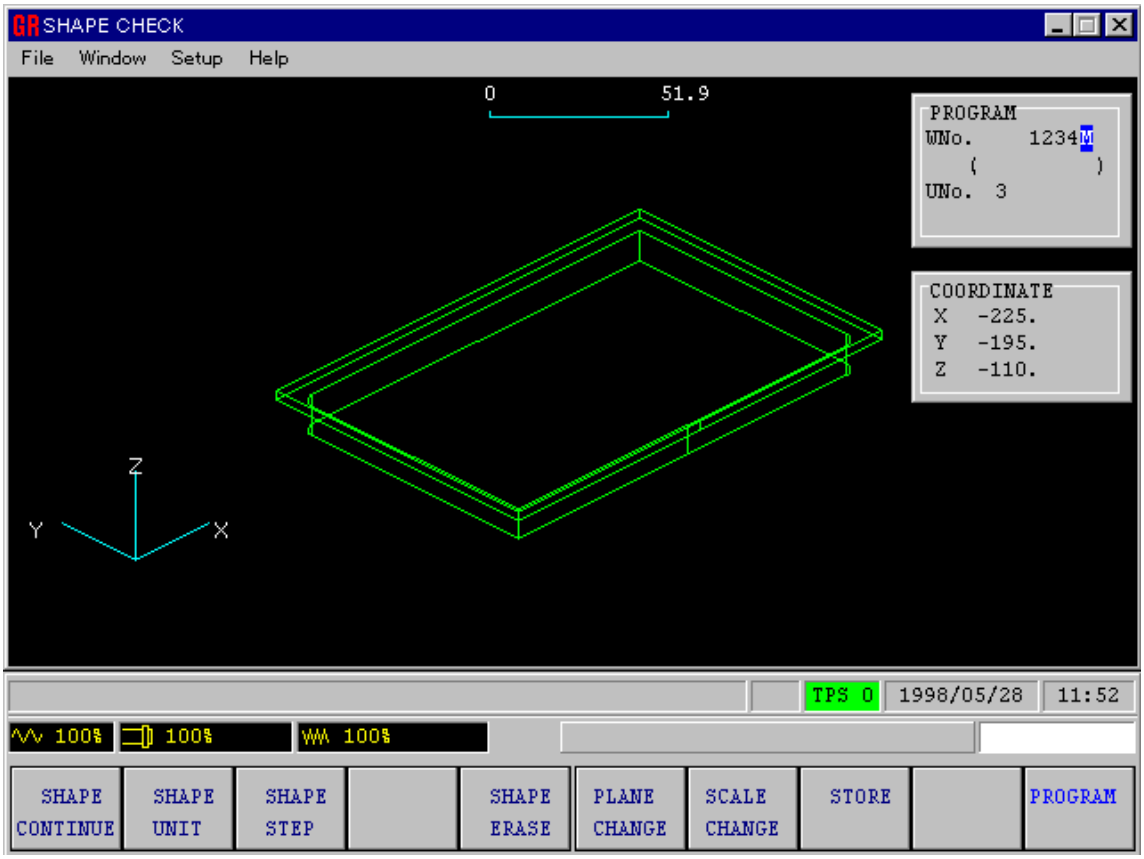
(6) Press the menu key **PLANE CHANGE**.

(7) Press the menu key **3-D**.

➔ A shape as seen obliquely from above is selected.

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

➔ The shape will be displayed in stereographic form as shown below.



D735P0031E

(9) Press the menu key **PLANE CHANGE**.

(10) Press the menu key **XY** to change the display to that of XY plan.

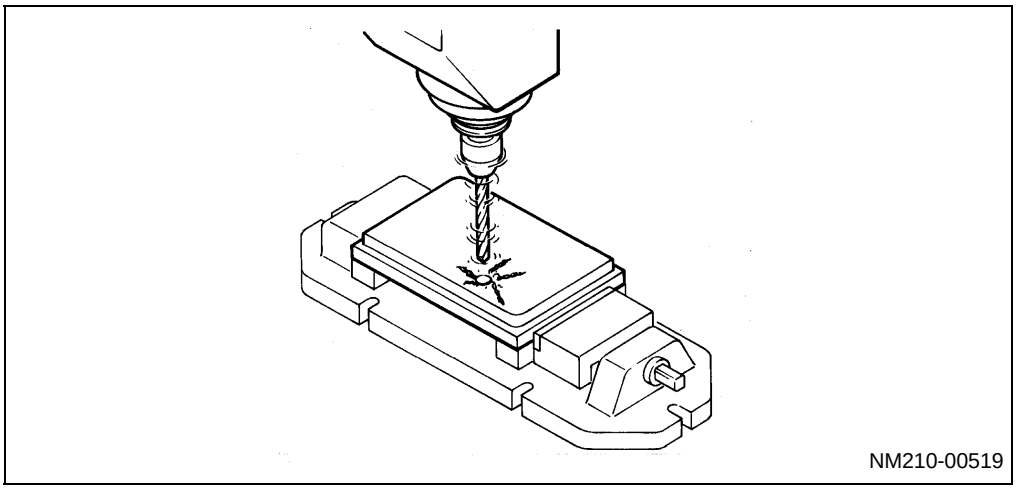
Note: The selected plane maintains its shape until another plane has been selected.

(11) Press the menu key **PROGRAM**. The **PROGRAM** display is resumed.

This completes an outside linear machining unit.

4-4-9 Point machining unit (1)

Let us create a program unit that is used to drill a 10-mm diameter through-hole. This machining unit is referred to as a drilling unit.



NM210-00519

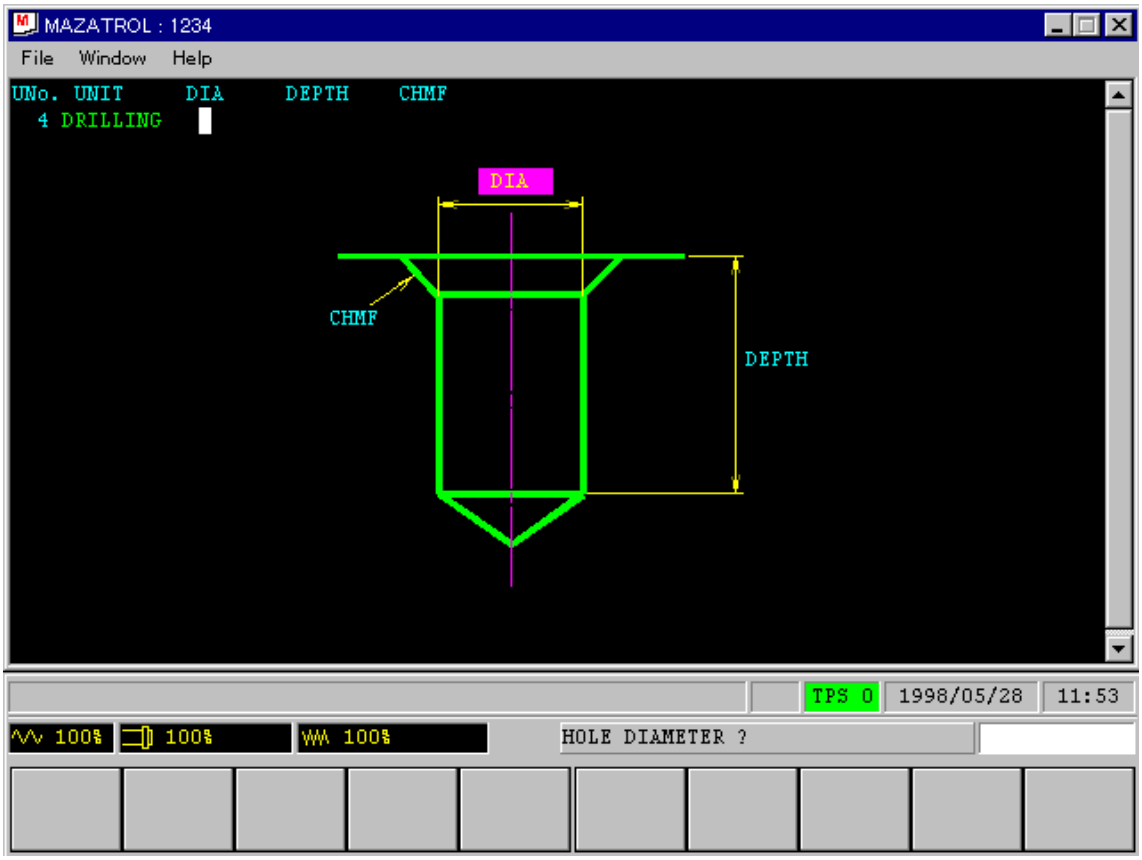
1. Creation of point machining unit (1)

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

(2) **MACHINING UNIT <MENU>?**..... **DRILLING**

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

➔ The following help window will be presented.



D735P0032E

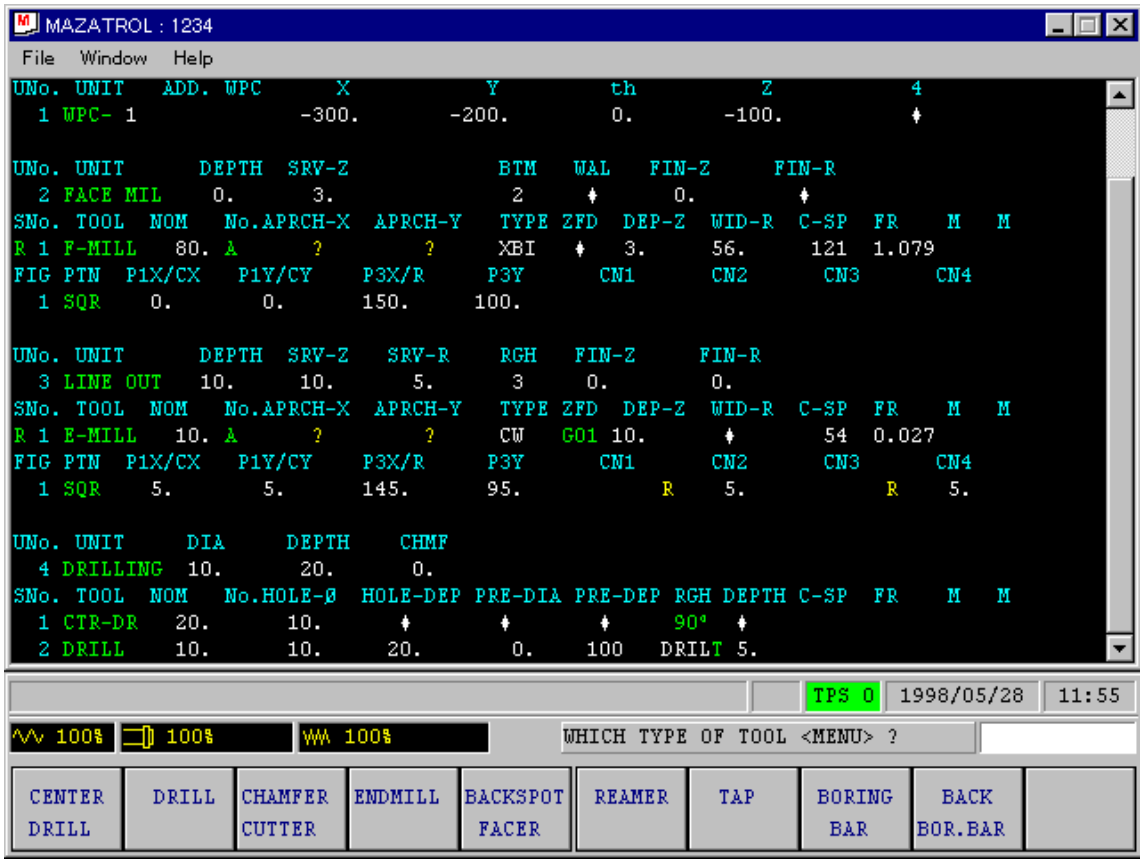
Now, start creating the point machining unit. The data for the item DIA, DEPTH and CHMF will be specified here.

(4) **HOLE DIAMETER?**..... 1 0 ⏎

(5) **HOLE DEPTH?**..... 2 0 ⏎

(6) **CHAMFER WIDTH?**..... 0 ⏎

A machining program 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.



D735P0033E

2. Creation of 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 article C-SP by pressing the cursor key → six 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 → eight times in succession to move the cursor to the position under the article C-SP.

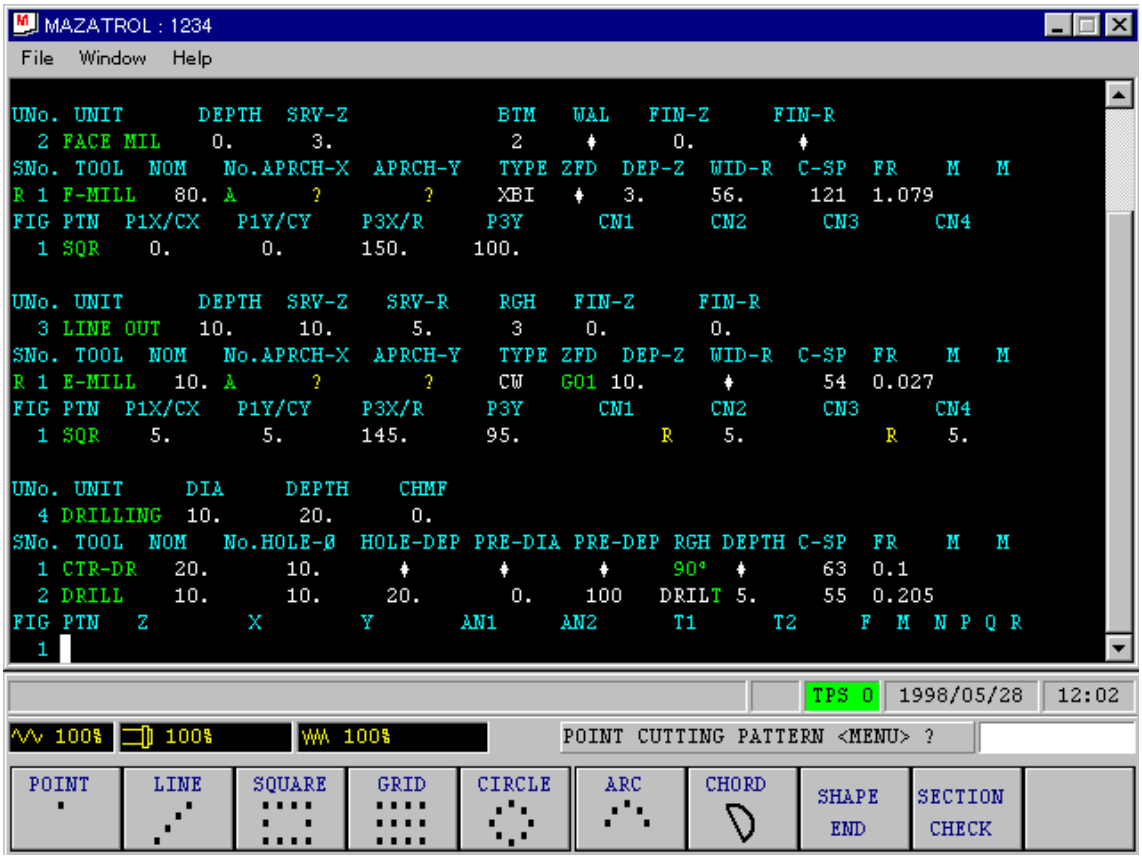
(7) **CUTTING SPEED, AUTO → <MENU>?** ..... **HSS AUTO**

(8) **FEEDRATE, AUTO → <MENU>?** ..... **HSS AUTO**

(9) **M CODE?** ..... →

(10) **M CODE?** ..... →

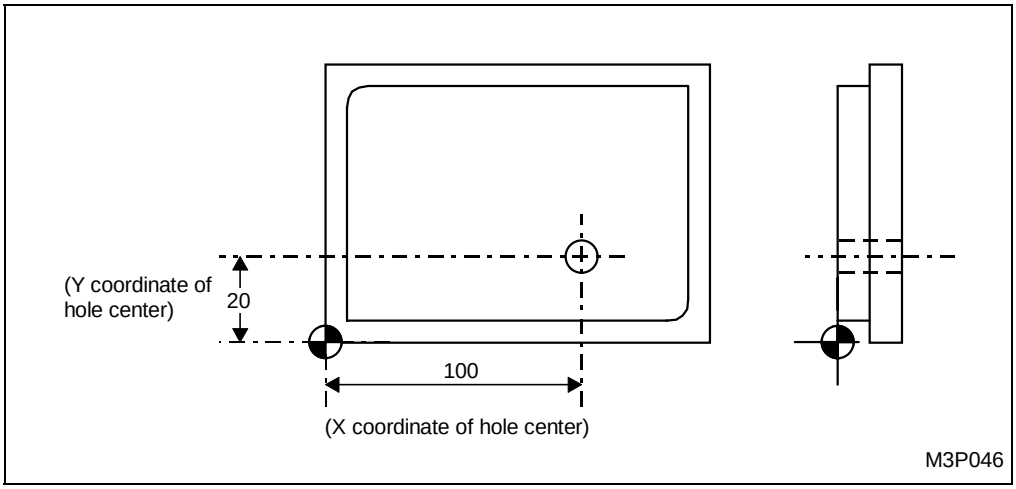
A tool sequence such as that shown below is now completed.



D735P0034E

3. Creation of shape sequence

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



M3P046

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

(2) **Z VALUE OF WORK SURFACE?** ..... 0 ⏎

(3) **HOLE POSITION X?** ..... 1 0 0 ⏎

(4) **HOLE POSITION Y?** ..... 2 0 ⏎

(5) **PATH <XY:0, Y→X:1, X→Y:2>?** 0

(The tool path to the next hole is to be selected from the three types shown below. If, however, the hole to which the tool is to be fed does not exist, input 0 since the selected function is unavailable.)

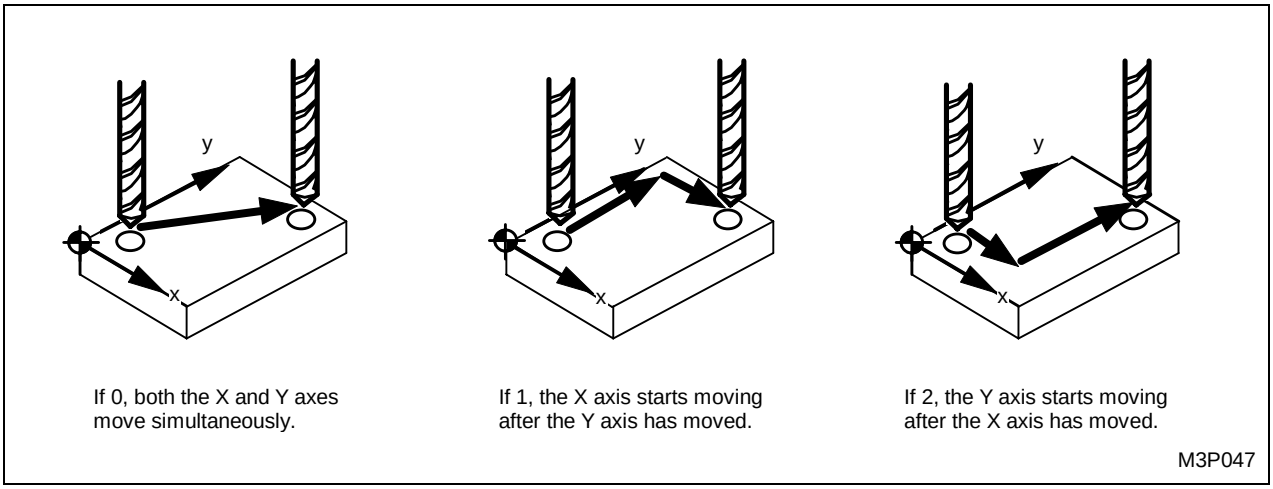


Fig. 4-12 Tool path

(6) **OMIT SPT MACHINING <Y:1, N:0>?** 0

(If 0, drilling occurs. If 1, positioning only occurs.)

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

The height of the tool path to the next hole is to be selected from the two types shown below. (If however, the hole to which the tool is to be fed does not exist, input 0 since the selected function is unavailable.)

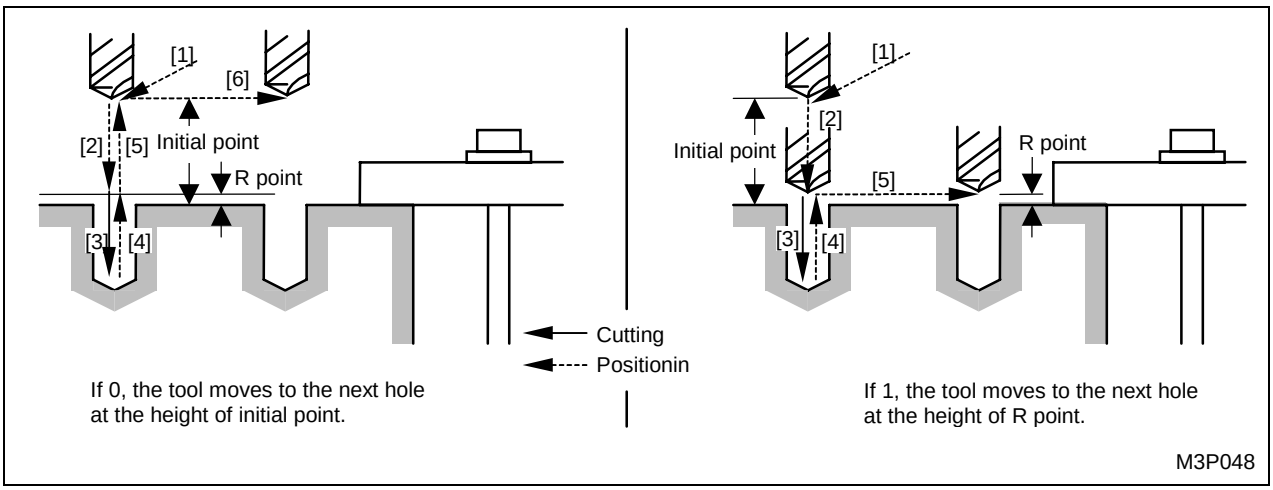
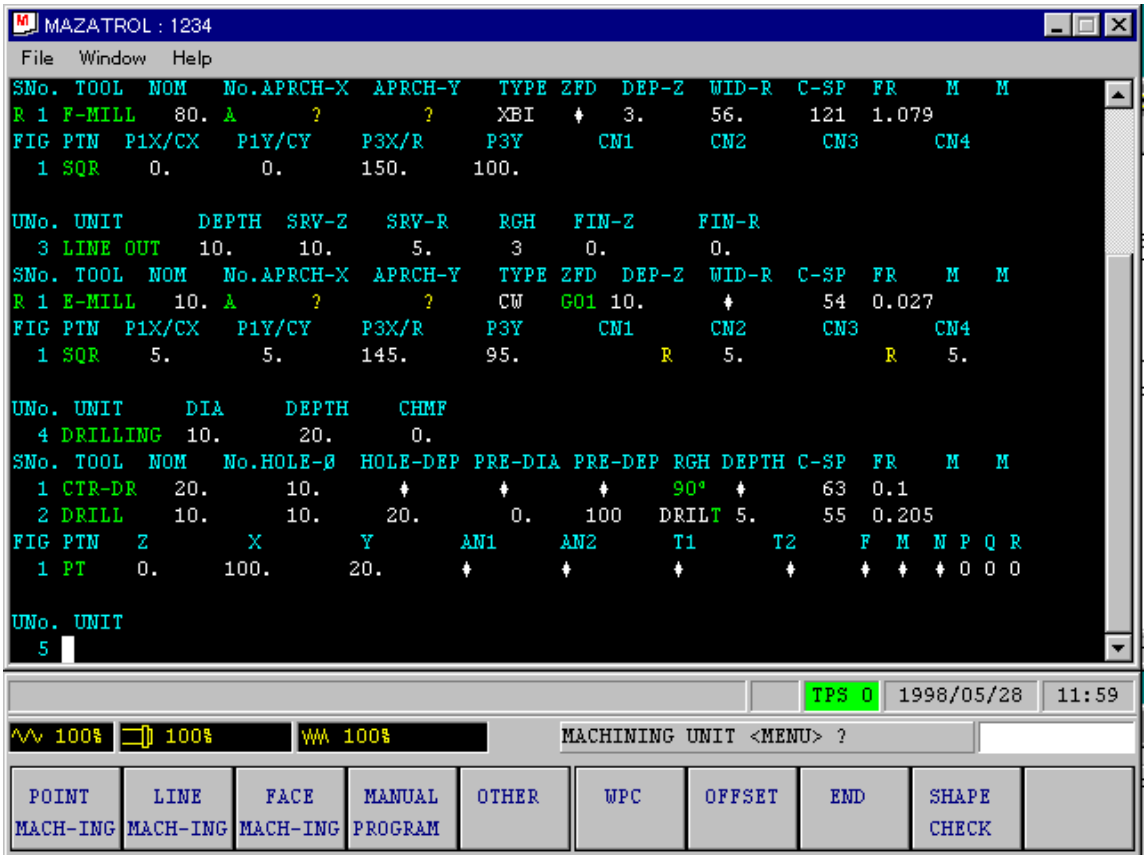


Fig. 4-13 Return position

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

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



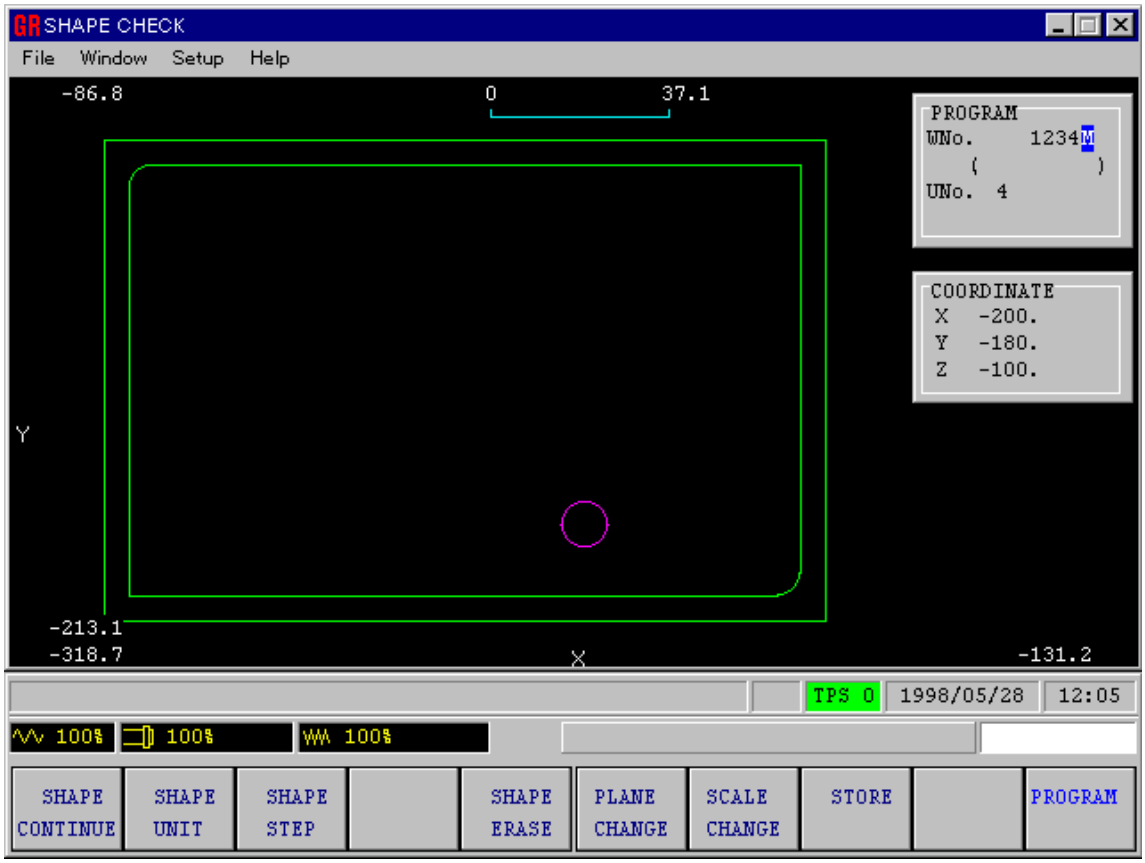
D735P0035E

4. Shape check

Let us call the **SHAPE CHECK** display on the screen to check the shapes.

(1) Press the menu key **SHAPE CHECK** to call the **SHAPE CHECK** display.

(2) Press the menu key **SHAPE CONTINUE**.



D735P0036E

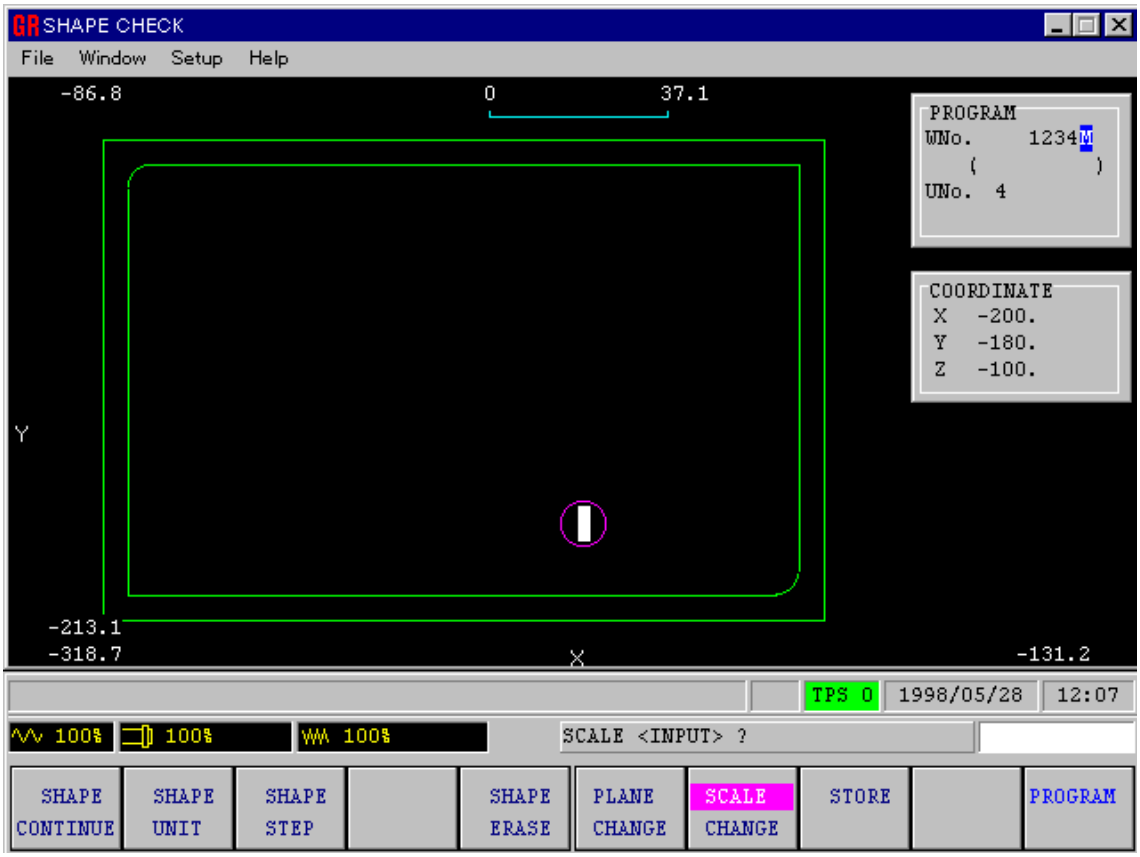
The SCALE CHANGE function is explained here. This function allows enlargement/reduction of shapes, as well as changing of the shape display position.

Enlargement of shapes

Let us display a 10 mm diameter hole in enlarged form using the SCALE CHANGE function.

(3) Press the menu key **SCALE CHANGE**.

(4) Using the cursor keys ↓ →, place the cursor on the 10 mm diameter hole position as shown below.



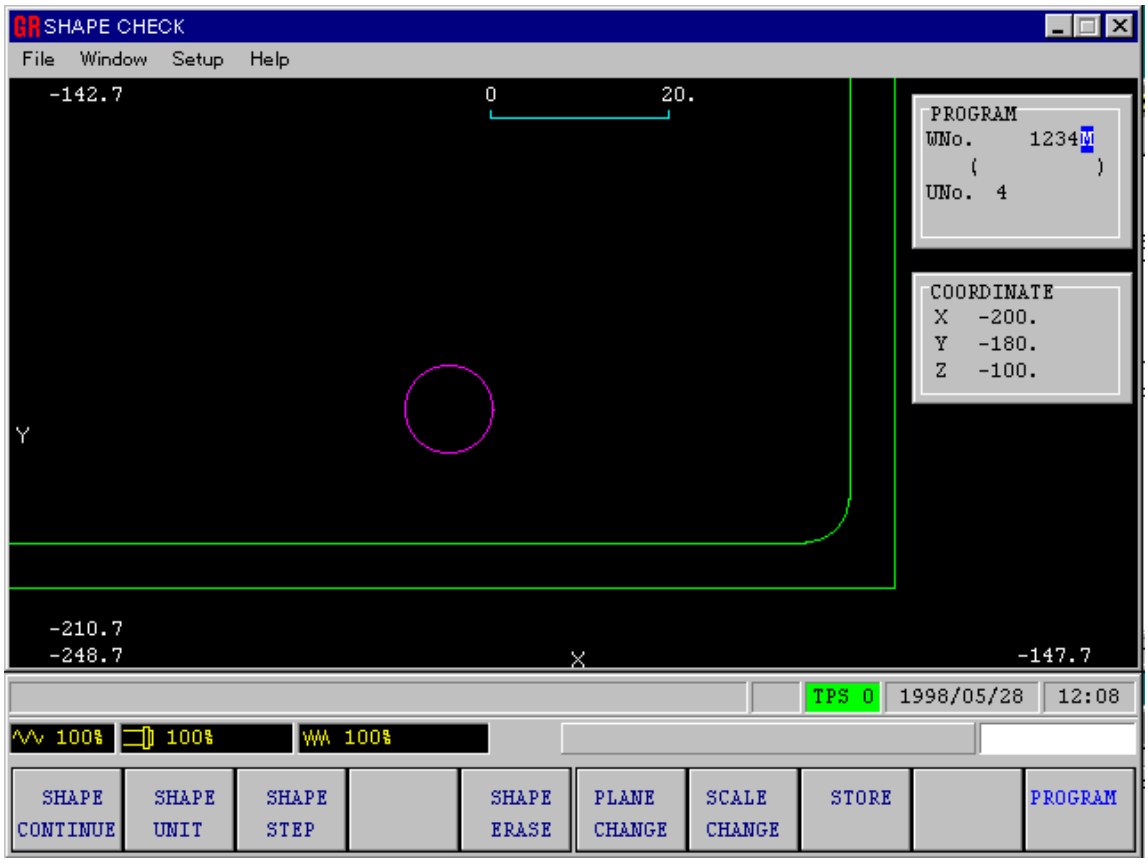
D735P0037E

(5) **SCALE <INPUT>?**..... 2 0 ⏎

(The size of the shape is changed so that the line in the center on the top of the screen becomes 20 mm in length.)

(6) Press the menu key **SHAPE CONTINUE**.

➔ As shown below, the shape has been enlarged with its center in the 10 mm diameter hole position.



D735P0038E

Reduction of shapes

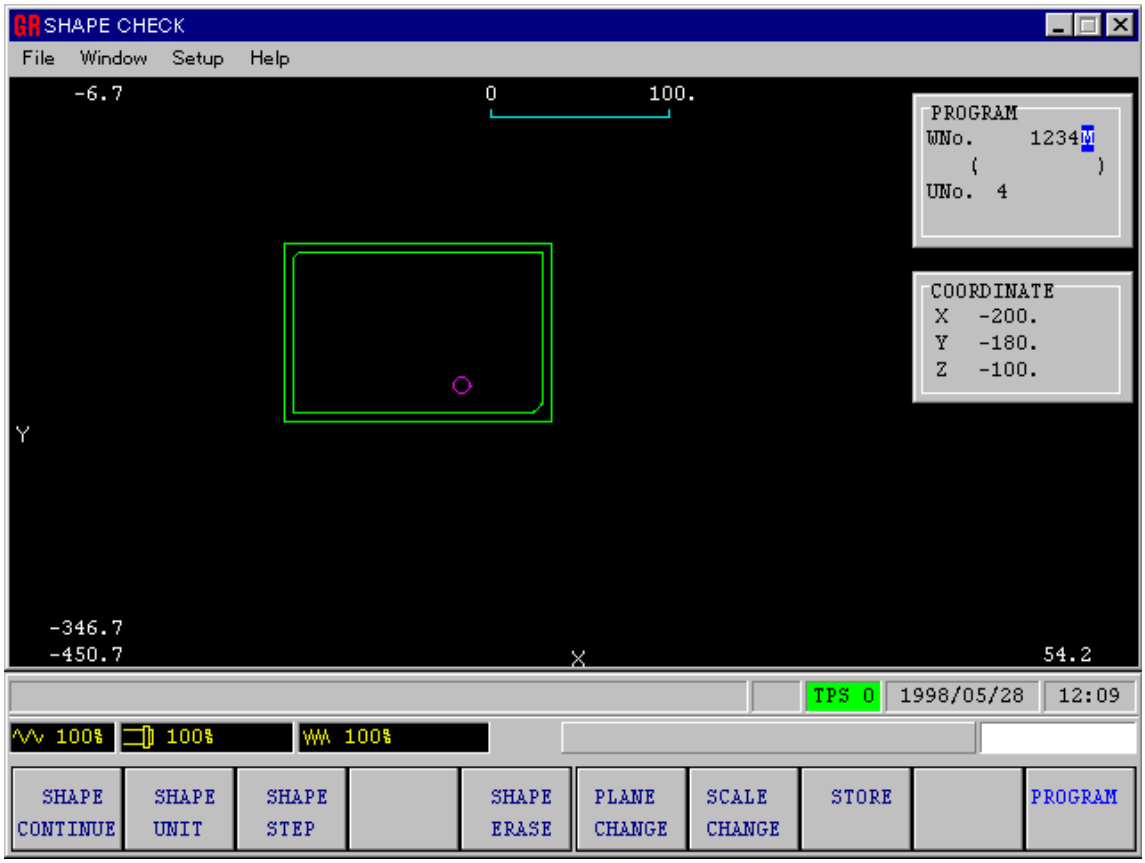
(7) Press the menu key **SCALE CHANGE**.

(8) **SCALE <INPUT>?**..... 1 0 0 ⏎

(The size of the shape is changed so that the line in the center on the top of the screen becomes 100 mm in length.)

(9) Press the menu key **SHAPE CONTINUE**.

➔ As shown below, the shape has been reduced in size.



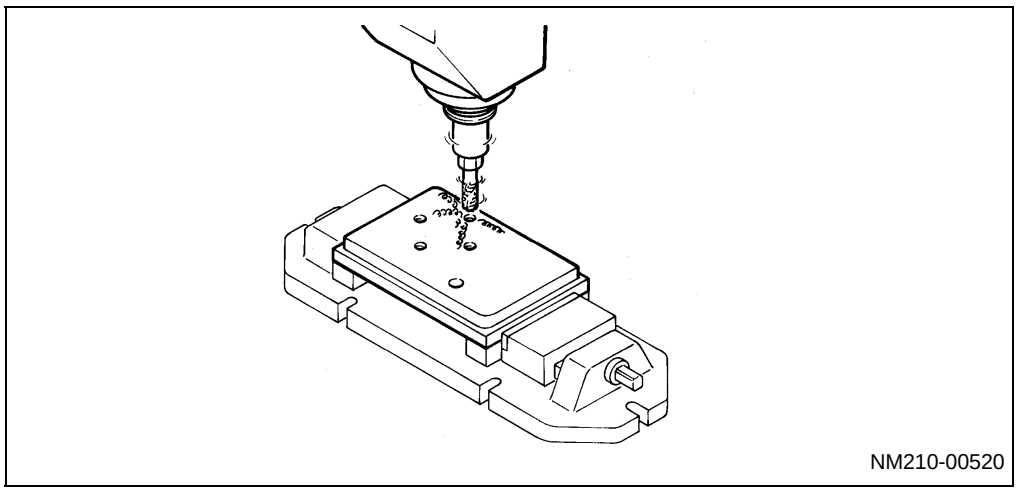
D735P0039E

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

This completes the creation of a point machining unit.

4-4-10 Point machining unit (2)

Let us create a unit that is used to drill four M8 tapped through-holes. This machining program unit is referred to as a tapping unit.



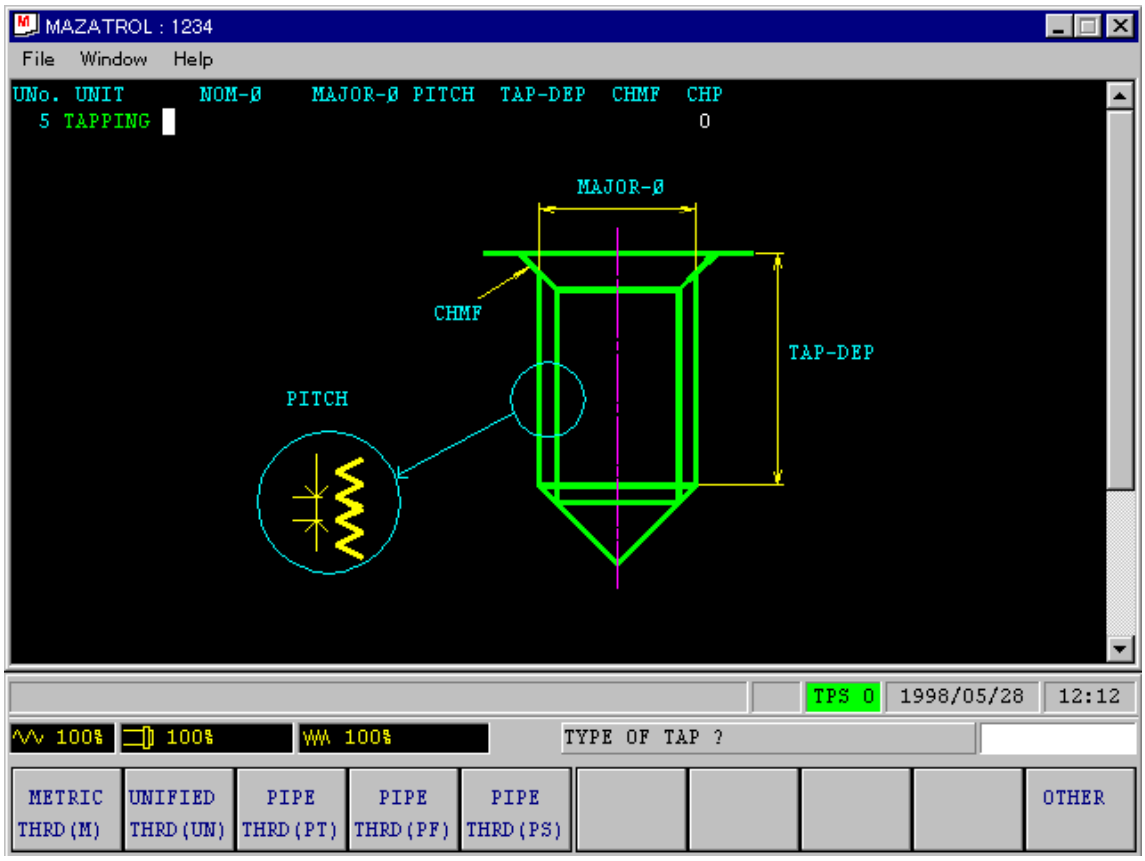
NM210-00520

1. Creation of point machining unit (2)

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

(2) **MACHINING UNIT <MENU>?**..... **TAPPING**

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



D735P0040E

Now, start creating the point machining unit. The data for the article NOM-ϕ, TAP-DEP will be specified here.

(4) **TYPE OF TAP?** ..... **METRIC THRD (M)**

(5) **ACTUAL DIAMETER OF TAP?** ..... 8 [→]

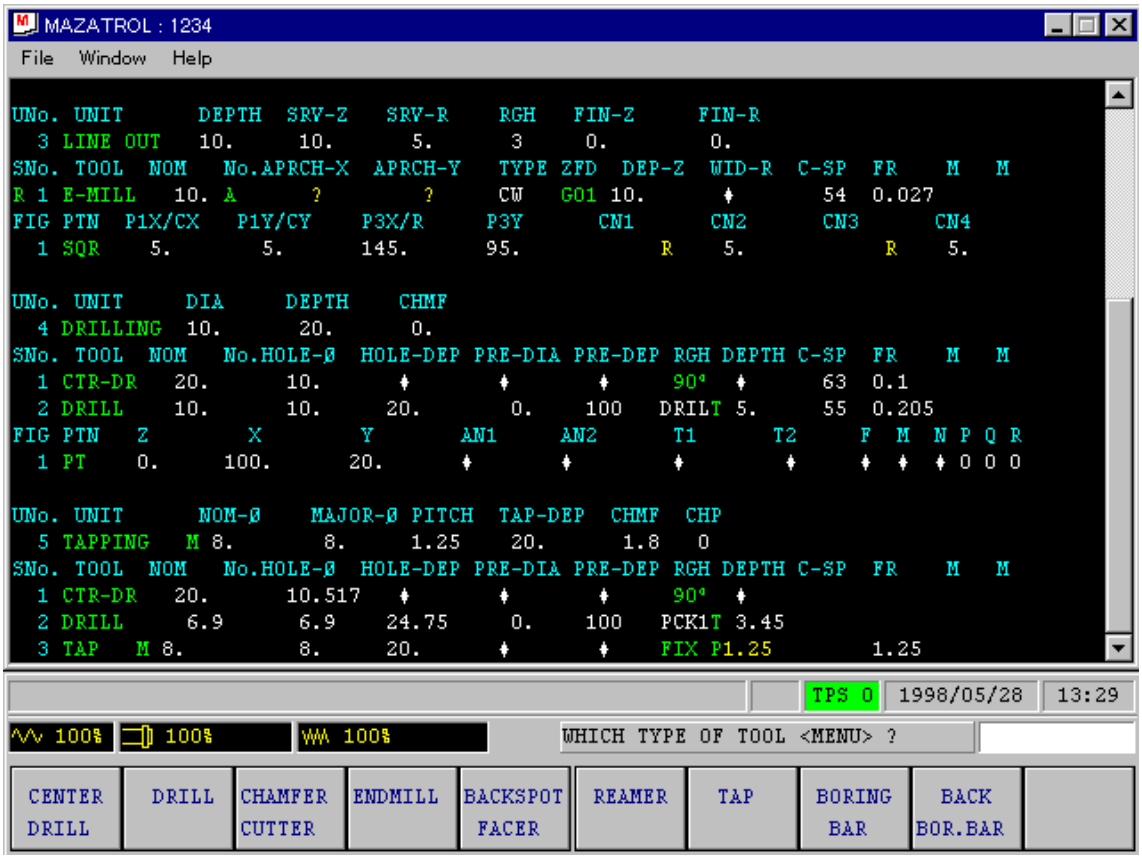
(Setting of the nominal diameter causes automatic setting of the outer diameter, the pitch and the amount of chamfering.)

(6) **TAPPING DEPTH?** ..... 2 0 [→]

(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.



D735P0041E

2. Creation of tool sequence

(1) Press the cursor key ➔ six times in succession to move the cursor to the position under the article C-SP.

(2) **CUTTING SPEED, AUTO → <MENU>?** ..... **HSS AUTO**

(3) **FEEDRATE, AUTO → <MENU>?** ..... **HSS AUTO**

(4) **M CODE?** ..... ➔

(5) **M CODE?** ..... ➔

(6) Press the cursor key ➔ eight times in succession to move the cursor to the position under the article 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 ➔ eight times in succession to move the cursor to the position under the article C-SP.

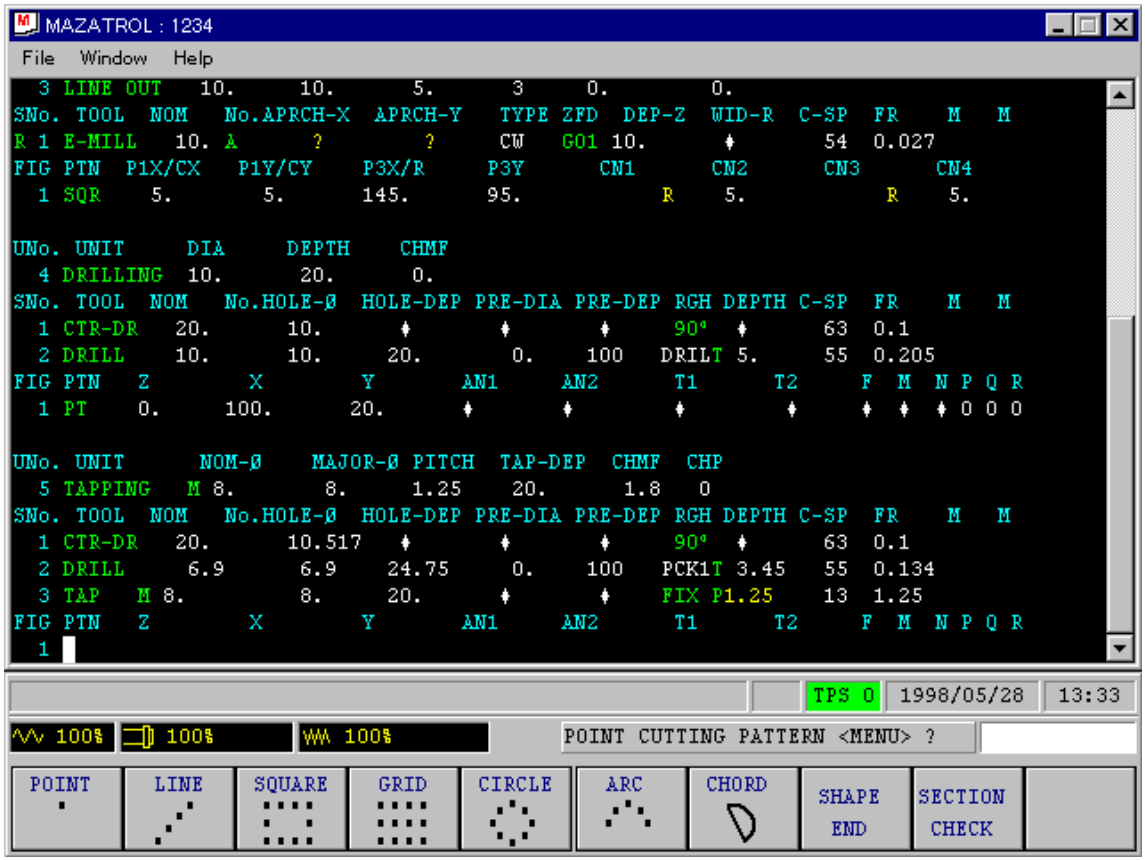
(12) **CUTTING SPEED, AUTO → <MENU>?** ..... **HSS AUTO**

(13) **FEEDRATE, AUTO → <MENU>?** ..... **HSS AUTO**

(14) **M CODE?** ..... ➔

(15) **M CODE?** ..... ➔

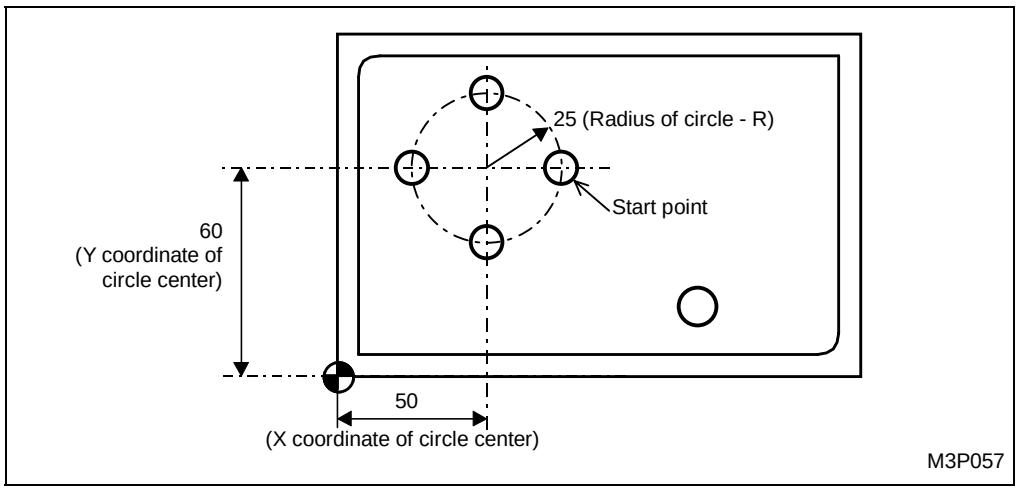
A tool sequence such as that shown below is now completed.



D735P0042E

3. Creation of shape sequence

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



M3P057

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

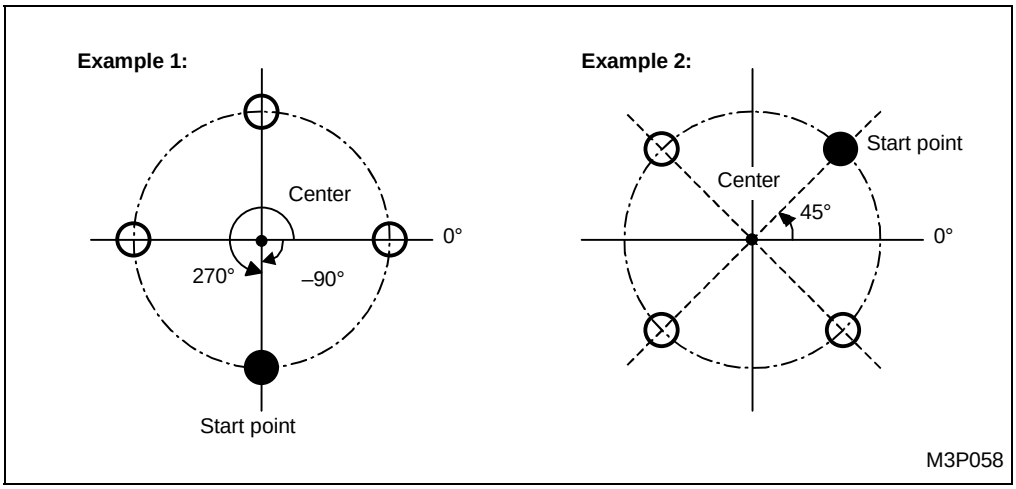
(2) **Z VALUE OF WORK SURFACE?** ..... 0 ➔

(3) **CIRCLE CENTER X?** ..... 5 0 ➔

(4) **CIRCLE CENTER Y?** ..... 6 0 →

(5) **ANGLE OF START PT FROM X AXIS?** ..... 0 →

(Input the angle of the line connecting the center of the circle and the first hole to be drilled with respect to the X axis.)



(6) **CIRCLE RADIUS R?** ..... 2 5 →

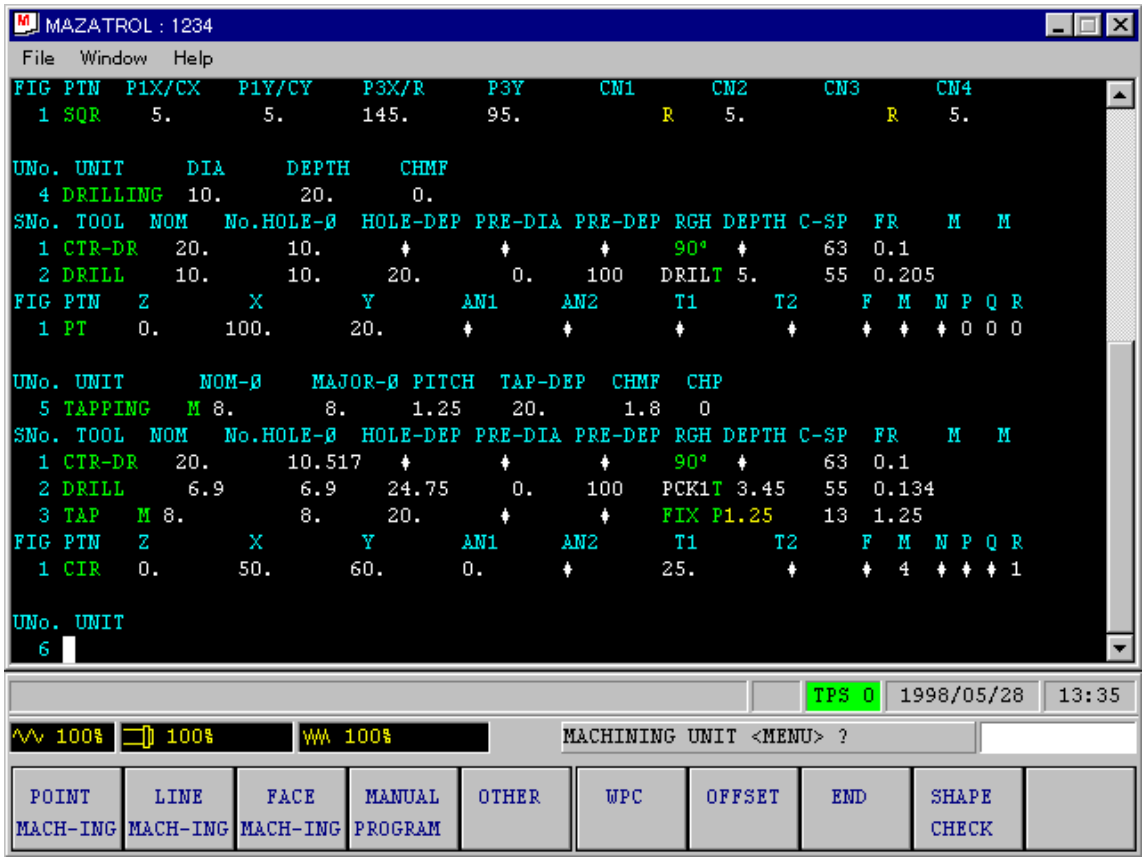
(7) **NUMBER OF HOLES?** ..... 4 →

(8) **RETURN POSITION <INIT:0, R:1>?** ..... 1 →

(For Return position see Fig. 4-13.)

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

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



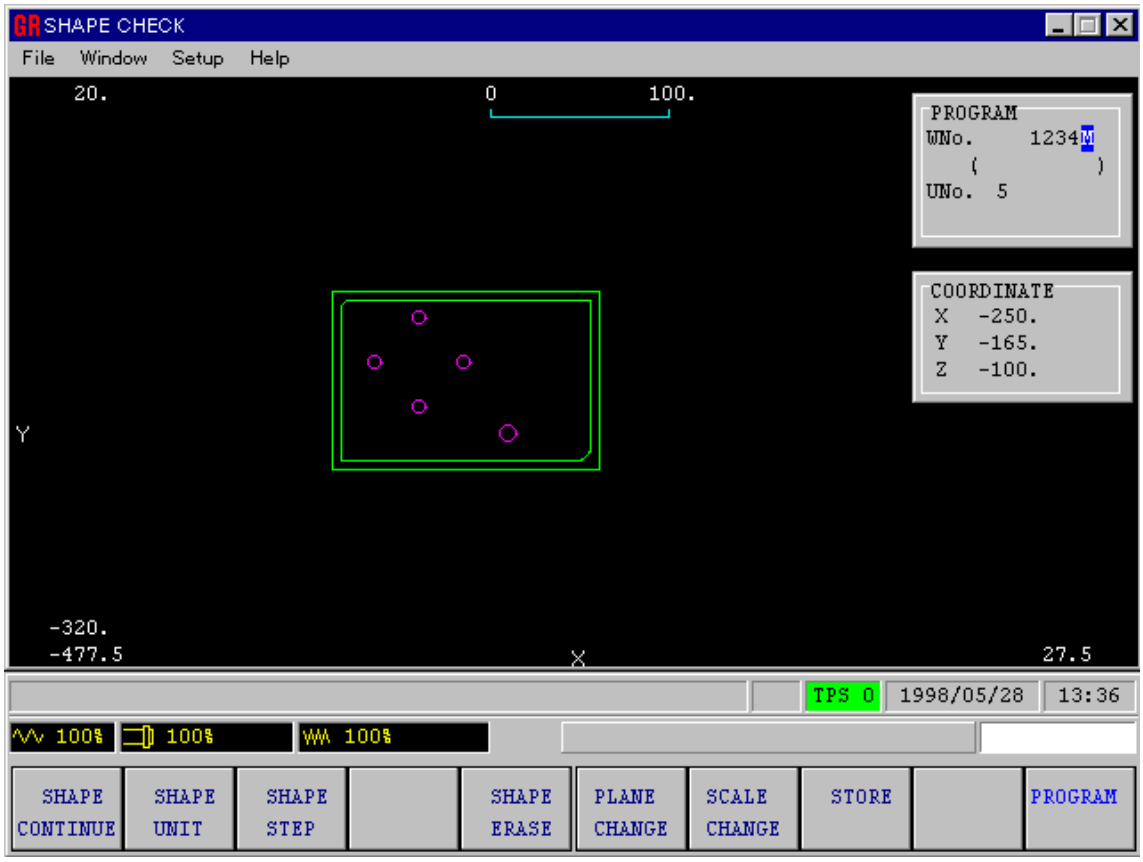
D735P0043E

4. Shape check

Let us call shapes on the **SHAPE CHECK** display.

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

(2) Press the menu key **SHAPE CONTINUE.**



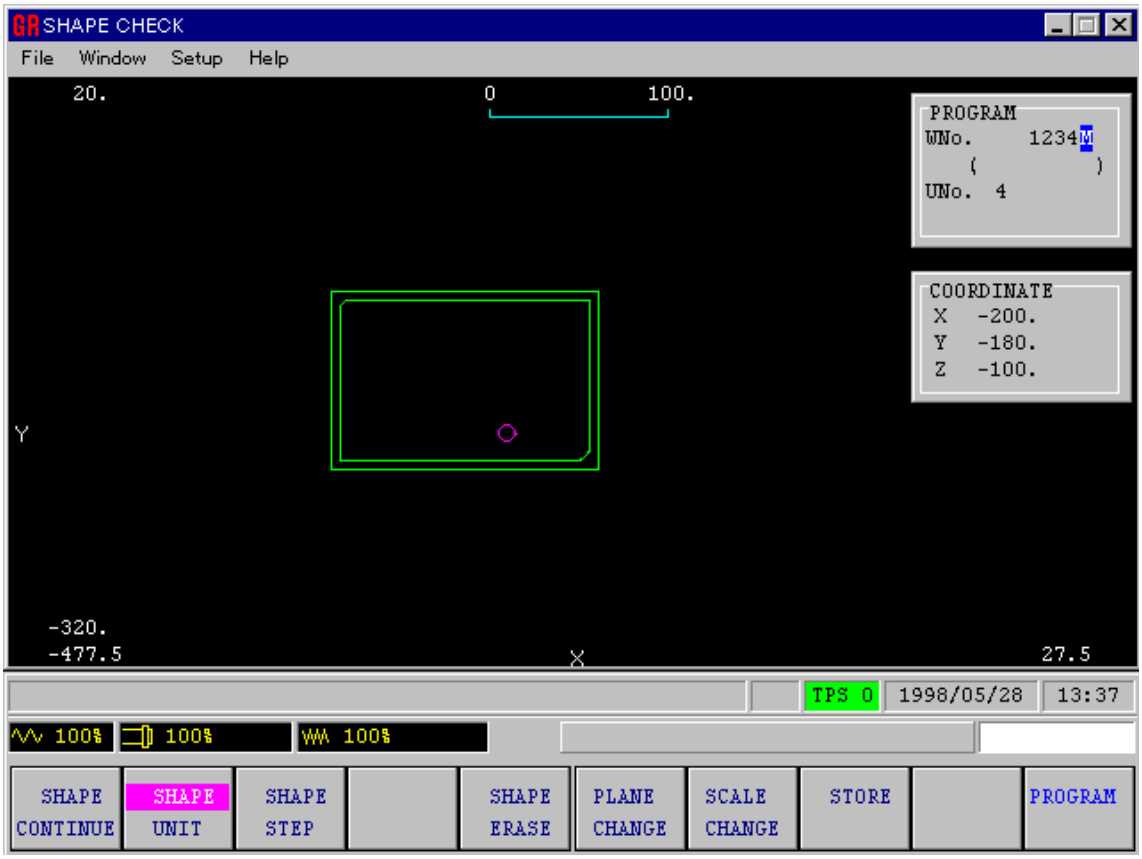
D735P0044E

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

(4) Press the menu key **SHAPE UNIT** several times to display shapes on a unit basis.

The reverse-display status of **SHAPE UNIT** is cleared when the final shape has been displayed.

The following display shows the unit number 4 shape of point machining unit (1).



D735P0045E

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

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

4-4-11 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**.

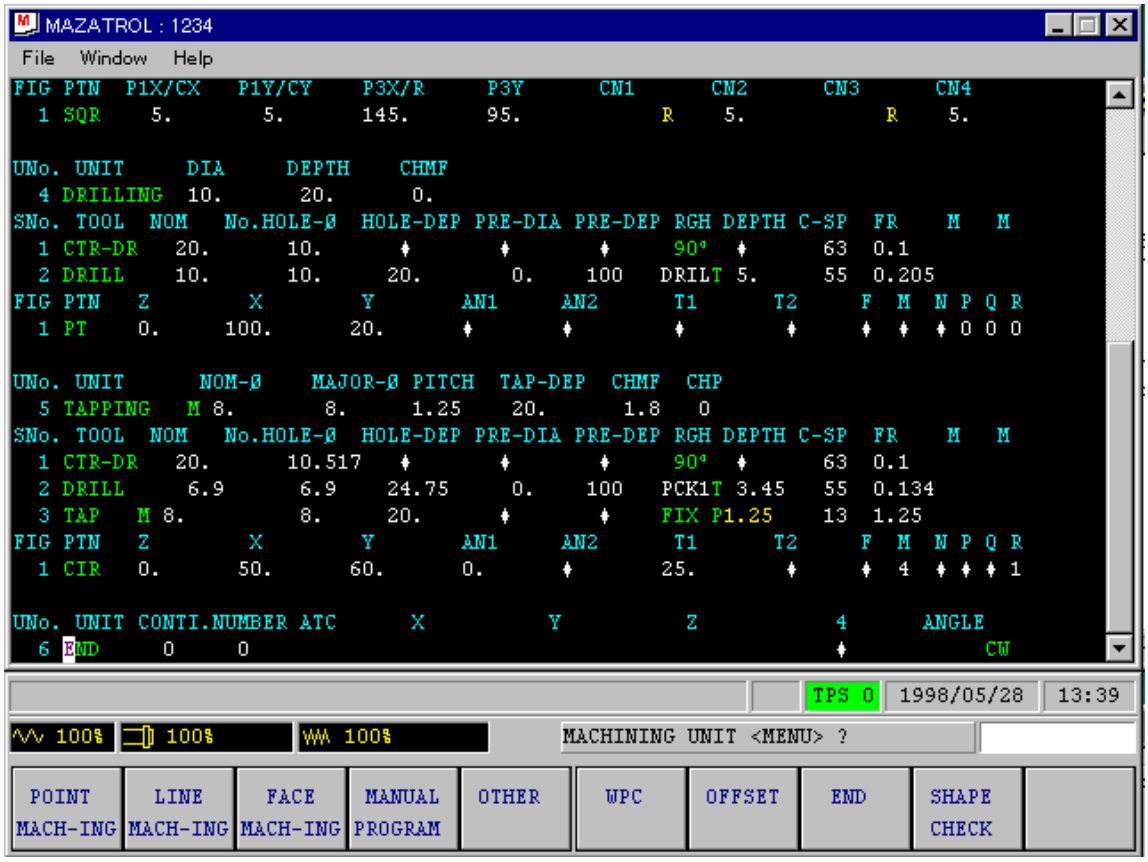
(2) **CONTINUE <Y:1, N:0>?** ..... 0 ⏎

(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>?** ..... 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.



D735P0046E

4-4-12 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.

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

```
UNo.    MAT      INITIAL-Z    ATC MODE    MULTI MODE      MULTI FLAG    PITCH-X  PITCH-Y
  0     S45C        20.           1           OFF              ◆           ◆        ◆
UNo.    UNIT     ADD. WPC      X        Y        th        Z        4
  1     WPC-1                -300.    -200.      0.      -100.      0.
UNo.    UNIT     DEPTH   SRV-Z          BTM   WAL    FIN-Z    FIN-R
  2    FACE MIL    0.      3.             2     ◆      0.       ◆
SNo.   TOOL    NOM-ϕ  No.  APRCH-X   APRCH-Y   TYPE   ZFD   DEP-Z  WID-R  C-SP   FR    M  M
 R1    F-MILL   80.A          ?         ?       XBI     ◆     3.     56.   121  1.079
 FIG    PTN     P1X/CX    P1Y/CY     P3X/R     P3Y     CN1   CN2   CN3   CN4
  1     SQR       0.        0.        150.     100.
UNo.    UNIT     DEPTH   SRV-Z   SRV-R   RGH   CHMF   FIN-Z    FIN-R
  3    LINE OUT    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    E-MILL   10.A          ?         ?       CW     G01   10.     ◆     54   0.027
 FIG    PTN     P1X/CX    P1Y/CY     P3X/R     P3Y     CN1   CN2   CN3   CN4
  1     SQR       5.        5.        145.     95.             R5.          R5.
UNo.    UNIT      DIA      DEPTH      CHMF
  4   DRILLING    10.       20.        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°     ◆     63   0.1
  2   DRILL      10.          10.       20.      0.      100    DRIL   T5.    55   0.205
 FIG    PTN     Z     X     Y    AN1  AN2   T1    T2    F     M     N     P     Q     R
  1      PT     0.  100.  20.    ◆    ◆     ◆     ◆     ◆     ◆     ◆     0     0     0
UNo.    UNIT    NOM-ϕ   MAJOR-ϕ   PITCH    TAP-DEP    CHMF      CHP
  5   TAPPING    M8.       8.       1.25       20.      1.8       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.       10.517      ◆        ◆       ◆     90°     ◆     63   0.1
  2    DRILL       6.9      6.9        24.75      0.     100.    PCK1   T3.45   55   0.134
  3     TAP       M8.       8.         20.        ◆       ◆      FIX    P1.25   13   1.25
FIG    PTN     Z     X     Y    AN1  AN2   T1    T2    F     M     N     P     Q     R
  1     CIR     0.   50.   60.   0.    ◆    25.    ◆     ◆     4     ◆     ◆     ◆     1
UNo.    UNIT    CONTI.    NUMBER    ATC    X     Y     Z     4     ANGLE
  6     END        0         0                                ◆      CW
```

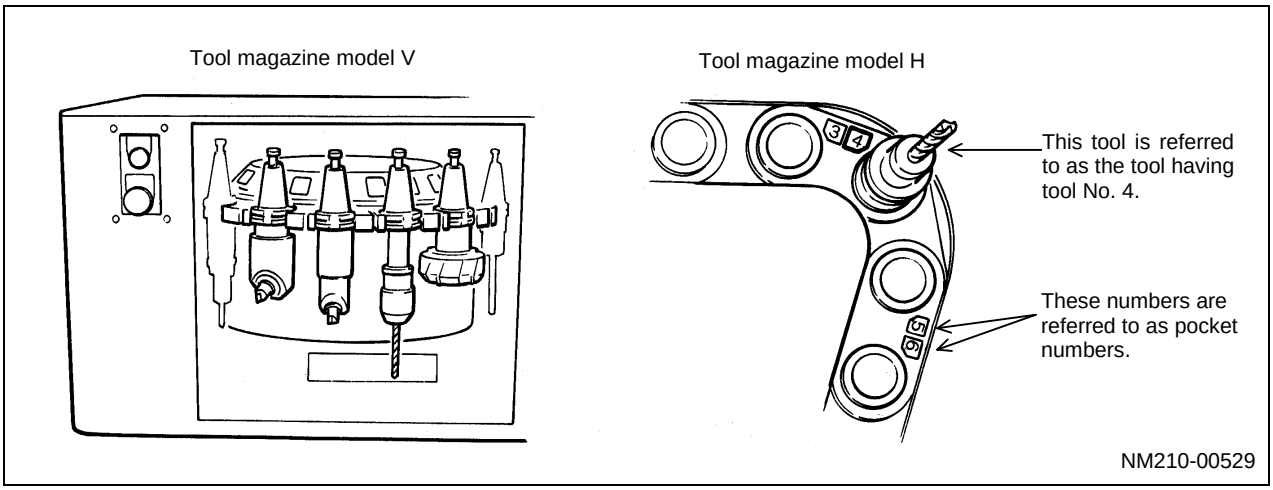
Fig. 4-14 Sample program

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 input. 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 selelctor key.

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

➔ The **TOOL DATA** display will then be presented 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.

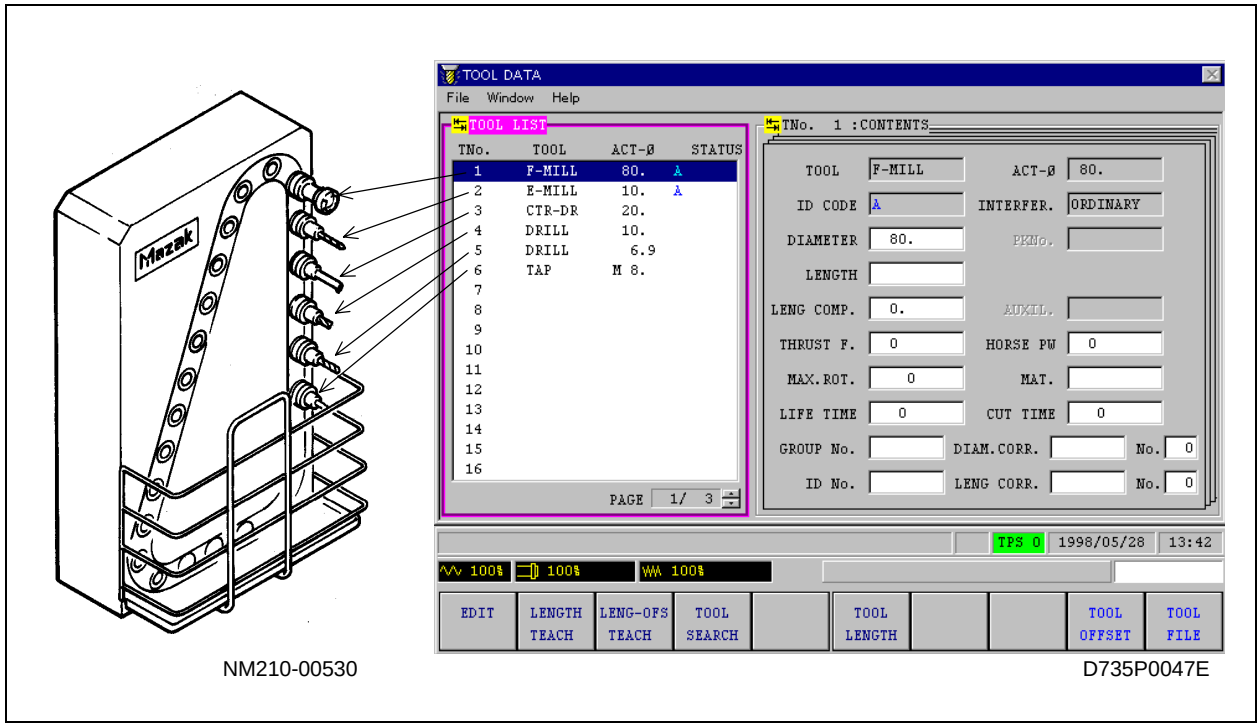


Fig. 4-15 Correspondence to the pocket numbers

- NOTE -

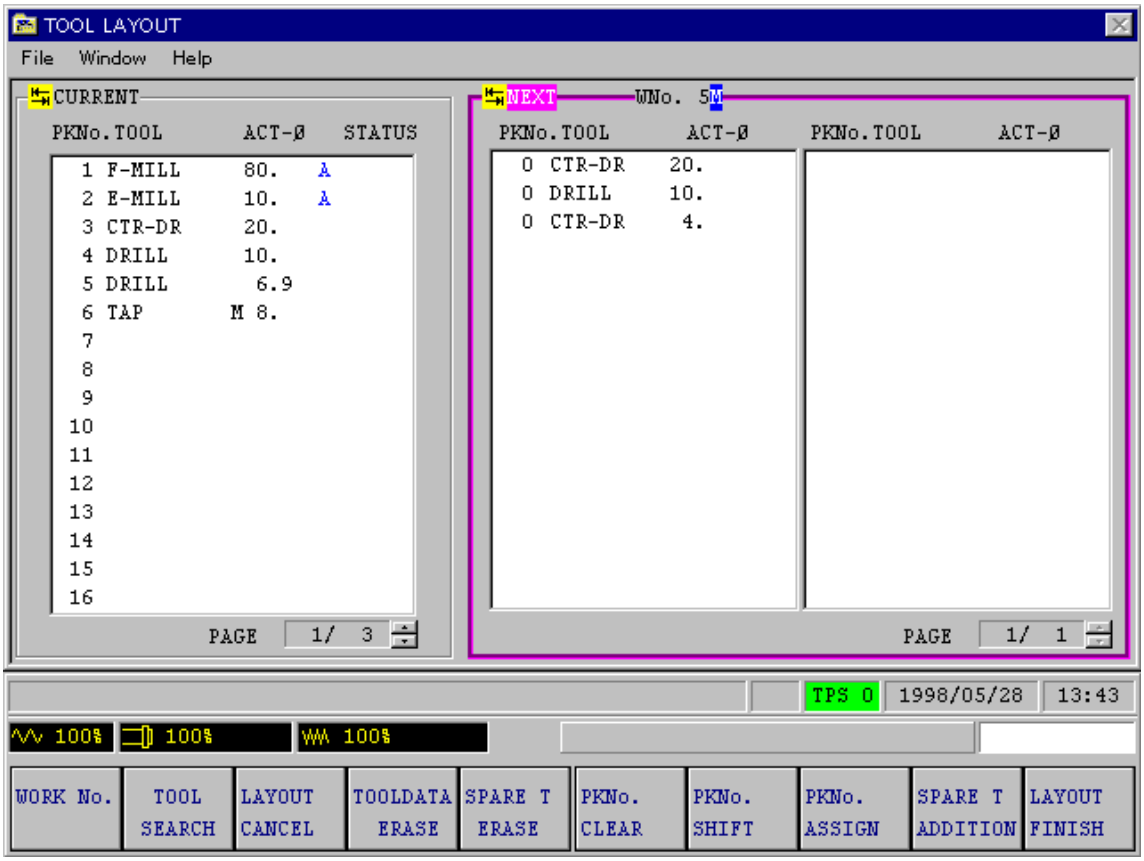
4-5-3 TOOL LAYOUT display

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 presented.



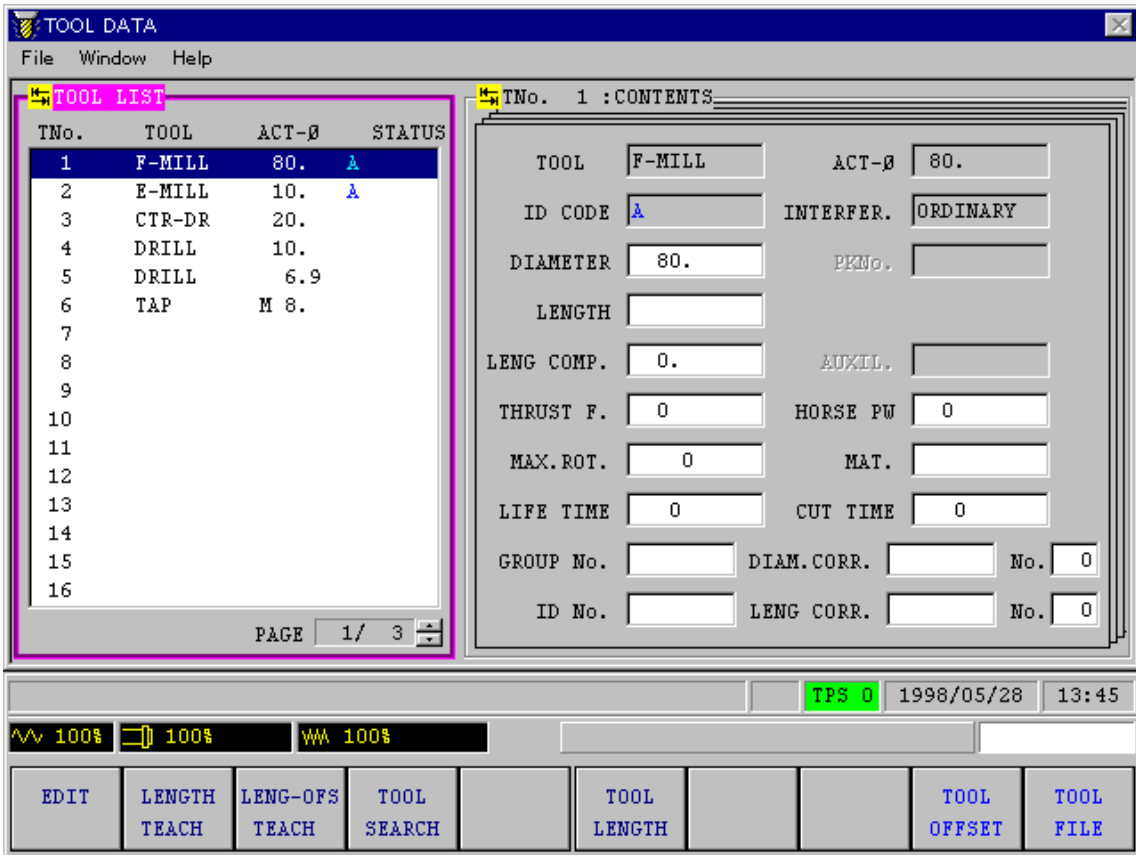
D735P0048E

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).

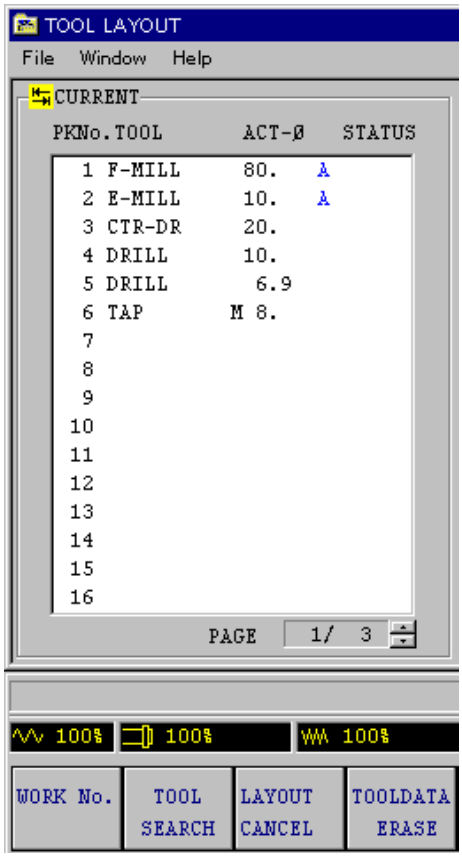
The following figure shows the relationship between the **PROGRAM** display, **TOOL DATA** display and **TOOL LAYOUT** display:



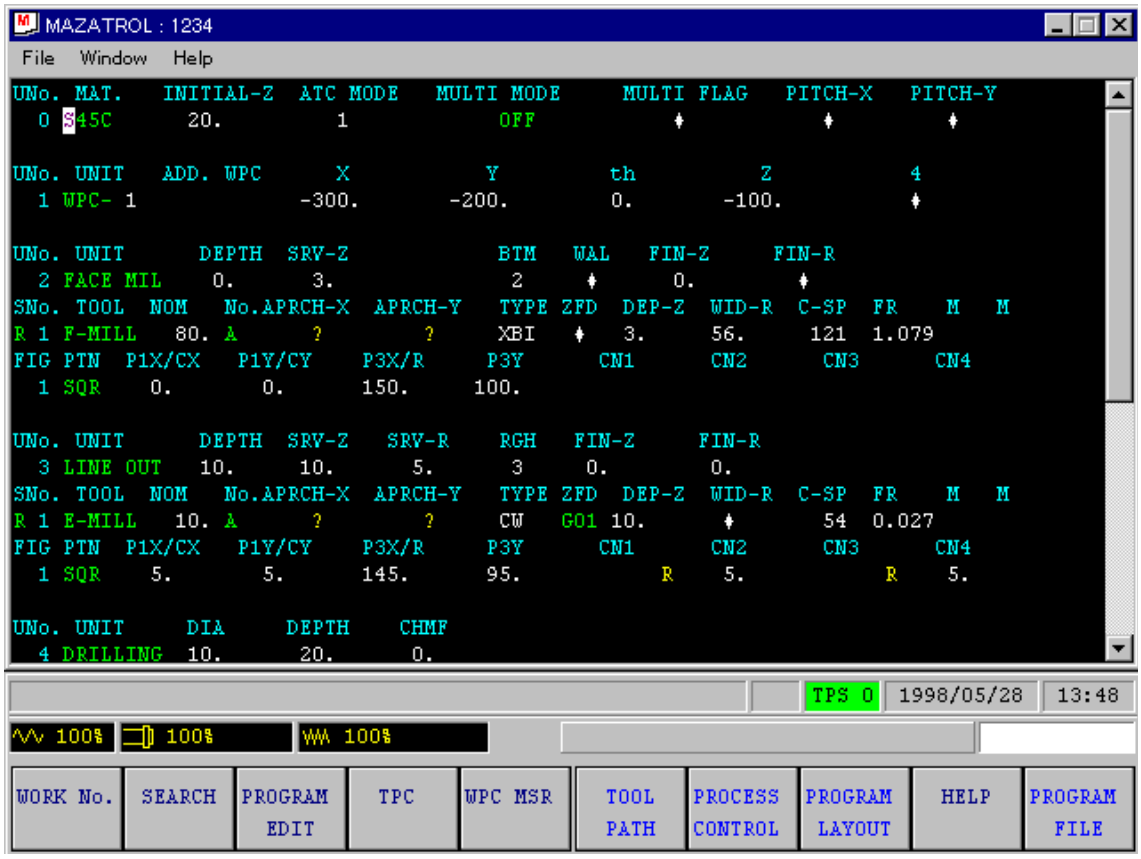
D735P0049E

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.

3 3



D735P0050LE



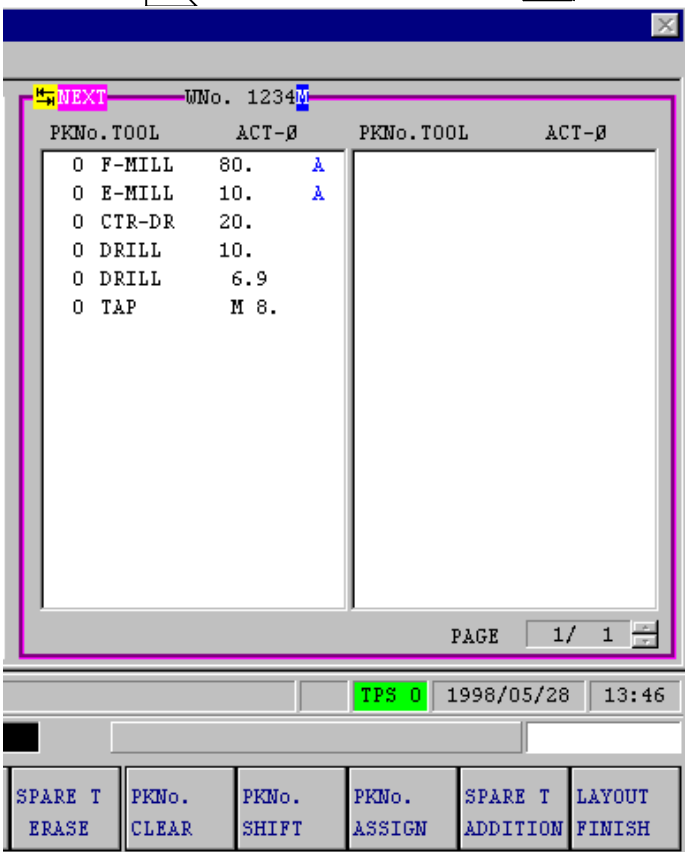
Registration of new tools

D735P0051E

2

1

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



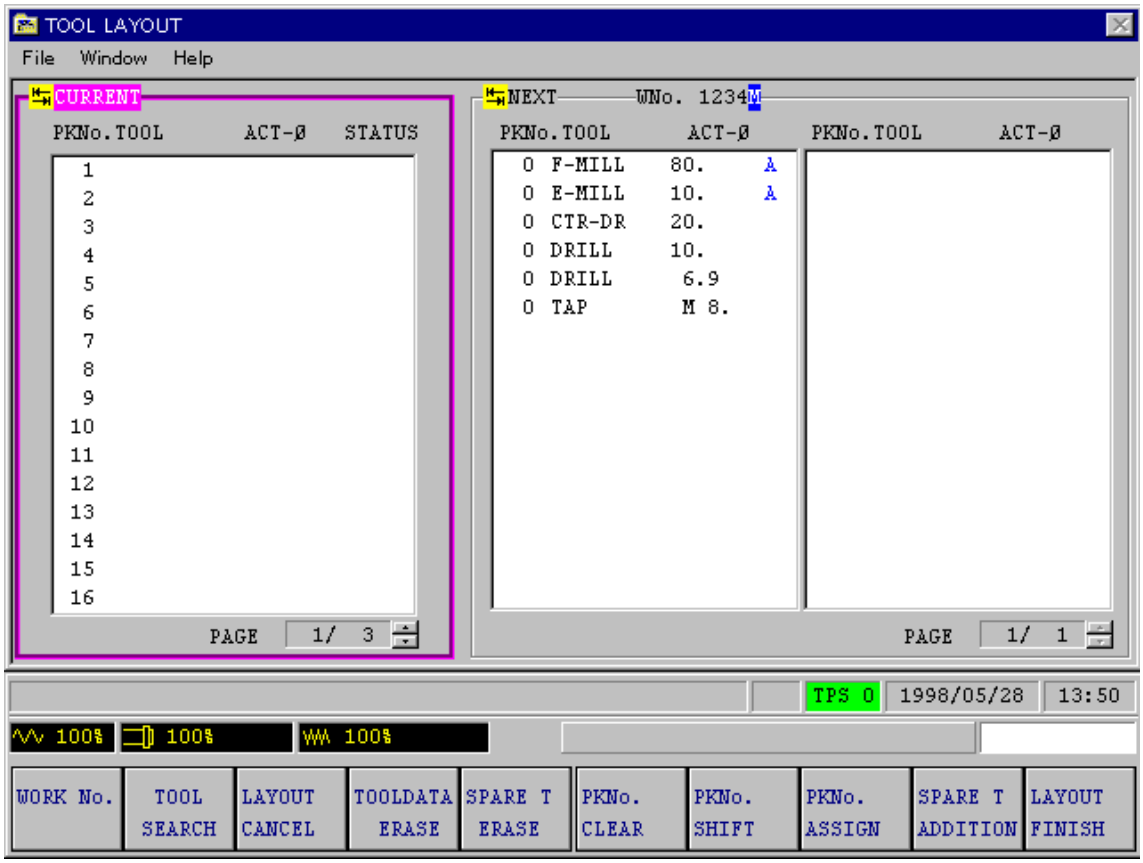
D835P0050RE

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. Input 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 ⇨



D735P0052E

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.

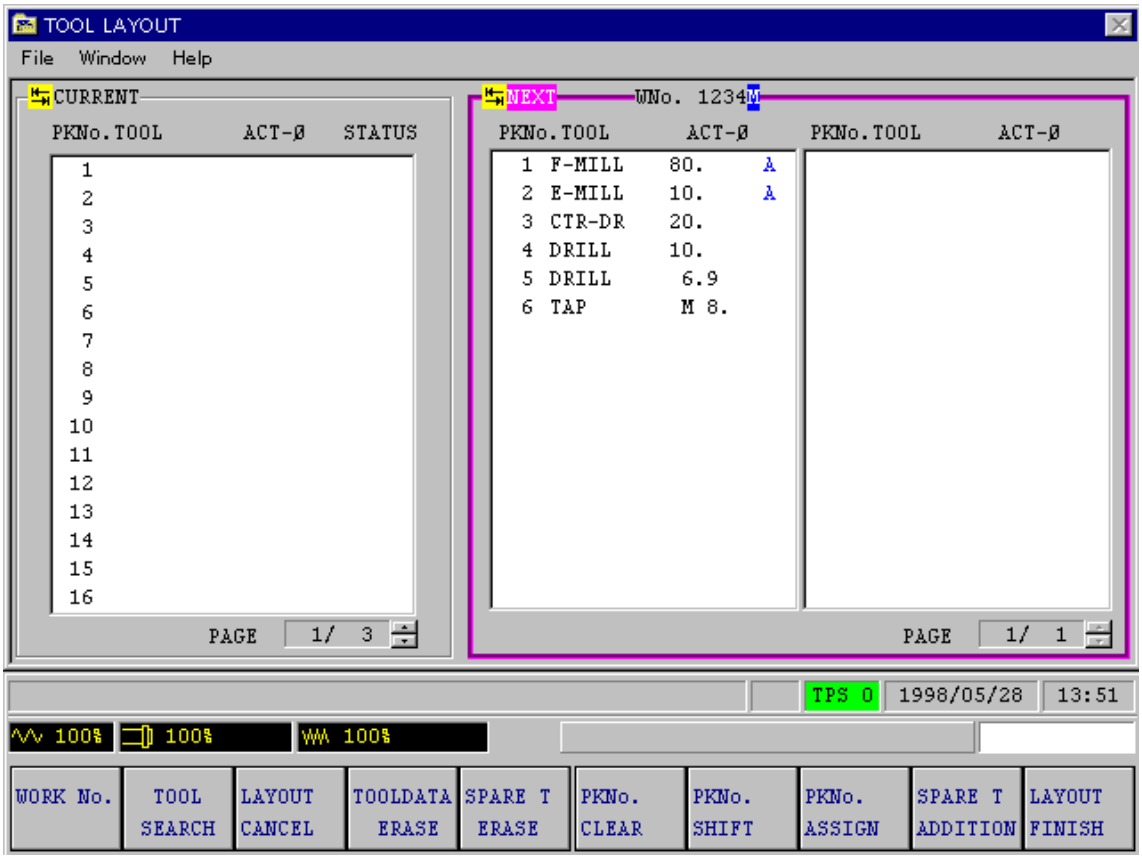
Three modes are available for assigning pocket numbers: automatic, manual, and shift. Automatic assignment, the simplest of these three modes, is described below.

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.



D735P0053E

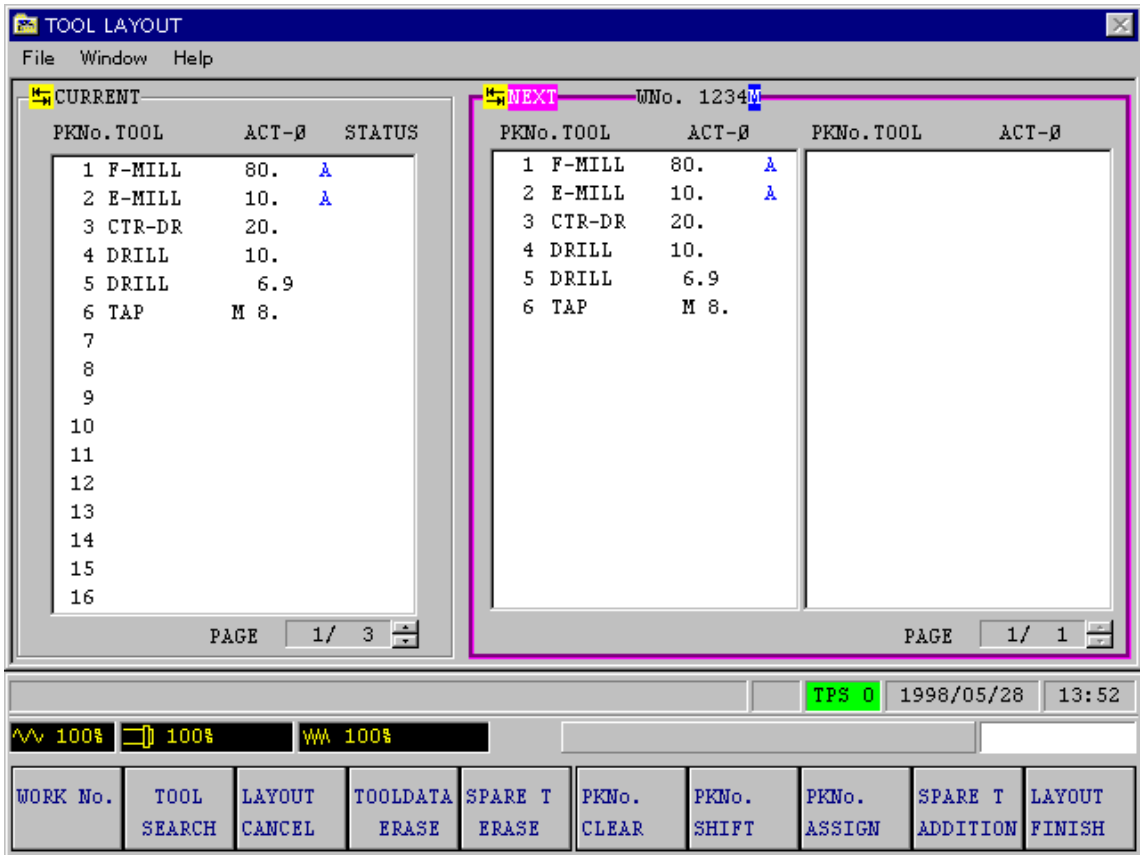
Next, carry out the following key operations to register the tools on the NEXT display onto the CURRENT display.

(4) **LAYOUT FINISH <MENU>?** ..... **LAYOUT FINISH**

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

(6) **LAYOUT FINISH**

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



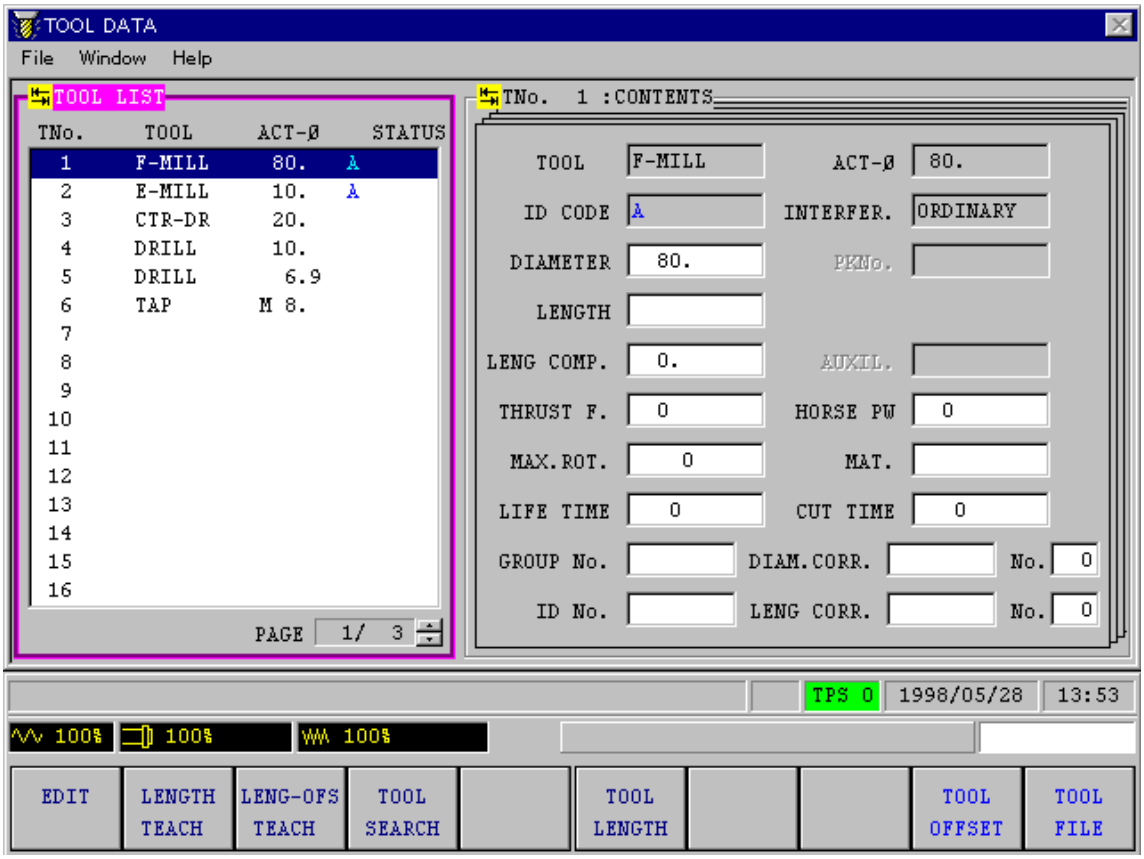
D735P0054E

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.



D735P0055E

4-6 Check of the Machining Path

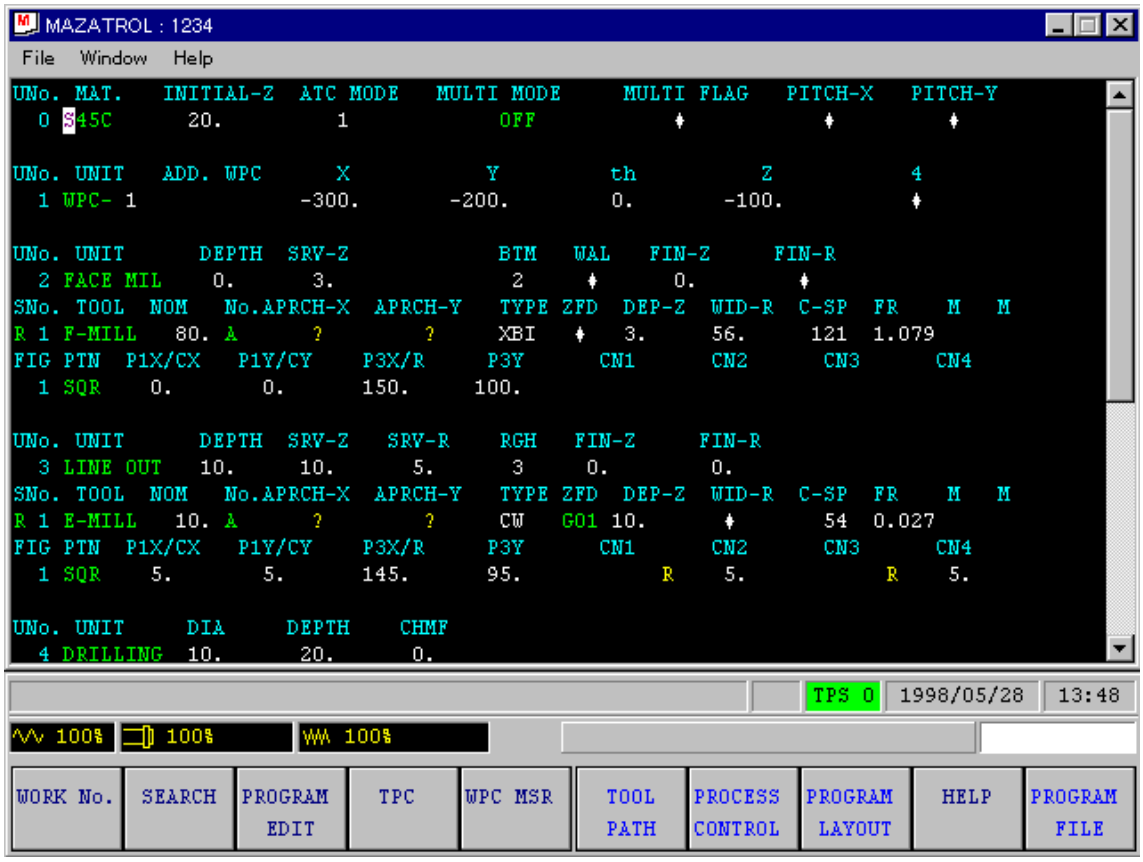
Let us return the **PROGRAM** display.

(1) Press the display selector key.

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

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

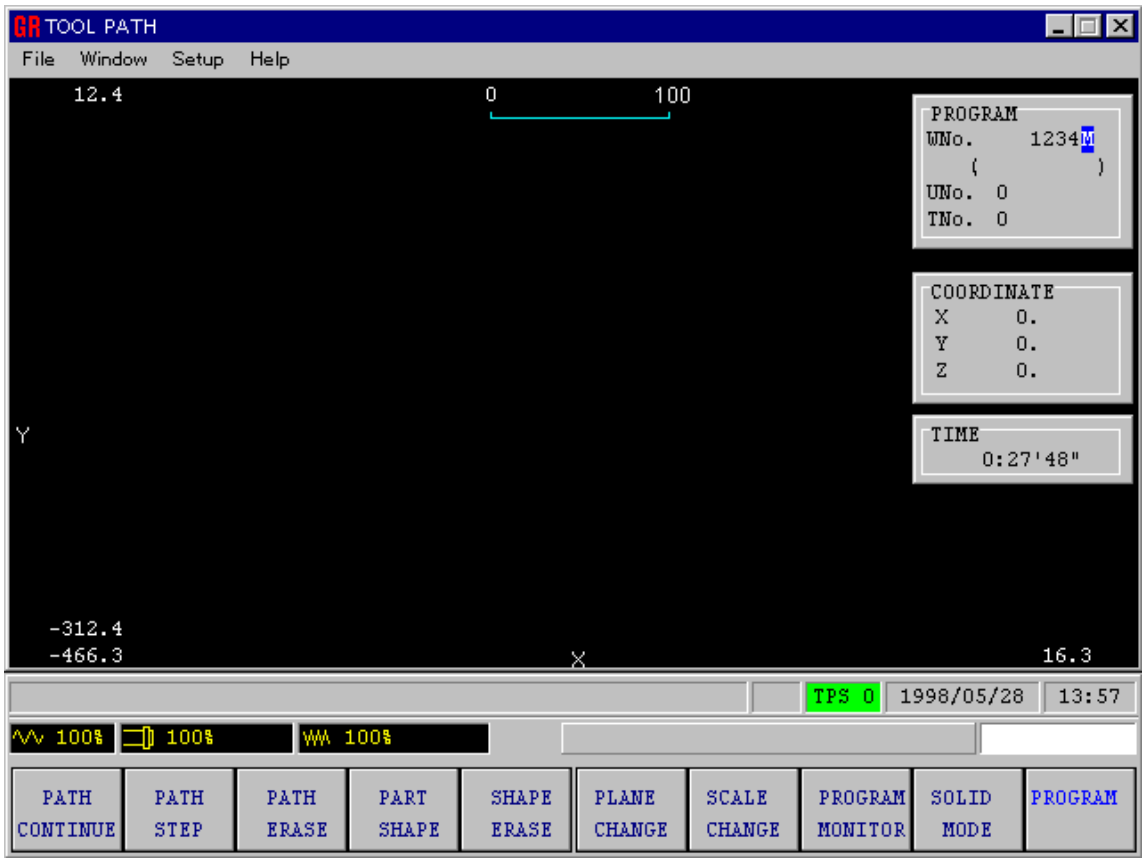
Next, let us call the **TOOL PATH** display on the screen and check the machining path on that display. See Section 6-2, “TOOL PATH CHECK Display” of the Operating Manual for details.



D735P0051E

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

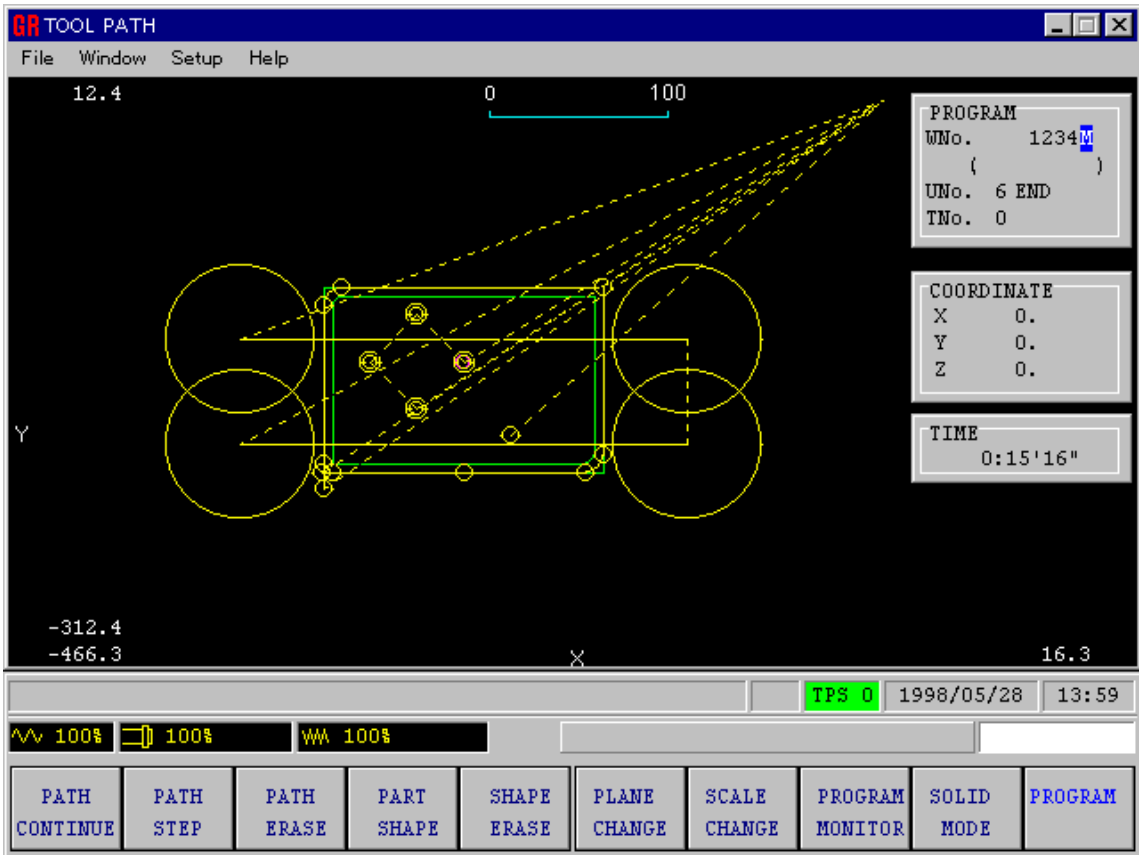
➔ The following **TOOL PATH** display will be presented.



D7350056E

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

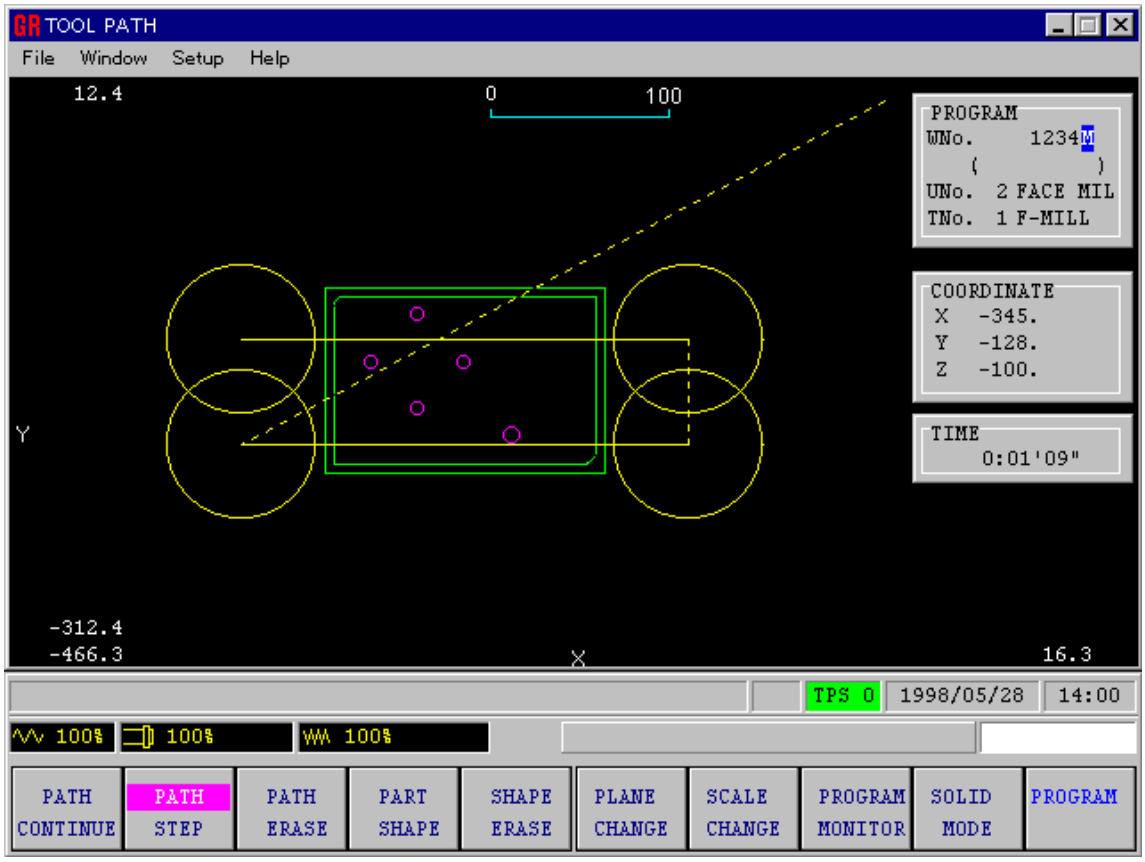
(5) Next, press the menu key **PATH CONTINUE** to display the machining path of the tool through to the end.



D735P0057E

(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.



D735P0058E

(8) To check the machining path in further details, use the PLANE CHANGE function (see Subsection 4-4-8) or the SCALE CHANGE function (see Subsection 4-4-9).

(9) When the all operation has been completed, press the display selector key and the menu key **POSITION** to resume the **POSITION** display.

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 2-12

2-2-5 Copy 2-15

2-2-6 End of program 2-21

3 WINDOW FUNCTIONS 3-1

3-1 MAZATROL Help Window 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 Types of Units 4-1

4-2 Common Unit 4-1

4-3 Basic Coordinates System Unit 4-6

4-4 Auxiliary Coordinates System Unit 4-6

4-5 Types of the Machining Unit 4-7

4-6 Point Machining Unit 4-7

4-6-1 Types of point machining units 4-8

4-6-2 Procedure for selecting point machining unit 4-8

4-6-3 Unit data and automatic tool development 4-9

4-6-4 Automatic tool development for cemented carbide drill 4-36

4-6-5 New tapping auto-setting scheme 4-37

4-6-6 Tool sequence data 4-42

4-6-7 Tool path 4-47

4-6-8 Shape sequence 4-87

4-7 Line Machining Units 4-104

4-7-1 Types of line machining units 4-104

4-7-2 Procedure for selecting line machining unit 4-105

4-7-3 Unit data, automatic tool development and tool path 4-105

4-7-4 Line machining tool sequence data 4-143

4-7-5 Line machining shape sequence data 4-146

4-7-6 Precautions in line machining 4-146

4-7-7 Automatic corner override 4-150

4-8 Face Machining Units 4-151

4-8-1 Types of face machining units 4-152
4-8-2 Procedure for selecting face machining unit 4-152
4-8-3 Unit data, automatic tool development and tool path 4-153
4-8-4 Face machining tool sequence data 4-199
4-8-5 Precautions in face machining 4-204
4-8-6 Override in case of the overall width cutting 4-210
4-8-7 Definitions of forms in line machining and face machining units 4-212
4-9 End Unit 4-233
4-10 Special Mode Unit 4-234
4-10-1 Procedure for calling up the special mode unit 4-234
4-10-2 M code unit 4-235
4-10-3 Sub-program unit 4-236
4-10-4 Basic coordinate shift unit (option) 4-237
4-10-5 Pallet changing unit 4-242
4-10-6 Indexing unit 4-242
4-10-7 Process end unit 4-244
4-11 Manual Program Mode Unit 4-246
4-11-1 Input procedure 4-246
4-11-2 Structure of the unit 4-246
4-11-3 Composition of the sequence 4-247
4-12 MMS Unit 4-248
4-12-1 Procedure for calling up the MMS unit 4-248
4-12-2 Composition of the unit 4-248
4-12-3 Composition of the MMS sequence 4-249

4-12-4 Measurement of length of feeler 4-250

4-12-5 Feeler calibration measurement 4-251

4-12-6 Type of measurement 4-253

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 Indexing 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 6-4

7 TPC DATA CREATION 7-1

7-1 Setting Tool Path Control (TPC) Data 7-1

7-2 Description of Each TPC Data Item 7-4

8 BACKGROUND PROGRAMMING 8-1

9 CASE OF APPEARANCE OF ALARM..... 9-1

10 THREE-DIGIT G-FORMAT 10-1

10-1 Outline 10-1

10-2 Detailed Description..... 10-1

10-3 Three-digit G-format of MAZATROL Program 10-2

10-4 Various Data Description Using G10 10-14

- 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
Selecting of program to be displayed	Program creation
Changeover to creating mode	Program editing (INSERT, ERASE, COPY)
Changeover to **TOOL PATH** display	Shape check function
Changeover to **PROGRAM FILE** display	Changeover to listing mode
Coordinate measuring function	
Checking of program during automatic operation	See the Notes below.
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 message **WORK No. (NAME SEARCH <?INP>)?** is displayed on the screen.

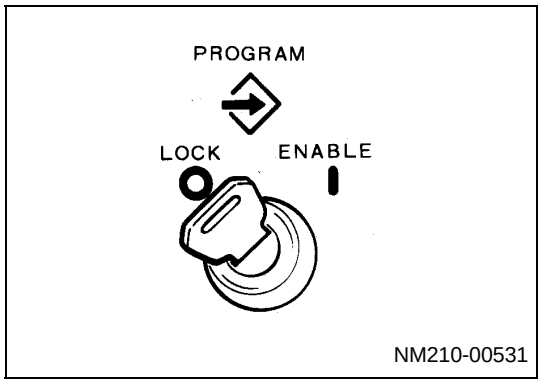
(2) Enter the workpiece program number by means of numeric keys.

Example: Workpiece No. 1000

Press the keys [1] [0] [0] [0] [⏎] 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 4-4-12, “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		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

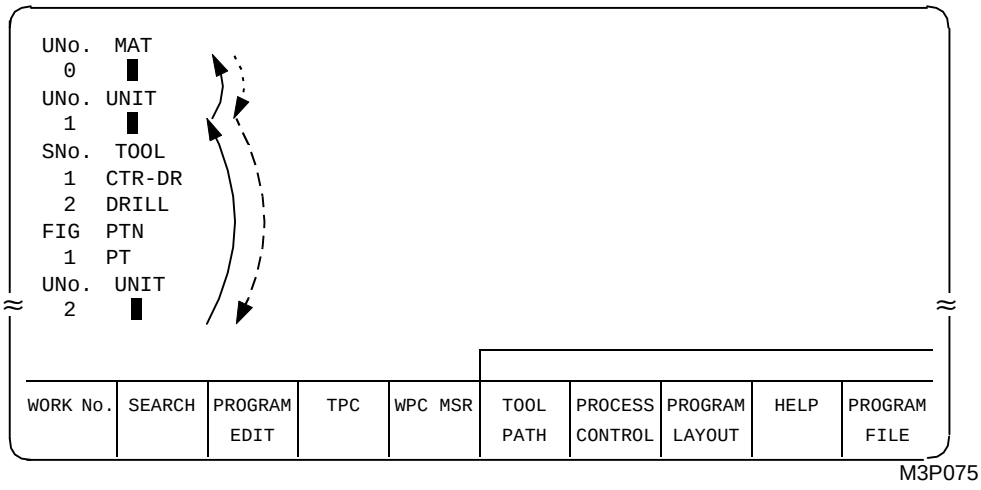
2-1 Cursor Movement

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 (⇓, ⇑)

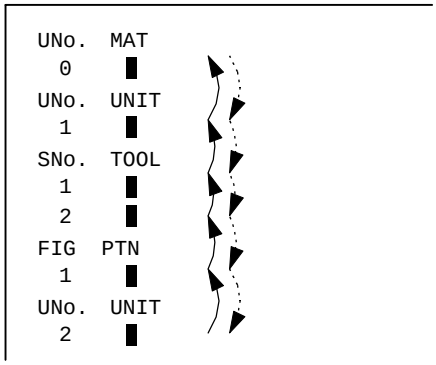
The cursor moves unit by unit in the direction of the arrows.



M3P075

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:

- Search .....Used to display the intended unit or tool sequence
- Insertion .....Used to insert a unit, a tool sequence or a shape sequence
- Deletion.....Used to erase the unit, tool sequence or shape sequence
- Copy .....Used to copy the program, unit or shape
- Window .....Unit data is displayed in the help window graphically. (See Chapter 3, “WINDOW FUNCTIONS.”)
- End of program .....Used to return from creating mode to listing mode

Note: Only search and window 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		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 workpiece number.

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		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: [1] [0] [⏎]

➔ The cursor moves to the number of the unit entered and the unit is displayed on the screen.

```
UNo.  UNIT
  10  ▌
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		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
       ▌
```

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/OTHER** → 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		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**.

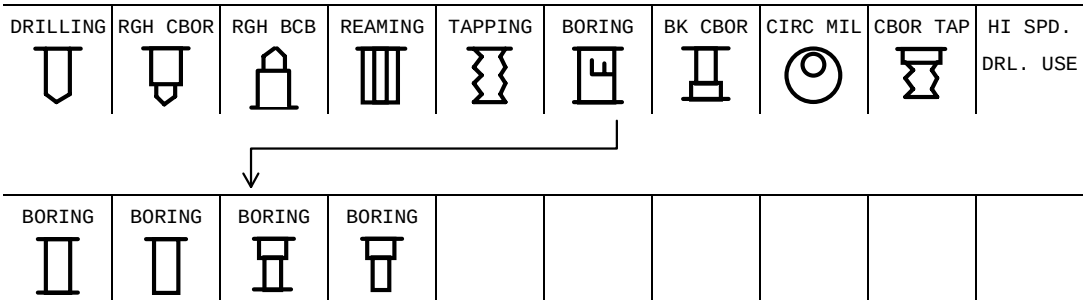
➔ The unit menu is displayed and the message **UNIT NAME SEARCH <INPUT>?** is indicated on the screen.

					UNIT NAME SEARCH <INPUT>?				
POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END		
(a)	(b)	(c)		(d)					

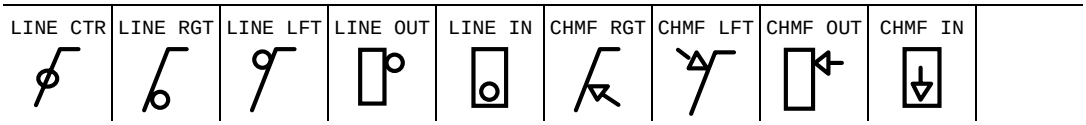
(4) Select the name of the unit to be found.

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

- Pressing of the menu key **BORING** results in the display of the boring unit menu.

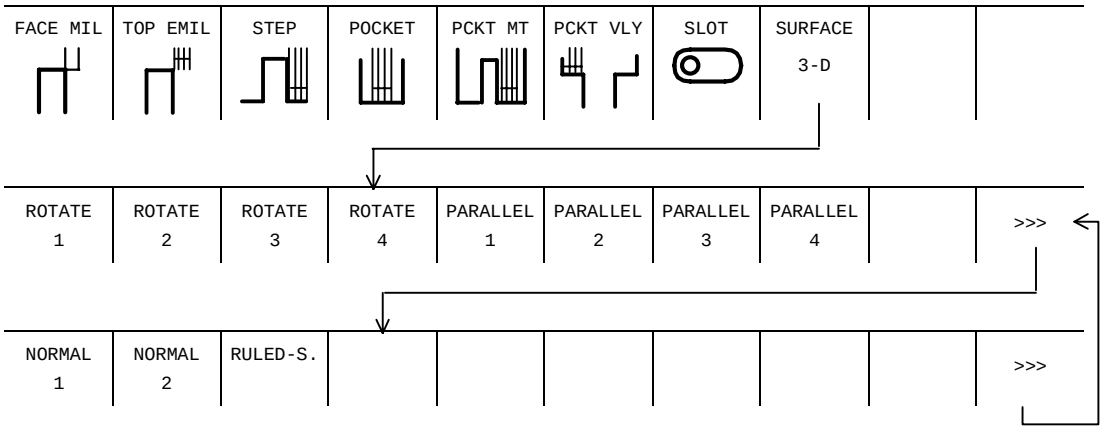


(b) Pressing of the menu key **LINE MACH-ING** results in the display of the linear machining unit menu.

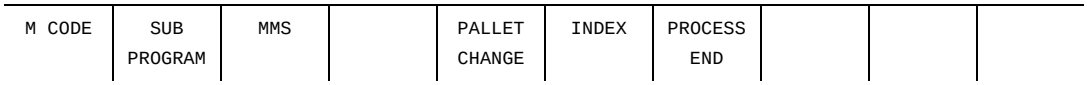


(c) Pressing of the menu key **FACE MACH-ING** results in the display of the face machining unit menu.

- Pressing of the menu key **SURFACE 3-D** results in the display of the three-dimensional surface machining unit menu.
- Pressing of the menu key **>>>** permits changing the three-dimensional surface machining unit menu.

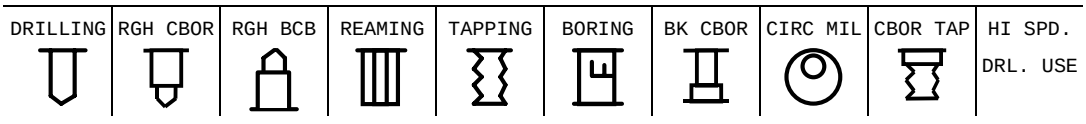


(d) Pressing of the menu key **OTHER** results in the display of the special unit menu.



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

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
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
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 function, TOOL SEARCH, 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		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 tool names.

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR.BAR	CHIP VACUUM	>>>	

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    CHF-C
```

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    E-MILL
```

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

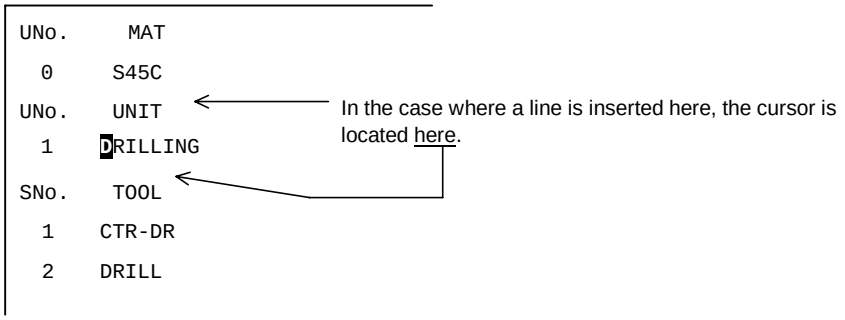
The line to be inserted (unit, tool) is determined depending on the cursor position.

1. Insertion of a unit

[Menu selection] **INSERT**

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

Example:



(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		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

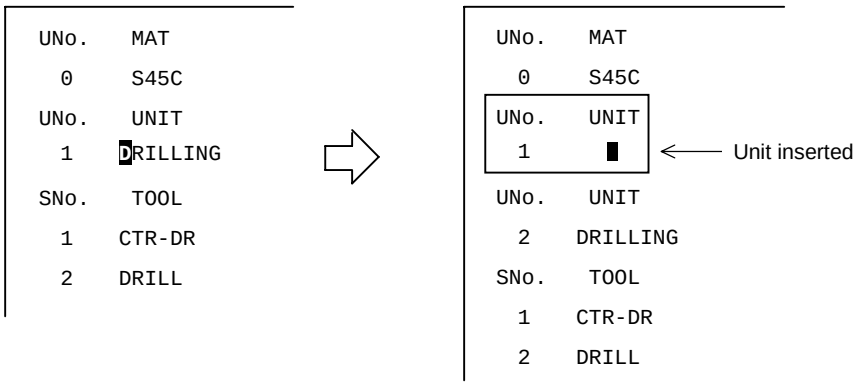
(3) Press the menu key **INSERT**.

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

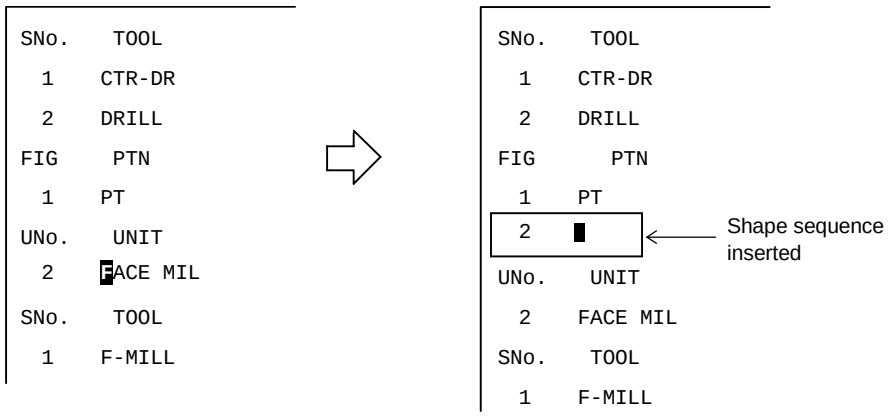
(4) Press the input key.

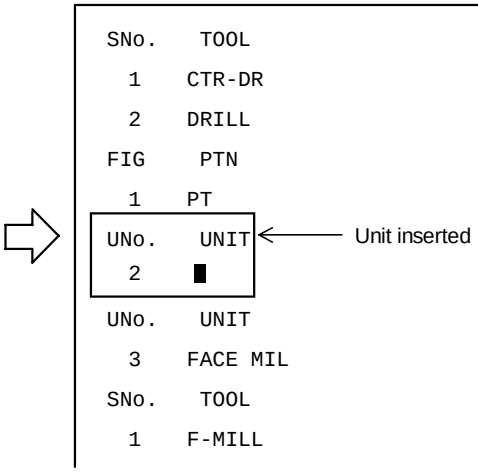
➔ This causes the insertion of the empty unit.

Example:



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.

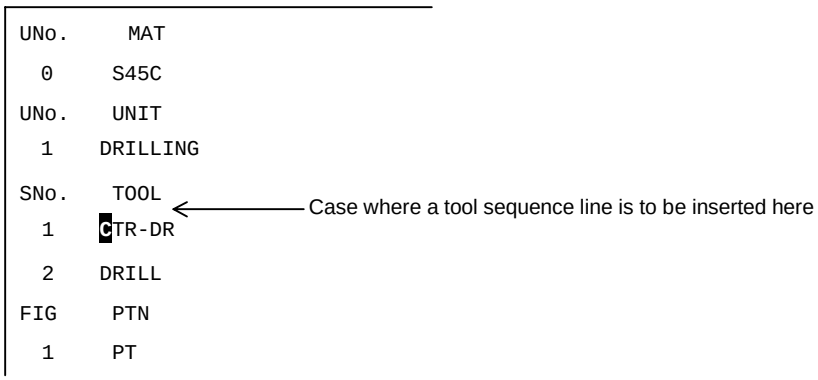
(5) Enter the data. Refer to the Chapter 4, “PROGRAM CREATION” for the selection of each unit and data setting.

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		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

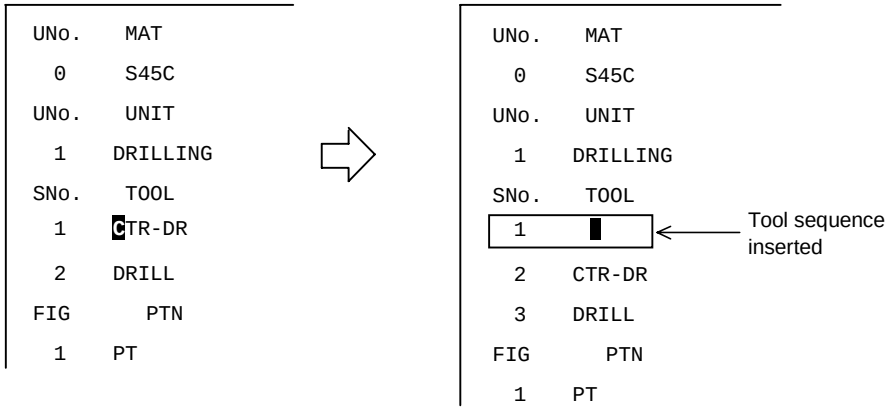
(3) Press the menu key **INSERT**.

➔ 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.

Example:



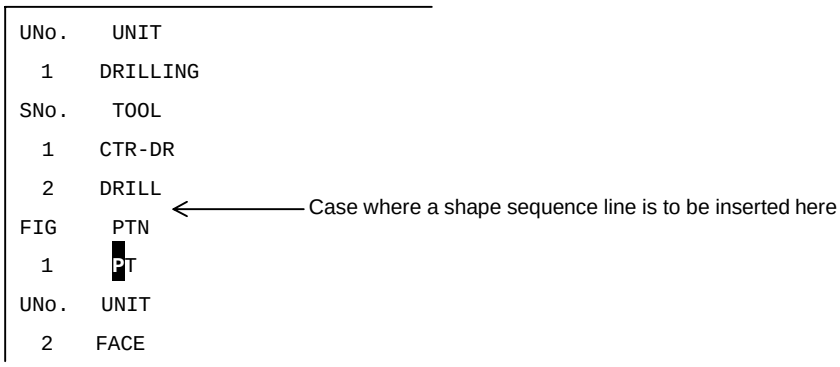
(5) Enter the data. Refer to the Chapter 4, “PROGRAM CREATION” for the selection of each unit and data setting.

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		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

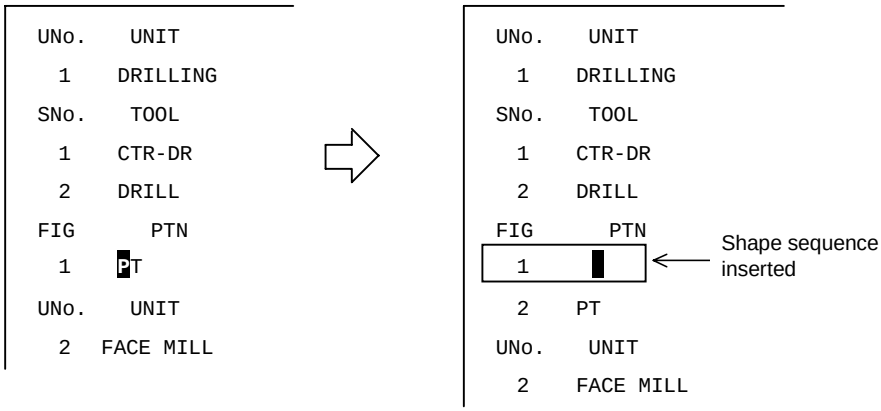
(3) Press the menu key **INSERT**.

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

(4) Press the input key.

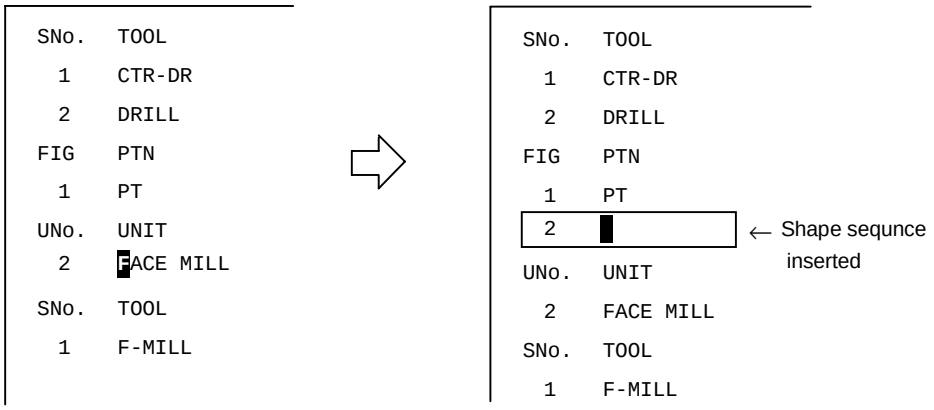
➔ This causes the insertion of the empty shape sequence line.

Example:



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.

Example:



(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:

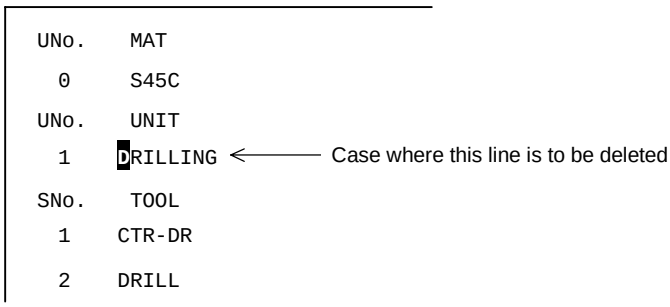
- Deletion of the unit
- Deletion of the tool sequence
- Deletion of the shape sequence

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.



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

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

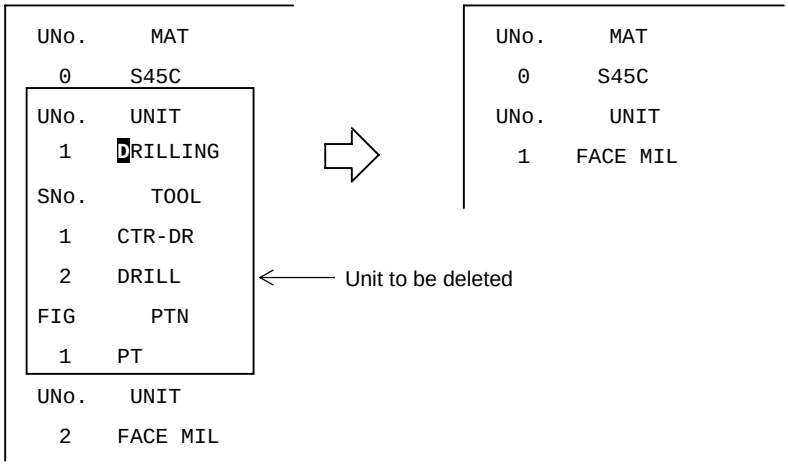
PROGRAM COMPLETE	SEARCH		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(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 deleted.

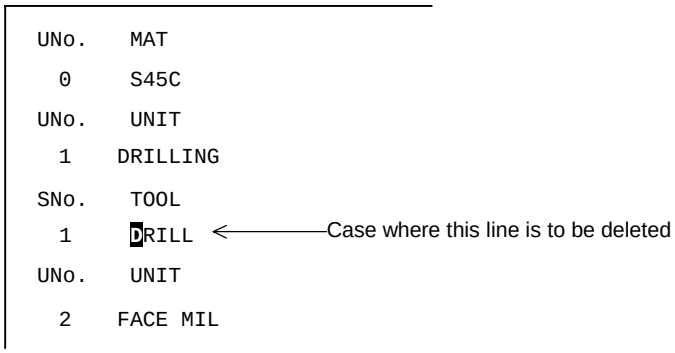


Note: When the deletion operation is done for UNo. 0 (common unit), the alarm **410 ILLEGAL DELETION** is displayed.

2. Deletion of the tool sequence

[Menu selection] **ERASE**

(1) Place the cursor on the tool sequnece 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		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

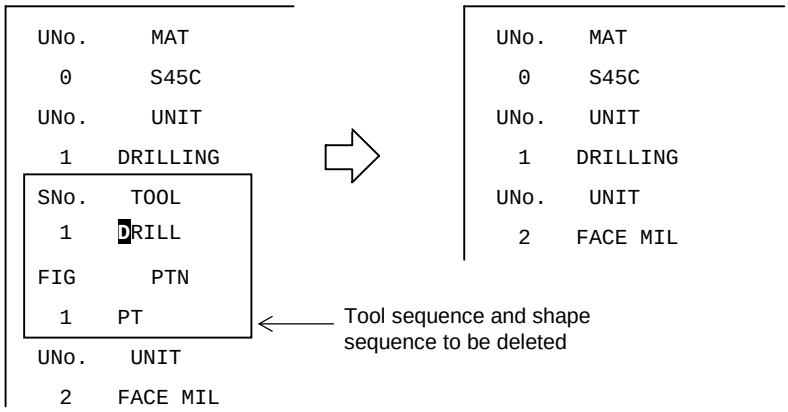
(3) Press the menu key **ERASE**.

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

(4) Press the input key .

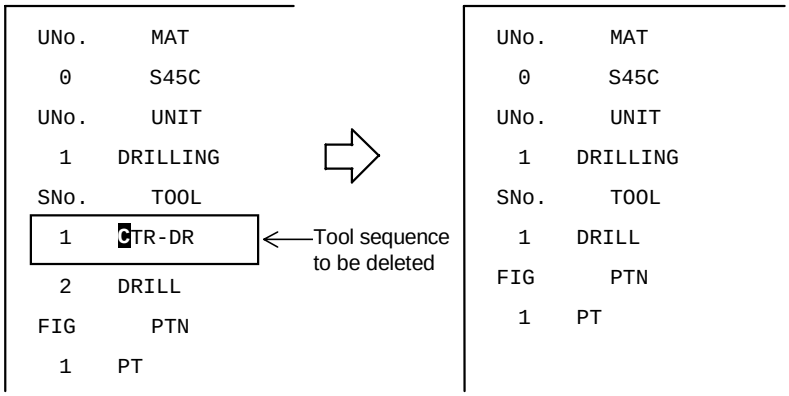
- If there is only one tool sequence line, the tool sequence line in which the cursor is located and the shape sequence which follows will be deleted at the same time.

Example:



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

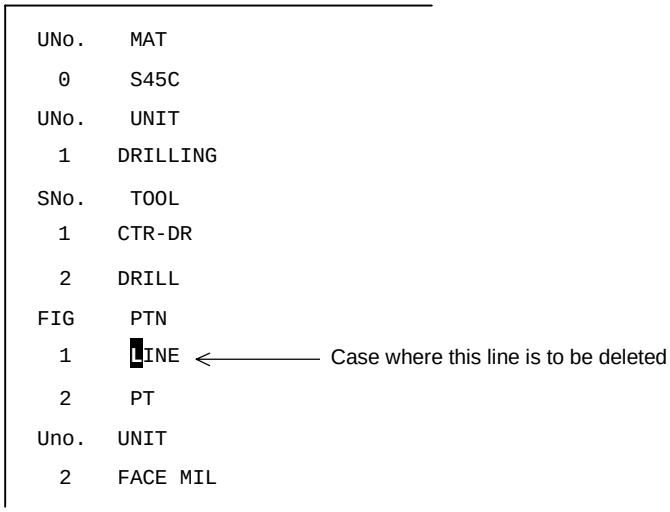
Example:



3. Deletion of the shape sequence

[Menu selection] **ERASE**

(1) Place the cursor on the shape sequnece 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.

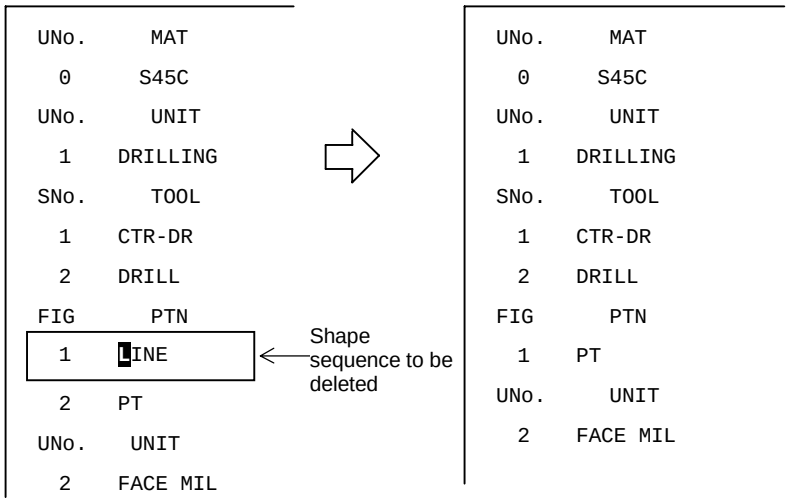
PROGRAM COMPLETE	SEARCH		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

(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

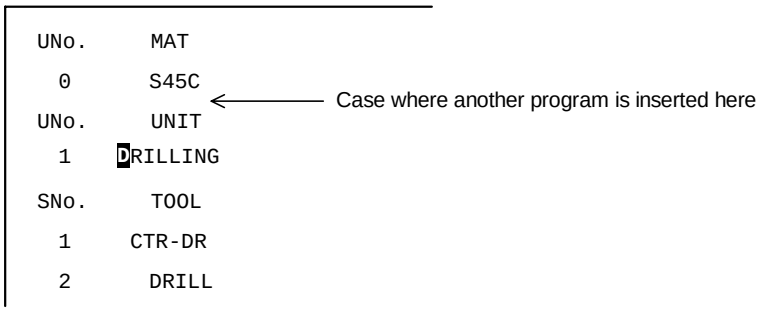
1. Copying of a program

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		TPC	INSERT	ERASE	SHAPE COPY	UNIT COPY	PROGRAM COPY	HELP

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

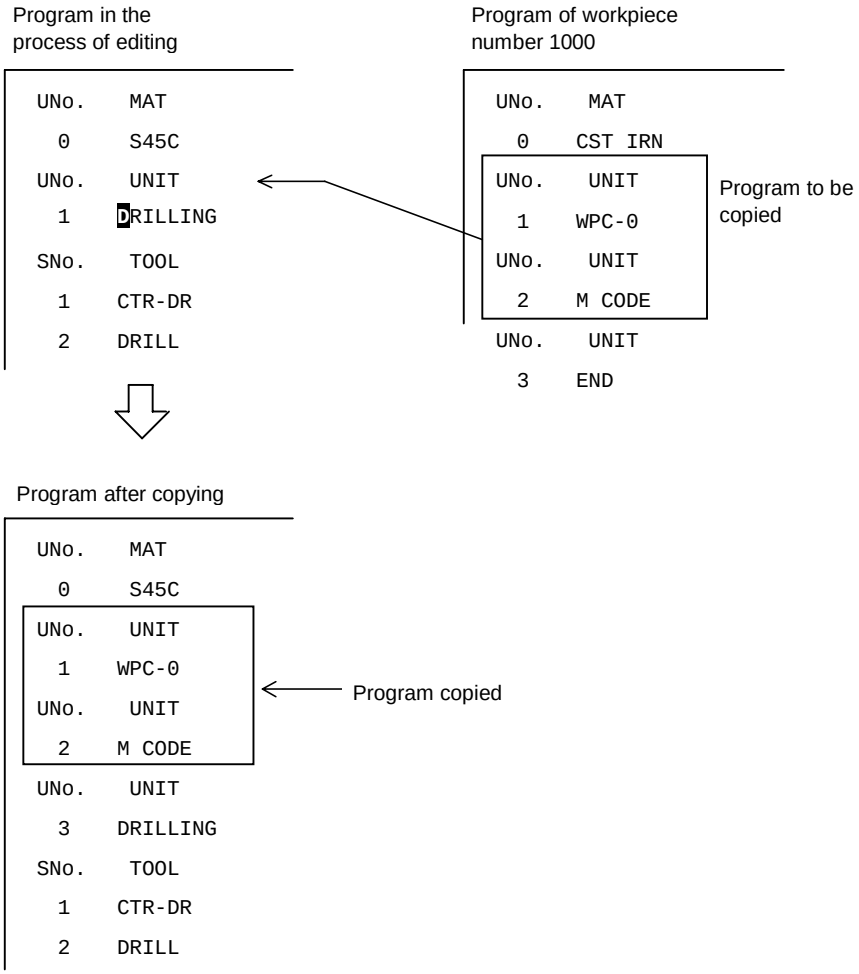
➔ The display of PROGRAM COPY is then reversed and the screen displays the message **WORKPIECE PROGRAM NUMBER?**.

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

Example: Workpiece number 1000

Press the following keys: [1] [0] [0] [0] [⏎]

➔ The program of workpiece number 1000 is then copied.



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 workpiece number which is not recorded is entered, the alarm **405 PROGRAM No. NOT FOUND** is displayed.

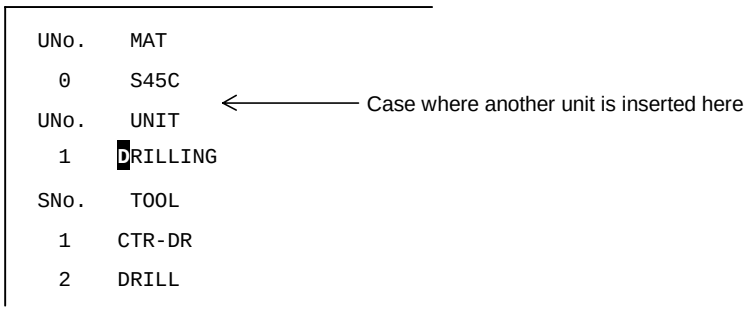
Note 3: When an EIA/ISO program workpiece number is entered, the alarm **440 EIA/ISO PROGRAM DESIGNATED** is displayed.

2. Copying of a unit

In the process of creating or editing a program, this UNIT COPY function is used to perform the copying, unit by unit from the program or from another program. The unit and also the tool sequence and the shape sequence which follow are copied.

[Menu selection] **UNIT COPY**

(1) Move the cursor to the line on which a unit is copied.



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		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 message: **WORKPIECE PROGRAM NUMBER?**.

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

Example: Workpiece number 1000

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

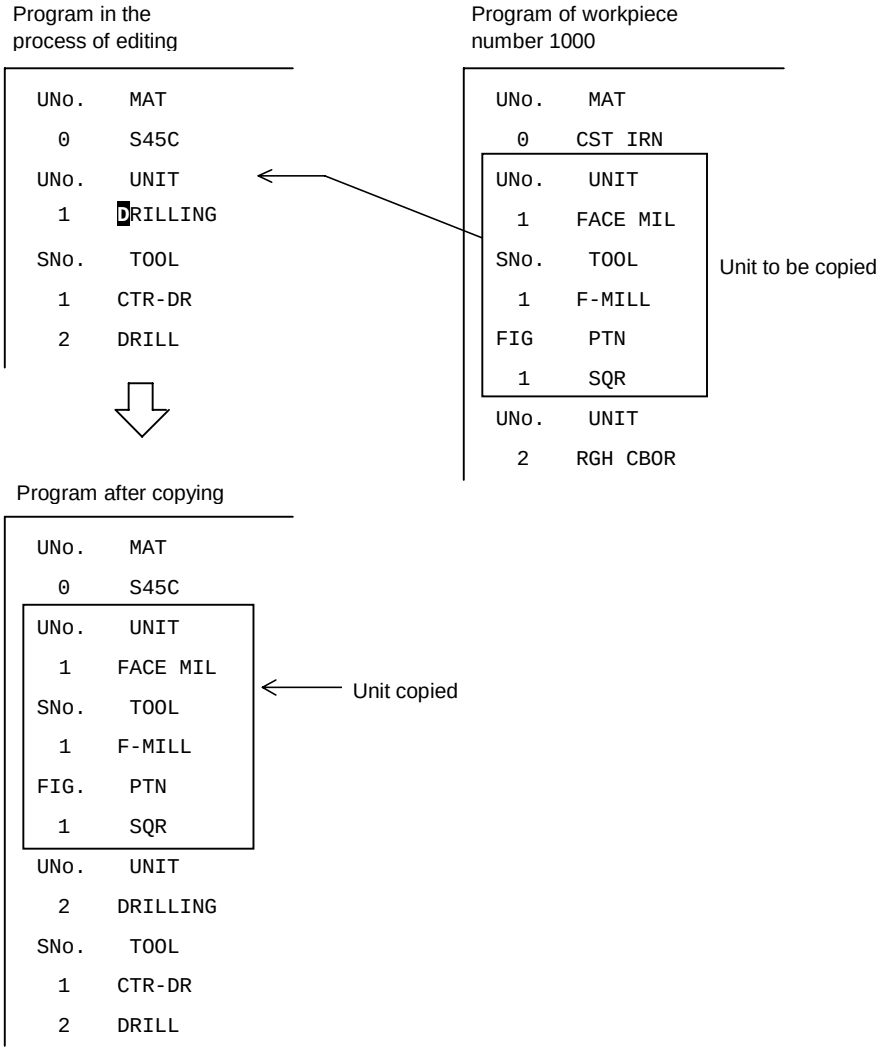
➔ When the workpiece number is entered, the screen displays the message **UNIT NUMBER <INPUT>?**.

(5) Enter the number of the unit to be copied.

Example: Unit number 1

Press the following keys: 1 [→]

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



Note 1: The common unit UNo. 0 cannot be copied. Any attempt to make such a copy will cause the alarm **402 ILLEGAL NUMBER INPUT** to be displayed.

Note 2: When an unregistered workpiece number is entered, the alarm **405 PROGRAM No. NOT FOUND** is displayed.

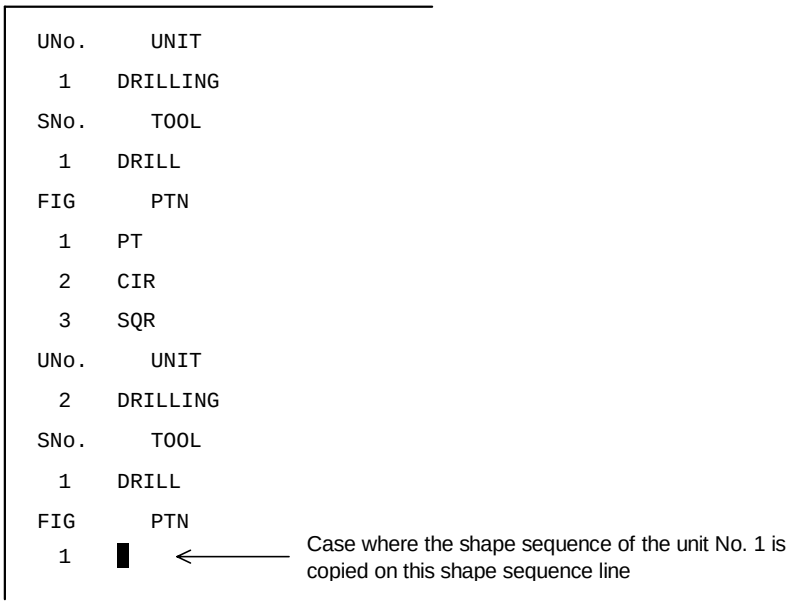
Note 3: When a workpiece number of the EIA/ISO program is entered, the alarm **440 EIA/ISO PROGRAM DESIGNATED** is displayed.

3. Copying of shape

This SHAPE COPY function is used to copy the shape sequence in the process of creation or editing of a program. However, it is impossible to perform the copying if the shape sequence line has already been filled with data.

[Menu selection] **SHAPE COPY**

(1) Move the cursor to the position in which the shape sequence is to be copied.



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		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>?**.

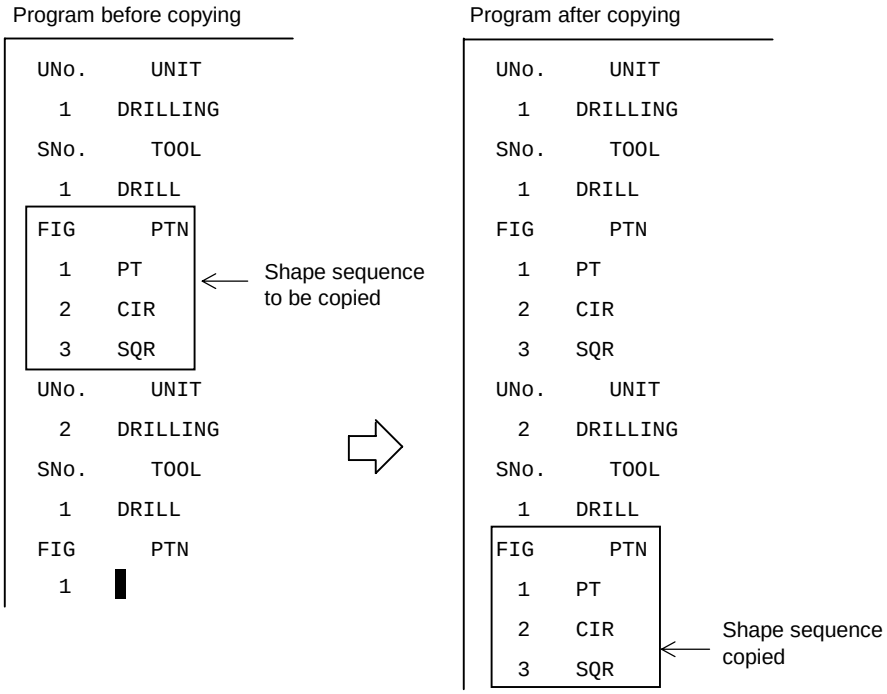
(4) Enter the number of the unit containing the shape sequence to be copied.

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**

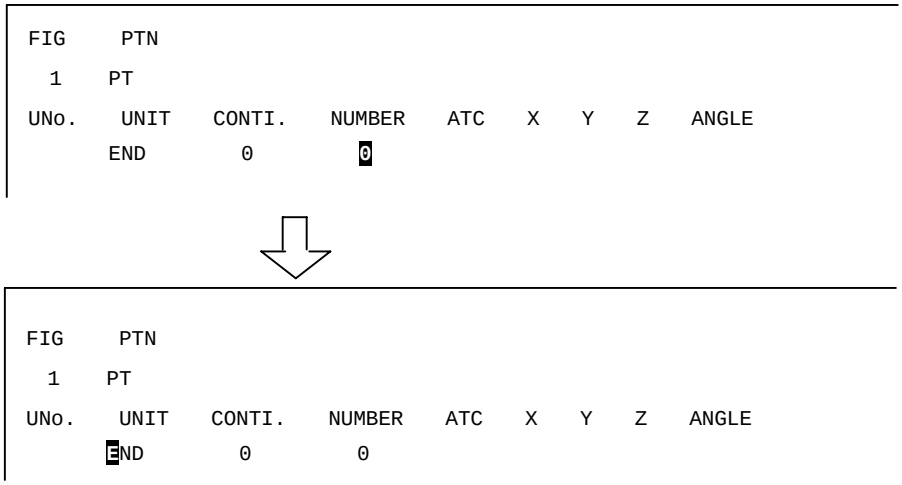
(1) In creating mode, press the menu selector key.

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

PROGRAM COMPLETE	SEARCH		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 WINDOW FUNCTIONS

<Functions>

The window display functions described below are provided to make it easy for data to be set during creation and editing of programs.

Windows can be displayed by pressing the corresponding menu key.

3-1 MAZATROL Help Window

The MAZATROL help window 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:

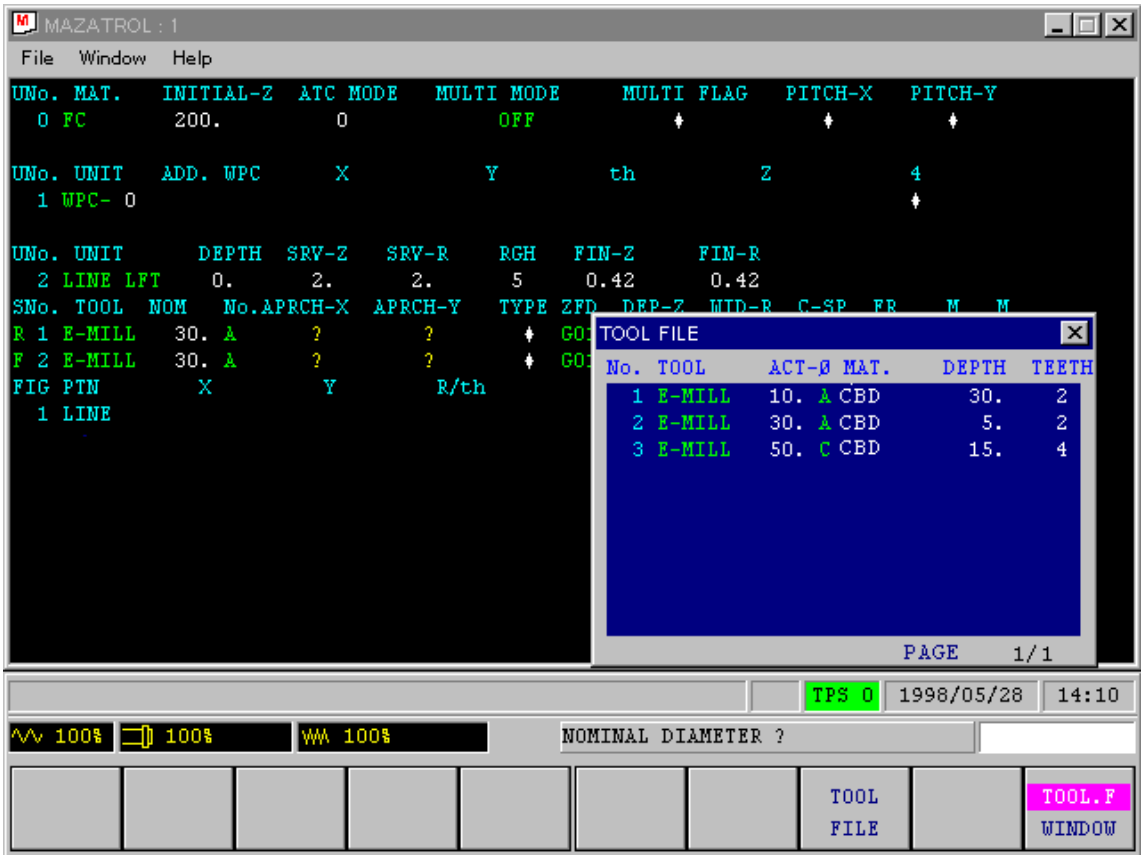
- Common unit
- Machining unit
- Basic coordinates system unit (WPC)
- Auxiliary coordinates system unit (OFFSET)
- MMS unit tool sequence (except for patterns X-Y-th CNR and CALIBR.)

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” of a line/face machining unit, or an end mill or chamfering cutter data item “NOM” 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:



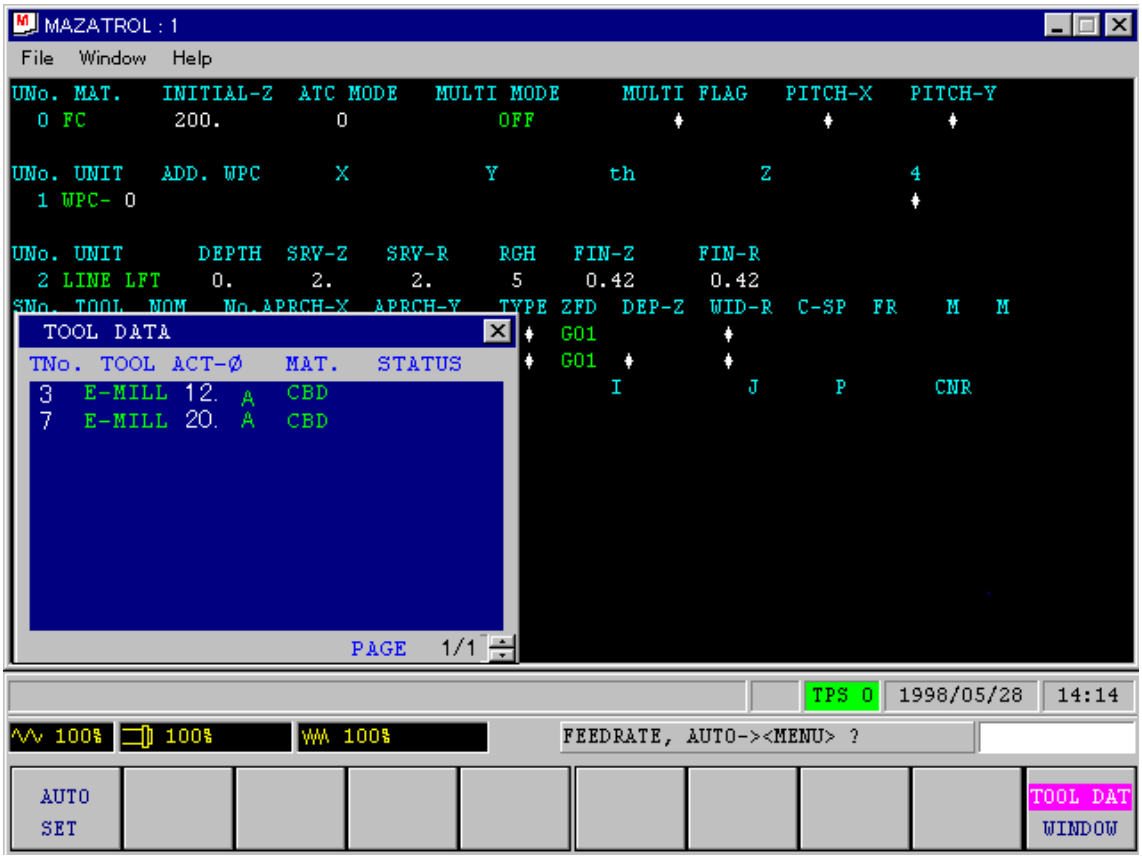
D735P0059E

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:



D735P0060E

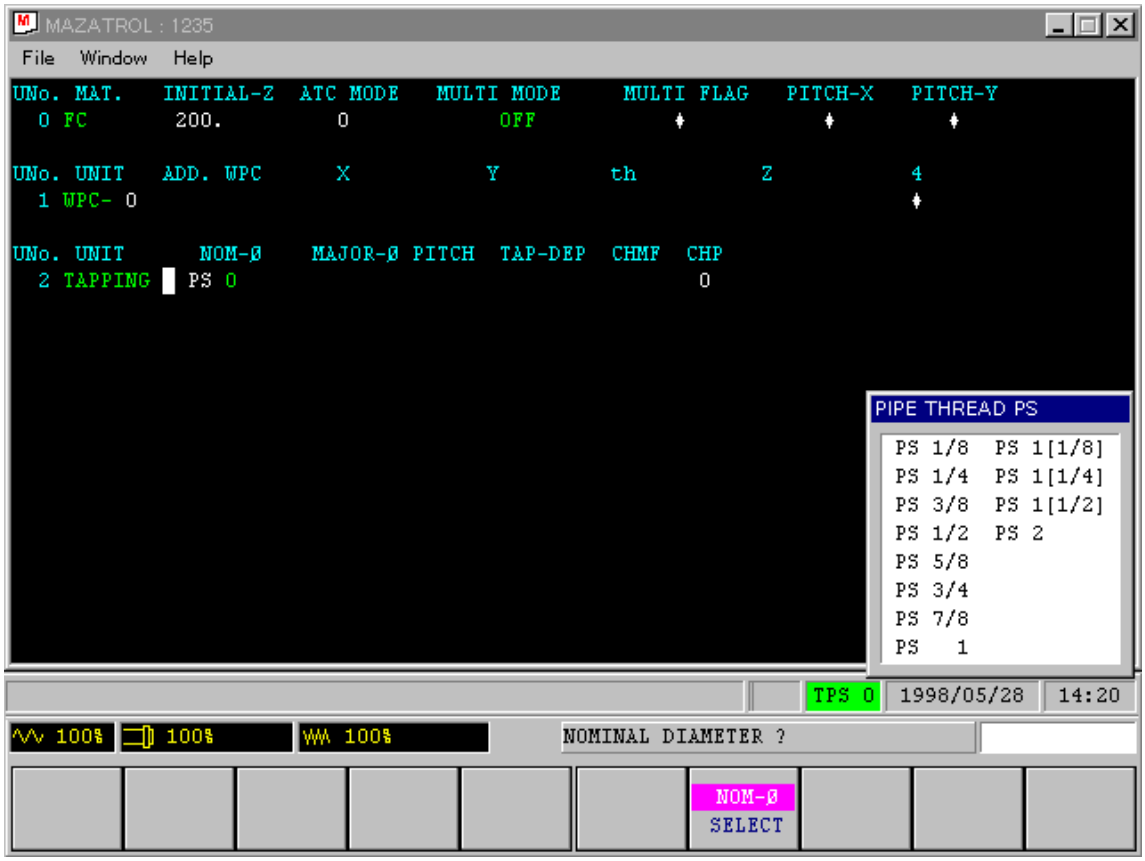
3-4 Tap Nominal Diameter Window

The tap nominal diameter window can be displayed by pressing the menu key **NOM-ϕ SELECT** while the cursor remains set to “NOM-ϕ” 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:



D735P0061E

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 Types of Units

As shown previously in Part 1, Chapter 1, “MAZATROL PROGRAMMING PROCEDURES,” a MAZATROL program must consist basically of the four units listed below [numbers in parentheses ( ) below denote the section numbers within this manual].

Common unit:	A unit that must be created at the beginning of a program (4-2)
Basic coordinates unit:	A unit used to set basic coordinates (4-3)
Machining unit:	A unit used for workpiece machining (4-5 through 4-8)
End unit:	A unit that must be set at the end of a program (4-9)

In addition, the four units listed below can be created as required.

Auxiliary coordinates unit:	Used to set auxiliary coordinates (4-4).
Special mode unit:	Used to make the machine perform operations other than machining (4-10)
Manual program mode unit:	Used to create a program corresponding to an EIA/ISO program (4-11)
MMS unit:	Used for automatic measurement of basic coordinates (4-12)

The functions of each of the eight units listed above are described below.

4-2 Common Unit

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

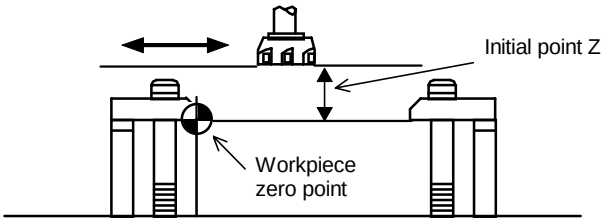
Data for the common unit are set on the **PROGRAM** display in creating mode.

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

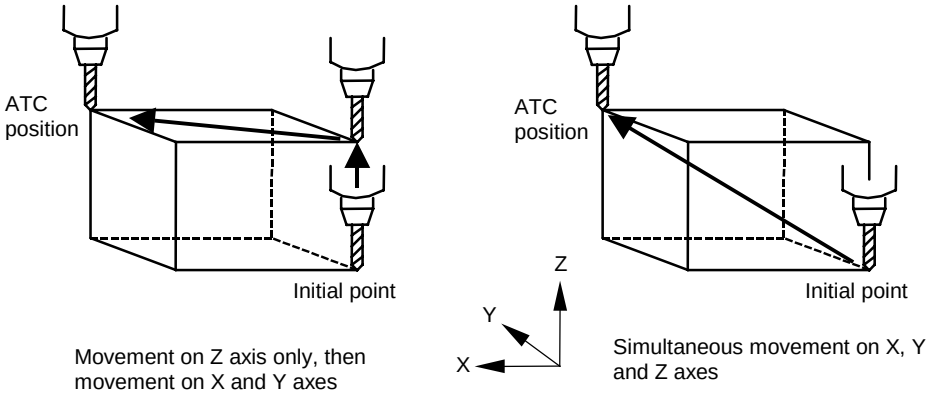
UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	□	□	□	□	□	□	□

Cursor position	Entering
MAT	Specify the material of the workpiece using the menu key to perform the automatic determination of the cutting conditions. 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. Also, these names of workpiece materials are already registered in the system and they are MAZAK-recommended ones. To register new workpiece material names, refer to Section 8-1, “CUTTING CONDITION (W.-MAT./T.-MAT.) Display,” of the relevant Operating Manual. **Example:**
INITIAL-Z	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. 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.

FC	FCD	S45C	SCM	SUS	AL	CU		>>>	



M3P077

Cursor position	Entering
ATC MODE	Specify the mode of movement from the initial point to the ATC position at the time of ATC. <Independent movement on each axis: 0> <Simultaneous movement on all axes: 1>  Movement on Z axis only, then movement on X and Y axes Simultaneous movement on X, Y and Z axes M3P078 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. **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.
MULTI MODE	Specify the multi-workpiece machining mode. MULTI OFF | MULTI 5 * 2 | OFFSET TYPE - 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
	(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.
	(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.

Cursor position	Entering
(MULTI MODE)	

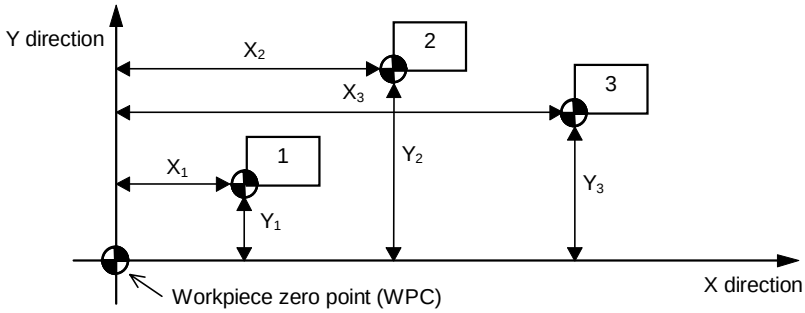
UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	S45C	50	0	OFFSET TYPE	◆	◆	◆

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
↓	↓	↓	↓	↓

It is possible to set a maximum of 10 OFS numbers.

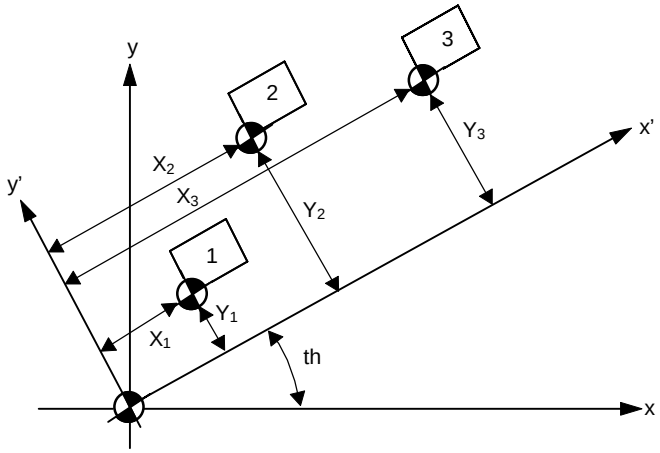
The relation between the position of the workpiece and the coordinates is as follows:

- Case of th = 0

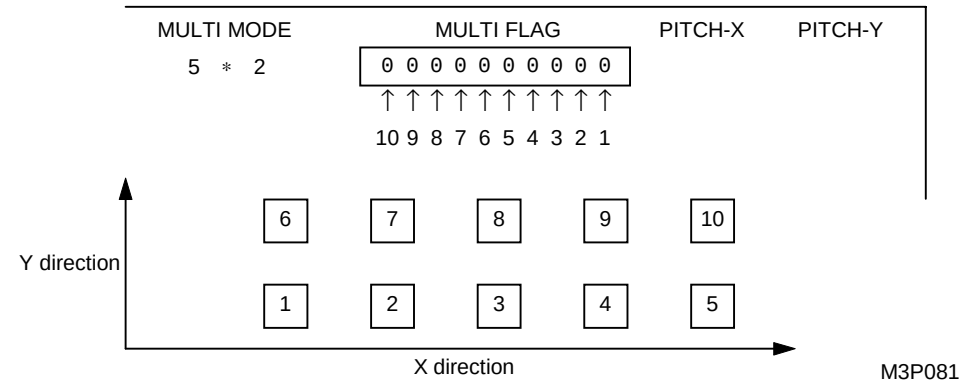


M3P079

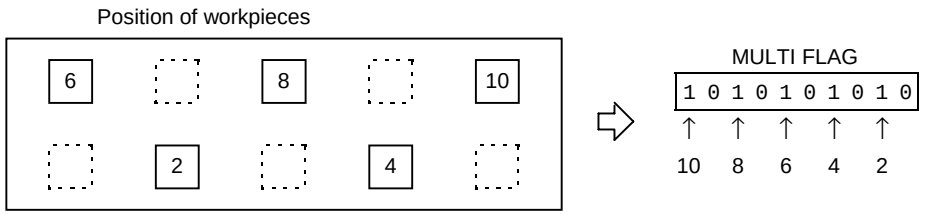
- Case of designation of th

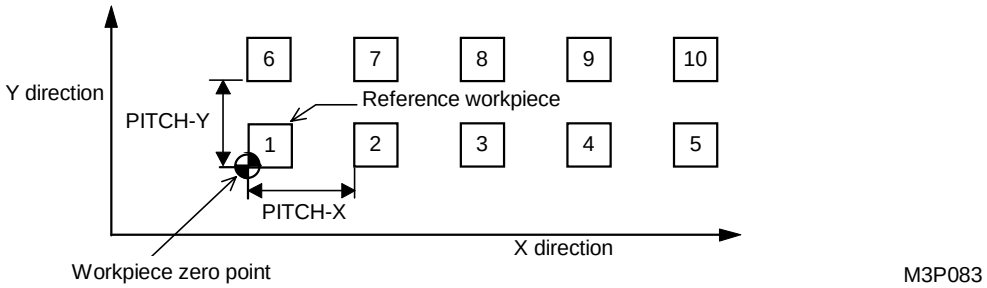


M3P080

<table>
<tr><th>Cursor position</th><th>Entering</th></tr>
<tr><td>MULTI FLAG</td><td>
Specify the execution or non-execution of the machining for each workpiece in case of MULTI 5 ∗ 2.
Non-execution 0
Execution 1
A maximum of 10 identical workpieces can be machined with only one program.
The relation of the value of 10 figures in the MULTI FLAG column and the workpiece positions is as follows:


M3P081

Example of entering:


M3P082
</td></tr>
<tr><td>PITCH-X
PITCH-Y</td><td>
Specify the pitch in the directions X and Y between the located workpieces in case of MULTI 5 × 2.


M3P083

The workpieces are arranged with an equal distance of the values PITCH-X, PITCH-Y from the position of the reference workpiece.
Note: Minus data cannot be entered in the articles PITCH-X, -Y.
</td></tr>
</table>

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.

4-3 Basic Coordinates System Unit

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.

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

Press the menu key **WPC**.

UNo.	UNIT	ADD. WPC	X	Y	th	Z	*4*	*5*
1	WPC- ☐	☐	☐	☐	☐	☐	☐	☐

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

Cursor position	Entering
UNIT	Several basic 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 basic 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 ±999.999)
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 ±999.999)

Note: During the execution of a MAZATROL program, the modal status display on the **POSITION** display will remain set to G54.1 P0.

4-4 Auxiliary Coordinates System Unit

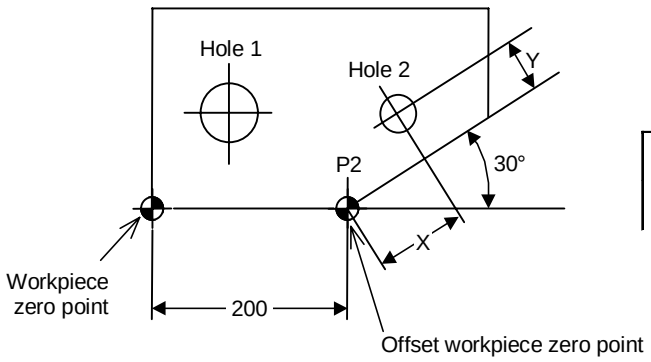
In the process of programming, the workpiece zero point can be offset to any position in order to facilitate the entering of data. In order to do this, it is necessary to select this auxiliary coordinates system unit. Press the menu key **OFFSET**.

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

UNo.	UNIT	U (X)	V (Y)	D (th)	W (Z)
	OFFSET	☐	☐	☐	☐

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)

Example:



UNo.	UNIT	U (X)	V (Y)	D (th)	W (Z)
	OFFSET	200.	0.	30.	0.

M3P084

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).

4-5 Types of the Machining Unit

The machining unit is available in the following three types:

- Point machining unit ..... used for drilling of holes
- Line machining unit ..... used for a contour machining
- Face machining unit ..... used for machining an area and machining form

Each machining unit includes tool sequence and shape sequence.

4-6 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.

4-6-1 Types of point machining units

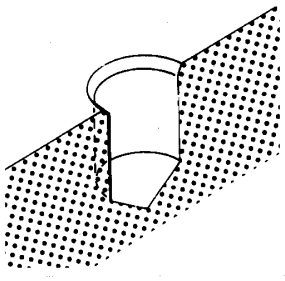
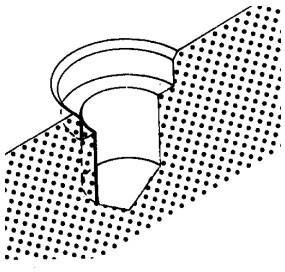
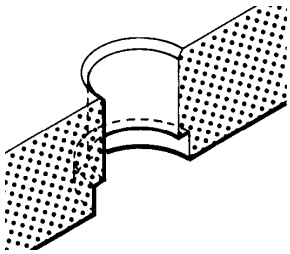
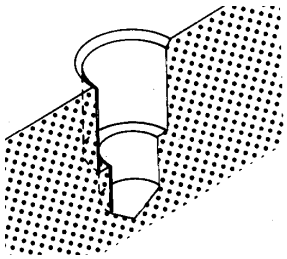
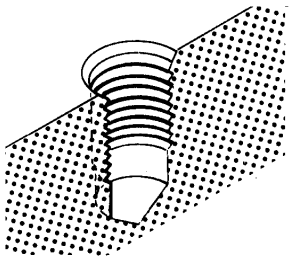
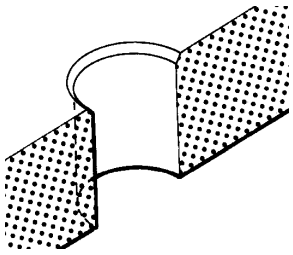
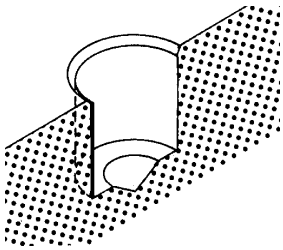
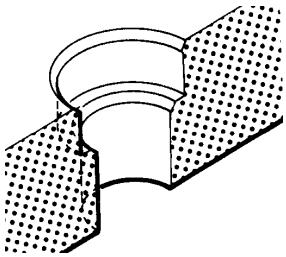
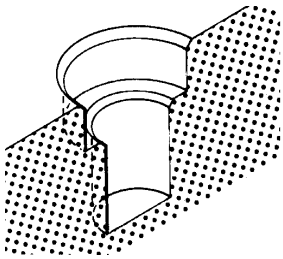
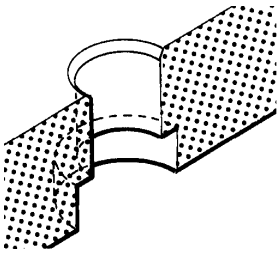
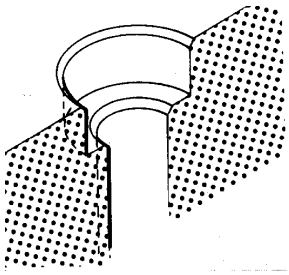
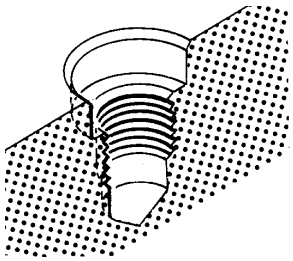
1. Drilling	2. RGH CBOR machining	3. RGH BCB machining	4. Reaming
 NM210-00532	 NM210-00533	 NM210-00534	 NM210-00535
5. Tapping	6-(1) Boring of through hole	6-(2) Boring of non-through hole	6-(3) Boring of stepped through hole
 NM210-00536	 NM210-00537	 NM210-00538	 NM210-00539
6-(4) Boring of stepped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

Fig. 4-1 Types of point machining units

4-6-2 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	MANUAL PROGRAM	OTHER	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 4-6-4 “Automatic tool development for cemented carbide drill.”

4-6-3 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

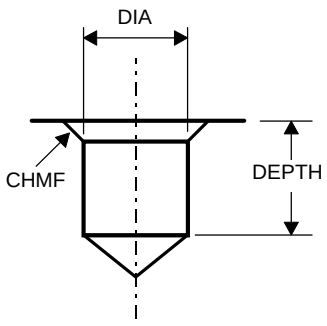
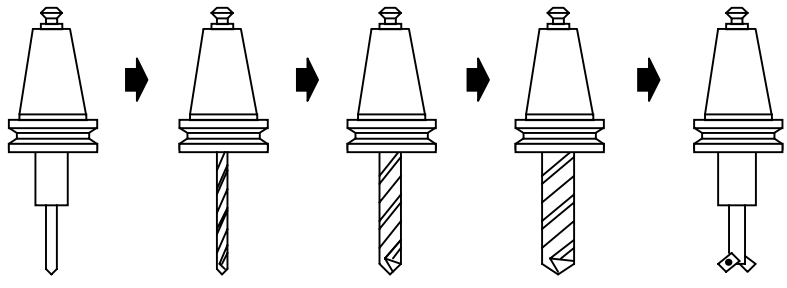
UNo.	UNIT	DIA		DEPTH	CHMF							
2	DRILLING	999.999		999.999	99.9							
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D			
2	DRILL	○		○	○	○	○	○	○			
3	DRILL	○		○	○	○	○	○	○			
4	DRILL	○		○	○	○	○	○	○			
5	CHF-C			○	○	○	○	◆	○			

○ : 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

DRILLING unit	Tool sequence
 M3P085	 M3P086

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0

The bold codes represent parameter addresses.

Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- DEPTH < CHMF
- DIA < 0.05
- **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

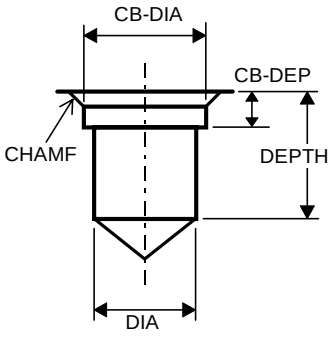
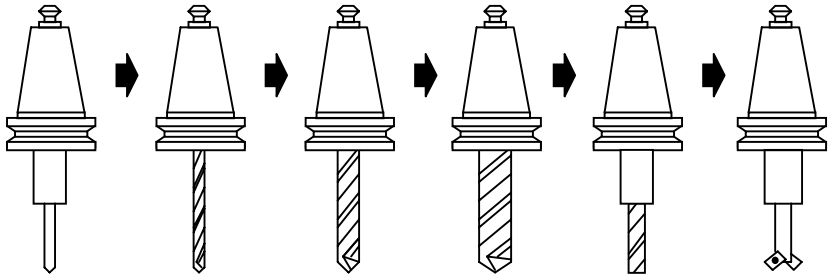
UNo.	UNIT	CB-DIA		CB-DEP	CHMF	BTM	DIA	DEPTH					
2	RGH CBOR	999.999		999.999	99.9		999.999	999.999					
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	E-MILL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

○ : 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

RGH CBOR unit	Tool sequence
 M3P087	 M3P088

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
End mill	Development is always executed.
Chamfering cutter	Development is not executed in the following casses: CHMF = 0 DIA + (CB-DEP × 2) ≥ CB-DIA + (CHMF × 2) < **D13**

The bold codes represent parameter addresses.

Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- CB-DIA < DIA
- DEPTH < CB-DEP
- DEPTH < CHMF
- **D10** < 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	CHF-C			○	○	○	○	◆	○				
6	BK FACE	○		○	○	◆	○	◆	◆				

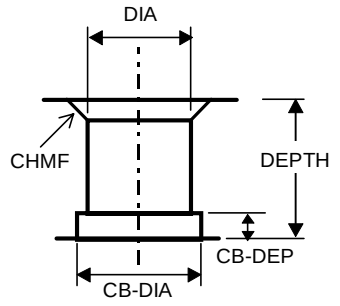
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : Data are not necessary to be set here.

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

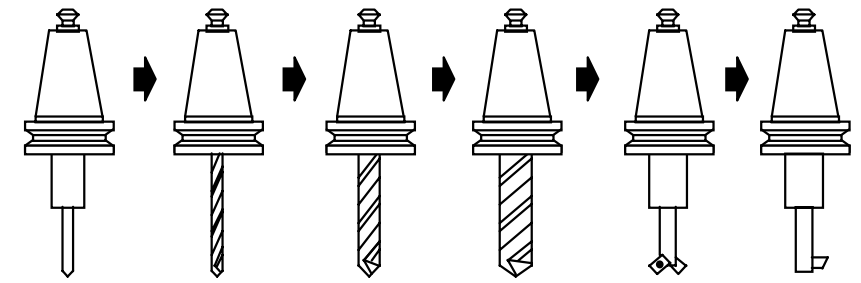
Note 2: The tool sequence represents the case of the maximum development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

RGH BCB unit



M3P089

Tool sequence



M3P090

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** DIA + (CHMF × 2) ≤ **D13** CHMF = 0
Back facing tool	Development is always executed.

The bold codes represent parameter addresses.

Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- CB-DIA < DIA
- DEPTH < CB-DEP
- DEPTH < CHMF
- **D10** < 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.

A. Data setting

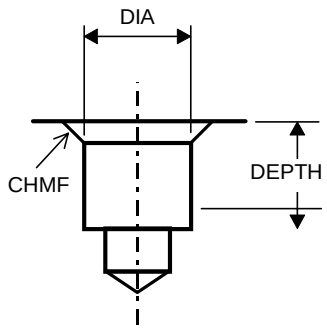
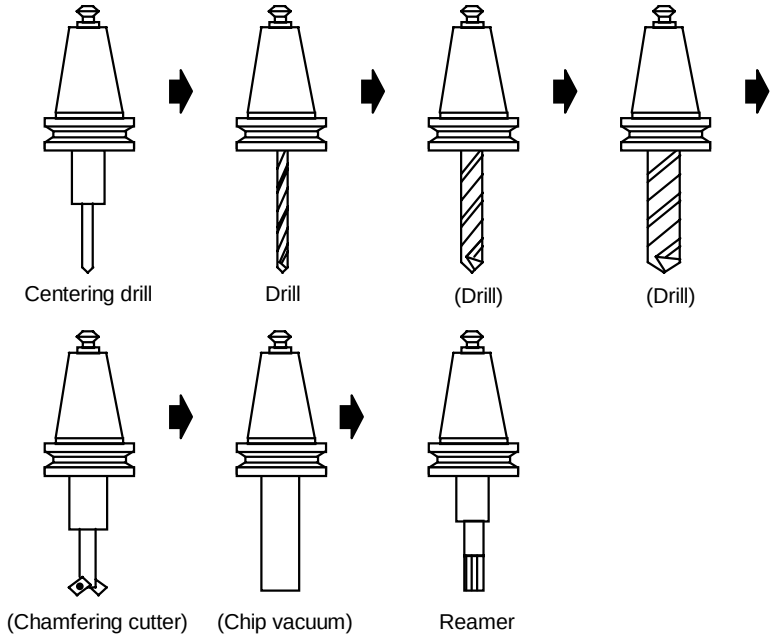
UNo.	UNIT	DIA		DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING	999.999		999.999	99.9								
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	CHF-C			○	○	○	○	◆	○				
6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

○ : 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

REAMING unit	Tool sequence
 M3P091	 M3P092

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA – **D35** ≤ **D8**: Development of one tool **D8** < DIA – **D35** ≤ **D9**: Development of two tools **D9** < DIA – **D35** ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Chip vacuum	Development is not executed if chip removal is not required.
Reamer	Development is always executed.

The bold codes represent parameter addresses.

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

Case of preceding process = boring

A. Data setting

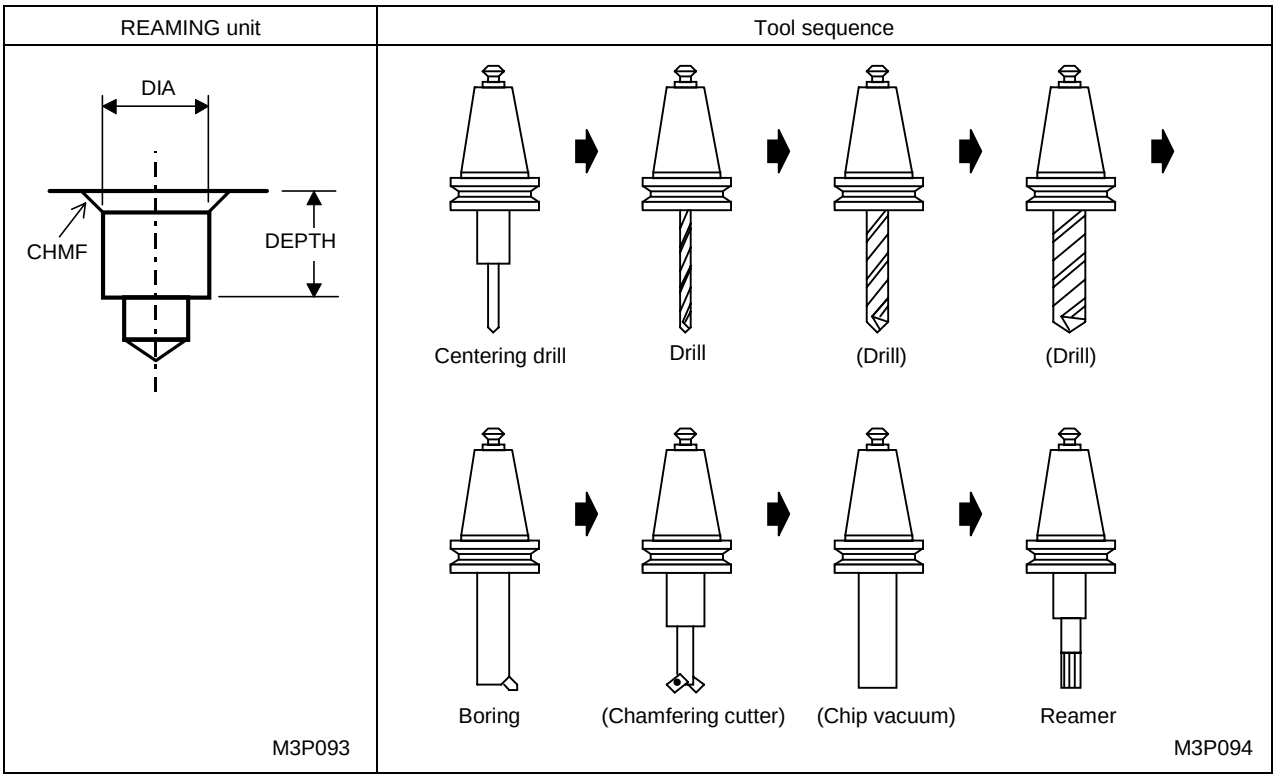
UNo.	UNIT	DIA		DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING	999.999		999.999	99.9								
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○		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

○ : 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.



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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA – **D36** ≤ **D8**: Development of one tool **D8** < DIA – **D36** ≤ **D9**: Development of two tools **D9** < DIA – **D36** ≤ **D10**: Development of three tools
Boring tool	Development is always executed.
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Chip vacuum	Development is not executed if the chip removal is not required.
Reamer	Development is always executed.

The bold codes represent the parameter addresses.

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

Case of preceding process = end mill

A. Data setting

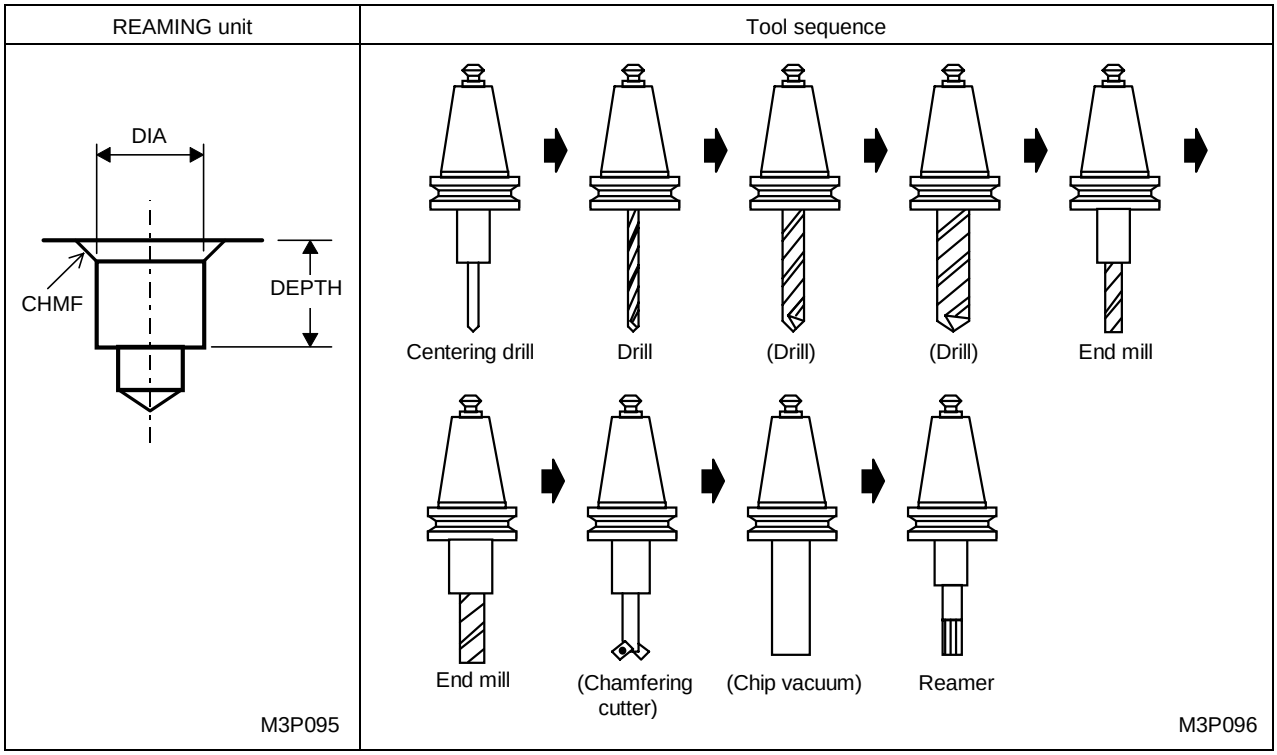
UNo.	UNIT	DIA		DEPTH	CHMF	PRE-REAM	CHP						
2	REAMING	999.999		999.999	99.9								
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	DRILL	○		○	○	○	○	○	○				
4	DRILL	○		○	○	○	○	○	○				
5	E-MILL			○	○	○	○	○	○				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

○ : 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.



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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ DIA – **D37** ≤ **D8**: Development of one tool **D8** < DIA – **D37** ≤ **D9**: Development of two tools **D9** < DIA – **D37** ≤ **D10**: Development of three tools
End mill	Development of two tools is executed.
Chamfering cutter	Development is not executed in the following cases: DIA + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Chip vacuum	Development is not executed if the chip removal is not required.
Reamer	Development is always executed.

The bold codes represent the parameter addresses.

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

5. Tapping unit (TAPPING)

Select this unit for performing tapping.

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	CHF-C			○	○	○	○	◆	○				
6	CHP 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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

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>

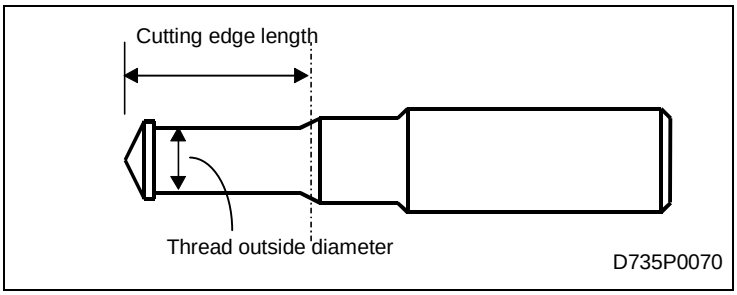
Example: For PT 3/8 screws: Press the menu key **E (1/8) EIGHTH,** and then press the keys [3] and [→] in this order.

For PF 1 screws: Press the keys [1] and [→] in this order.

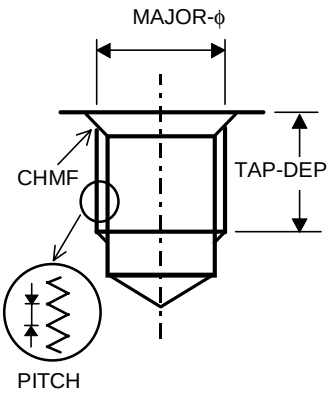
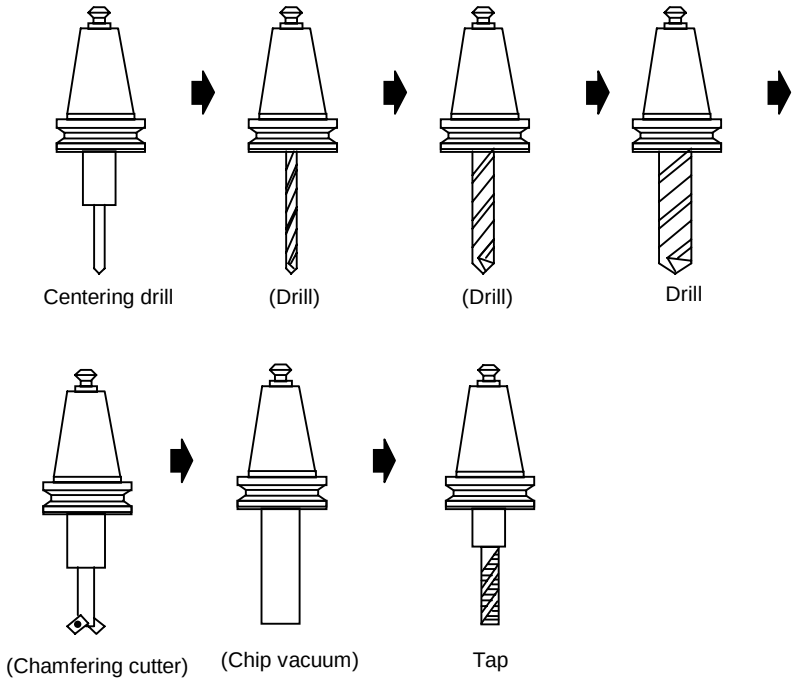
Note 1: The thread depths of PT screws or PS screws are set automatically according to MAZAK specifications.

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 “4-6-5 New tapping auto-setting scheme.”

TAPPING unit	Tool sequence
	

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	A maximum of three tools are developed depending on the diameter of the hole. 0.05 ≤ Diameter of pre-hole drilling ≤ **D8**: Development of one tool **D8** < Diameter of pre-hole drilling ≤ **D9**: Development of two tools **D9** < Diameter of pre-hole drilling ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: Diameter of hole + (CHMF × 2) ≤ **D2** – **D4** CHMF = 0
Chip vacuum	Development does not take place if the chip removal is not required.
Tap	Development always takes place.

The bold codes represent the parameter addresses.

Note: In the following cases the alarm **416 AUTO PROCESS IMPOSSIBLE** will be displayed.
- TAP-DEP < CHMF
- Case of designation of threading other than the JIS standard threading (however, this can be used for forced insertion).

6. Boring unit (BORING)

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

Through hole boring unit (BORING T1)

Select this unit for performing through-hole boring.

A. Data setting

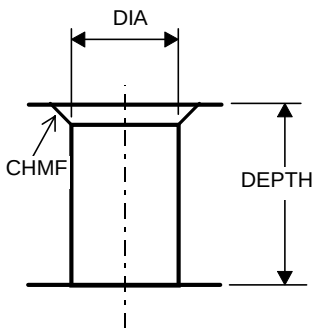
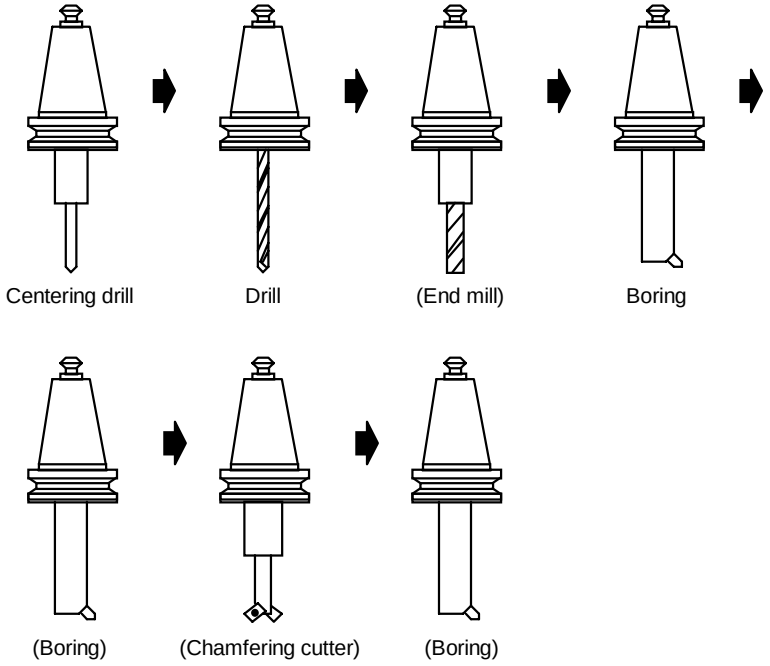
UNo.	UNIT	DIA		DEPTH	CHMF	WAL							
2	BORE T1	999.999		999.999	99.9								
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	○				
3	E-MILL			○	○	○	○	○	○				
4	BOR BAR	○		○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	CHF-C			○	○	○	○	◆	○				
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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

BORING T1 unit	Tool sequence
 M3P099	 M3P100

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

The bold codes represent the parameter addresses.

Note: In the following cases the alarm **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

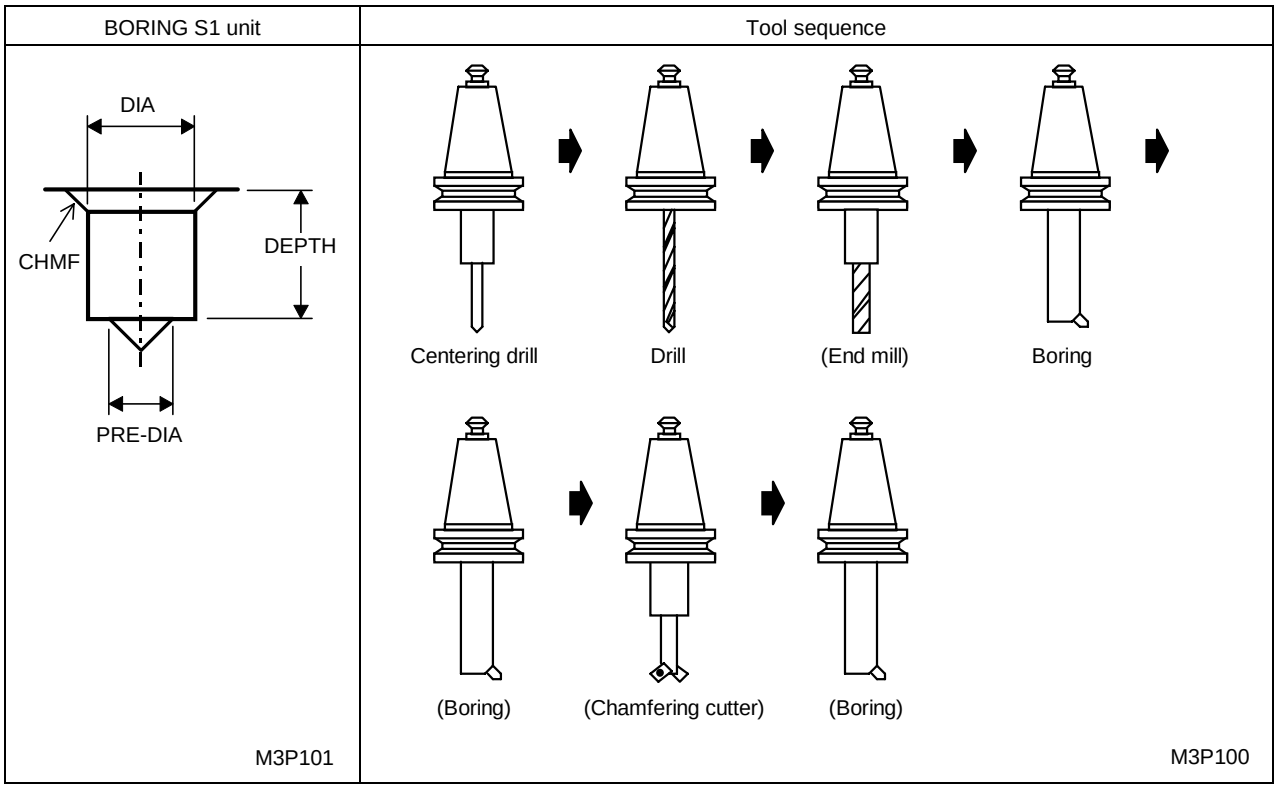
UNo.	UNIT	DIA		DEPTH	CHMF	BTM	WAL	PRE-DIA						
2	BORE S1	999.999		999.999	99.9			999.999						
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA		PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆		◆	90°	CTR-D				
2	DRILL	○		○	○	○		○	○	○				
3	E-MILL			○	○	○		○	○	○				
4	BOR BAR	○		○	○	○		○	○	○				
5	BOR BAR	○		○	○	○		○	○	○				
6	CHF-C			○	○	○		○	◆	○				
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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.



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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

The bold codes represent the parameter addresses.

Note: The alarm **416 AUTO PROCESS IMPOSSIBLE** is given in the following cases:
- DIA < PRE-DIA
- DIA ≤ 6.0
- DEPTH < CHMF
- PRE-DIA = 0 → Depth of hole < (A/3.328558 + **D12**)
- PRE-DIA ≠ 0 → Depth of hole < (A – PRE-DIA)/3.328558

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

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

Stepped through hole boring unit (BORING T2)

Select this unit for performing stepped through hole boring.

A. Data setting

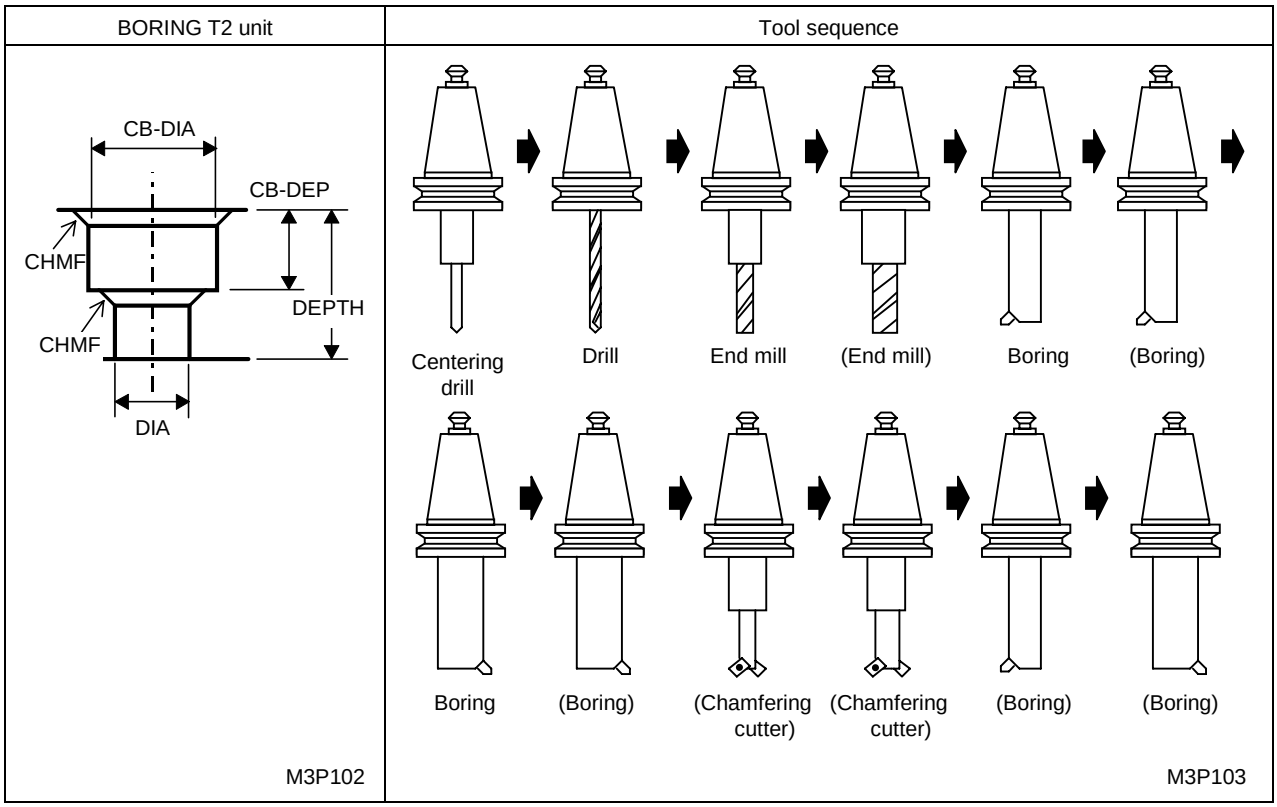
UNo.	UNIT	CB-DIA		CB-DEP	CHMF	BTM	WAL	DIA	DEPTH	CHMF	WAL		
2	BORE T2	999.999		999.999	99.9			999.999	999.999	99.9			
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA		PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆		◆	90°	CTR-D				
2	DRILL	○		○	○	○		○	○	○				
3	E-MILL			○	○	○		○	○	○				
4	E-MILL			○	○	○		○	○	○				
5	BOR BAR	○		○	○	○		○	○	○				
6	BOR BAR	○		○	○	○		○	○	○				
7	BOR BAR	○		○	○	○		○	○	○				
8	BOR BAR	○		○	○	○		○	○	○				
9	CHF-C			○	○	○		○	◆	○				
10	CHF-C			○	○	○		○	◆	○				
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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.



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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

The bold codes represent the parameter addresses.

Note: The alarm **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			

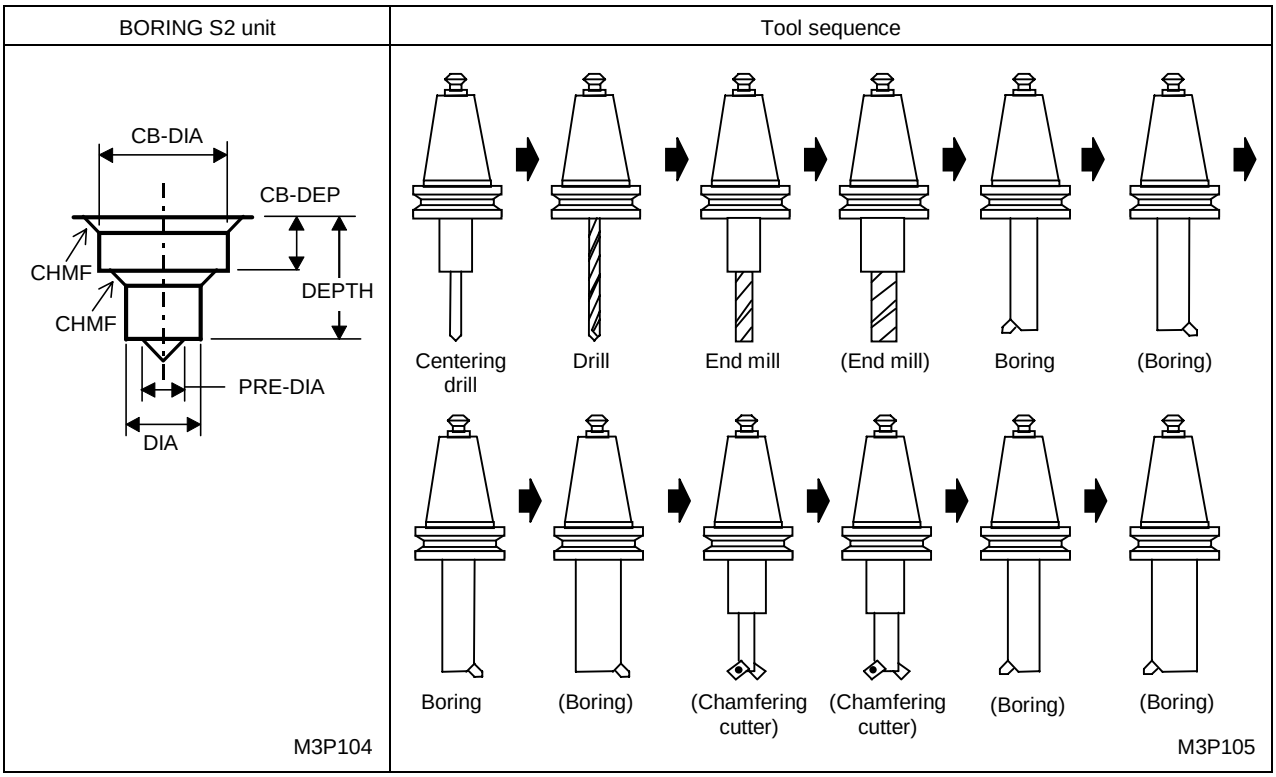
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆	◆	◆	90°	CTR-D				
2	DRILL	○		○	○	○	○	○	◆				
3	E-MILL			○	○	○	○	○	○				
4	E-MILL			○	○	○	○	○	○				
5	BOR BAR	○		○	○	○	○	○	○				
6	BOR BAR	○		○	○	○	○	○	○				
7	BOR BAR	○		○	○	○	○	○	○				
8	BOR BAR	○		○	○	○	○	○	○				
9	CHF-C			○	○	○	○	◆	○				
10	CHF-C			○	○	○	○	◆	○				
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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.



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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

The bold codes represent the parameter addresses.

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

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

B: DIA – 6.0 (in case of DIA – 6.0 < **D8**),
B: **D8** (in case of **D8** ≤ DIA – 6.0),
B: DEPTH – A/3.328558 – **D12** (in case of CB-DEP = 0) or
B: DEPTH – (A – PRE-DIA)/3.328558 (in case of PRE-DIA ≠ 0)
A: DIA – 6.0 (in case of DIA – 6.0 < **D8**) or
A: **D8** (in case of **D8** ≤ DIA – 6.0)

7. Back boring unit (BK-CBORE)

Select this unit for performing back boring.

A. Data setting

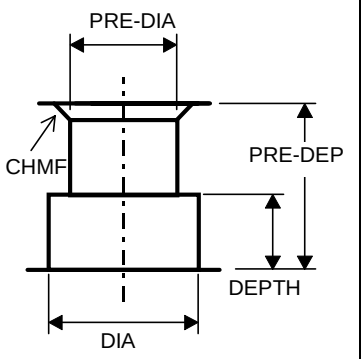
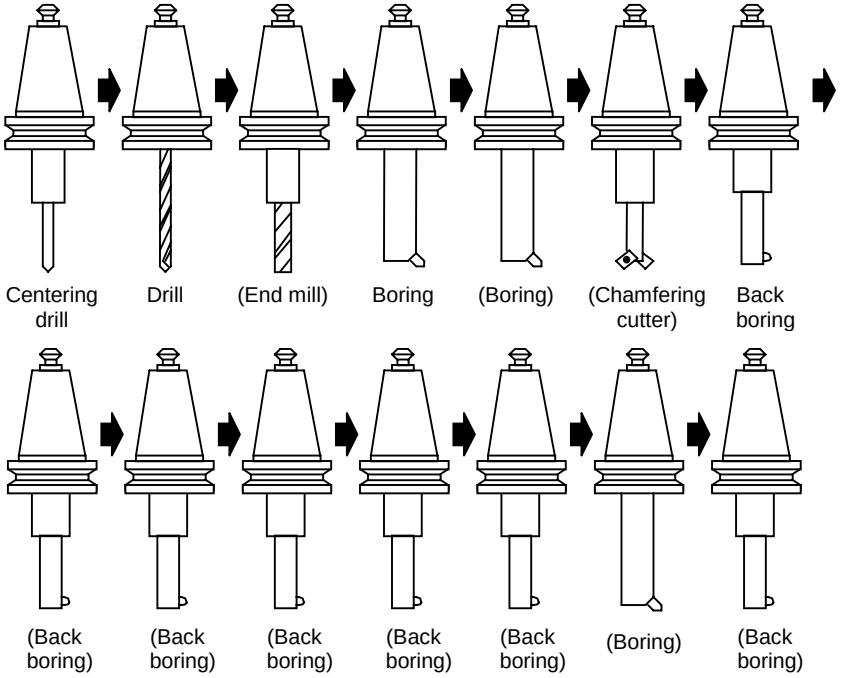
UNo.	UNIT	DIA		DEPTH	BTM	WAL	PRE-DIA	PRE-DEP			CHMF	WAL		
2	BK-CBORE	999.999		999.999			999.999	999.999			99.9			
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP		PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	CTR-DR	○		○	◆		◆	◆	90°	CTR-D				
2	DRILL	○		○	○		○	○	○	○				
3	E-MILL			○	○		○	○	○	○				
4	BOR BAR	○		○	○		○	○	○	○				
5	BOR BAR	○		○	○		○	○	○	○				
6	CHF-C			○	○		○	○	◆	○				
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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

BK-CBOR unit	Tool sequence
 M3P106	 M3P107

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

The bold codes represent the parameter addresses.

Note: The alarm **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.

$$N = \frac{(DBBL - DP)}{6}$$ (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	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	◆	◆

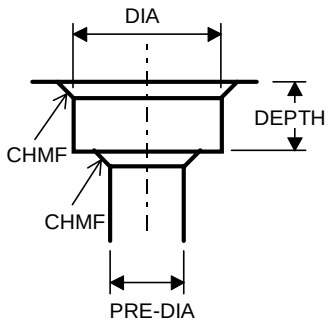
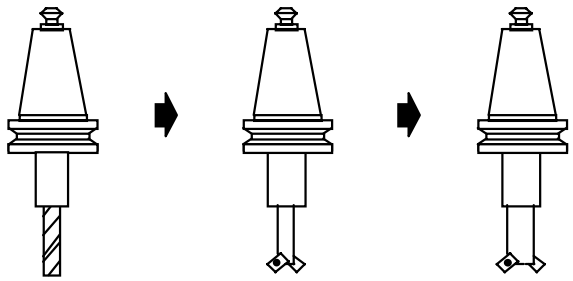
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	E-MILL			○	○	○	○	○	○				
2	CHF-C			○	○	○	○	◆	○				
3	CHF-C			○	○	○	○	◆	○				

○ : 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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

CIRC MIL unit	Tool sequence
 M3P108	 End mill (Chamfering cutter) (Chamfering cutter) M3P109

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 4-6-6.

B. Automatic tool development

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

<Development patterns>

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

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

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

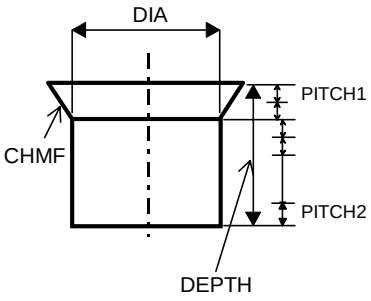
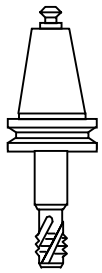
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
1	E-MILL			○	○	○	○	○	○				

○ : 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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

CIRC MIL unit	Tool sequence
 D735P0063	 End mill (Thread mill) D735P0064

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
End mill	Development is always executed.

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	E-MILL			○	○	○	○	○	○				

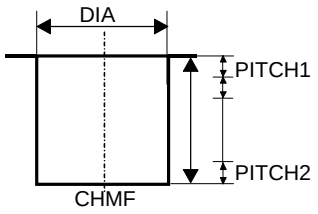
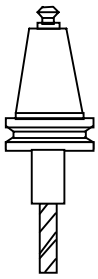
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : 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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

CIRC MIL unit	Tool sequence
	 End mill D735P0071

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
End mill	Development is always executed.

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			□	□	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	E-MILL			○	○	○	○	○	○				
6	CHF-C			○	○	○	○	◆	○				
7	CHF-C			○	○	○	○	◆	○				
8	CHP VAC			◆	◆	◆	◆	◆	◆	◆	◆		
9	TAP	○		○	○	○	◆	○	○				

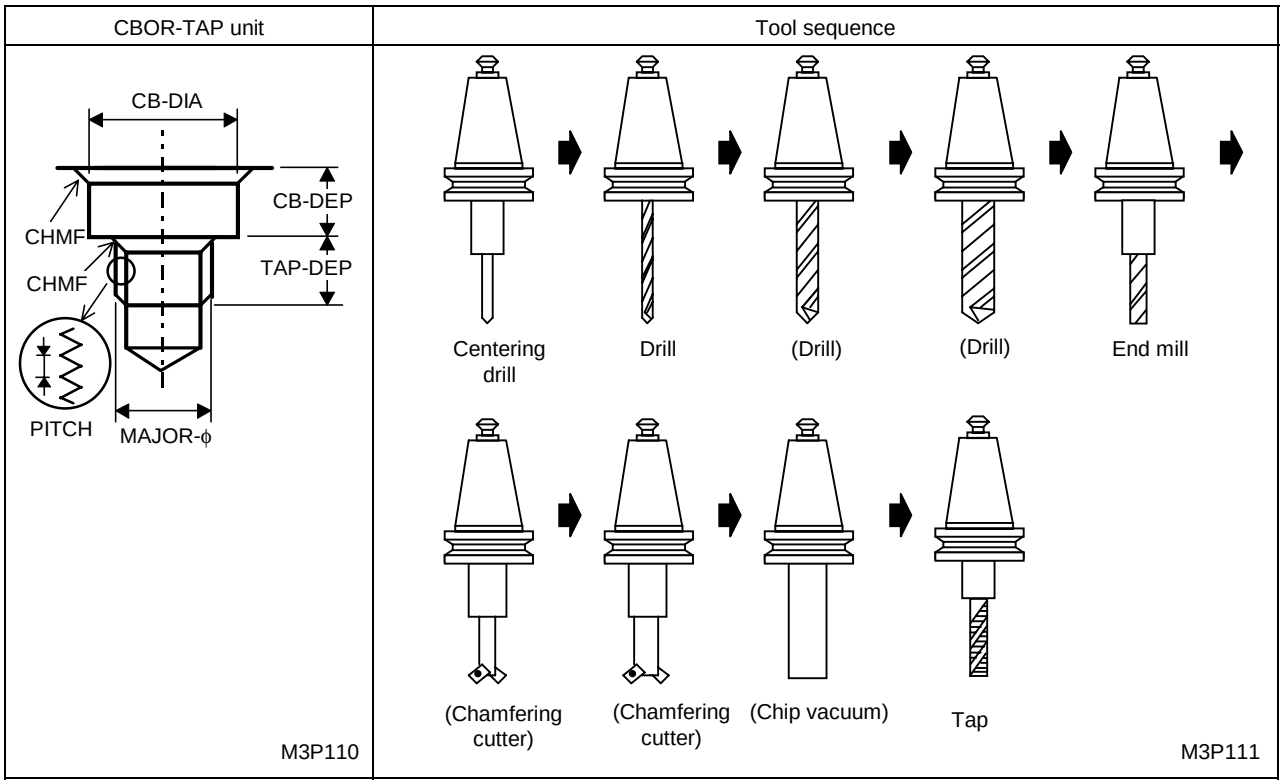
○ : The data displayed here are automatically determined by the automatic tool development function.

◆ : 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 tool development. For data setting in the tool sequence refer to Subsection 4-6-6.

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 “4-6-5 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 4-6-6.

B. Automatic tool development

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

<Development patterns>

Tool	Development patterns
Centering drill	Development is always executed.
Drill	The development of a maximum of three tools is executed depending on the diameter of the hole. 0.05 ≤ DIA ≤ **D8**: Development of one tool **D8** < DIA ≤ **D9**: Development of two tools **D9** < DIA ≤ **D10**: Development of three tools
Chamfering cutter	Development is not executed in the following cases: CHMF (faced hole) = 0 CHMF (threaded hole) = 0
Chip vacuum	The development is not executed 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:
- CB-DIA < MAJOR-ϕ
- (CB-DIA – MAJOR-ϕ)/2 < CHMF (threaded hole)
- PRE-DEP < CHMF (faced hole)
- TAP-DEP < CHMF (threaded hole)

4-6-4 Automatic tool development for cemented carbide drill

Subsection 4-6-3 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:

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

Automatic tool development for drilling with cemented carbide drills 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	○		○	○	○	○	DRIL	○				
2	CHF-C	○		○	○	○	○	◆	○				

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 development of tools. For data setting in the tool sequence refer to Subsection 4-6-6.

4-6-5 New tapping auto-setting scheme

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

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

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

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

1. Tapping for metric thread/unified thread

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

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

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

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

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

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

A. Text file format

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

B. Text data items

- Pre-hole diameter (Setting unit: 1/10000 mm)
  This item denotes the auto-setting values for NOM-ϕ and HOLE-ϕ in the last drill tool sequence whose automatic tool development will be conducted for the tapping unit/counterbore-tapping unit.

C. Editing procedure

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

(2) After copying “TapPrDia.org” (an auto-setting model file for metric thread/unified thread tapping) within the “C:\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”]
(Chamfering) = {(Tap outside diameter) + (Thread pitch) × 2 – (Prehole diameter)}/2
[If parameter **D44** is set to “1” (M32 scheme)]
(Chamfering) = {(Tap outside diameter) – (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 development. (For paramter **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"]
(Chamfering) = {(Tap outside diameter) + (Thread pitch) × 2 – (Prehole diameter)}/2
[If parameter **D44** is set to "1" (M32 scheme)]
(Chamfering) = {(Tap outside diameter) – (Prehole diameter)}/2

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

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

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

(6) Close "Explorer."

D. Example of editing

For "PT1/16", 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.

4-6-6 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	○	○	○	○	○	○	○	○	○	○	○	○	○	○
CHF-C	○	○	○	○	○	○	○	○	◆	○	○	○	○	○
E-MILL	○	○	○	○	○	○	○	○	○	○	○	○	○	○
BK FACE	○	○	○	○	○	○	◆	○	◆	◆	○	○	○	○
REAM	○	○	○	○	○	○	◆	◆	○	○	○	○	○	○
TAP	○	○	○	○	○	○	○	◆	○	○	○	○	○	○
BOR BAR	○	○	○	○	○	○	○	○	○	○	○	○	○	○
B-B BAR	○	○	○	○	○	○	○	○	○	○	○	○	○	○
CHP VAC	○	○	○	○	◆	◆	◆	◆	◆	◆	◆	◆	○	○
Reference item	1	2	3	4	5	6	7	8	9	10	11	12	13	13

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

Note 1: CHP 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 article 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.

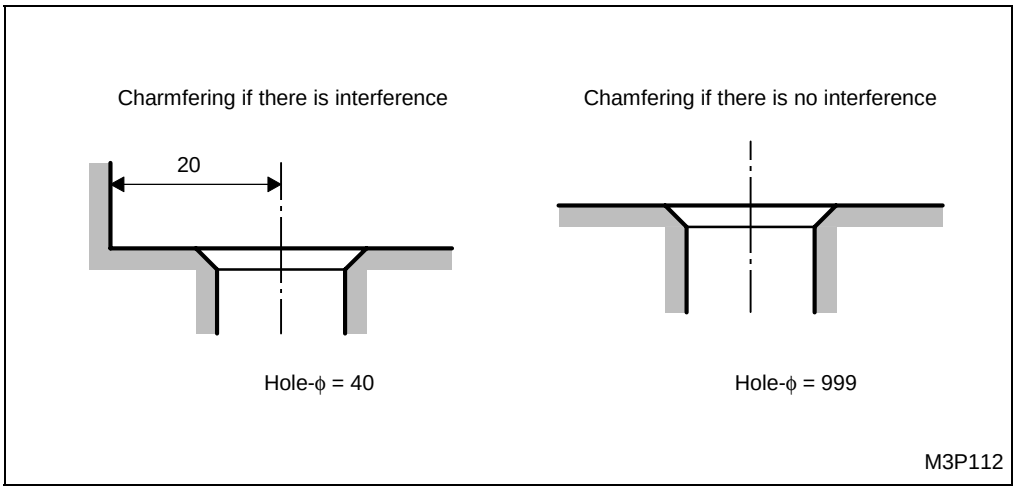


Fig. 4-2 Specification of diameter of machining hole

6. Depth of machining hole: HOLE-DEP

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

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

Note 1: For the chamfering cutter or the chamfering cycle of the centering drill, this article is specified as illustrated below.

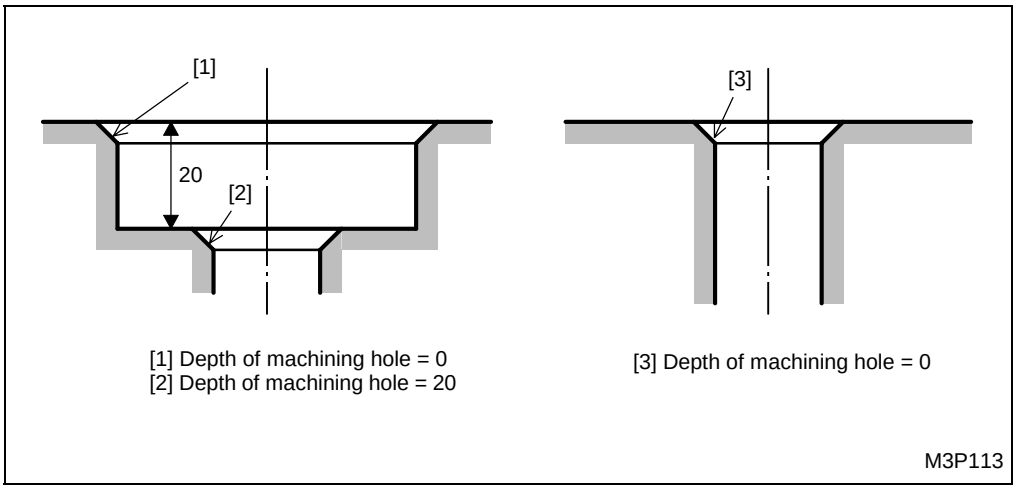
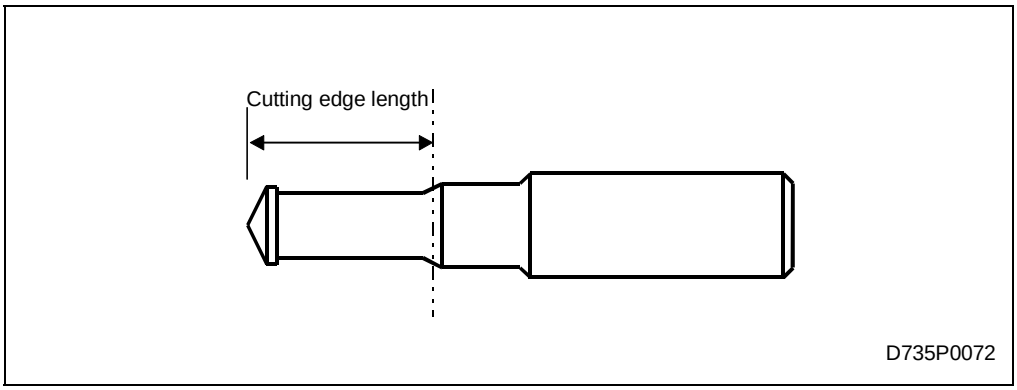


Fig. 4-3 Specification of depth of machining hole

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 article can be modified by means of numeric keys.

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

CYCLE 1	CYCLE 2	CYCLE 3							

For details, refer to Subsection 4-6-7 “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 4-6-7 “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 article can be modified by means of numeric keys.

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

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

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

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

CW CUT	CCW CUT								

For the 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 4-6-7 “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	AUTOPECK CYCLE			DECREME PECKING CYCLE 1	DECREME PECKING CYCLE 2		

For details, refer to Subsection 4-6-7 “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 tool automatic development mode CUT G01 (cutting feed) is selected.

CUT G01	RAPID G00								

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

- Thread pitch in the case of tap.
- Cutting depth in the radial direction in the case of boring bar and back boring tool.
- In the case of the centering drill, the machining cycle can be selected from the menu. In the tool automatic development mode CTR-DRIL CYCLE (centering drilling cycle) is selected. For deteils refer to Subsection 4-6-7 “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.

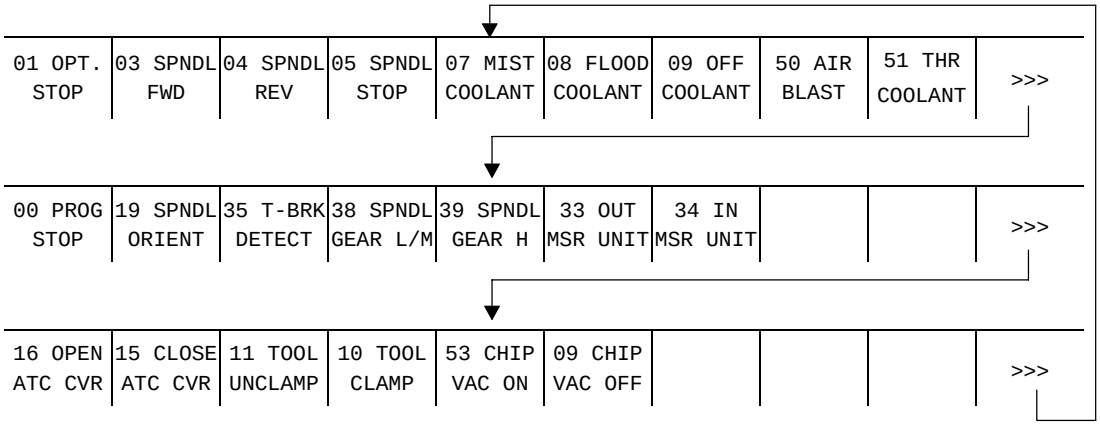
Same as the circumferential speed, the entry of data is done by means of menu keys or numeric keys.

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.

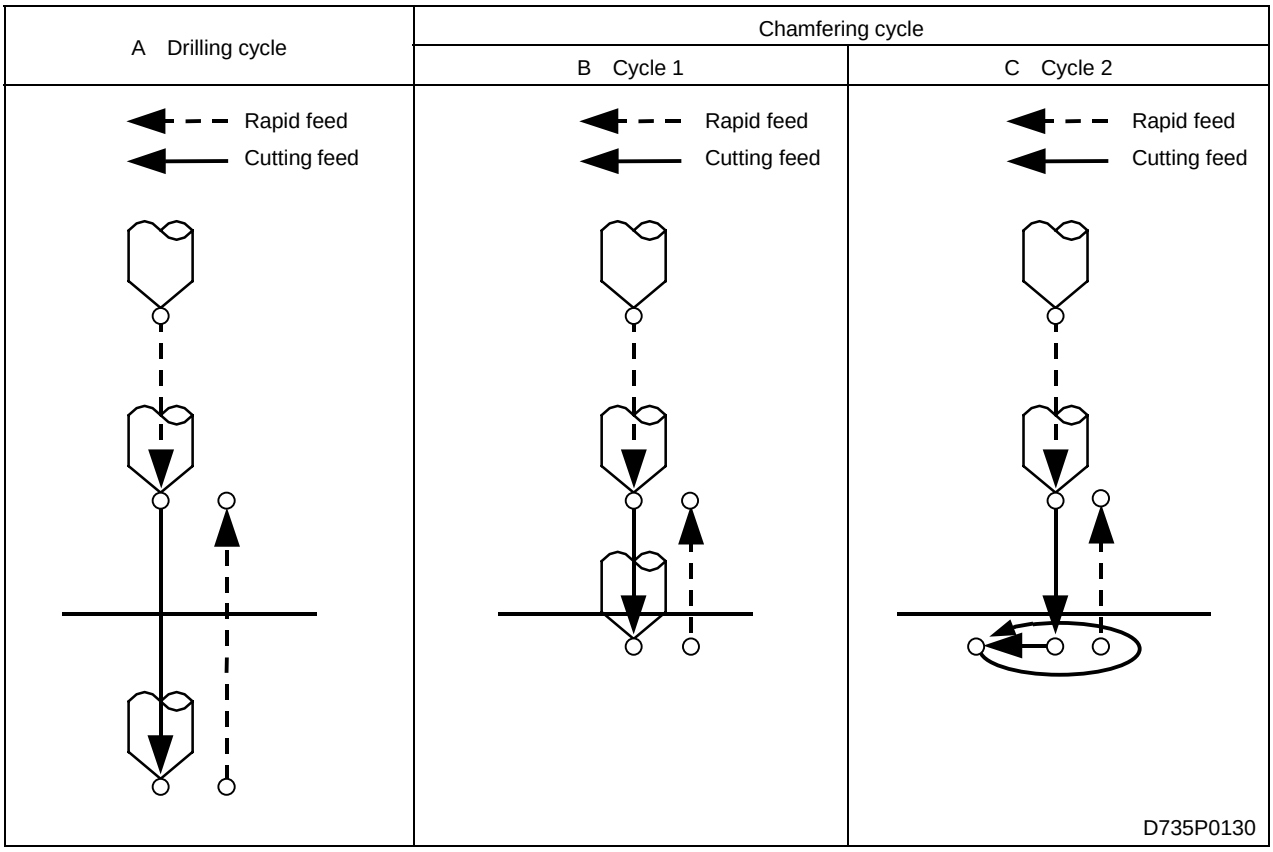


Note: In case of entry of data by the menu key, two figures of M code can be specified and in case of entry of data by the numeric keys, two figures or three figures of M code can be specified in the left or right data item respectively.

4-6-7 Tool path

1. Centering drill

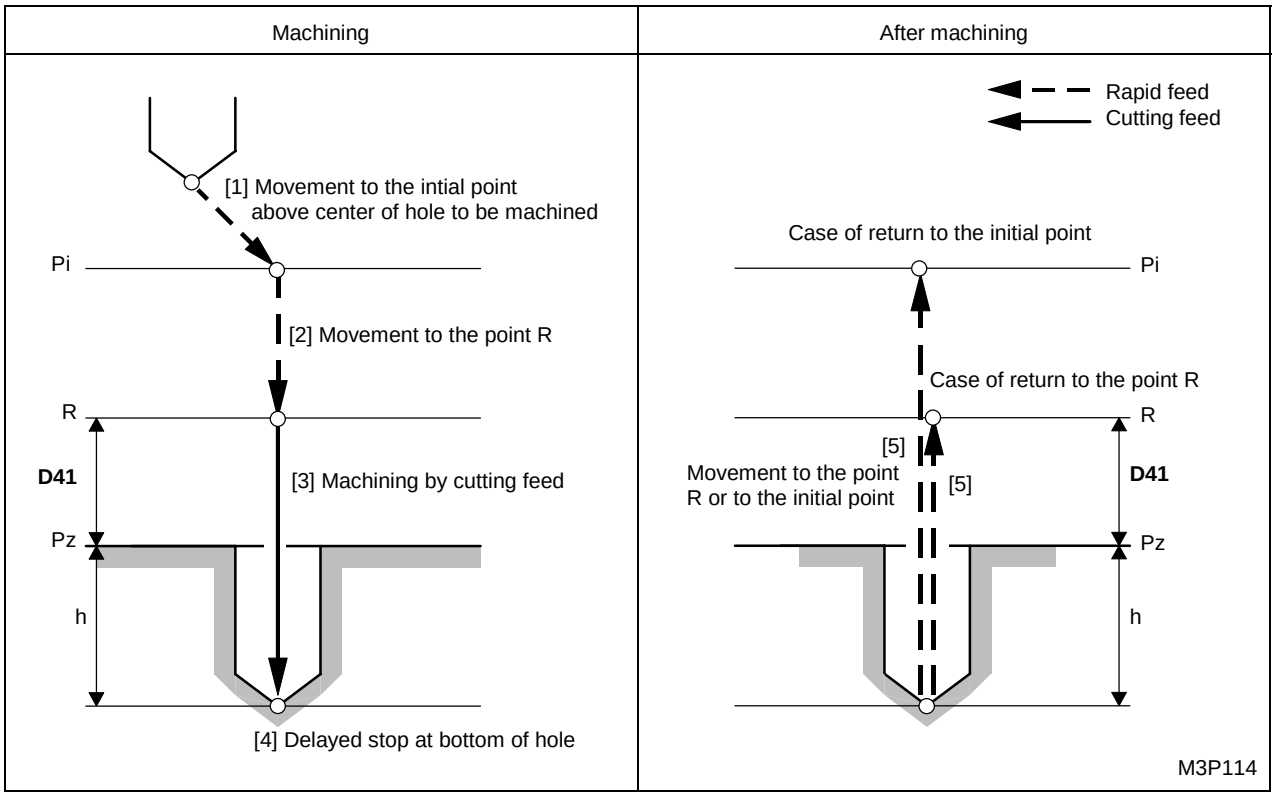
The cycle of machining with a centering drill is available in the following three types.



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 to be determined by the data INITIAL-Z entered in the common unit

Pz: Z coordinate of machining surface to be entered in the shape sequence

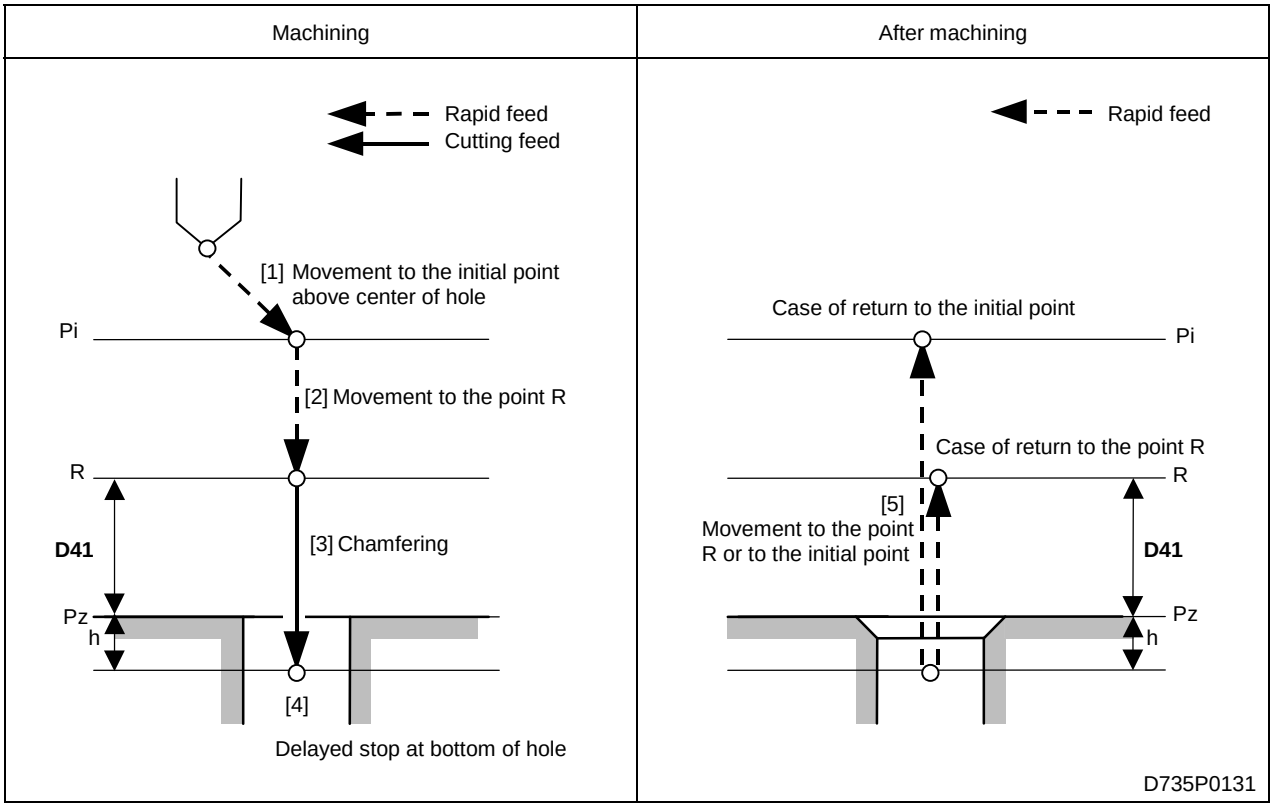
R: Safety clearance above the point Pz (parameter **D41**)

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

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

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 to be determined by the data INITIAL-Z in the common unit

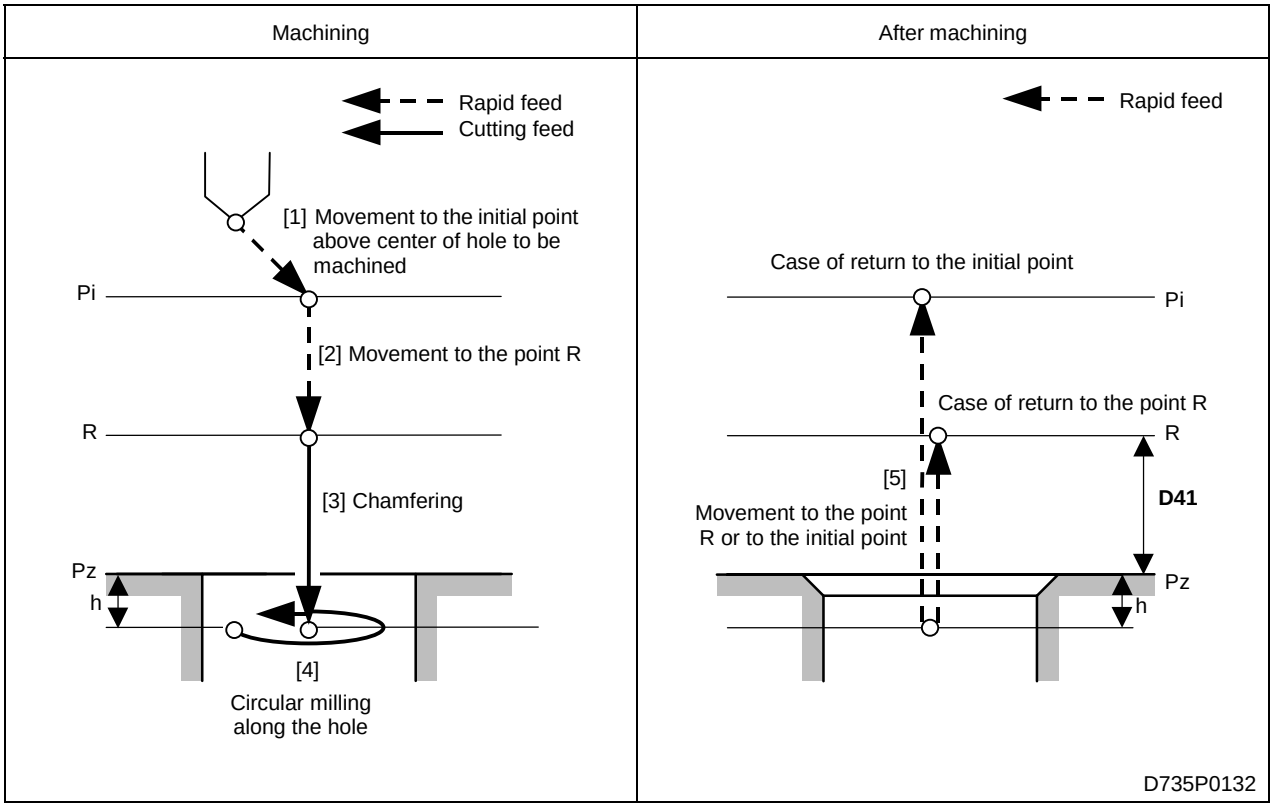
Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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 to be determined by the data INITIAL-Z entered in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

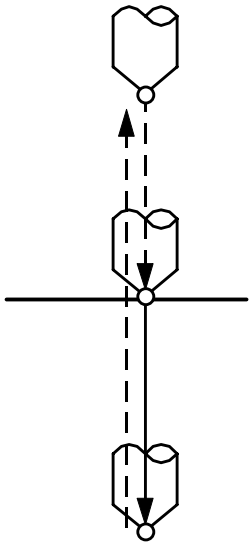
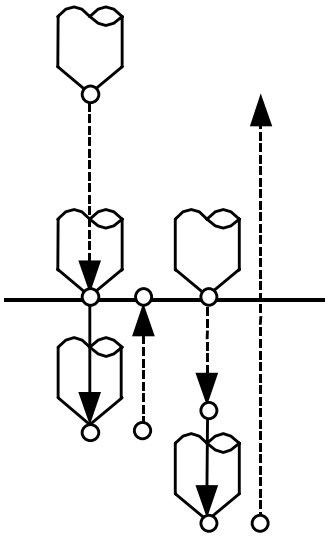
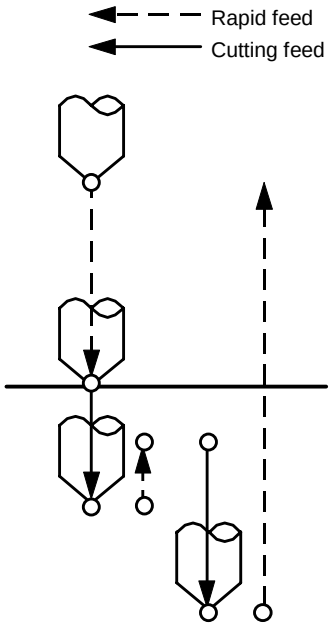
Note: When the following condition is fulfilled, the clearance R before machining will be equaled to the parameter **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.
- Case where the bit 7 of parameter **D91** is 1.

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

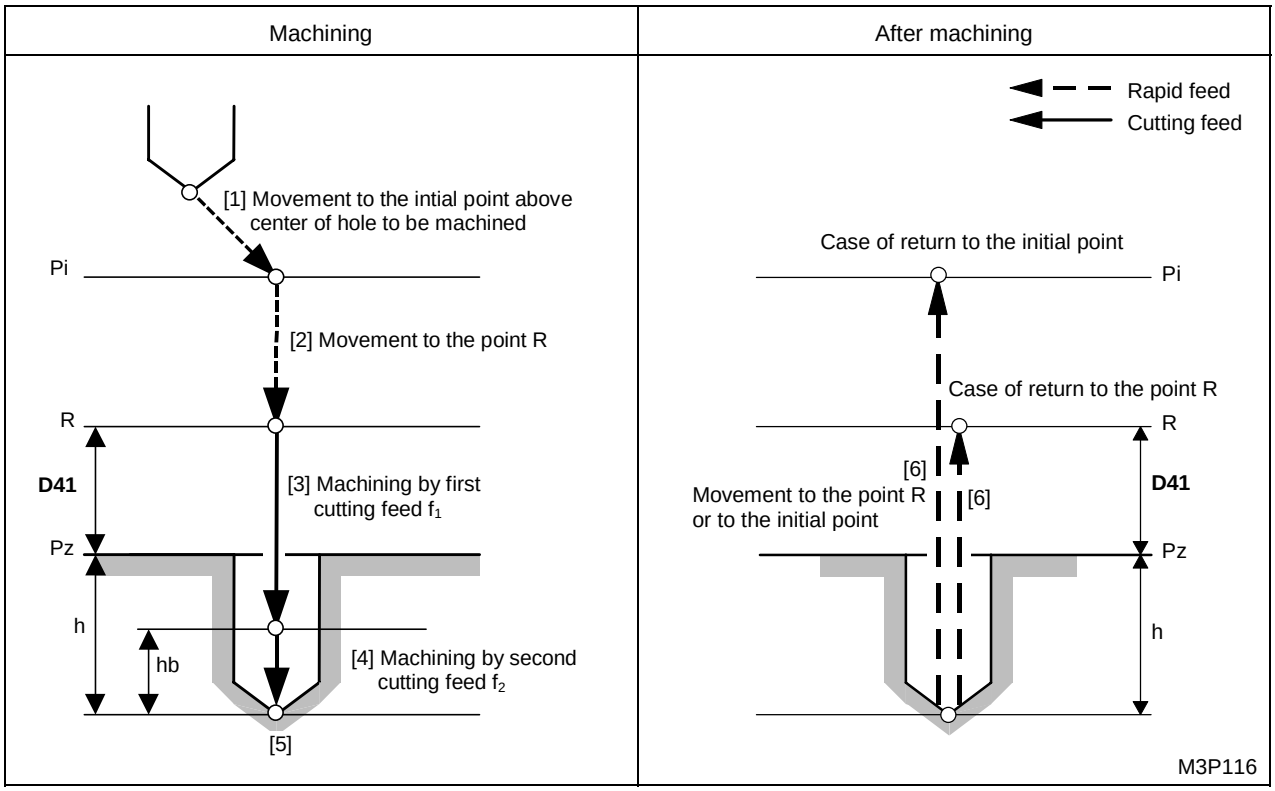
The cycle of machining with drill is available in the following six types.

A. DRILLING CYCLE	B. PECKING CYCLE 2 D. AUTOPECK CYCLE E. DECREME PECKING CYCLE 2	C. PECKING CYCLE 1 F. DECREME. PECKING CYCLE 1
		

M3P115

Remark: The cutting depths with the auto-pecking cycle of the cutting load detection type (AUTOPECK CYCLE), gradual depth reduction cycle (DECREME PECKING CYCLE 2), and rapid deep-hole machining cycle for gradual depth reduction (DECREME. PECKING CYCLE 1), can be varied. See Items A to F for the tool paths in each cycle.

A. Drilling cycle



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.

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

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

h = Depth of machining hole + Tool correction

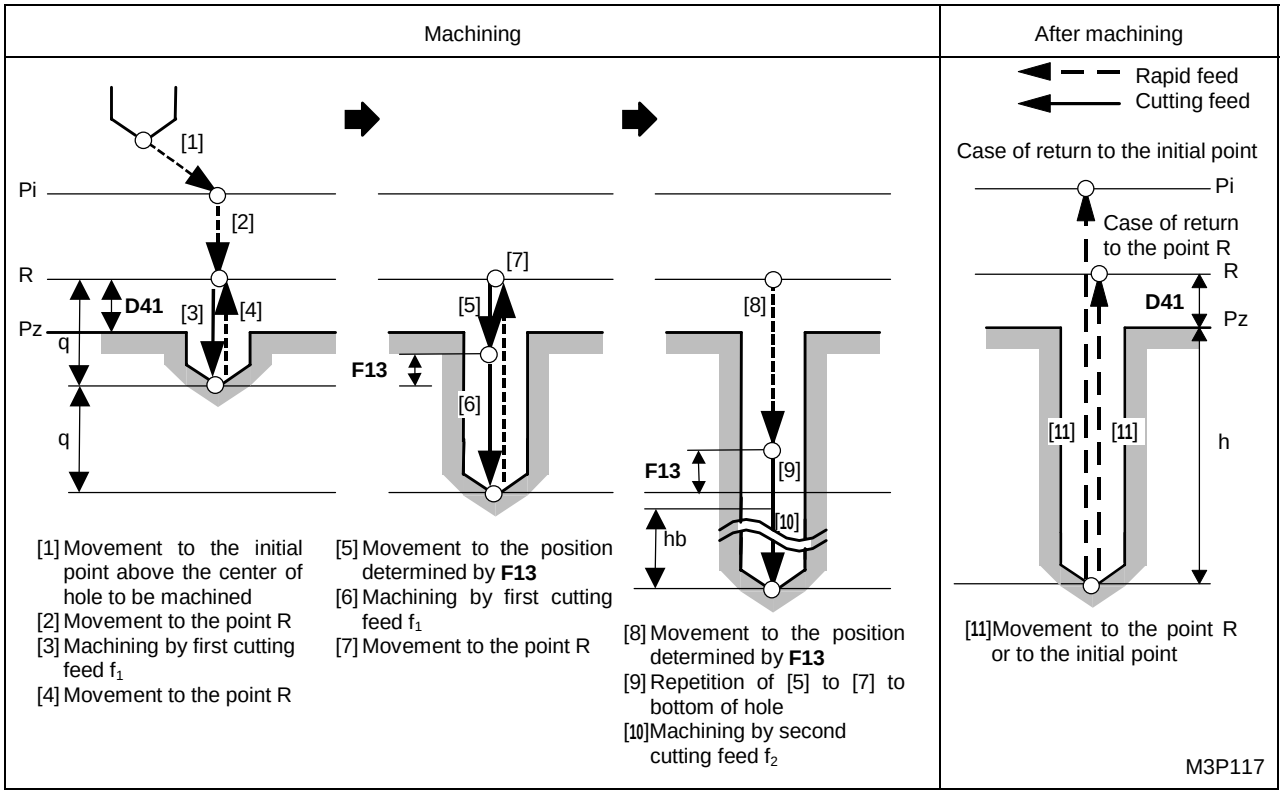
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

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

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

$f_2 = f_1 \times$ Feedrate updating rate

B. Deep hole drilling cycle (PECKING CYCLE 2)



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit.

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.
- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

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

h = Depth of machining hole + Tool correction

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

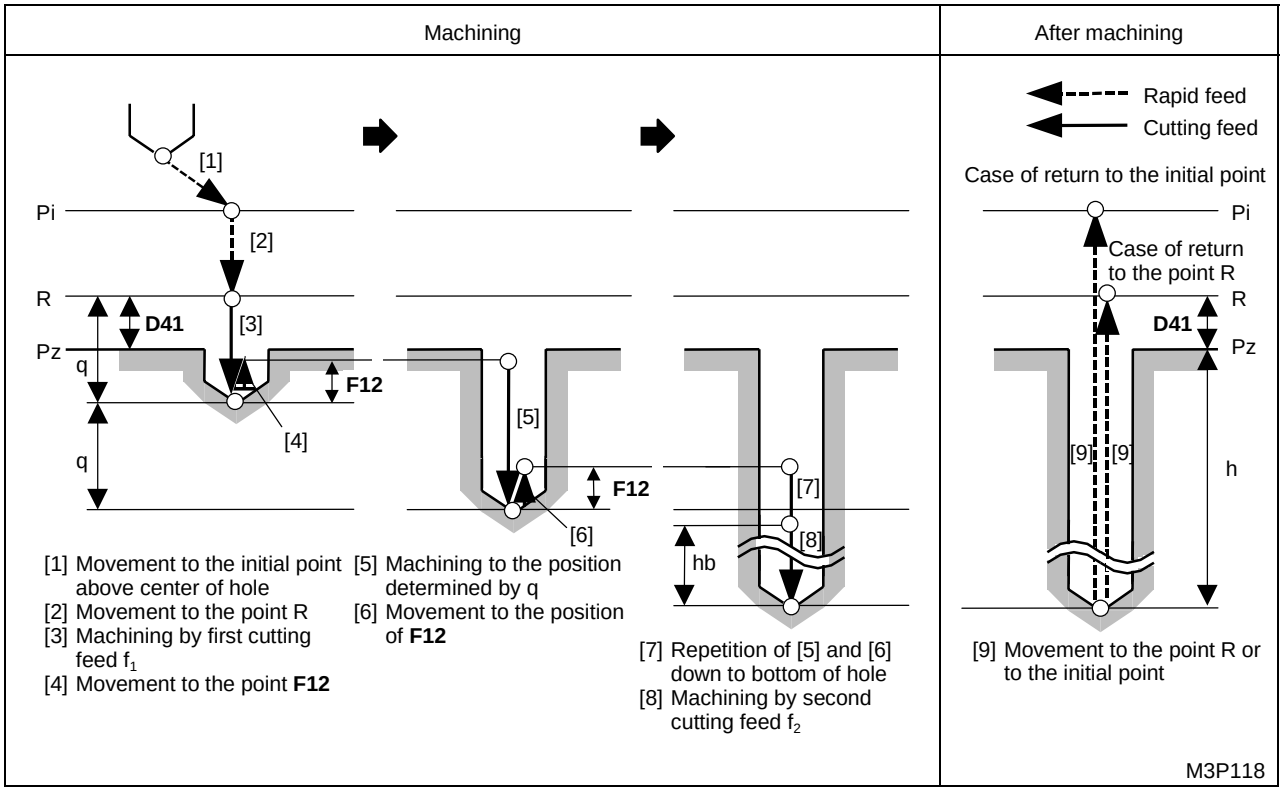
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

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

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

$f_2 = f_1 \times$ Feedrate updating rate

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



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.

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

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

h = Depth of machining hole + Tool correction

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

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

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

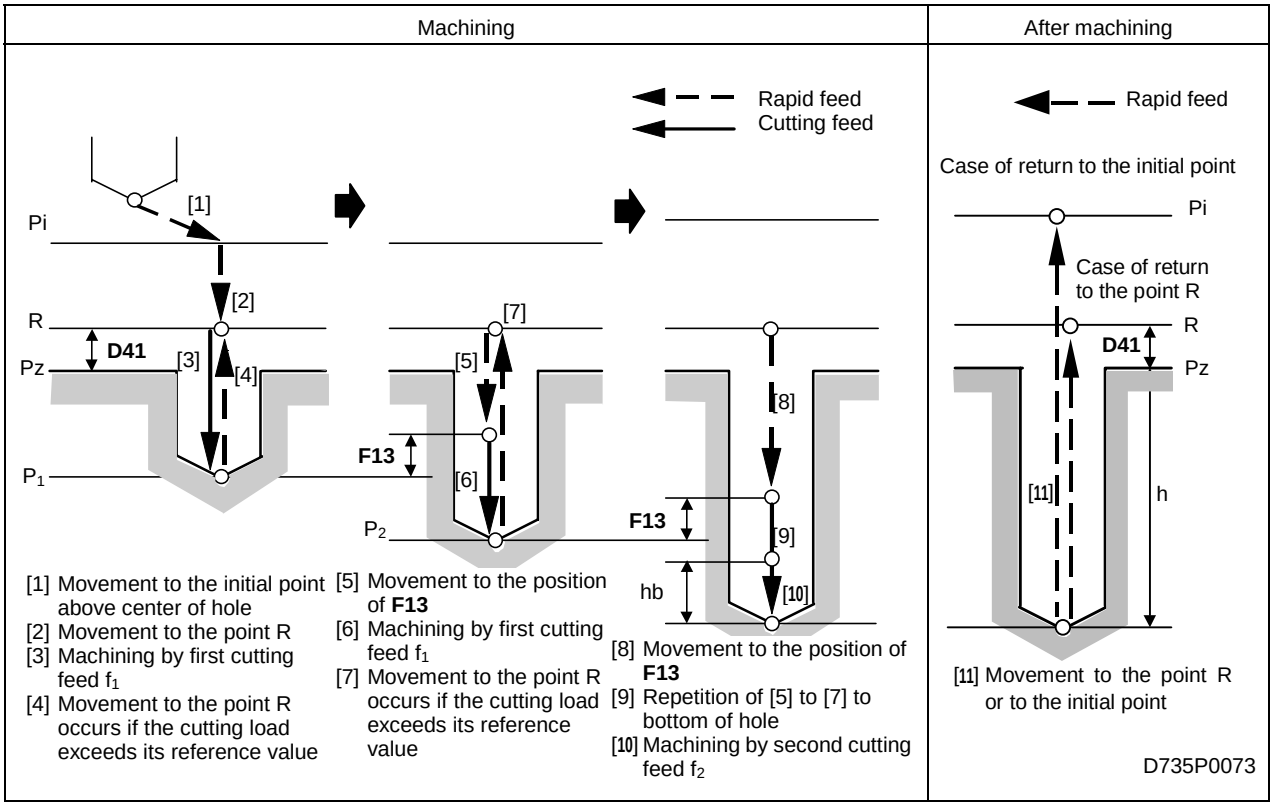
f_2: Feedrate to be modified by the data PRE-DEP (feedrate updating rate)

$f_2 = f_1 \times$ Feedrate updating rate

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

D. 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 to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

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

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.

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

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

h = Depth of machining hole + Tool correction

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

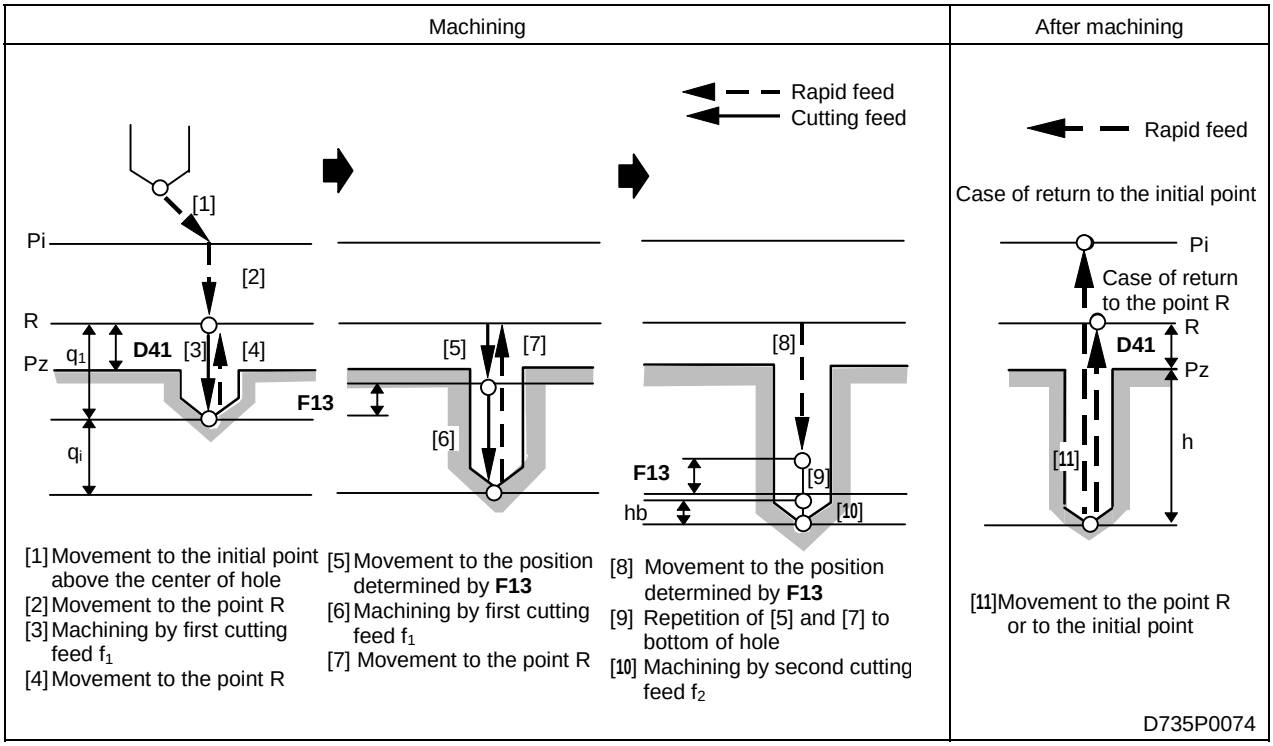
f_1: Feedrate (FR) to be set for the tool sequence

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

$f_2 = f_1 \times$ Feedrate updating rate

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

E. Gradual depth reduction cycle (DECREME PECKING CYCLE 2)



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit.

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

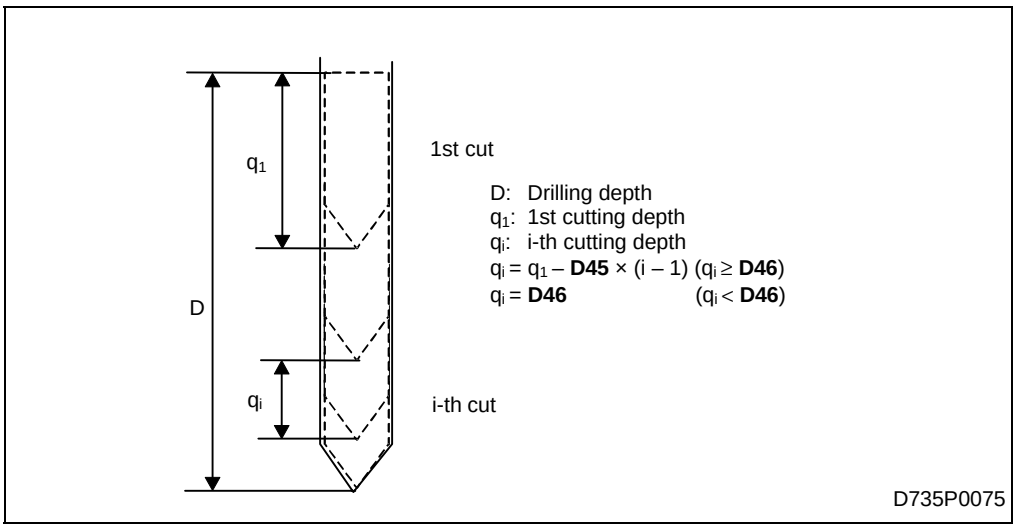
Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.
- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

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

h = Depth of machining hole + Tool correction

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

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



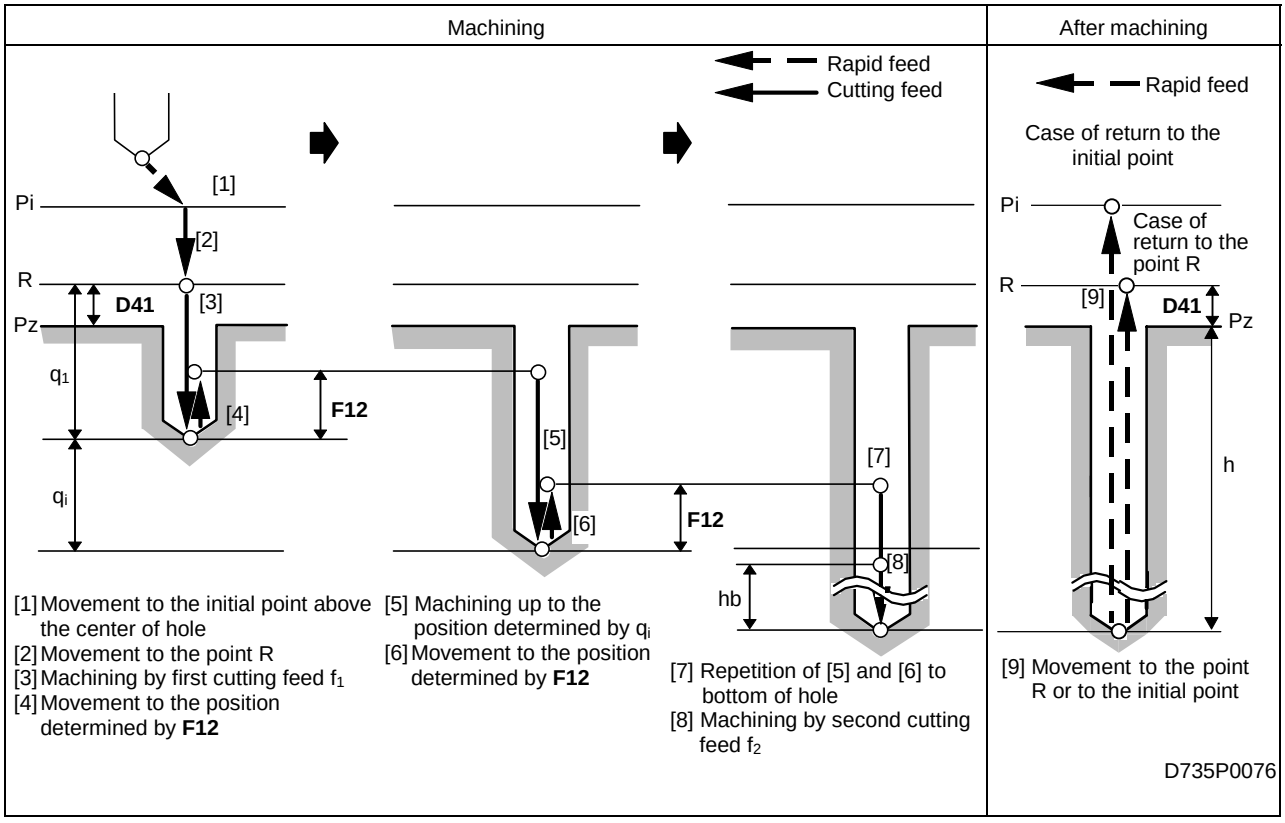
hb: Feedrate override distance from the hole bottom to be determined by the data PRE-DIA to be set for the tool sequence

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

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

$f_2 = f_1 \times$ Feedrate updating rate

F. Rapid deep-hole machining cycle for gradual depth reduction (DECREME PECKING CYCLE 1)



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit.

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

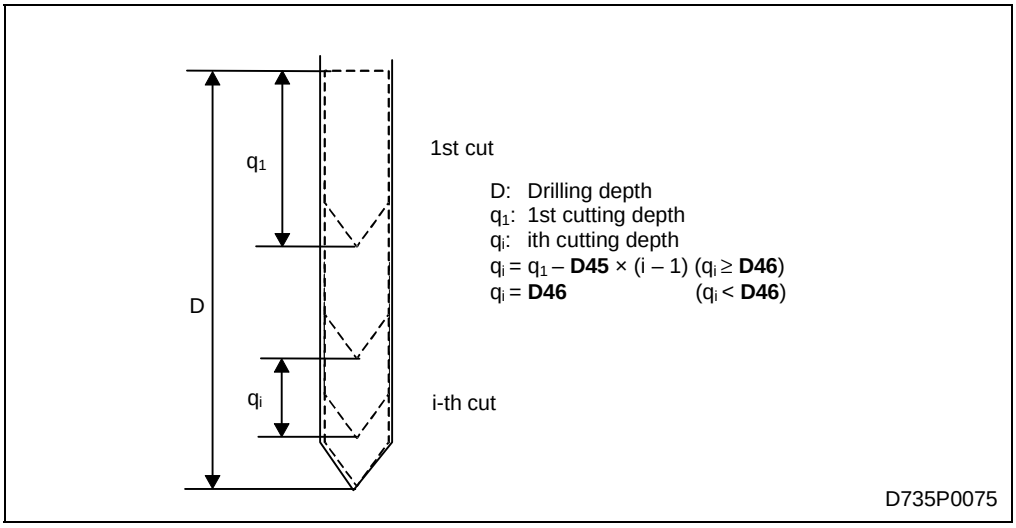
Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1** or **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.
- Case where the bit 6 of parameter **D91** is 1.
- Case where the respective tool sequence contains a centering drill (**D1**) or a drill (**D42**) as pre-machining tool.

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

h = Depth of machining hole + Tool correction

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

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



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

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

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

$f_2 = f_1 \times$ Feedrate updating rate

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

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.

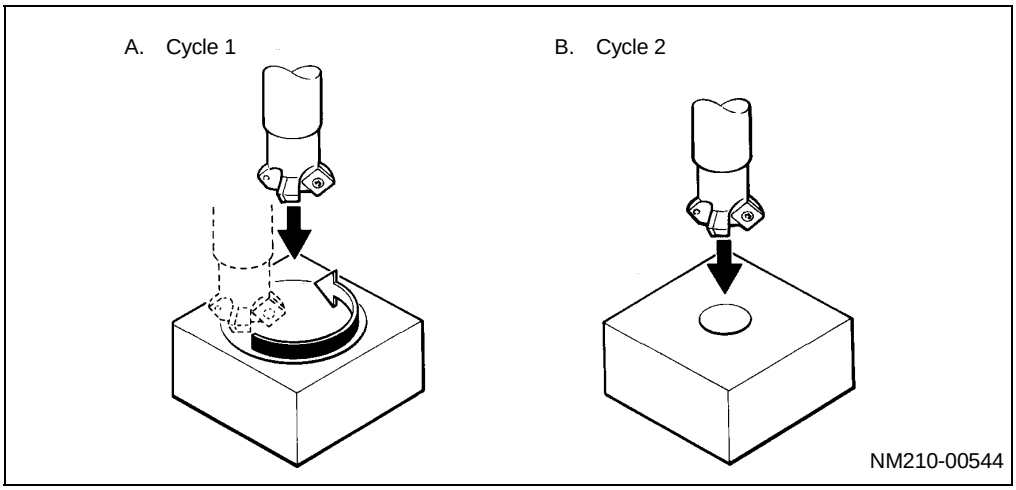
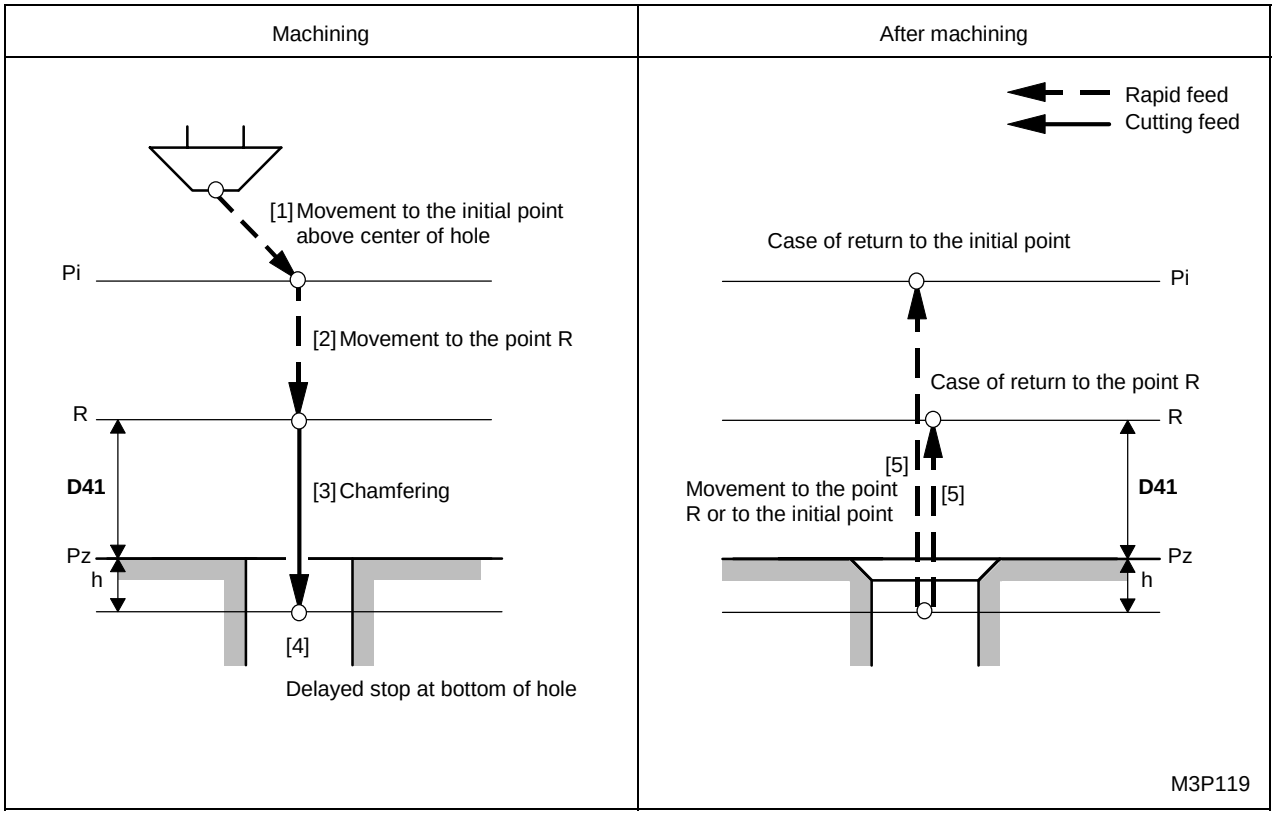


Fig. 4-4 Cycle 1 and cycle 2

A. Cycle 1



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

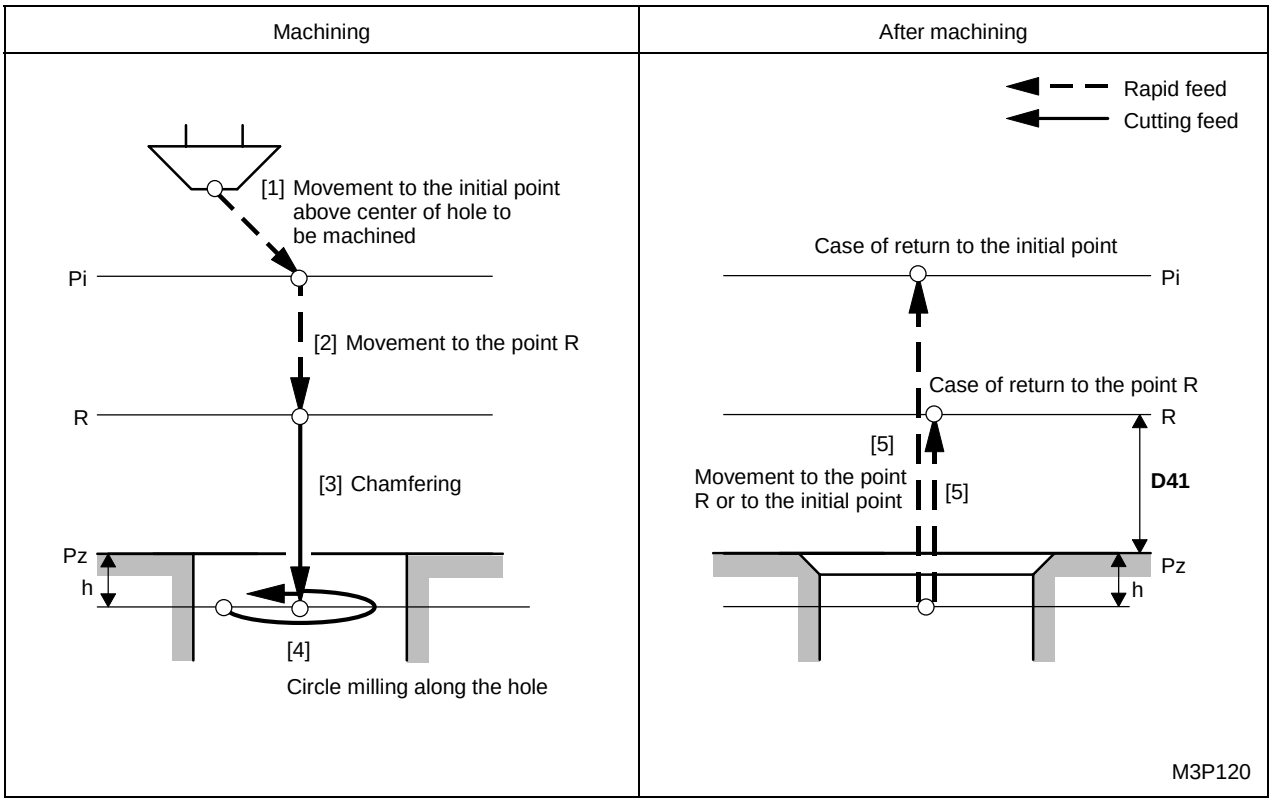
Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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 to be determined by the data INITIAL-Z entered in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following condition is fulfilled, the clearance R before machining will be equaled to the parameter **D42**.
However, clearance R after machining is always equaled to the parameter **D41**.
- Case where the bit 7 of parameter **D91** is 1.

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

Note 1: The time of delayed stop of the feed on Z axis at bottom of hole is set by the parameter **D16**.

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

4. End mill

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

TORNA.: 0..... circular milling cycle

1..... circular tornado milling cycle

2..... precision rapid boring tornado cycle

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

<In case of circular milling cycle>

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

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

1. For RGH CBOR and CBOR-TAP units
   - Diameter of machining hole = Nominal diameter (Cycle 1)
   - “Diameter of machining hole > Nominal diameter” and “Diameter of pre-hole > (Tool diameter + Safety clearance)” (Cycle 2)
   - “Diameter of machining hole > Nominal diameter” and “Diameter of pre-hole ≤ (Tool diameter + Safety clearance)” (Cycle 3)

2. For units other than those mentioned above
   - Diameter of machining hole = Tool diameter (Cycle 1)
   - “Diameter of machining hole > Tool diameter” and “Diameter of pre-hole > (Tool diameter + Safety clearance)” (Cycle 2)
   - “Diameter of machining hole > Tool diameter” and “Diameter of pre-hole ≤ (Tool diameter + Safety clearance)” (Cycle 3)

Note: The safety clearance is determined by the parameter **D23**.

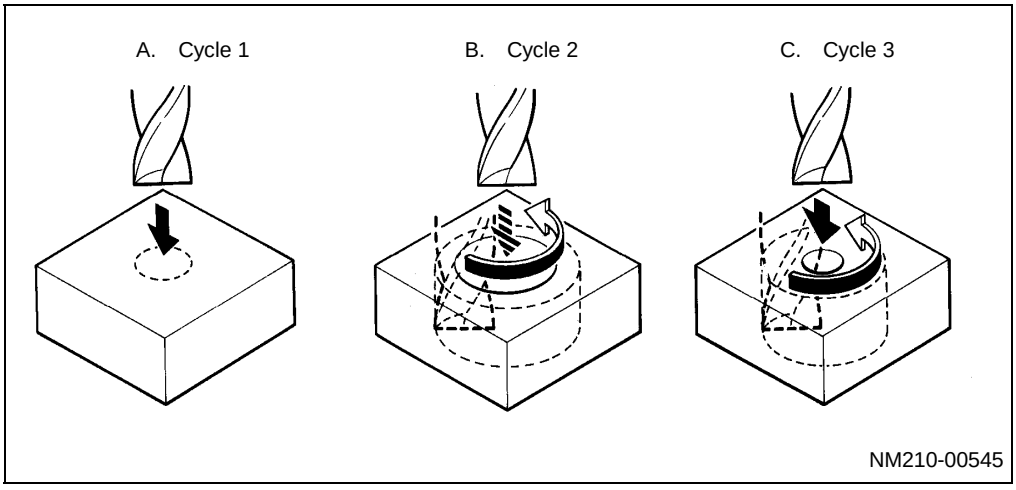
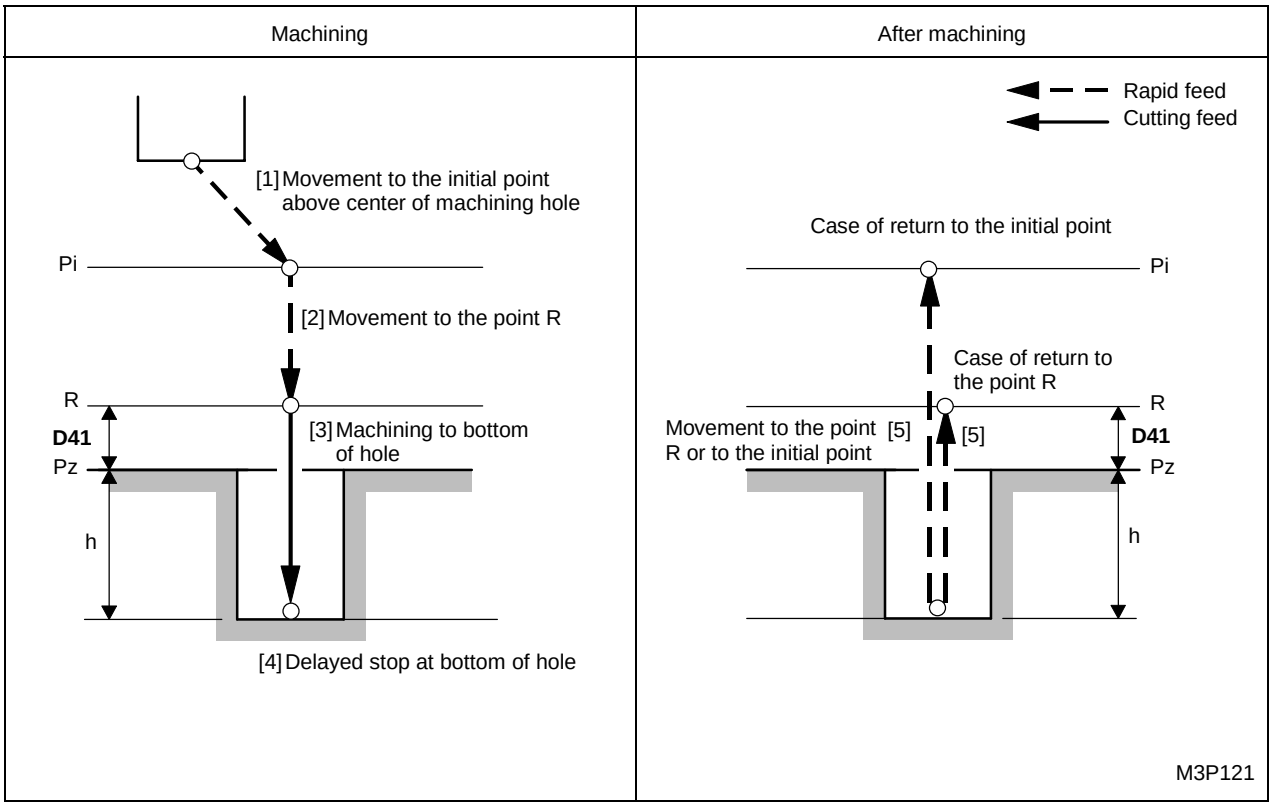


Fig. 4-5 Circular milling cycles 1, 2 and 3

A. Cycle 1



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

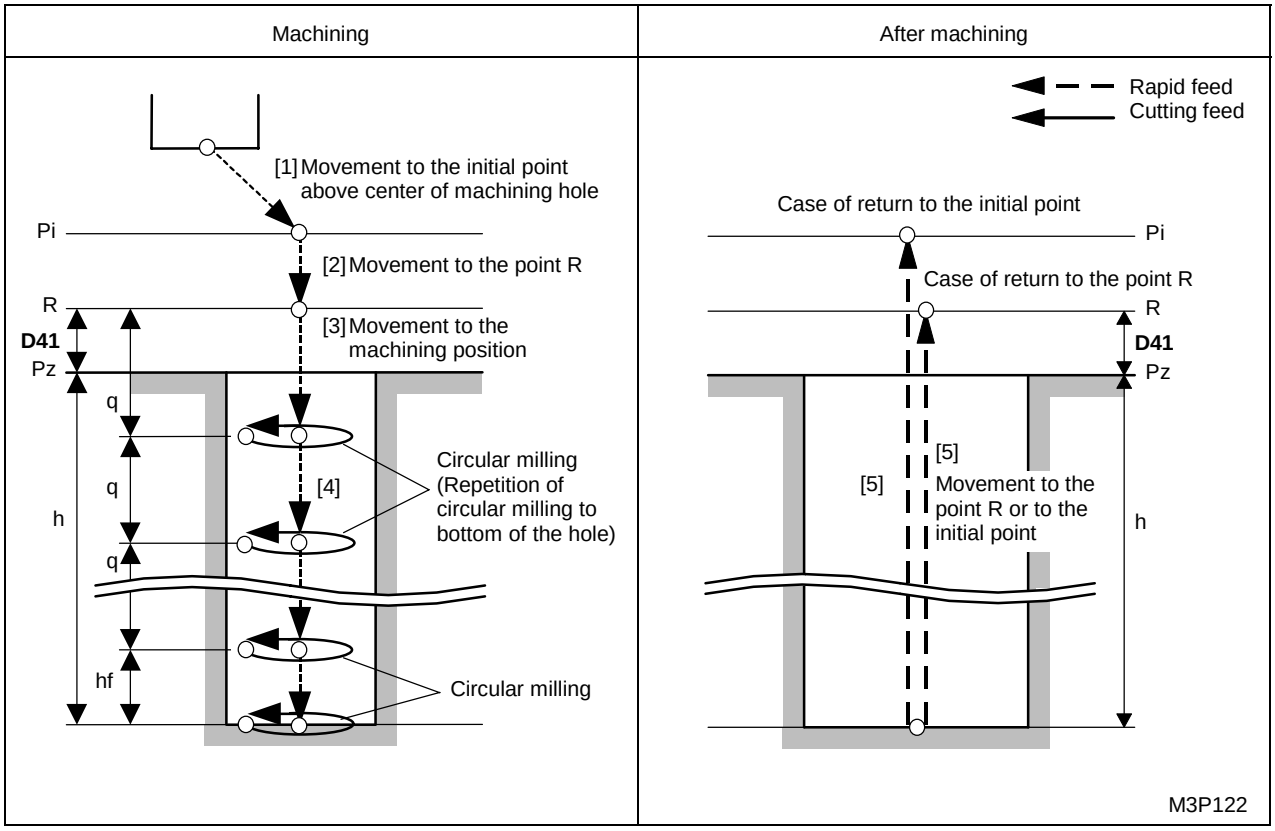
Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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**

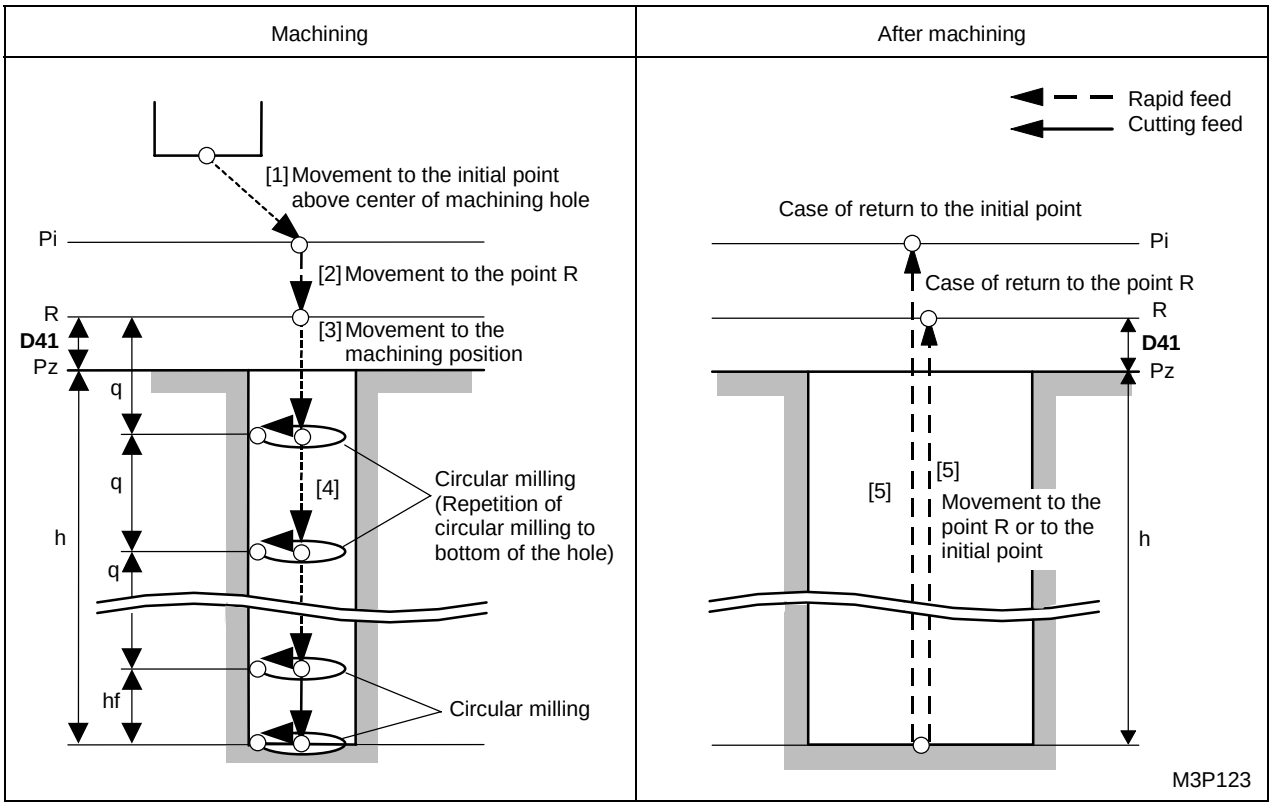
q: Cutting depth on Z per pass to be determined by:

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

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

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

C. Cycle 3



Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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.

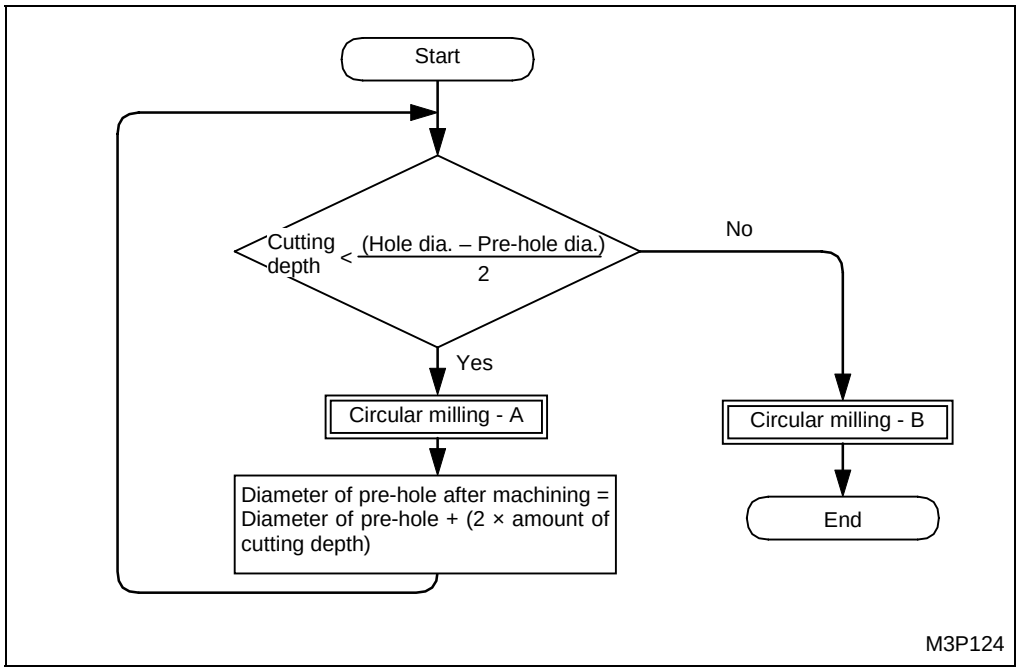


Fig. 4-6 Circular milling

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.

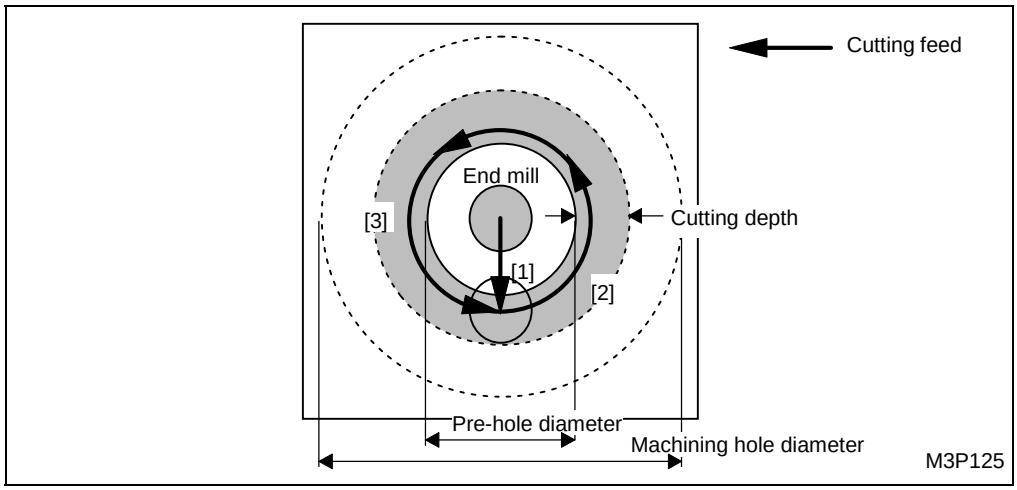


Fig. 4-7 Circular milling-A

- 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.

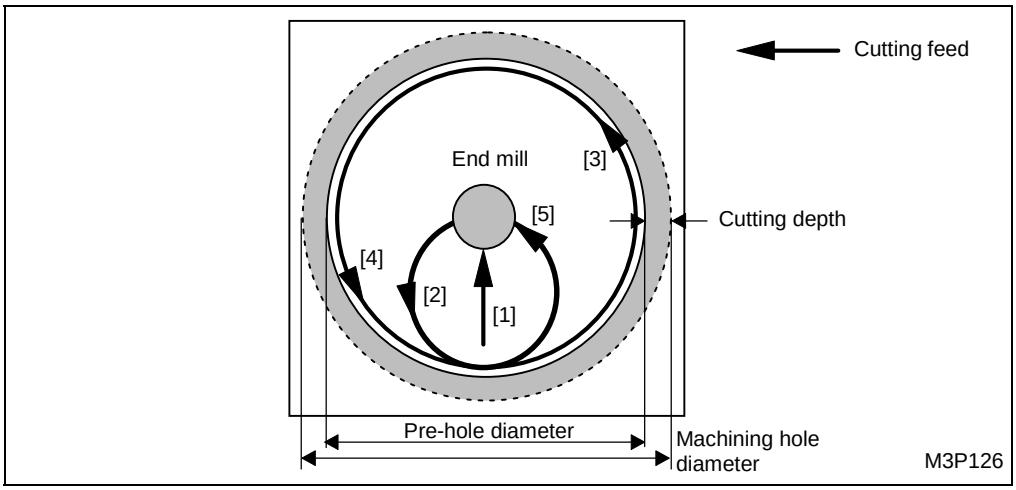


Fig. 4-8 Circular milling-B

- 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.

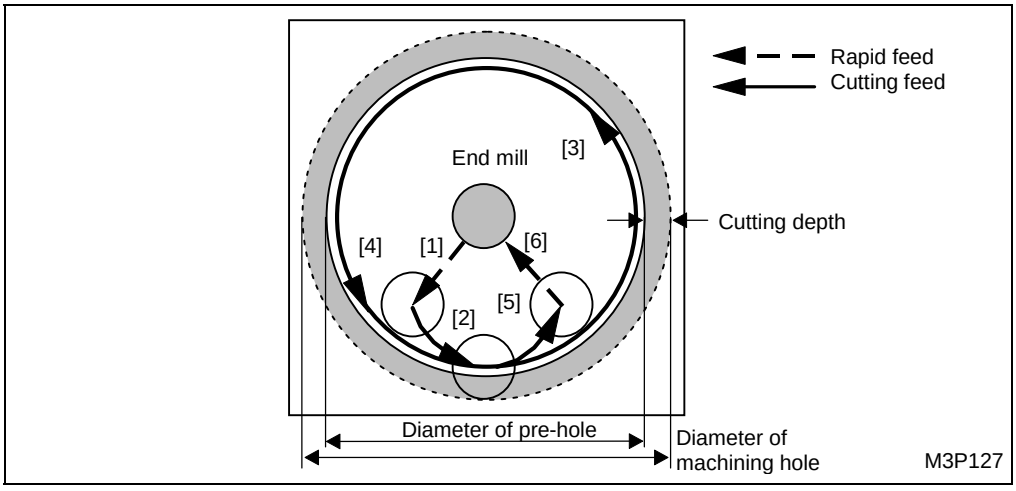
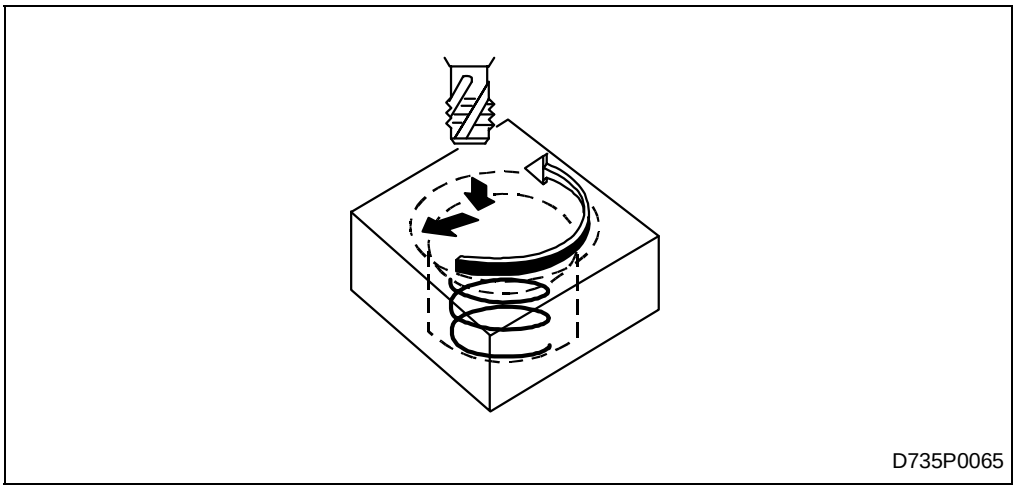
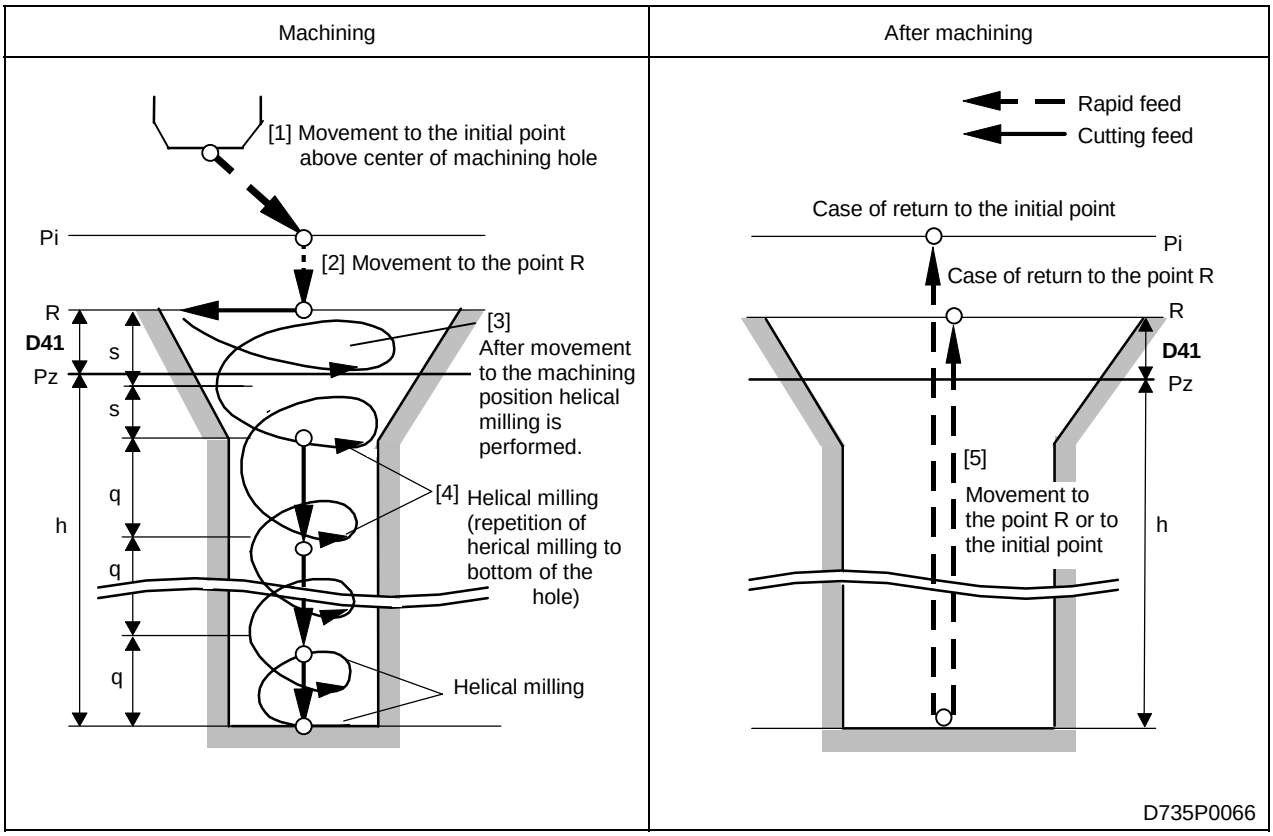


Fig. 4-9 Circular milling-B (case of shortening in chamfering)

- 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>





The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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.

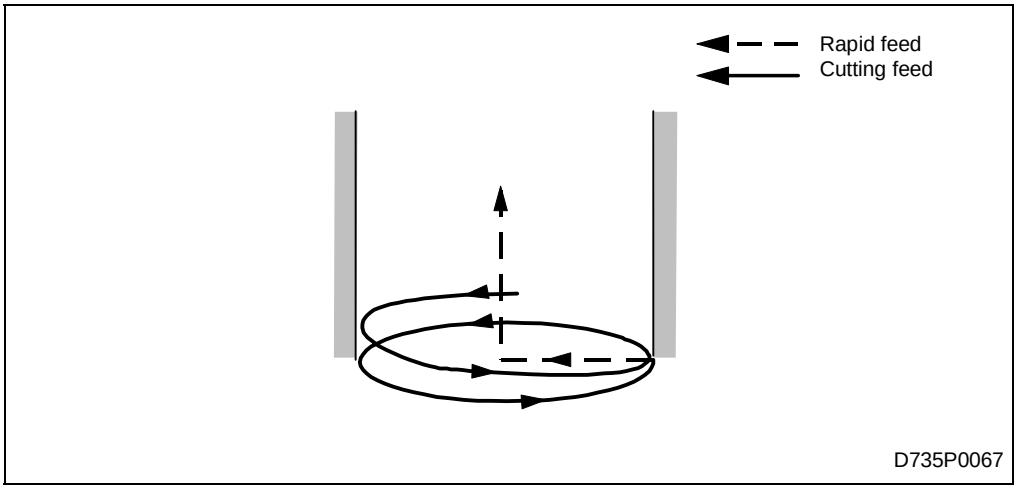


Fig. 4-10 Circular helical processing (with bottom finishing)

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 point R 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.

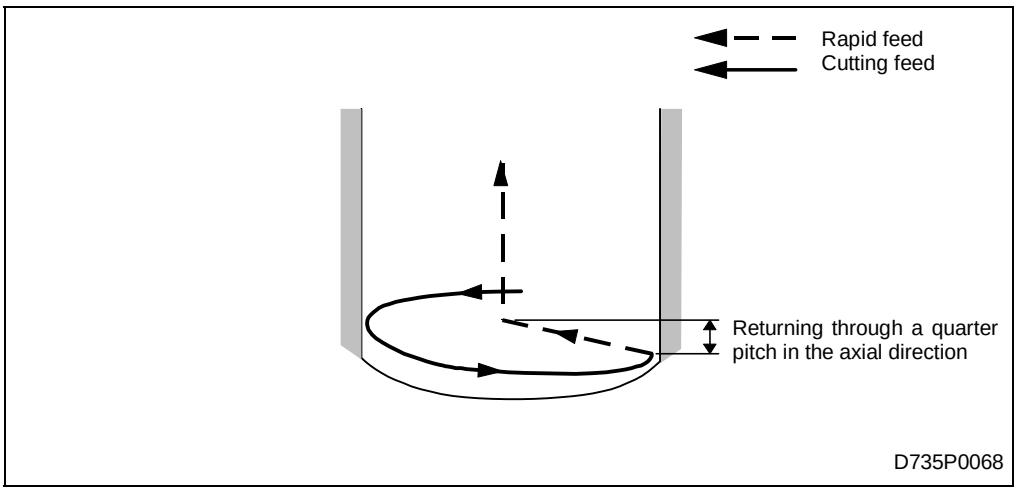
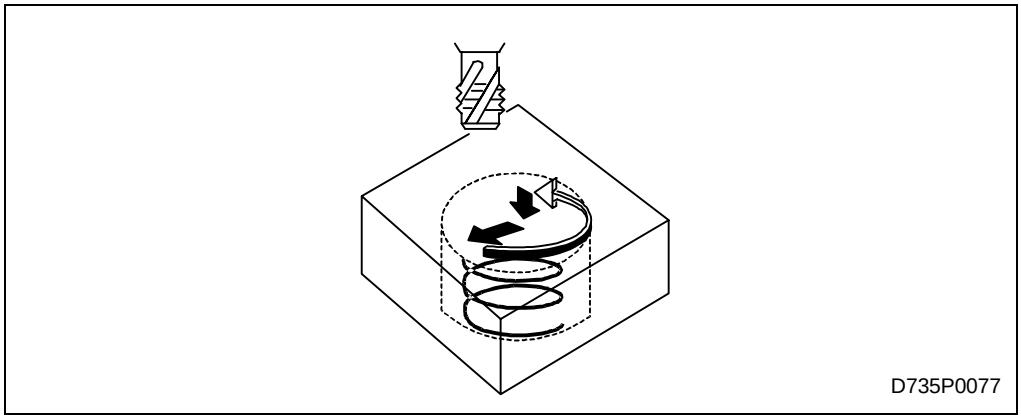


Fig. 4-11 Circular helical processing (without bottom finishing)

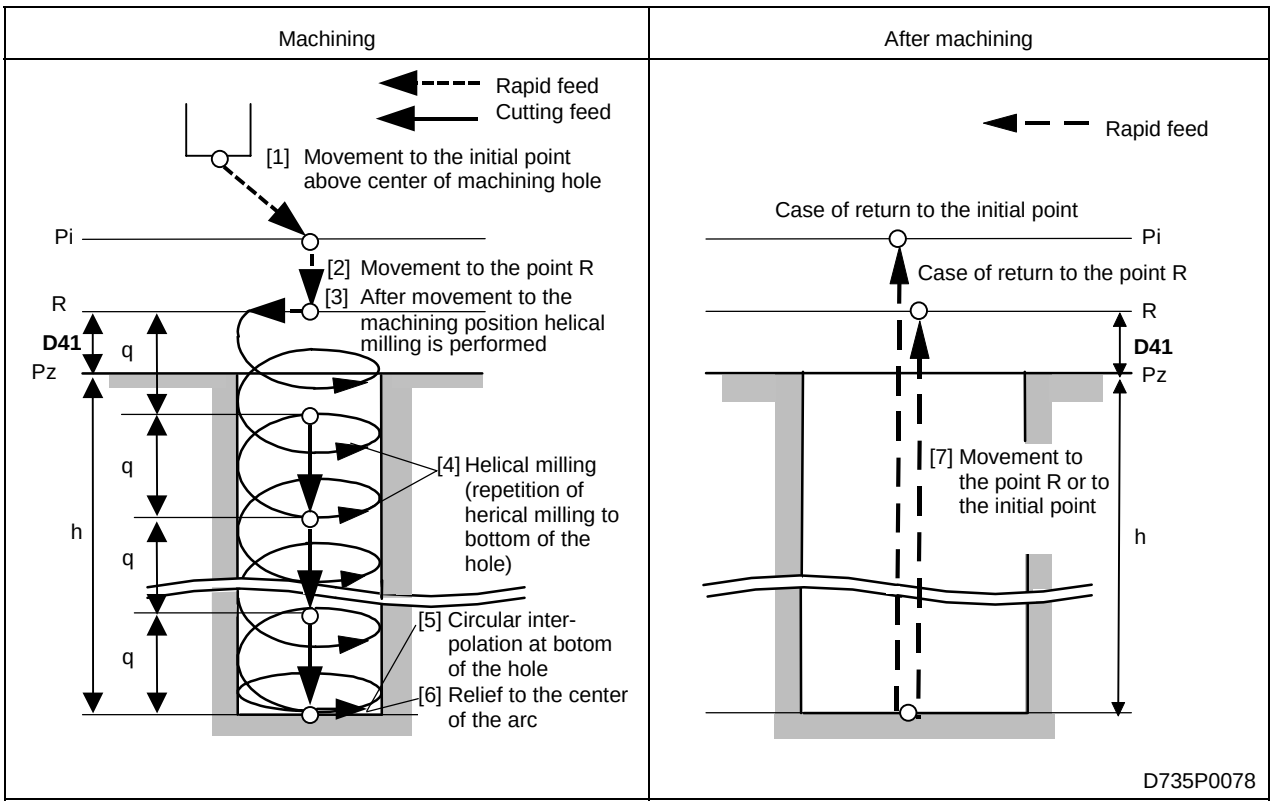
After helical interpolation down to the bottom of the hole, the tool moves to the center of the hole by returning through a quarter pitch in the axial direction and then moves in rapid feed rate to its initial point or to point R in the axial direction.

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 to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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.

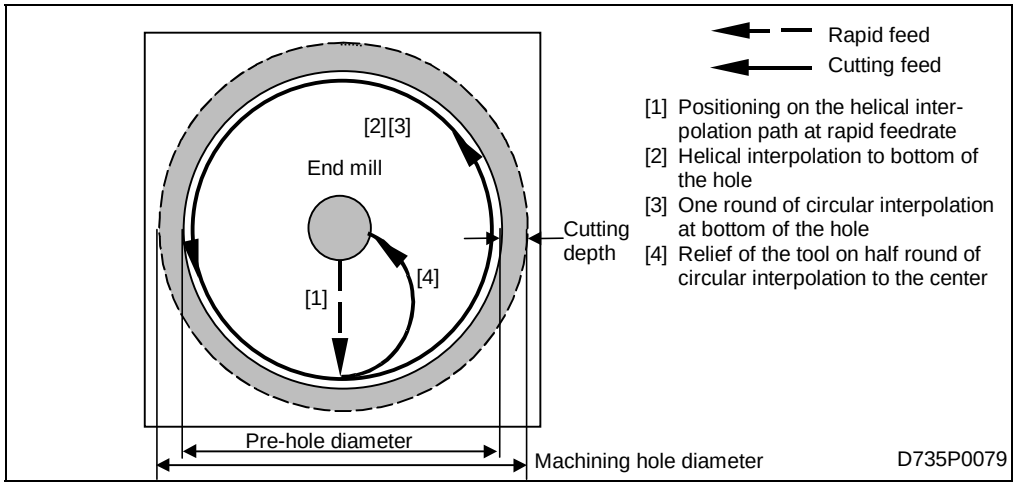


Fig. 4-12 Circular milling

- 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.

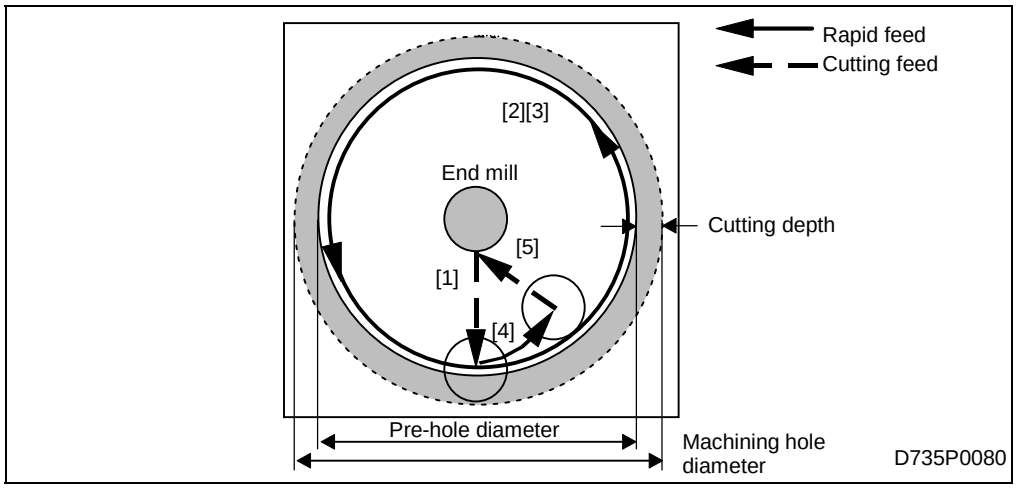
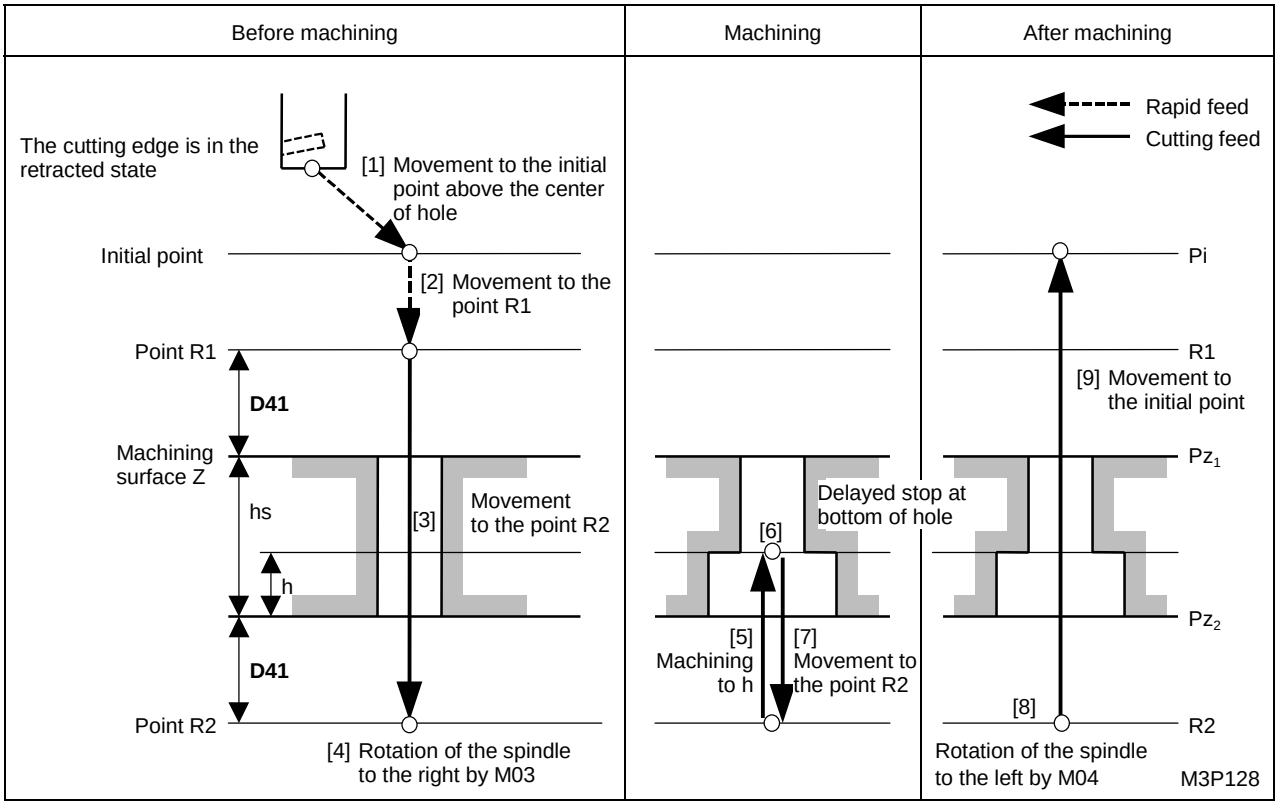


Fig. 4-13 Circular milling (case of shortening in chamfering)

- 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 to be determined by the data INITIAL-Z entered in the common unit

Pz_1: Z coordinate of the machining surface to be entered in the shape sequence

Pz_2: Position at a distance of hs from Pz_1

R1, R2: Safety clearance above the points Pz_1, Pz_2 respectively (parameter **D41**)

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 **D41**.

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

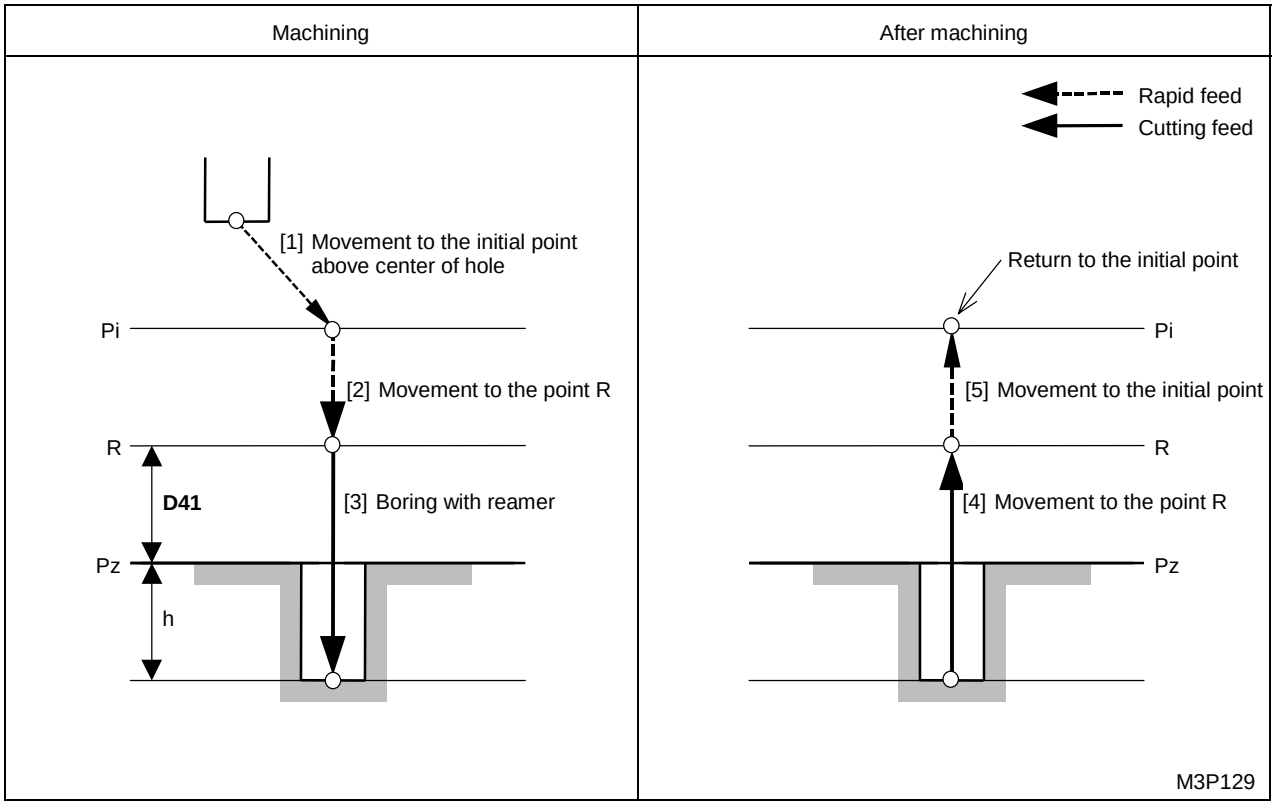
hs: Distance equal to the sum of the depth of pre-hole entered in the tool sequence and the tool data COMP. (tool correction) on the **TOOL DATA** display

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

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

Note 3: The rotation of the spindle to the right is performed by M03 entered in the tool sequence, whilst the rotation to the left is performed by M04.

6. Reamer



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1**.
However, clearance R after machining is always equaled to the parameter **D41**.

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

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

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

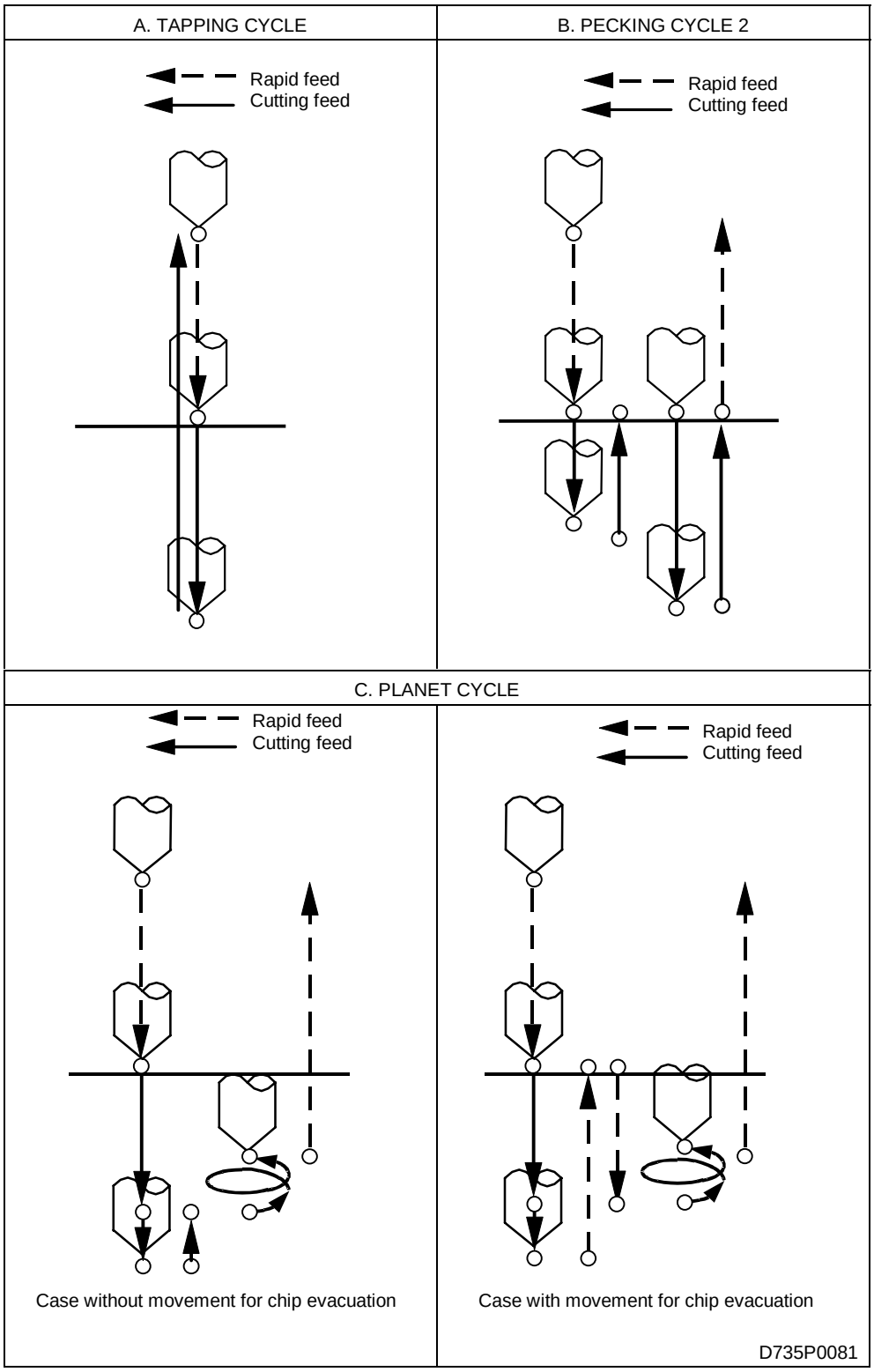
When menu key **CUT G01** is pressed ..... Parameter **D18**

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

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

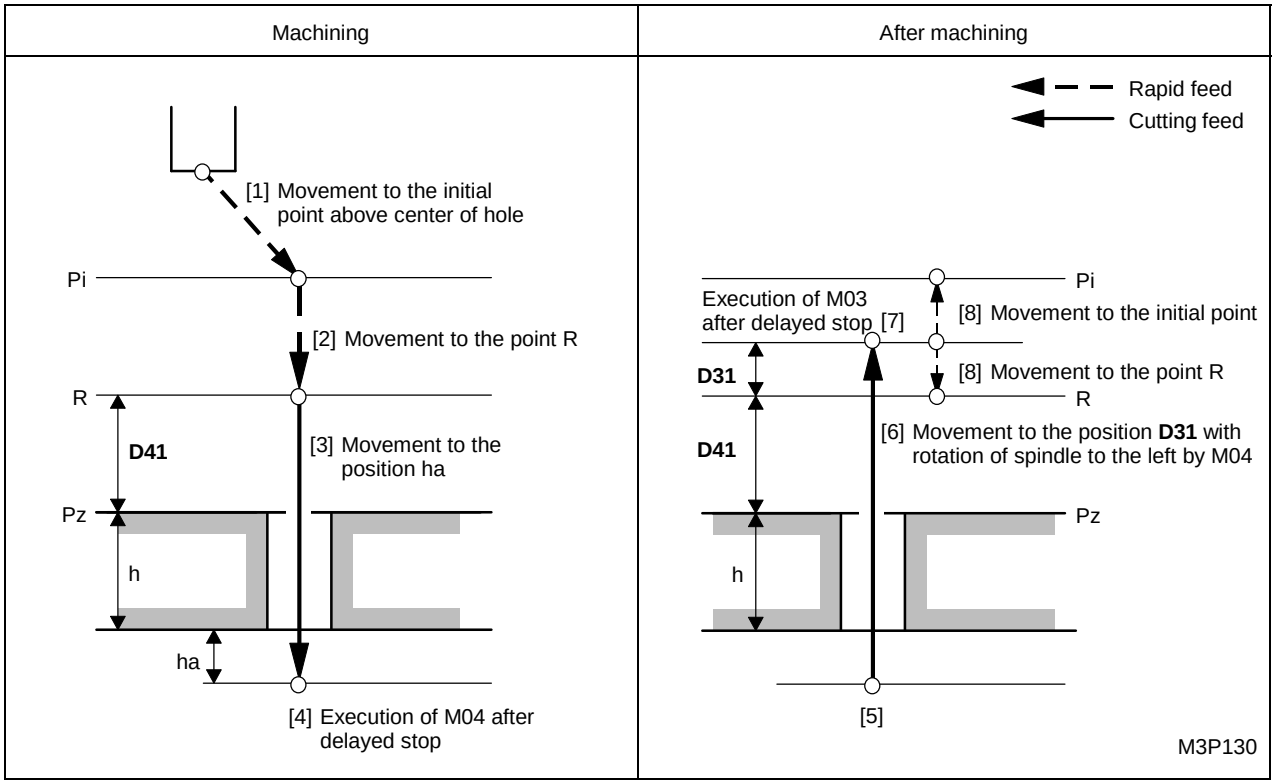
7. Tap

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



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

A. Tapping cycle (TAPPING CYCLE)



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1**.
However, clearance R after machining is always equaled to the parameter **D41**.
- 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 COMP. (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

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

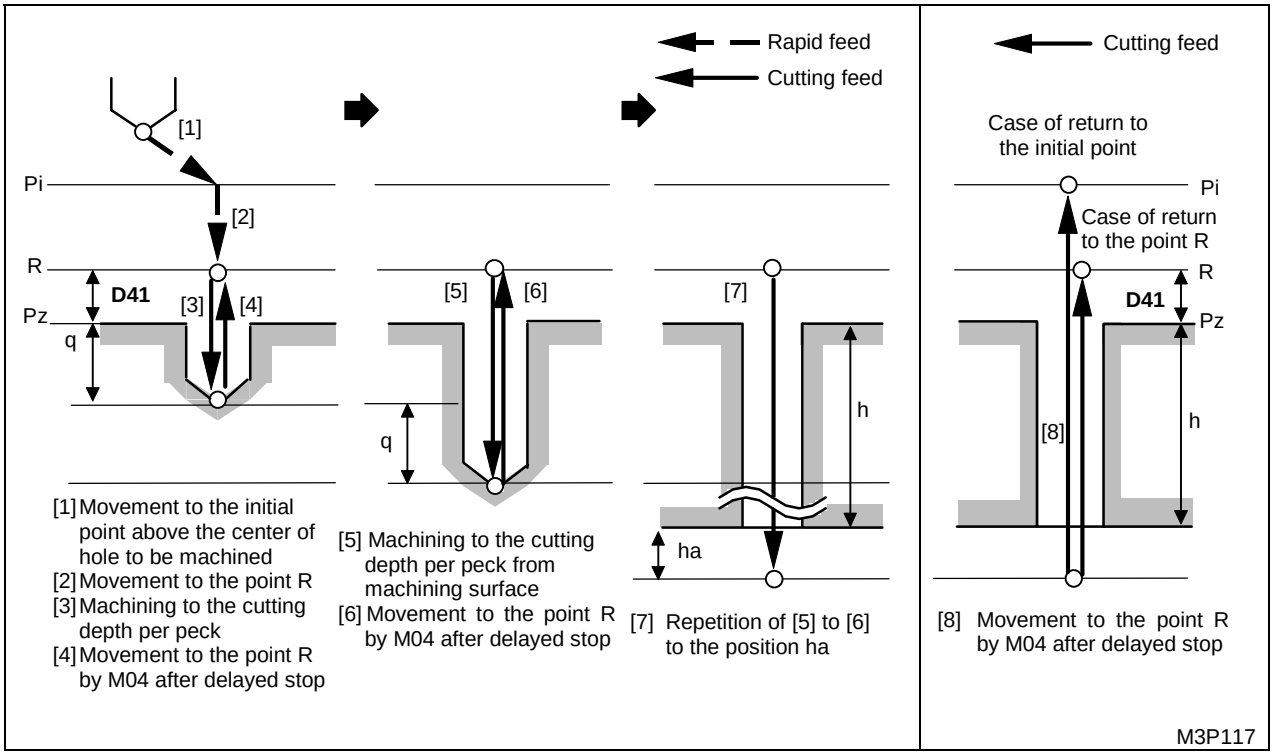
A .....**D30** when using metric and unified screws, **D43** when using pipe screws

Pt .....Pitch entered in the machining unit

Note 1: The entry of 1 in the bit 0 to 2 of parameter **D91** causes the following delayed stop.
Bit 0 ..... Delayed stop before execution of M04 at bottom of hole [4]
Bit 1 ..... Delayed stop after execution of M04 at bottom of hole [5]
Bit 2 ..... Delayed stop before execution of M03 after return [7]
Moreover, the delayed stop is entered in the article 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 to be determined by the data INITIAL-Z in the common unit.

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

Note: When the following two conditions are fulfilled, the clearance R before machining will be equaled to the parameter **D1**.
However, clearance R after machining is always equaled to the parameter **D41**.
- 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 COMP. (tool correction) on the **TOOL DATA** display

h = Depth of machining hole + Tool correction

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

A .....**D30** when using metric and unified screws, **D43** when using pipe screws

Pt .....Pitch entered in the machining unit

q: Cutting depth 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 article RGH in the tool sequence. If CYCLE FIX is selected, it will be determined by the parameter **D22**.

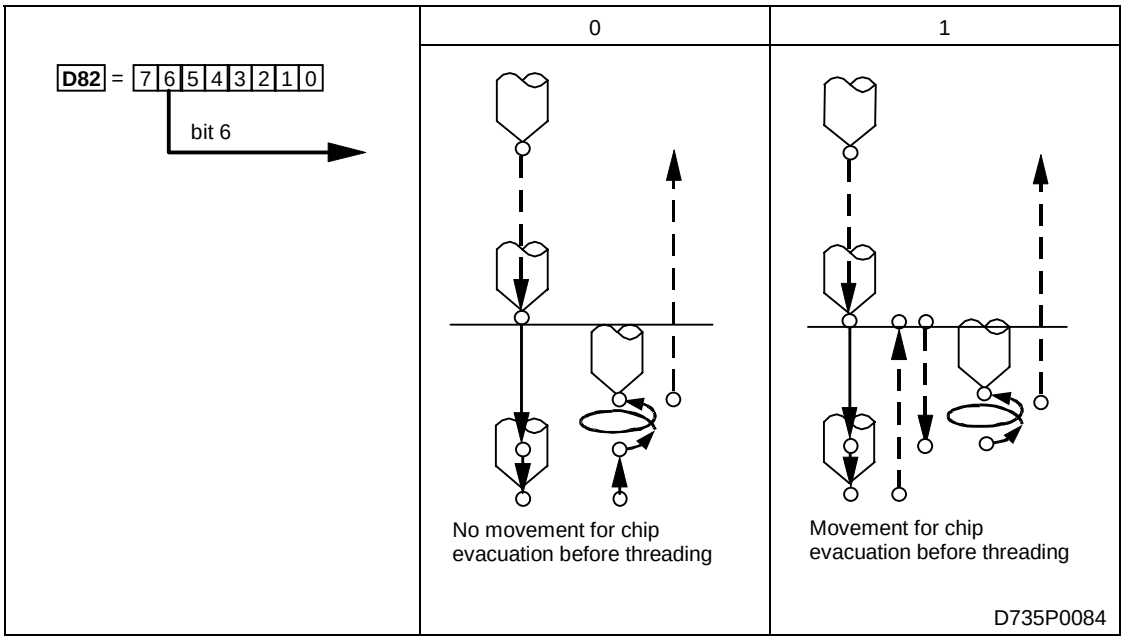
Note 2: When M04 is entered in the tool sequence, inversed tapping will be executed.

C. Planetary tapping (PLANET CYCLE)

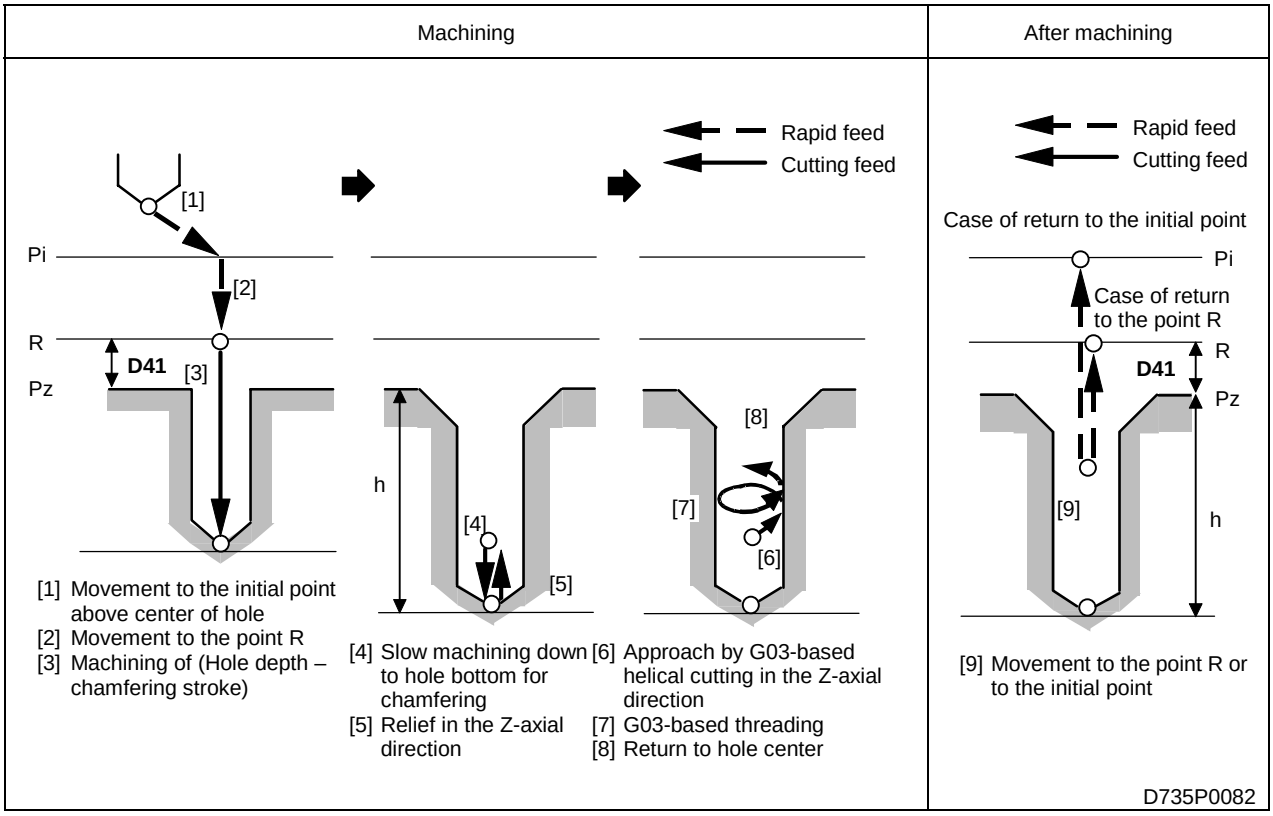
The planetary tapping cycle allows three types of machining (pre-hole machining, chamfering, and female threading) with one tool.

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 to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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

Note 1: The inversed tapping cannot be executed.

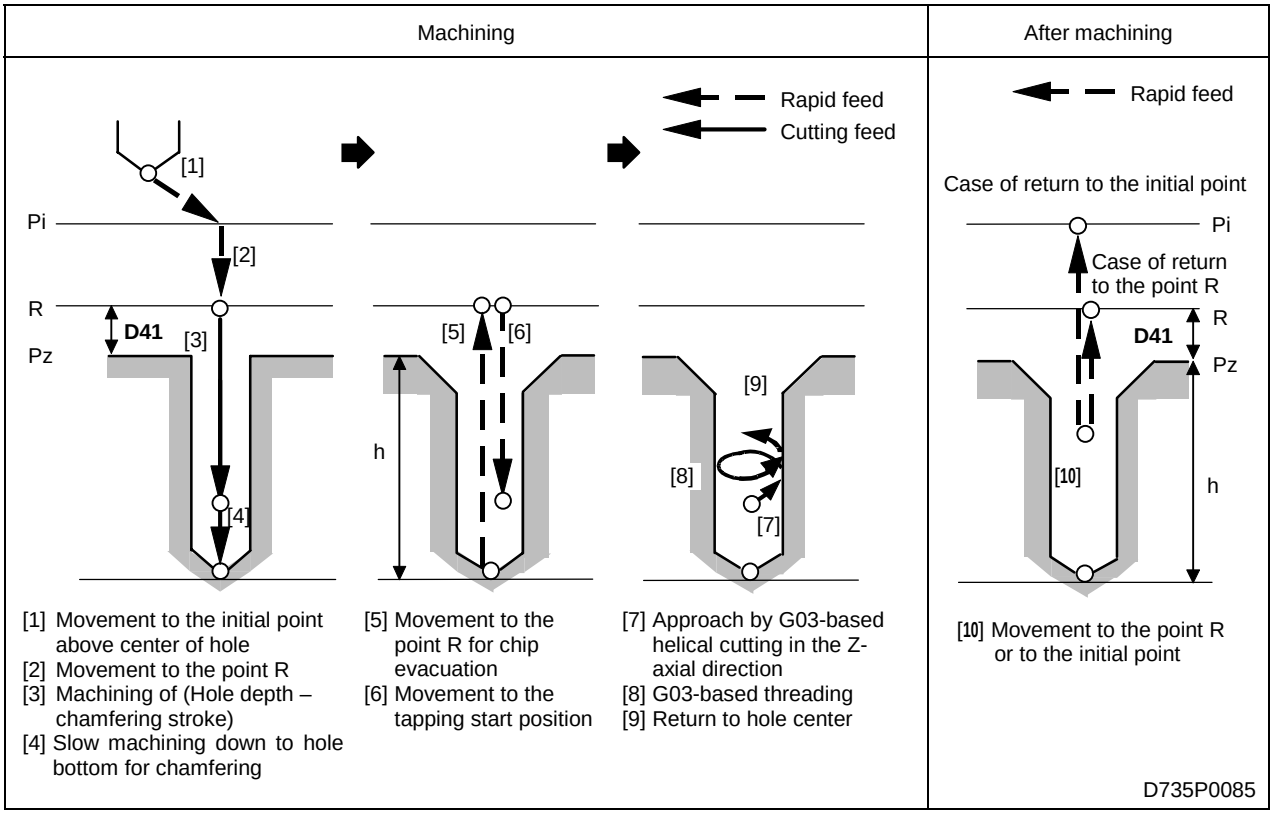
Note 2: The feed rate for chamfering on tool path [4] is calculated as follows:
Chamfering feed = Pre-hole machining feed × Chamfering feed override (parameter **D48**)/100

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

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

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

2. Case with movement for chip evacuation



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

R: Safety clearance above the point Pz (parameter **D41**)

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

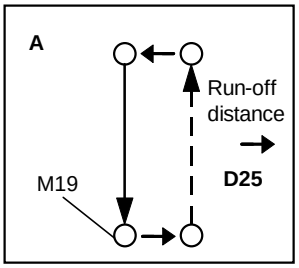
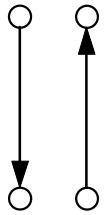
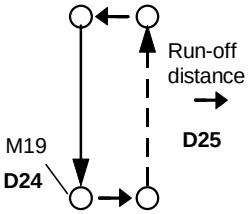
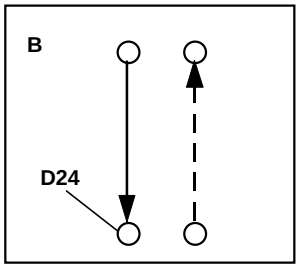
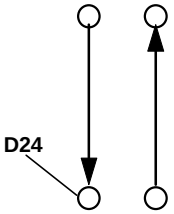
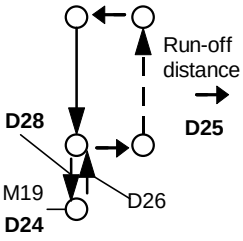
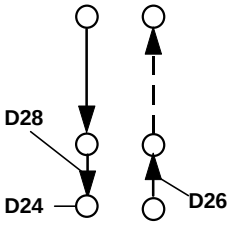
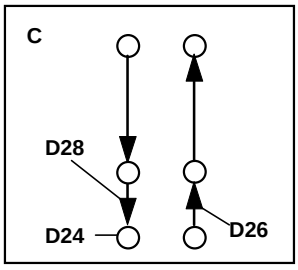
Note 4: The tool diameter in the tool data is modified for tapping diameter correction (fine-adjustment).

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	Cycle 2	Cycle 3
Roughness 0	No	No			
Roughness 1	No	Yes			
Roughness 2-9	Yes	Yes			
◄ – – Rapid feed ◄—— Cutting feed					M3P131

The bold codes represent the parameter addresses.

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

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

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

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

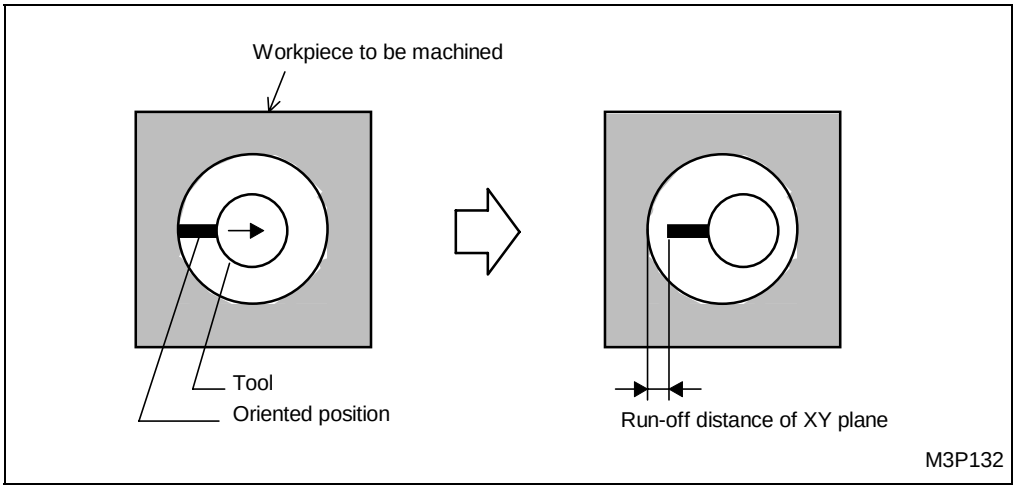
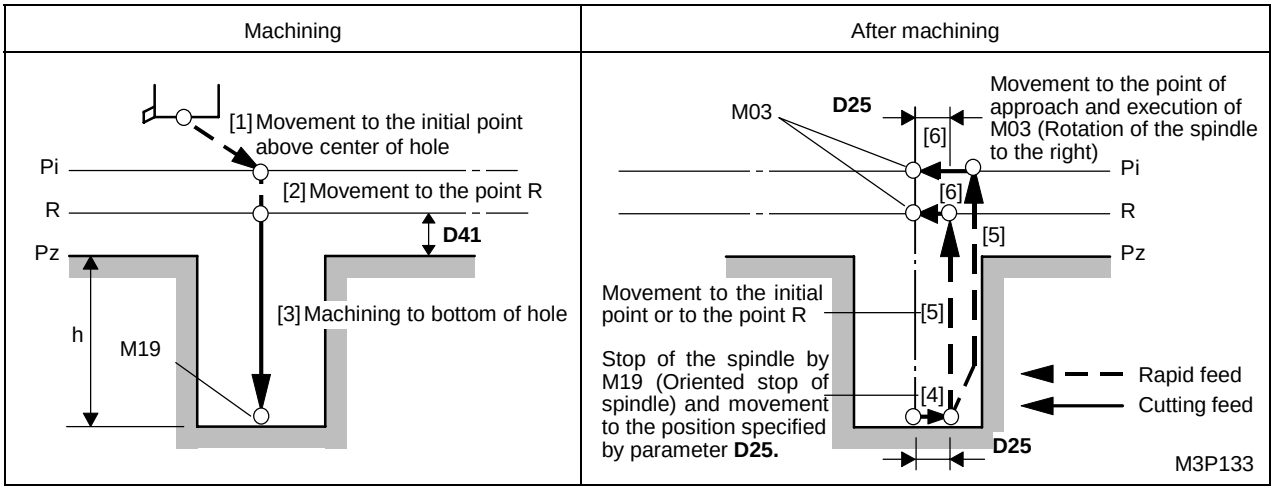


Fig. 4-14 Run-off distance on the X-Y plane

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 to be determined by the data INITIAL-Z entered in the common unit

Pz: Z coordinate of machining surfce to be entered in the shape sequence

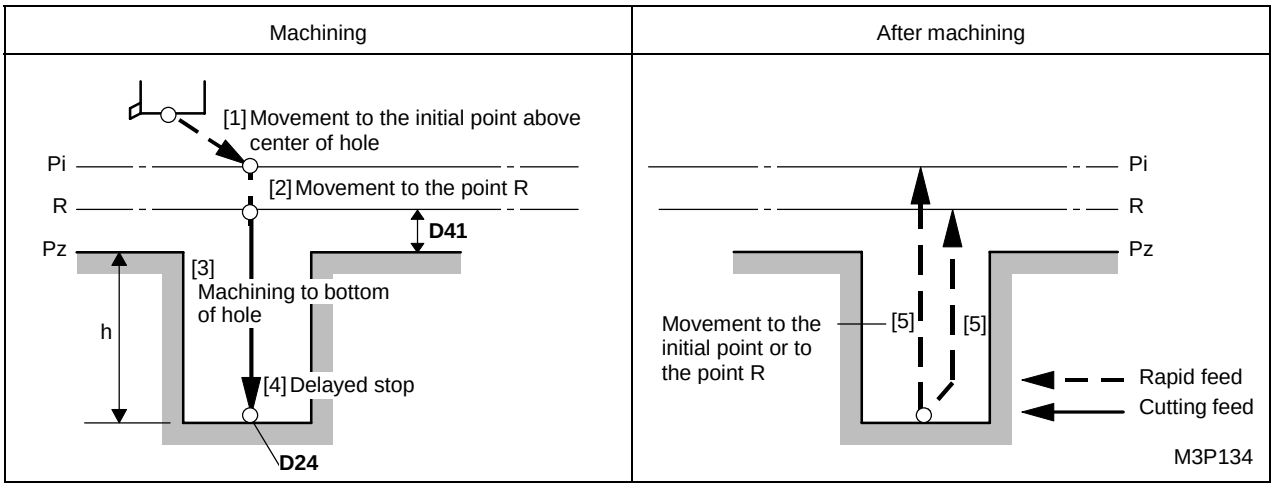
R: Safety clearance on Z axis (parameter **D41**)

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

Note 1: 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 to be determined by the data INITIAL-Z in the common unit

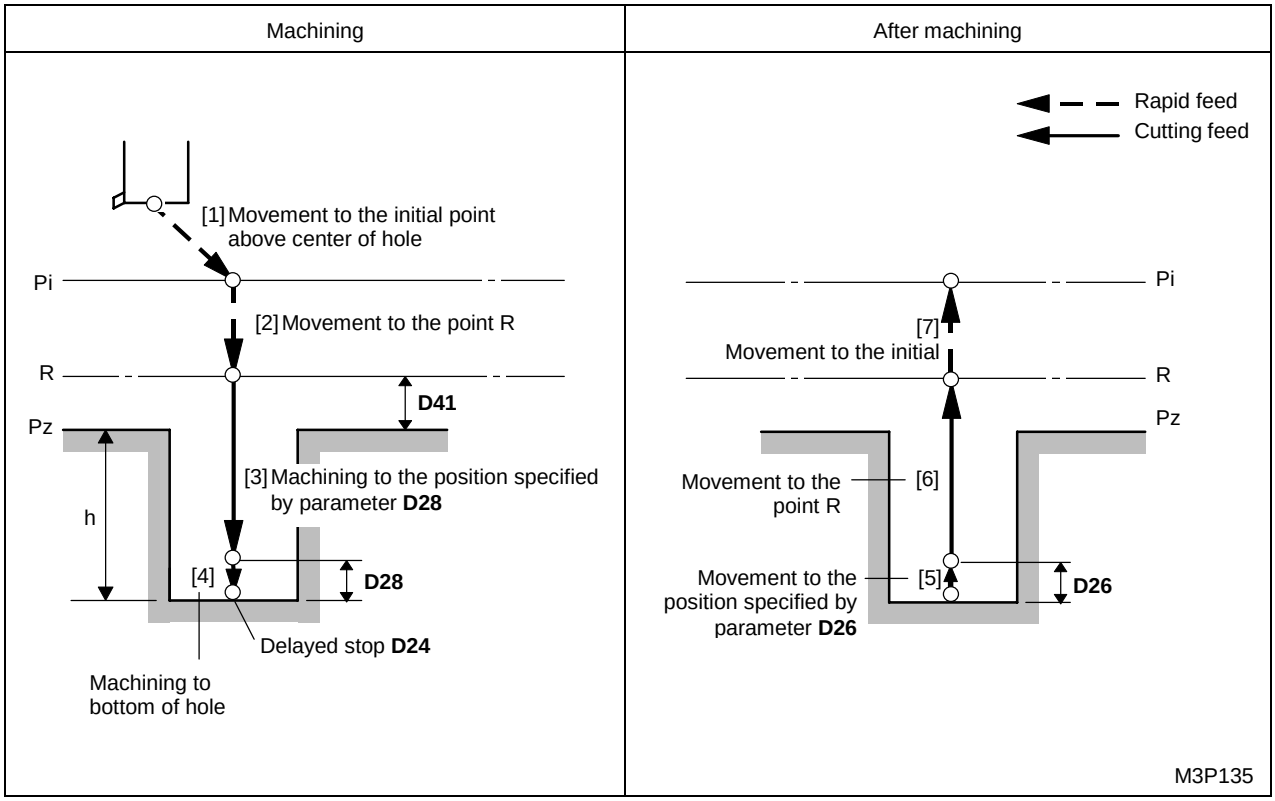
Pz: Z coordinate of machining surfce to be entered in the shape sequence

R: Safety clearance on Z axis (parameter **D41**)

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

Note: The delayed stop time of the feed 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 to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of machining surfce to be entered in the shape sequence

R: Safety clearance on Z axis (parameter **D41**)

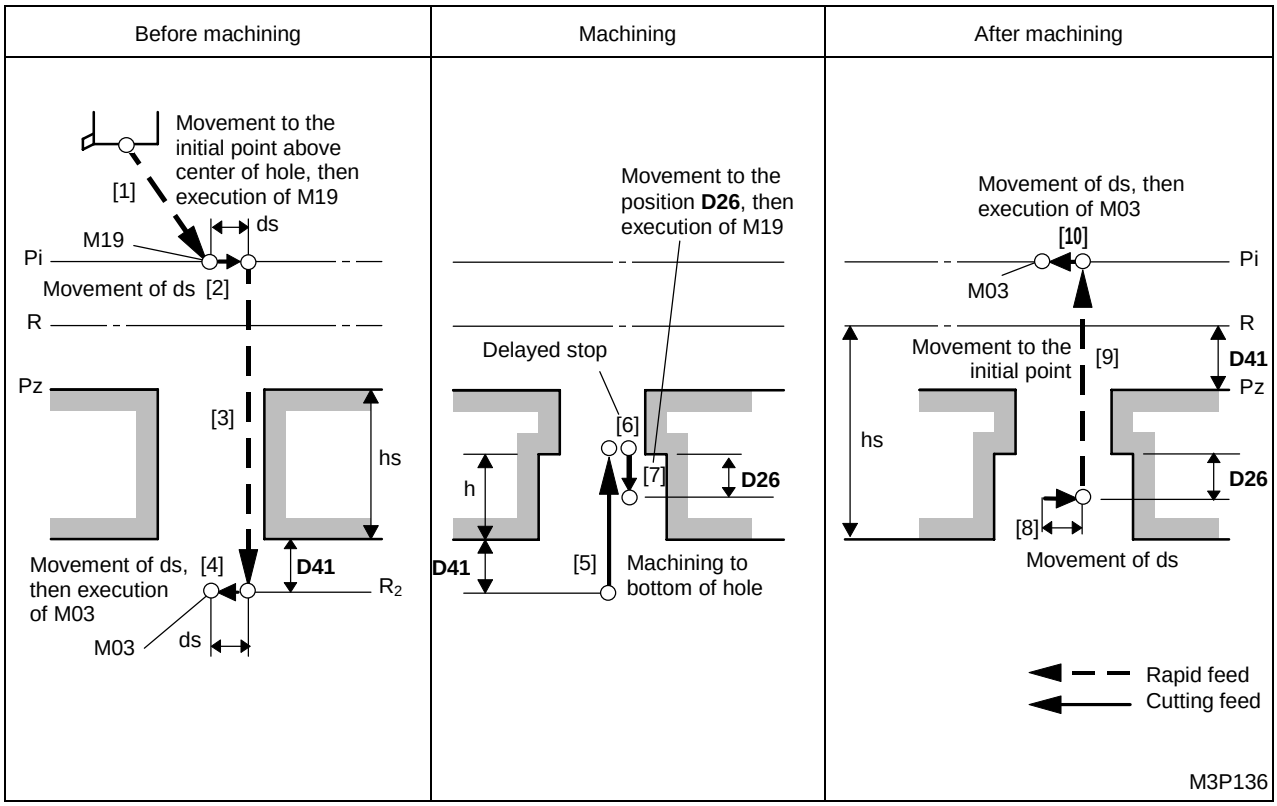
h: Distance equal to the sum of the depth of hole (HOLE-DEP) entered in the tool sequence and the data COMP. (tool correction) in the **TOOL DATA** display

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

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

Note 3: The delayed stop time of the feed 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 to be determined by the data INITIAL-Z in common unit

Pz: Z coordinate of machining surface to be entered in the shape sequence

R, R_2: Safety clearance on Z axis (parameter **D41**)

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

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

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

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

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

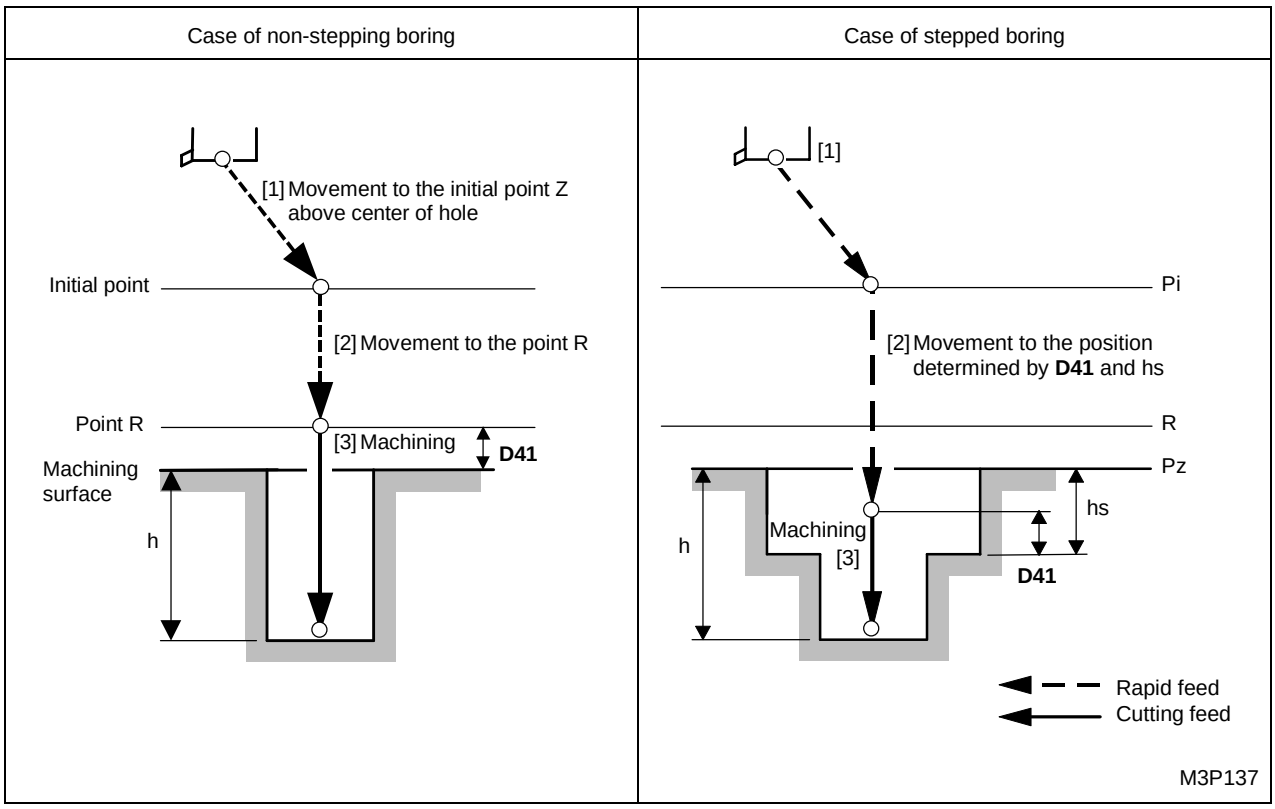
D33 ..... Movement on the XY plane entered in the parameter

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

Note 2: The delayed stop time of the 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 to be determined by the data INITIAL-Z entered in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

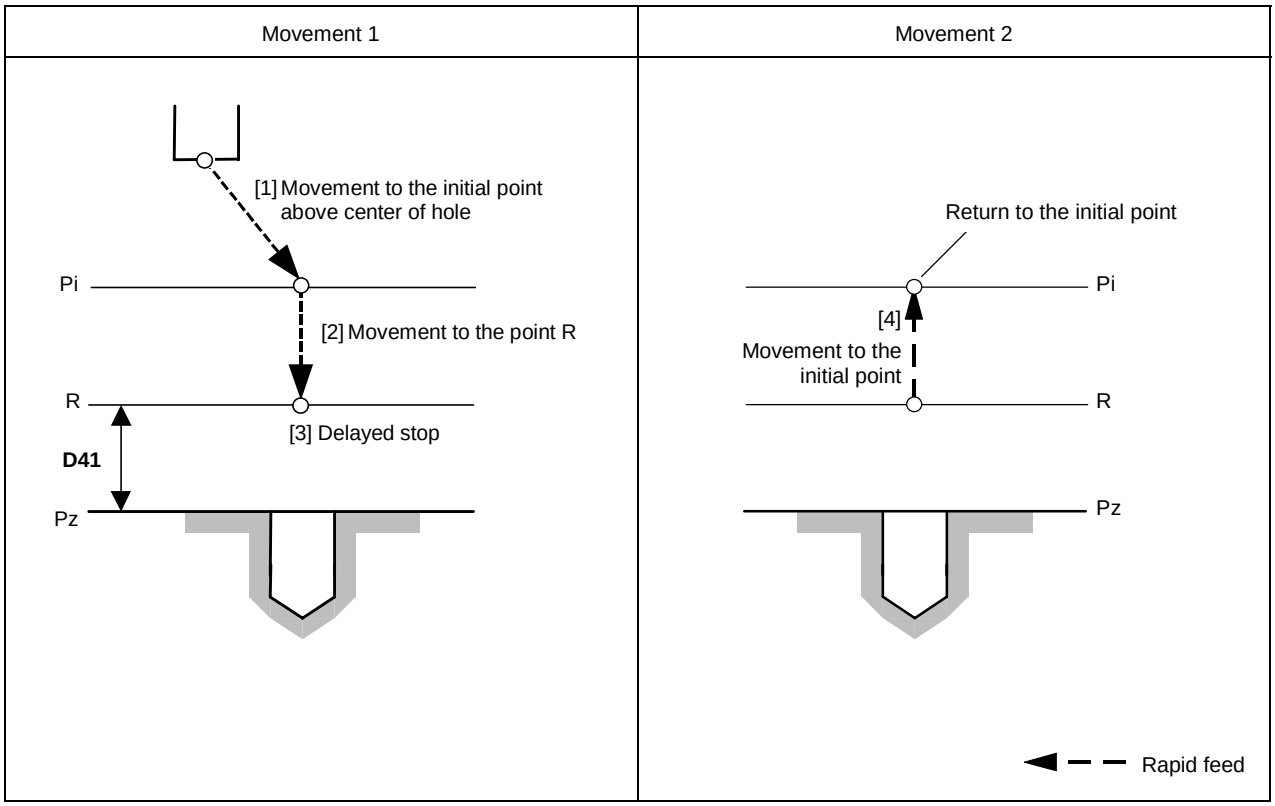
R: Safety clearance on Z axis (parameter **D41**)

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

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

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

10. Chip vacuuming tool (option)



The bold codes represent the parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pz: Z coordinate of the machining surface to be entered in the shape sequence

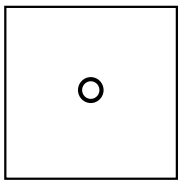
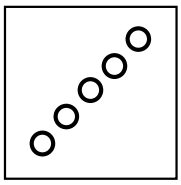
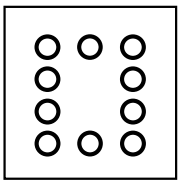
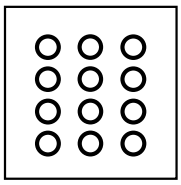
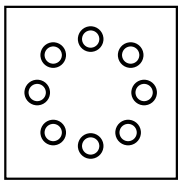
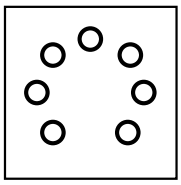
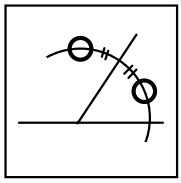
R: Safety clearance above the point Pz (parameter **D41**)

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

4-6-8 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 article Z is as shown below.

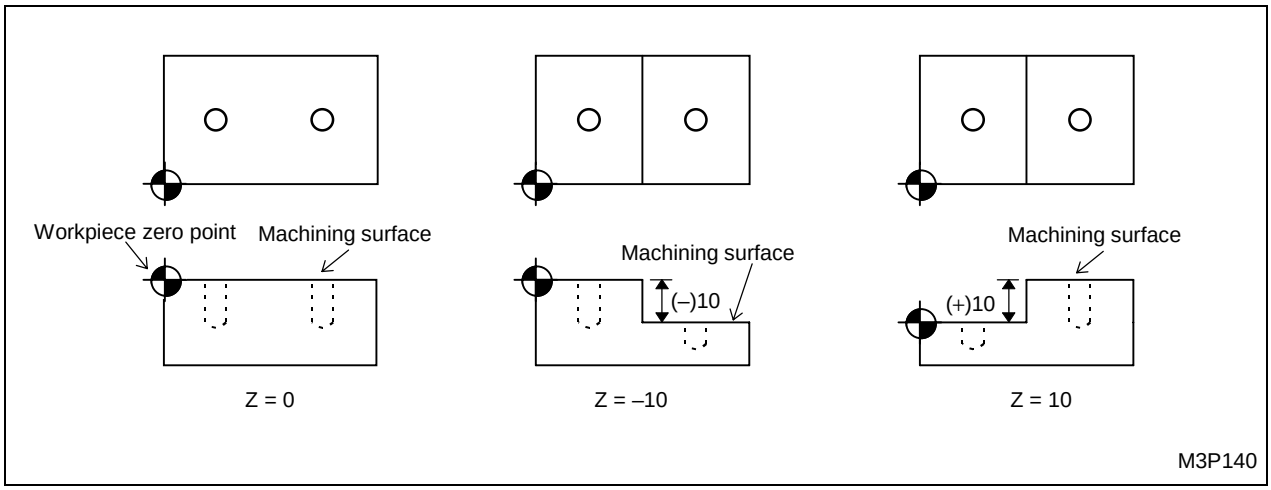
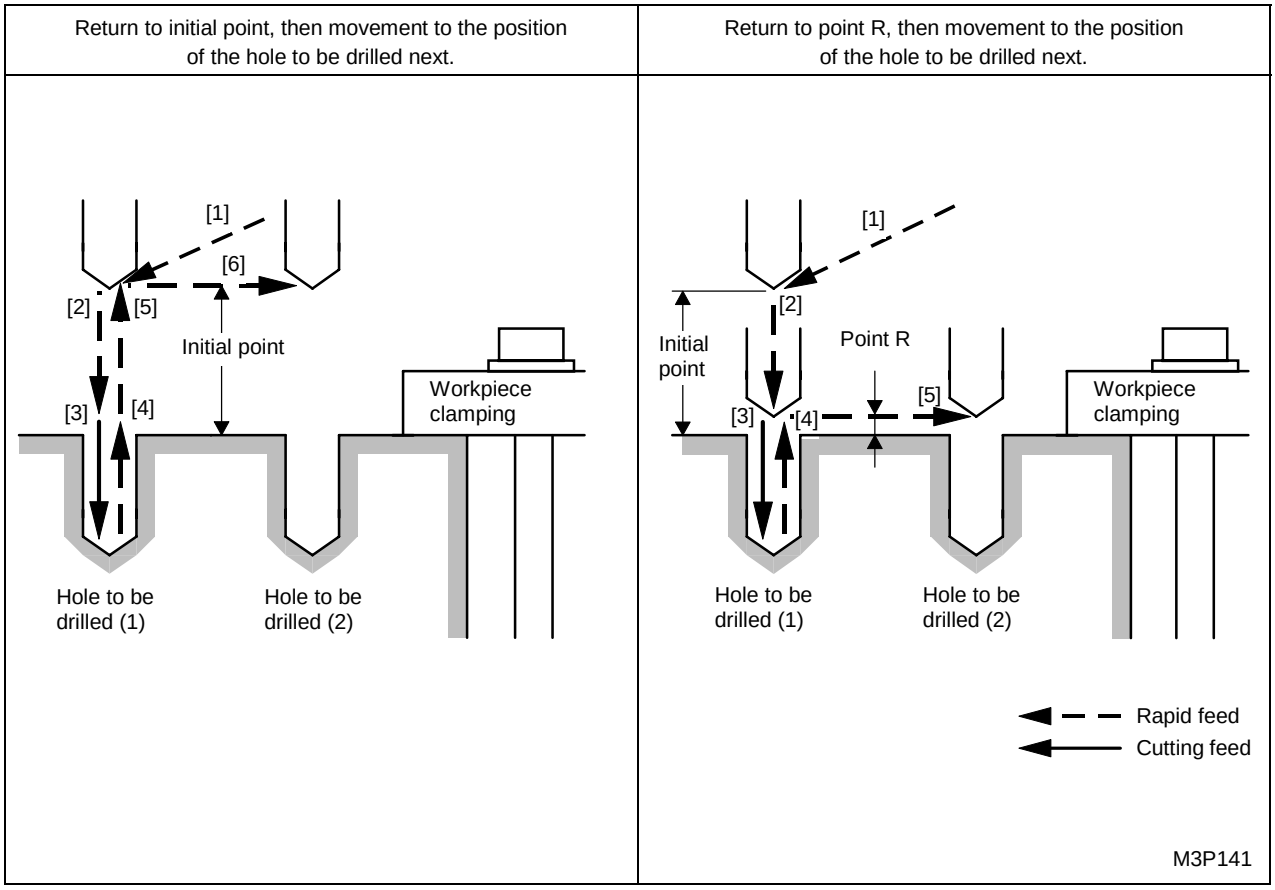


Fig. 4-15 Coordinate Z of the machining surface

(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.

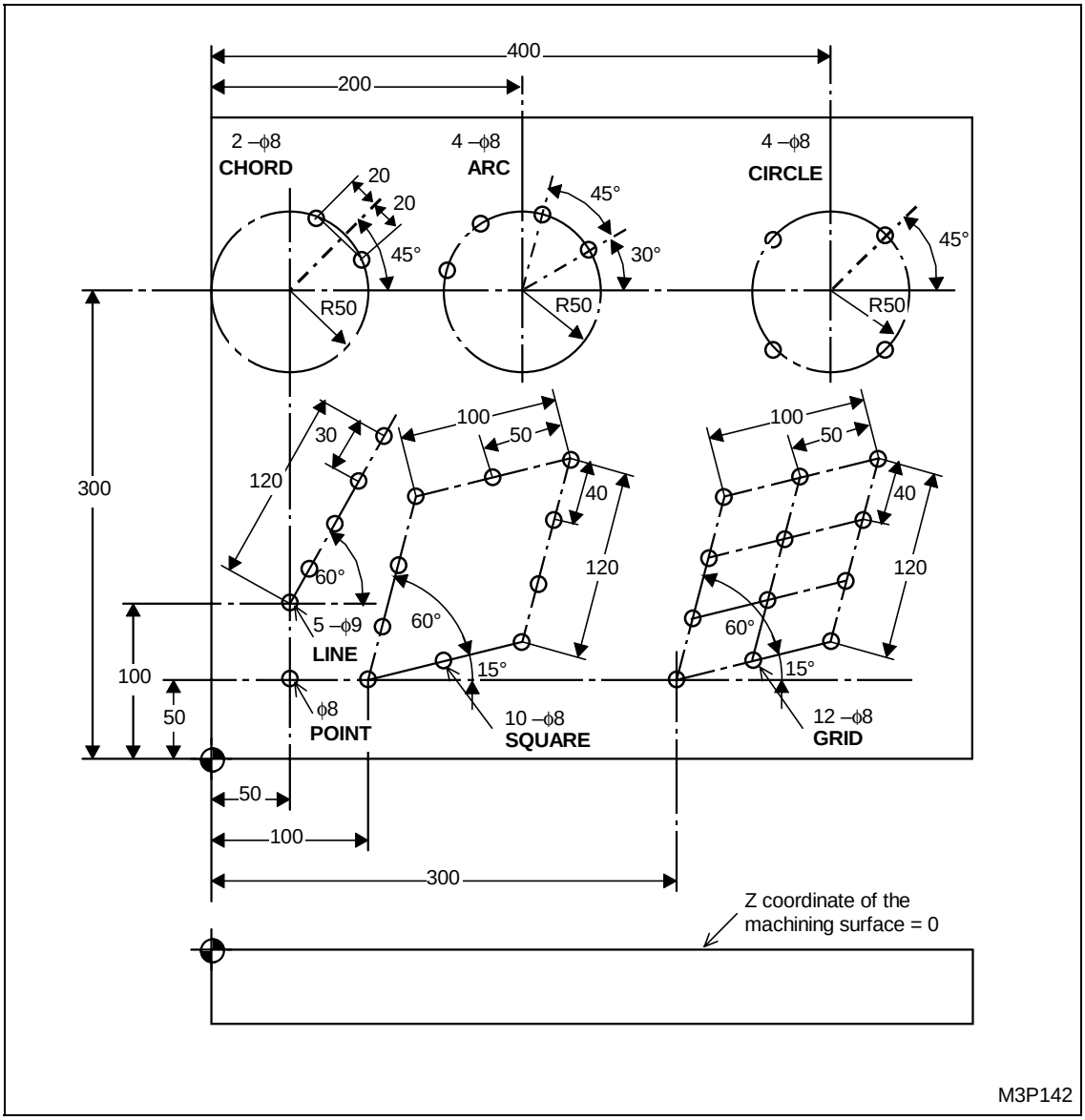
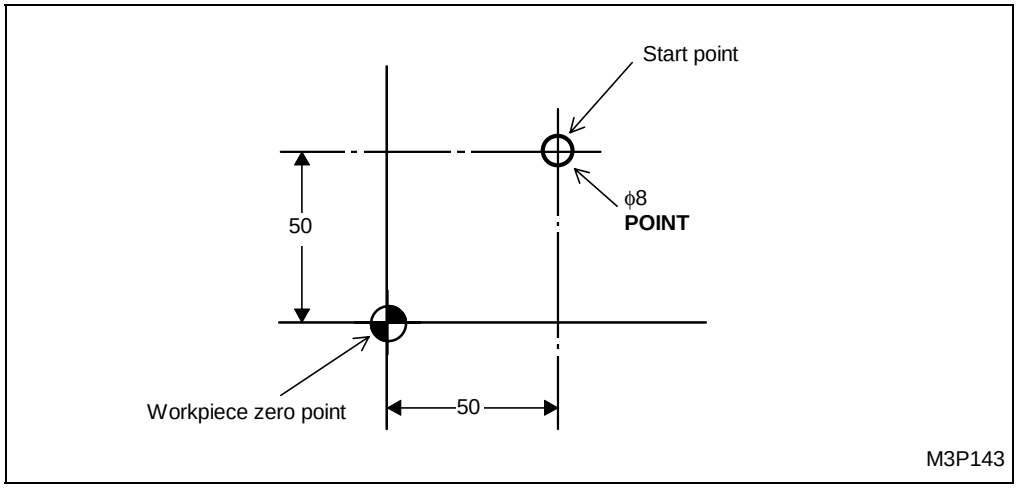


Fig. 4-16 Drawing for entry of shape sequence data

A. POINT



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

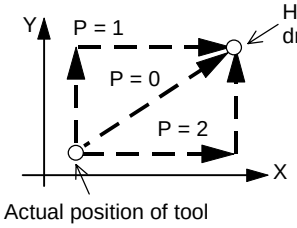
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE	CHECK
							END	

Press the menu key **POINT**.

(b) Data setting

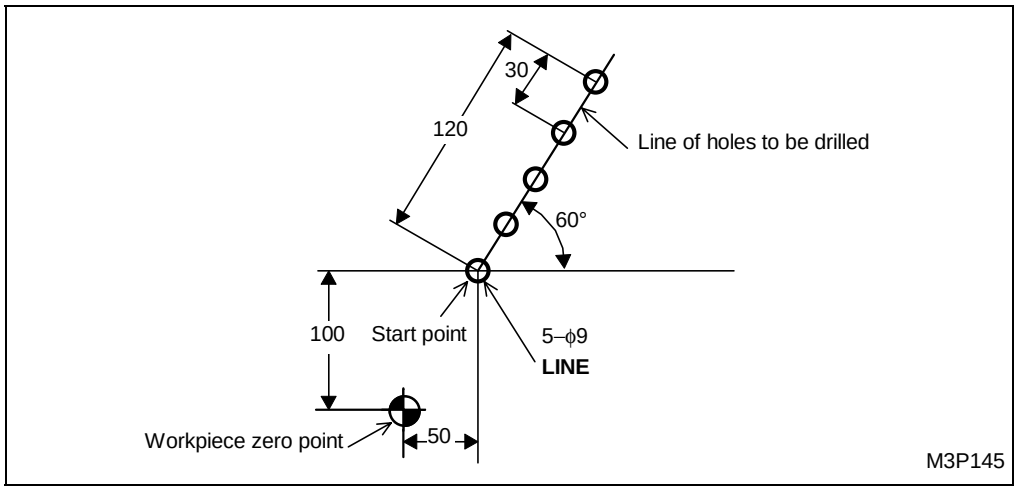
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	PT	0.	50.	50.	◆	◆	◆	◆	◆	◆	◆	0	0	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... 0 ⏎
X	Specify the X coordinate of the first hole to be drilled. ..... 5 0 ⏎
Y	Specify the Y coordinate of the first hole to be drilled. ..... 5 0 ⏎
P	Specify the tool path.  M3P144 Simultaneous movement on X and Y axes ..... 0 ⏎ Movement on Y axis, then on X axis ..... 1 ⏎ Movement on X axis, then on Y axis ..... 2 ⏎
Q	Specify if the machining at the start point is executed or not. Actual execution of machining ..... 0 ⏎ Only positioning without machining ..... 1 ⏎

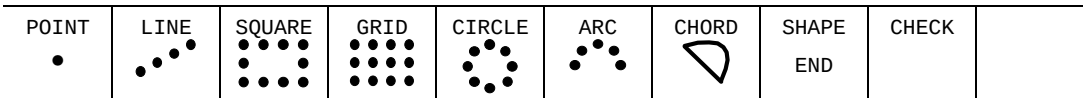
Cursor position	Description
R	Specify the position to which the tool returns after machining. Initial point ..... 0 ⏎ Point R ..... 1 ⏎

B. LINE



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.



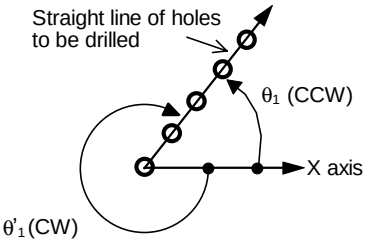
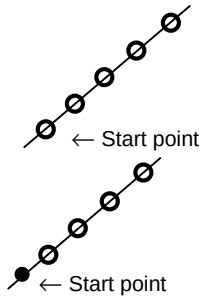
Press the menu key **LINE**.

(b) Data setting

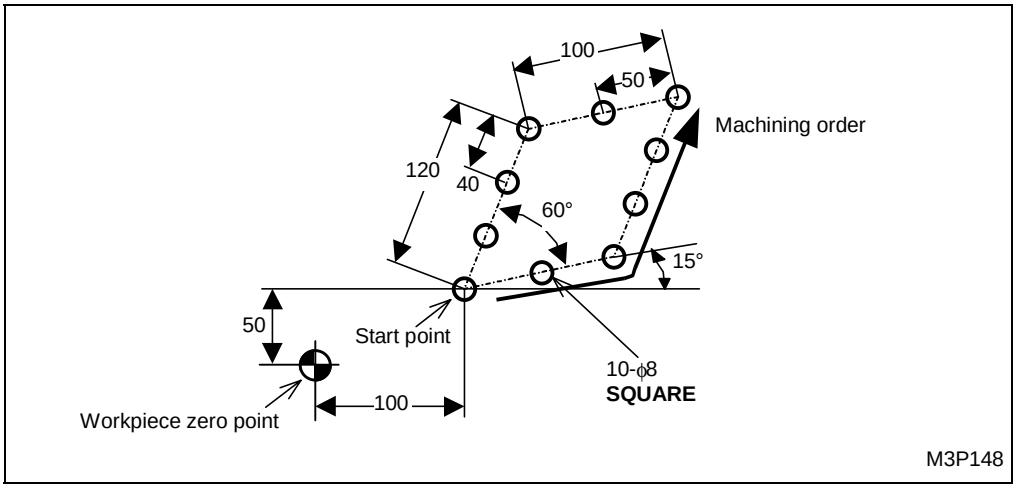
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
2	LIN	0.	50.	100.	60	◆	30.	◆	0	5	◆	◆	0	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... 0 ⏎
X	Specify the X coordinate of the first hole to be drilled. ..... 5 0 ⏎
Y	Specify the Y coordinate of the first hole to be drilled. ..... 1 0 0 ⏎

Cursor position	Description
AN1	Specify the angle θ_1 formed by the line of holes to be drilled and the X axis.  M3P146 There are two types of θ_1 CCW direction: $\theta_1 = 60°$ (to the left) (+)..... 6 0 → CW direction: $\theta'_1 = -300°$ (to the right) (–)..... – 3 0 0 → →
T1	Specify the spacing between holes to be drilled or the total distance between the first hole to be drilled and the last hole. ℓ_1: Spacing between holes ..... 3 0 → ℓ_2: Total distance between the first hole to be drilled and last hole ..... 1 2 0 →
F	Specify whether the data entered in T1, concern the spacing or the total distance. Spacing ..... 0 → Total distance ..... 1 →
M	Specify the number of holes to be drilled . ..... 5 →
O	Specify whether the machining at the start point is executed or not.  M3P147 Actual executionof the machining ..... 0 → Only positioningwithout machining ..... 1 →
R	Specify the position to which the tool returns after machining. Initial point ..... 0 → Point R ..... 1 →

C. SQUARE



(a) Menu selection

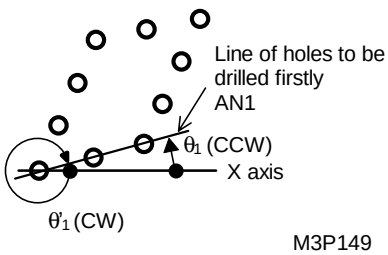
When the setting for the tool sequence has been finished, the following menu will be displayed.

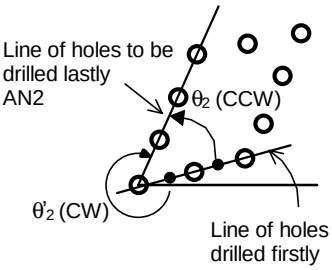
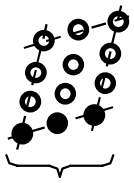
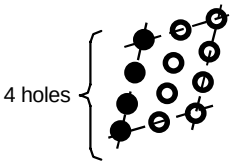
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE END	CHECK

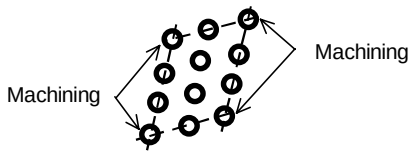
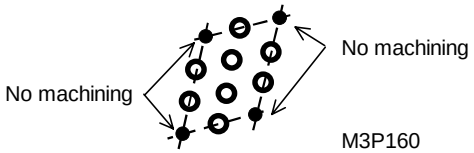
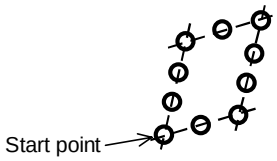
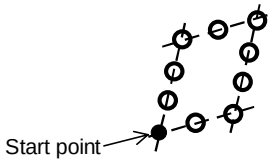
Press the menu key **SQUARE**.

(b) Data setting

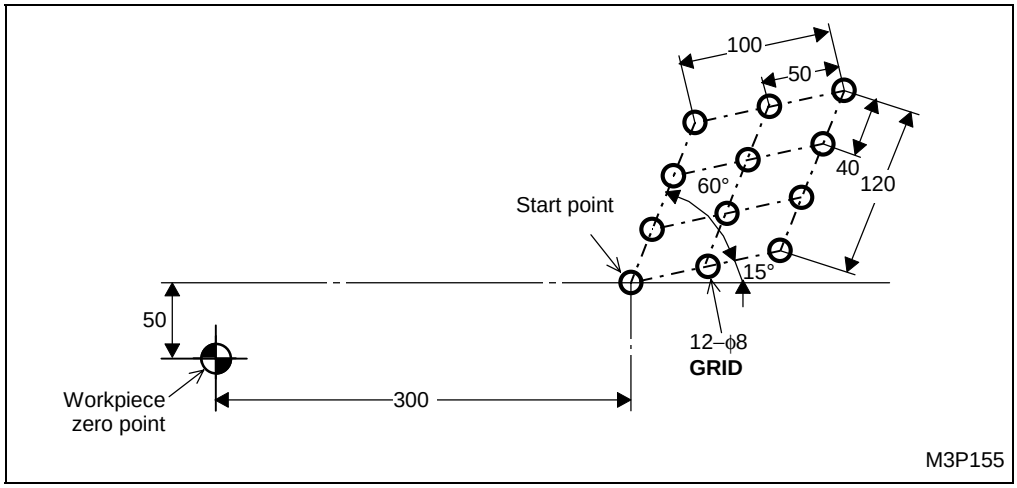
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
3	SQR	0.	100.	50.	15.	60.	50.	40.	0	3	4	0	0	0

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... **0** [→]
X	Specify the X coordinate of the first hole to be drilled. ..... **1** **0** **0** [→]
Y	Specify the Y coordinate of the first hole to be drilled. ..... **5** **0** [→]
AN1	Specify the angle θ'_1 formed by the line of holes to be drilled firstly and the X axis.  M3P149 There are two types of θ_1 CCW direction: $\theta_1 = 15°$ (to the left) (+) ..... **1** **5** [→] CW direction: $\theta'_1 = -345°$ (to the right) (–) ..... **–** **3** **4** **5** [→]

Cursor position	Description
AN2	Specify the angle θ'_2 formed by two lines of holes to be drilled.  M3P150 There are two types of θ_2 CCW direction: $\theta_2 = 60°$ (to the left) (+) ..... 6 0 → CW direction: $\theta'_2 = -300°$ (to the right) (–) ..... – 3 0 0 → →
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 firtly. ..... 5 0 → ℓ_2: Total length of the line of holes to be drilled firtly. ..... 1 0 0 → (Designate ℓ_1 or ℓ_2.)
T2	Spacify 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. ..... 4 0 → t_2: Total length of the line of holes to be drilled lastly. ..... 1 2 0 → (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 specing or the total length. Spacing ..... 0 → Total length ..... 1 →
M	Specify the number of holes on the line of holes to be drilled firstly. ..... 3 →  M3P158
N	Specify the number of holes on the line of holes to be drilled lastly. ..... 4 →  M3P159

Cursor position	Description
P	Specify if the machining at the four corners is executed or not. Machining at the four corners ..... 0 ⏎  No machining at the four corners ..... 1 ⏎  M3P160
Q	Specify if the machining at the start point is executed or not. Actual execution of the machining ..... 0 ⏎  Only positioning without machining ..... 1 ⏎  M3P154
R	Specify the position to which the tool returns after the machining. Initial point ..... 0 ⏎ Point R ..... 1 ⏎

D. GRID



(a) Menu selection

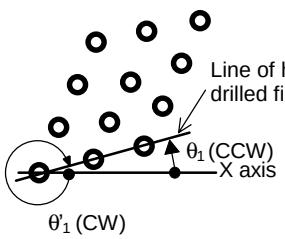
When the setting for the tool sequence has been finished, the following menu will be displayed.

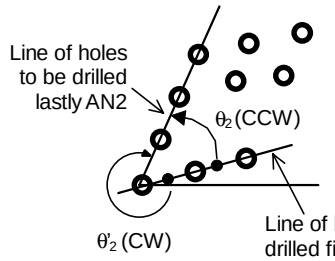
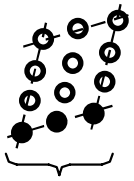
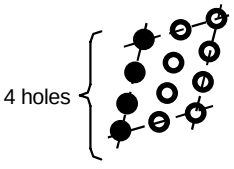
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE	CHECK
							END	

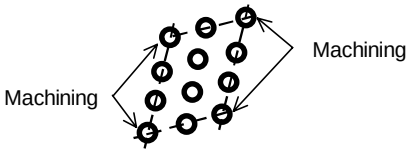
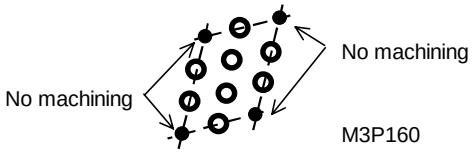
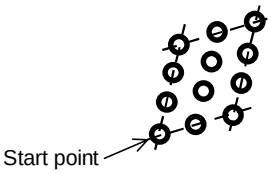
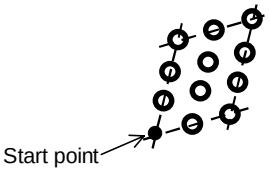
Press the menu key **GRID**.

(b) Data setting

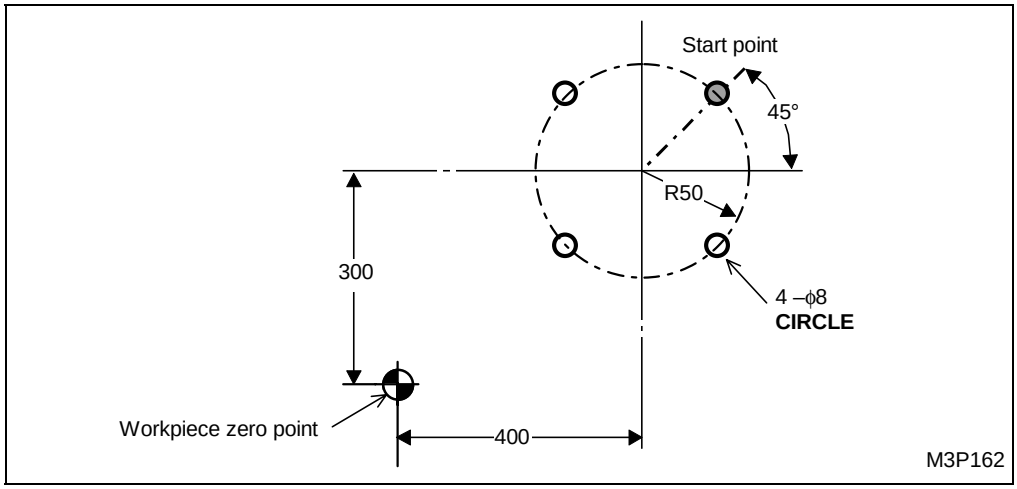
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
4	GRD	0.	100.	50.	15.	60.	50.	40.	0	3	4	0	0	0

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... [0] [→]
X	Specify the X coordinate of the first hole to be drilled. ..... [1] [0] [0] [→]
Y	Specify the Y coordinate of the first hole to be drilled. ..... [5] [0] [→]
AN1	Specify the angle θ_1 formed by the line of holes to be drilled firstly and the X axis.  M3P156 There are two types of θ_1 CCW direction: θ_1 = 15° (to the left) (+)..... [1] [5] [→] CW direction: θ'_1 = –345° (to the right) (–)..... [–] [3] [4] [5] [→]

Cursor position	Description
AN2	Specify the angle θ_2 formed by two lines of holes to be drilled.  M3P157 There are two types of θ_2 CCW direction: $\theta_2 = 60°$ (to the left) (+) ..... 6 0 → CW direction: $\theta'_2 = -300°$ (to the right) (–) ..... – 3 0 0 → →
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. ..... 5 0 → ℓ_2: Total length of the line of holes to be drilled firstly. ..... 1 0 0 → (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. ..... 4 0 → t_2: Total length of the line of holes to be drilled lastly. ..... 1 2 0 → (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 specing or the total length. Spacing ..... 0 → Total length ..... 1 →
M	Specify the number of holes on the line of holes to be drilled firstly. ..... 3 →  M3P158
N	Specify the number of holes on the line of holes to be drilled lastly. ..... 4 →  M3P159

Cursor position	Description
P	Specify if the machining at the four corners is executed or not. Machining at the four corners ..... 0 ⏎  No machining at the four corners ..... 1 ⏎  M3P160
Q	Specify if the machining at the start point is executed or not. Actual execution of the machining ..... 0 ⏎  Only positioning without machining ..... 1 ⏎  M3P161
R	Specify the position to which the tool returns after the machining. Initial point ..... 0 ⏎ Point R ..... 1 ⏎

E. CIRCLE



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

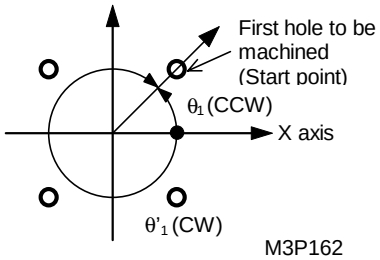
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE	CHECK
							END	

Press the menu key **CIRCLE**.

(b) Data setting

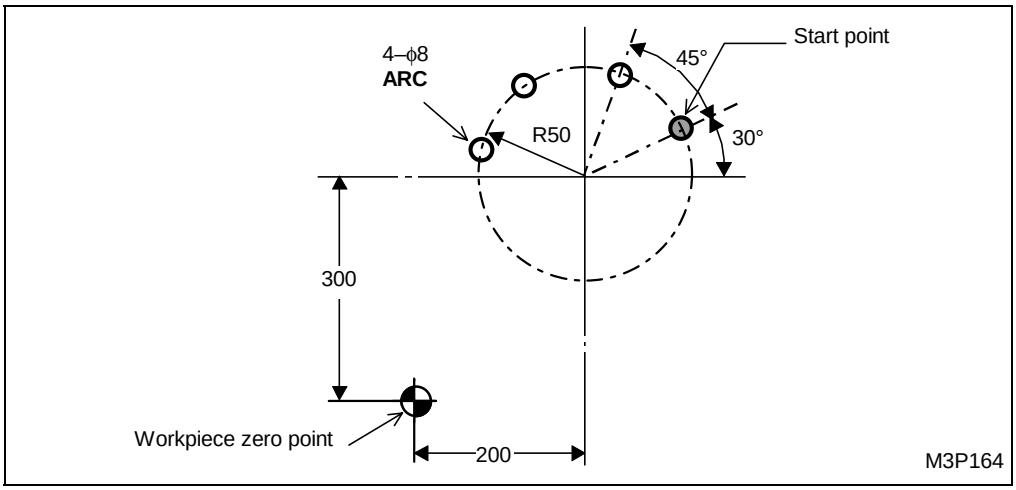
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
5	CIR	0.	400.	300.	45.	◆	50.	◆	◆	4	◆	◆	◆	0

◆: Data are not necessary to be set here.

Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... 0 →
X	Specify the X coordinate of the center of the circle. ..... 4 0 0 →
Y	Specify the Y coordinate of the center of the circle. ..... 3 0 0 →
AN1	Specify the angle θ_1 formed by the start point and the X axis.  M3P162 There are two types of θ_1 CCW direction: θ_1 = 45° (to the left) (+) ..... 4 5 → CW direction: θ'_1 = –315° (to the right) (–) ..... – 3 1 5 →
T1	Specify the radius of the circle. ..... 5 0 →
M	Specify the number of holes to be machined. ..... 4 →

Cursor position	Description
R	Spacify the position to which the tool returns after the machining. Initial point ..... 0 ⏎ Point R ..... 1 ⏎

F. ARC



(a) Menu selection

When the setting for the tool sequence has been finished, the following menu will be displayed.

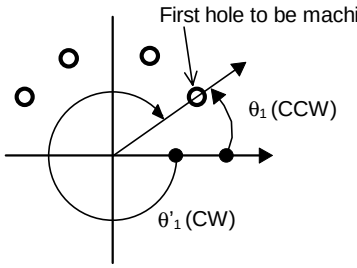
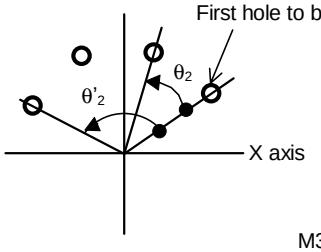
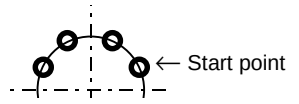
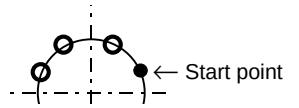
POINT	LINE	SQUARE	GRID	CIRCLE	ARC	CHORD	SHAPE END	CHECK	

Press the menu key **ARC**.

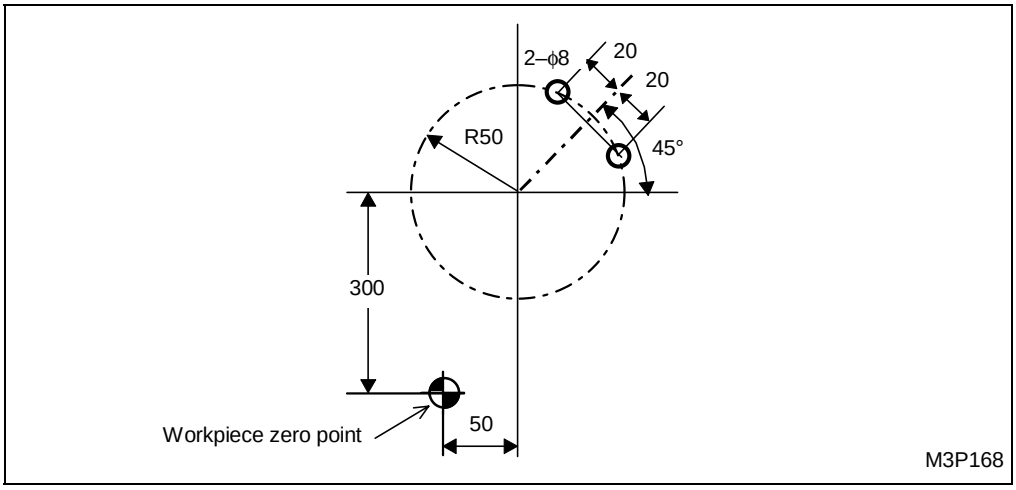
(b) Data setting

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. ..... 0 →
X	Specify the X coordinate of the center of the arc. ..... 2 0 0 →
Y	Specify the Y coordinate of the center of the arc. ..... 3 0 0 →
AN1	Specify the angle θ_1 formed by the start point and the X axis.  M3P165 There are two types of θ_1 CCW direction: θ_1 = 30° (to the left) (+) ..... 3 0 → CW direction: θ'_1 = –330° (to the right) (–) ..... – 3 3 0 →
AN2	Specify the angle of pitch between two adjacent holes or the angle between the first hole and the last hole.  M3P166 θ_2: Angle of pitch between two adjadent holes θ_2 = 45° ..... 4 5 → θ'_2: Total angle between the first hole and the last hole θ'_2 = 135° ..... 1 3 5 → (Designate θ_2 or θ'_2)
T1	Specify the radius of the arc ..... 5 0 →
F	Specify whether the data entered in AN2 concern the angle of pitch or the total angle. Angle of pitch ..... 0 → Total angle ..... 1 →
M	Specify the number of holes to be machined. ..... 4 →
Q	Specify the machining at the starting point is executed or not:  Actual execution of the machining ..... 0 →  Only positioning without machining ..... 1 → M3P167
R	Specify the position to which tool returns after machining. Initial point ..... 0 → Point R ..... 1 →

G. CHORD



(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

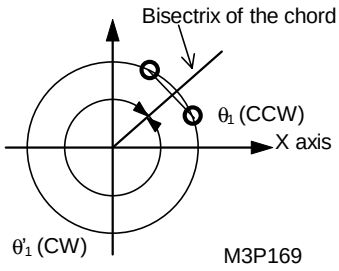
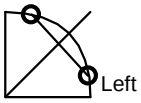
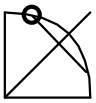
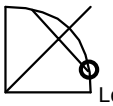
Press the menu key **CHORD**.

(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.

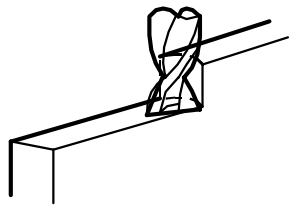
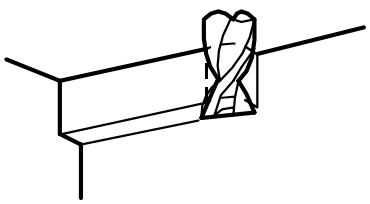
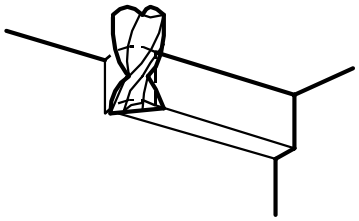
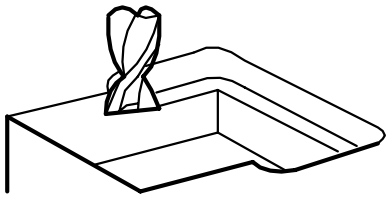
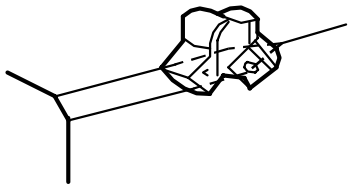
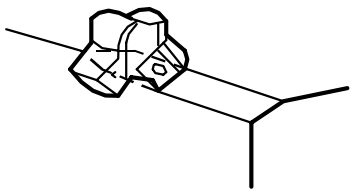
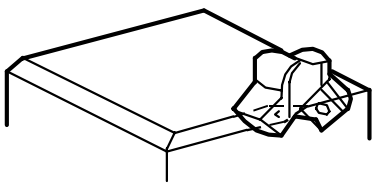
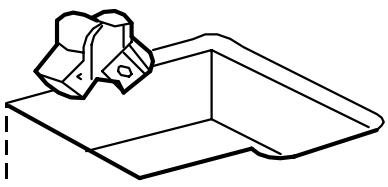
Cursor position	Description
Z	Specify the Z coordinate of the machining surface. ..... 0 →
X	Specify the X coordinate of the center of the circle. ..... 5 0 →
Y	Specify the Y coordinate of the center of the circle. ..... 3 0 0 →

Cursor position	Description
AN1	Specify the angle θ_1 formed by the bisectrix of the chord and the X axis.  M3P169 There are two types of θ_1 CCW direction: θ_1 = 45° (to the left) (+) ..... 4 5 → CW direction: θ'_1 = –315° (to the right) (–)..... – 3 1 5 → Specify the radius of the circle. ..... 5 0 →
T1	
T2	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. Total length ..... 4 0 → 1/2 of total length ..... 2 0 →
P	Specify the position of the hole to be machined. Right  Left Machining on both sides of the chord ..... 0 → Right  Machining of the right side of the chord ..... 1 →  Left M3P170 Machining of the left side of the chord ..... 2 →
R	Specify the position to which to tool returns after machining. Initial point ..... 0 → Point R ..... 1 →

4-7 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.

4-7-1 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

4-7-2 Procedure for selecting line machining unit

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

POINT MACH-ING	LINE MACH-ING	FACE MACH-ING	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE 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 article.

4-7-3 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	FIN-R			
1	LINE CTR	999.999	99.999	99.999	9			99.999	◆			
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL					◆			◆			
F2	E-MILL					◆		◆	◆			

Remark 1: Data in unit represent a maximum input value.

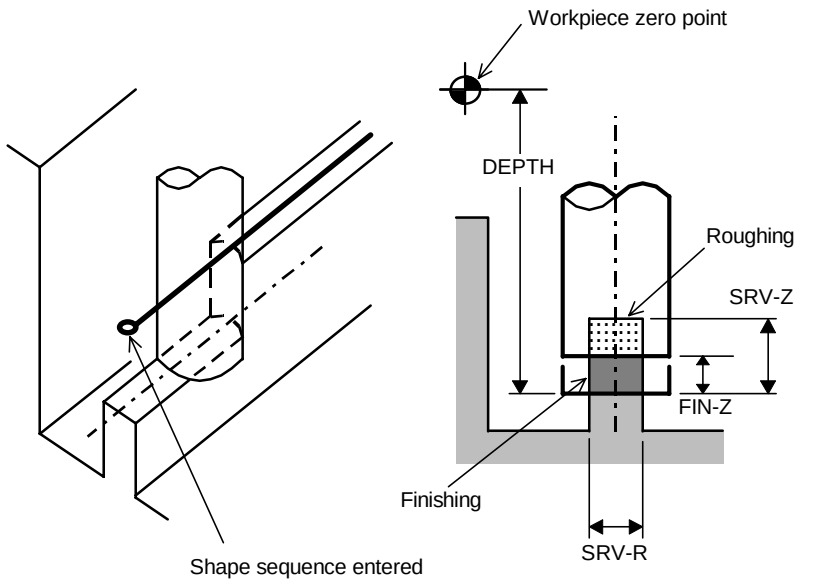
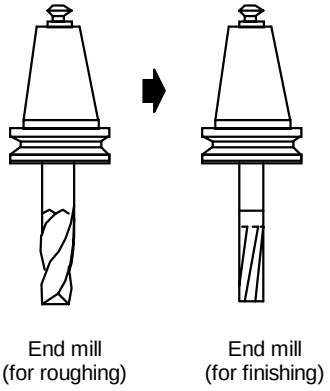
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, refer to Subsection 4-7-4.

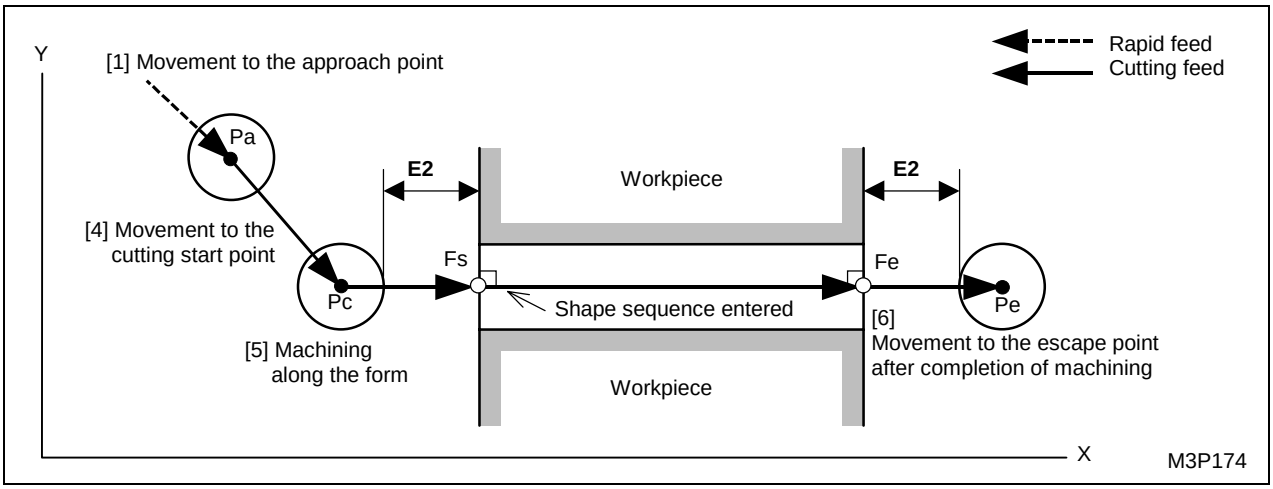
LINE CTR unit	Tool sequence
	
M3P172	M3P173

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

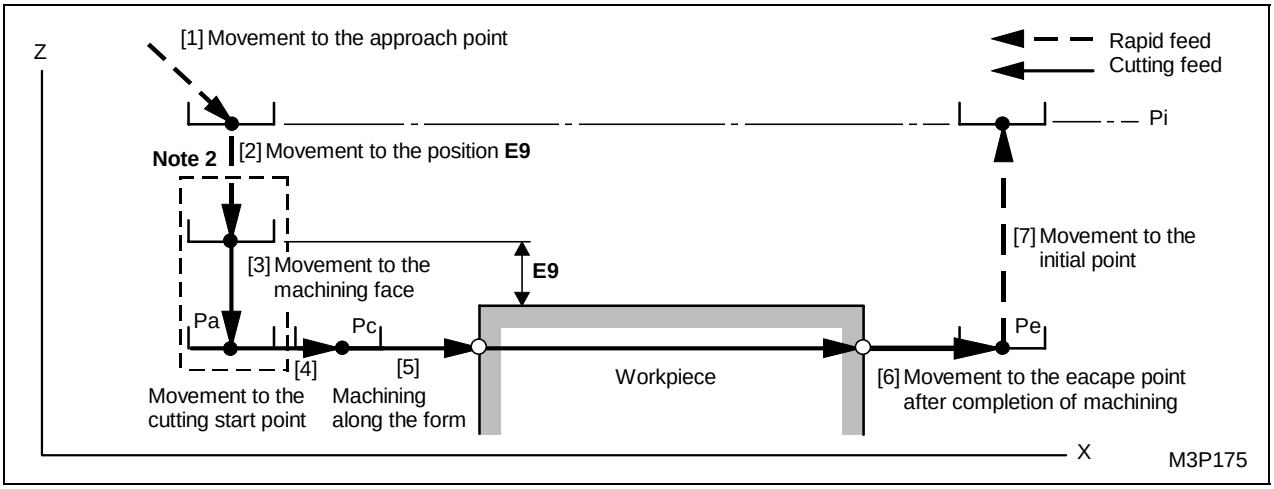
FIN-Z: A Z-axial finishing allowance is automatically entered once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

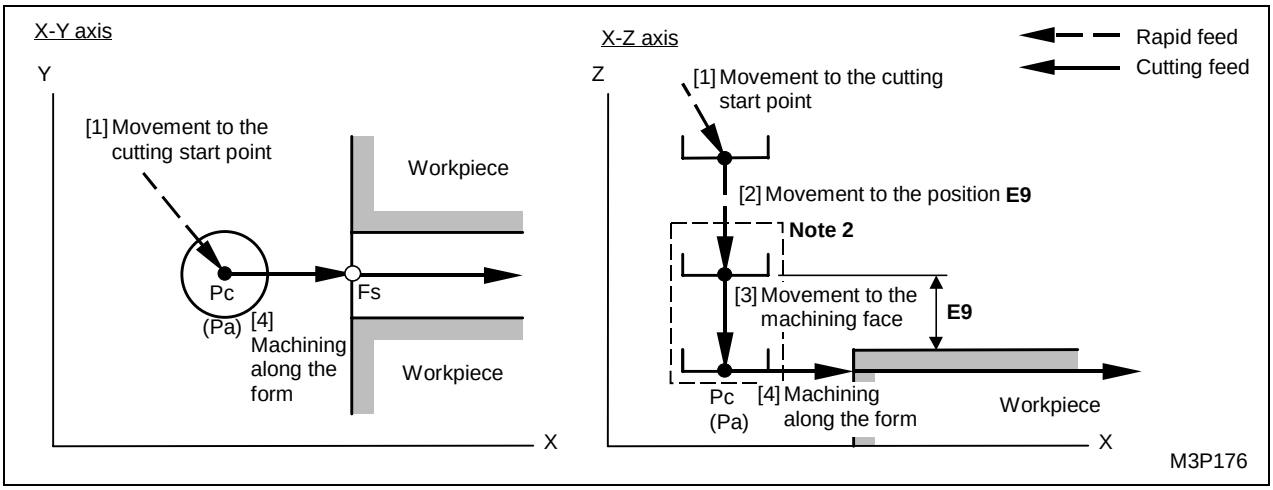
Pc: Cutting start point to be automatically established

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

Note 1: When ? is displayed in the articles 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 articles.



Note 2: See Subsection 4-7-6, “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	99.999	99.999	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	E-MILL					◆			◆				
F2	E-MILL					◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

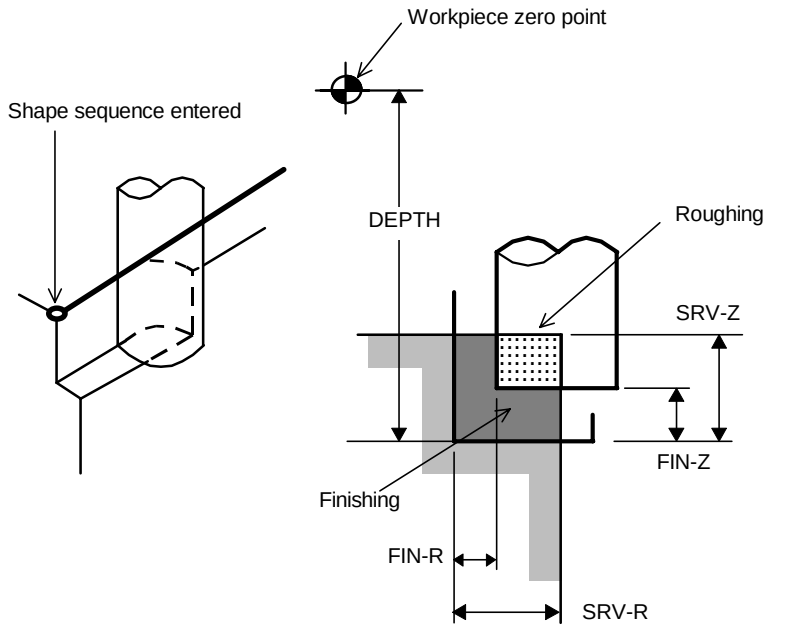
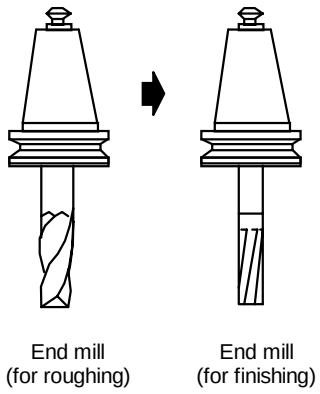
Remark 2: ◆: Data are not necessary to be set here.

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.

Q’ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z or SRV-R ≤ FIN-R
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, refer to Subsection 4-7-4.

LINE RGT unit	Tool sequence
 M3P177	 M3P173

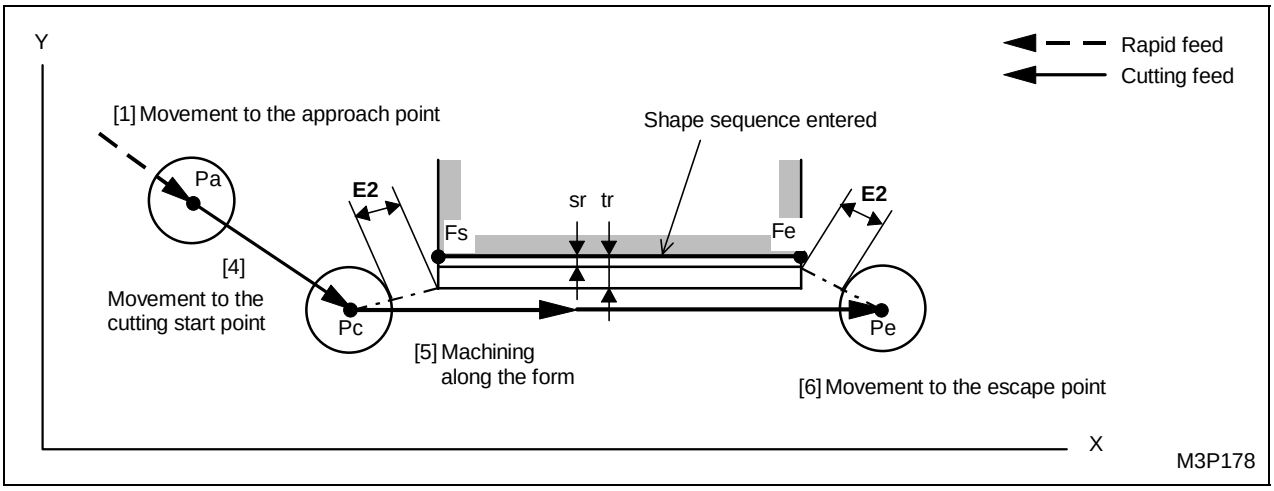
RGH: A roughness code should be selected out of the menu.

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

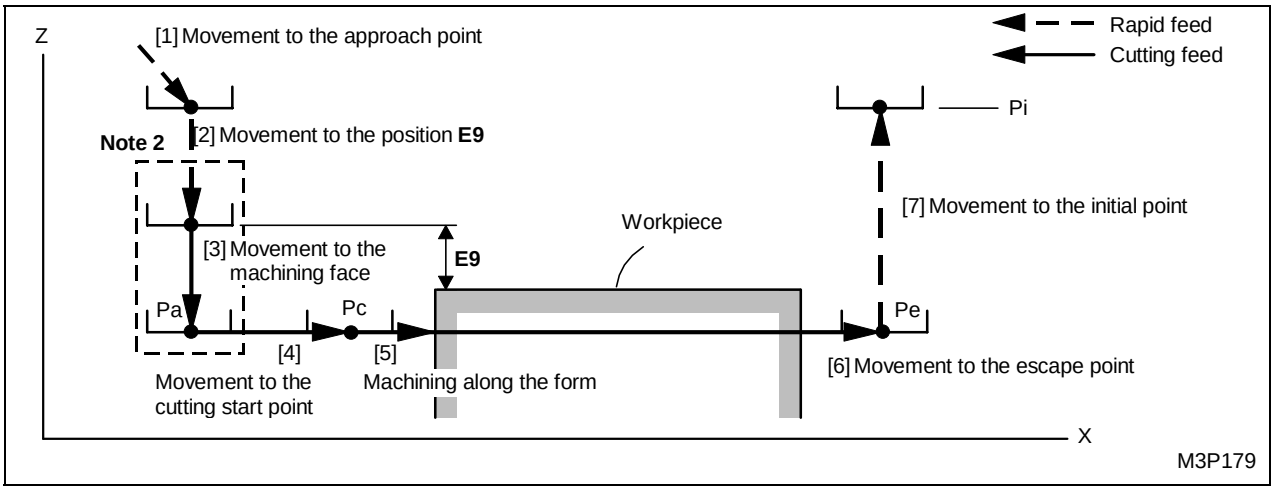
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data in the common unit

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

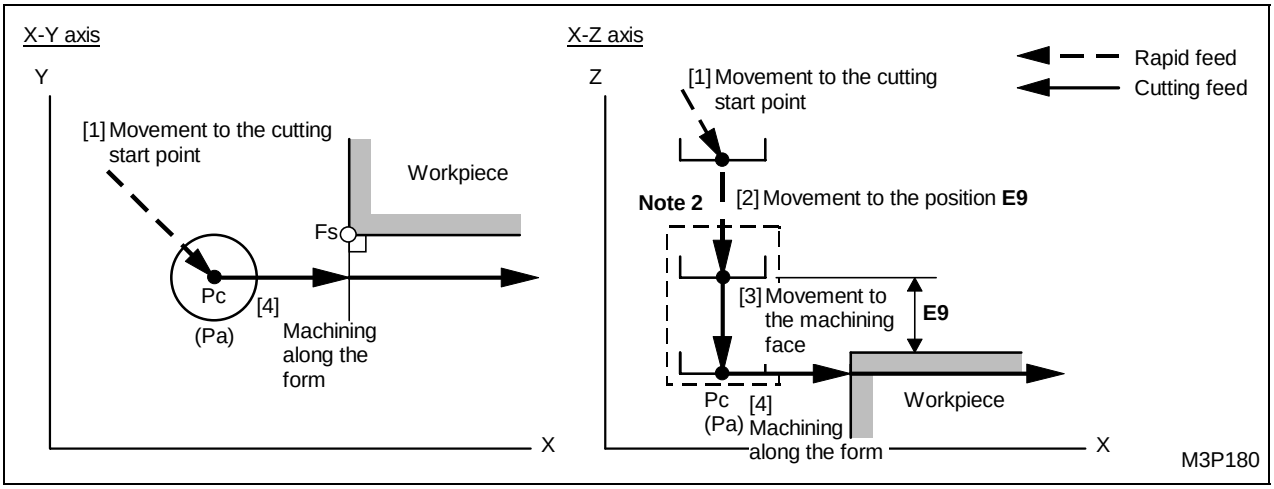
Fe: End point of form to be entered in the shape sequence

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

Note 1: When ? is displayed in the articles 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 articles.



Note 2: See Subsection 4-7-6 “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	99.999	99.999	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	E-MILL				◆			◆				
F2	E-MILL				◆		◆	◆				

Remark 1: Data in unit represent a maximum input value.

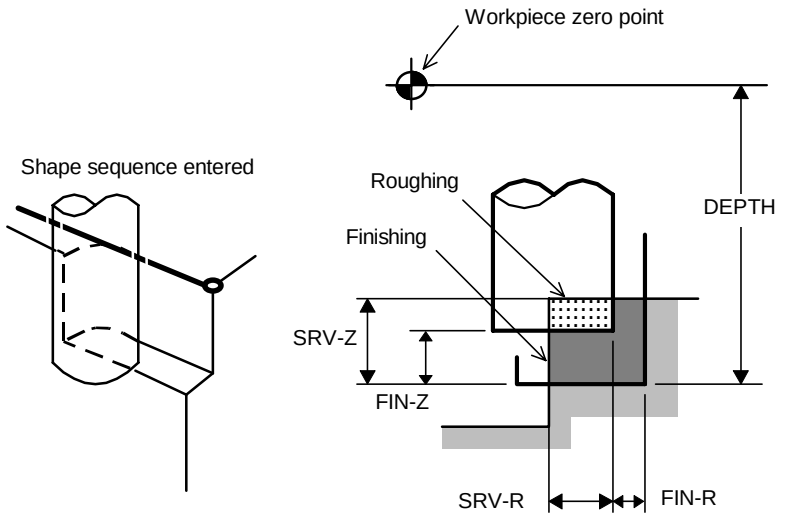
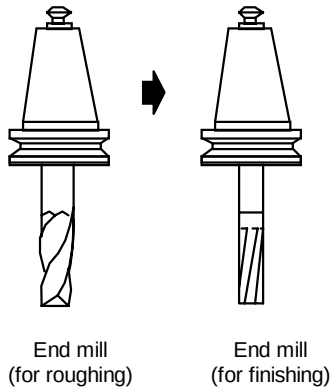
Remark 2: ◆: Data are not necessary to be set here.

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.

Q’ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z or SRV-R ≤ FIN-R
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, refer to Subsection 4-7-4.

LINE LFT unit	Tool sequence
 M3P181	 M3P173

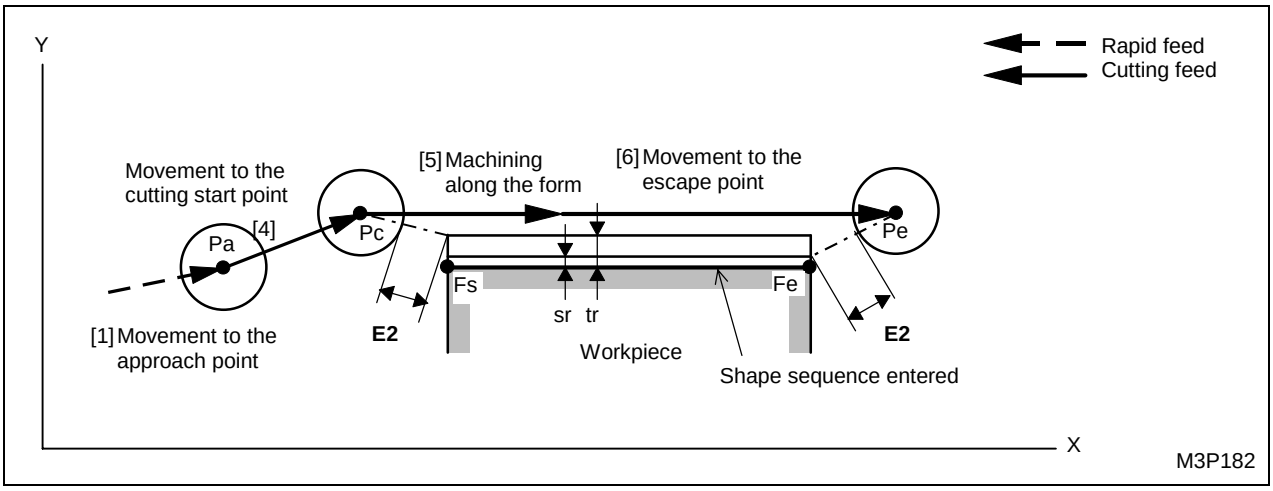
RGH: A roughness code should be selected out of the menu.

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

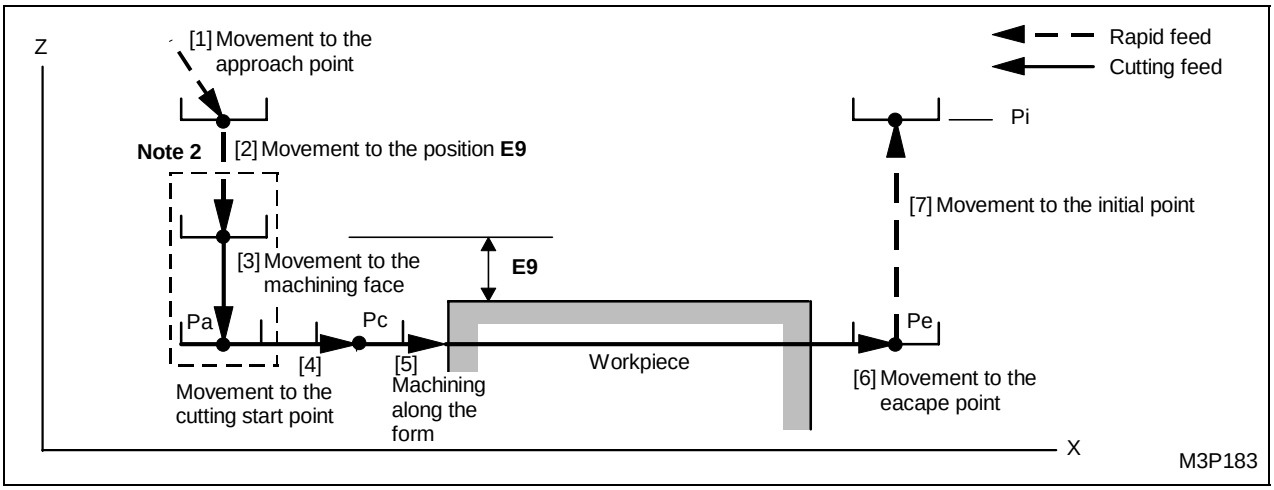
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

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

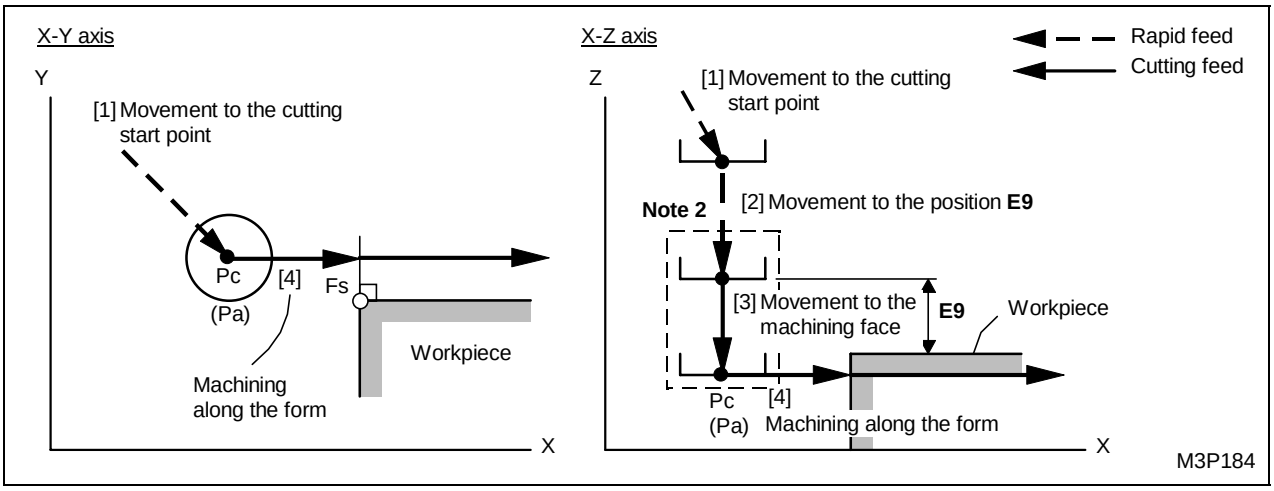
Fe: End point of form to be entered in the shape sequence

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

Note 1: When ? is displayed in the articles 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 articles.



Note 2: See Subsection 4-7-6, “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	99.999	99.999	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	E-MILL								◆				
F2	E-MILL							◆	◆				

Remark 1: Data in unit represent a maximum input value.

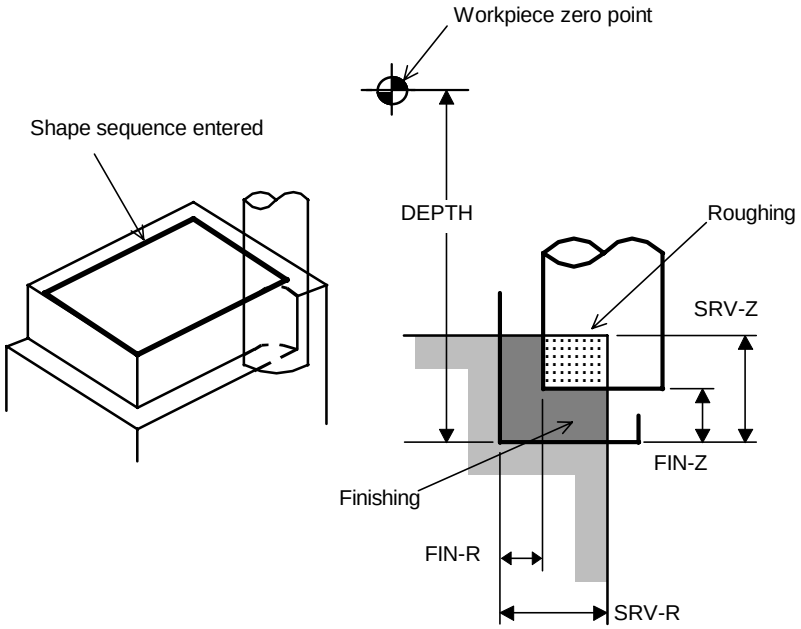
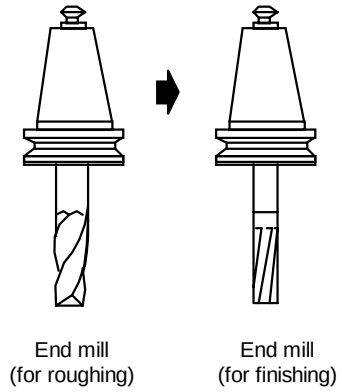
Remark 2: ◆: Data are not necessary to be set here.

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.

Q’ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z or SRV-R ≤ FIN-R
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, refer to Subsection 4-7-4.

LINE OUT unit	Tool sequence
	
M3P185	M3P173

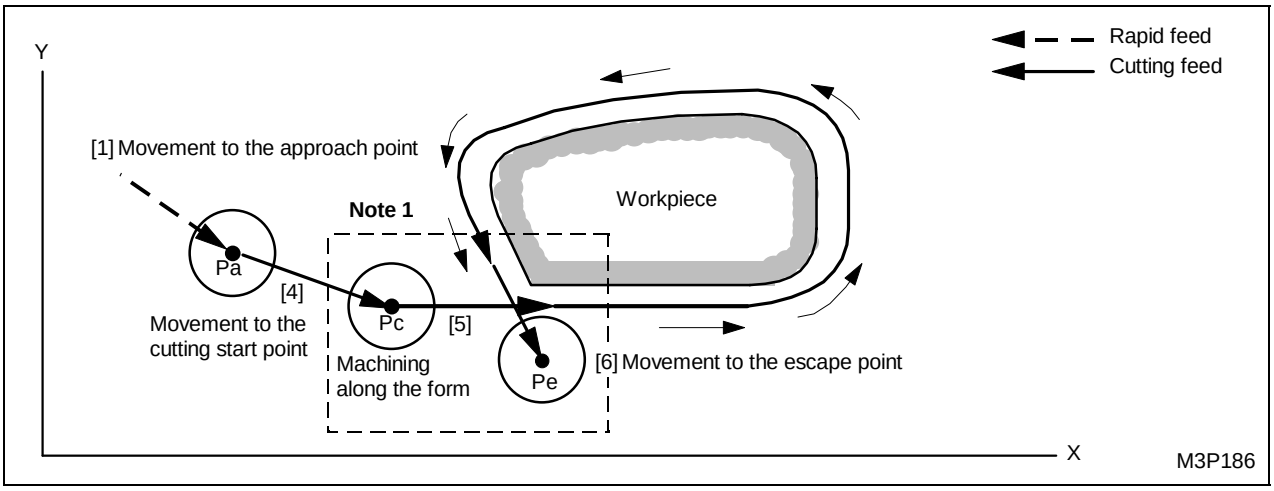
RGH: A roughness code should be selected out of the menu.

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

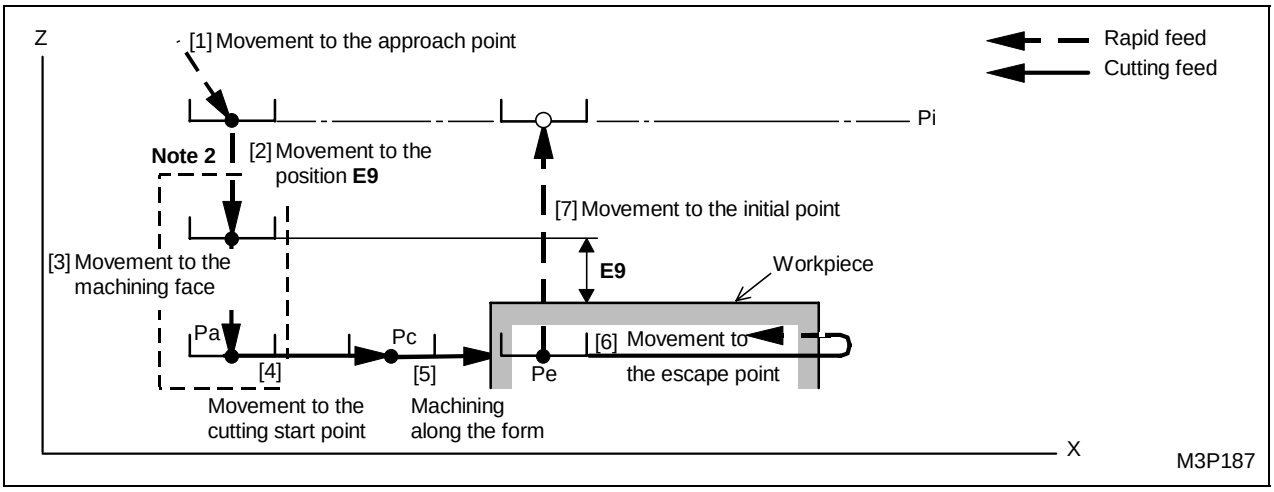
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

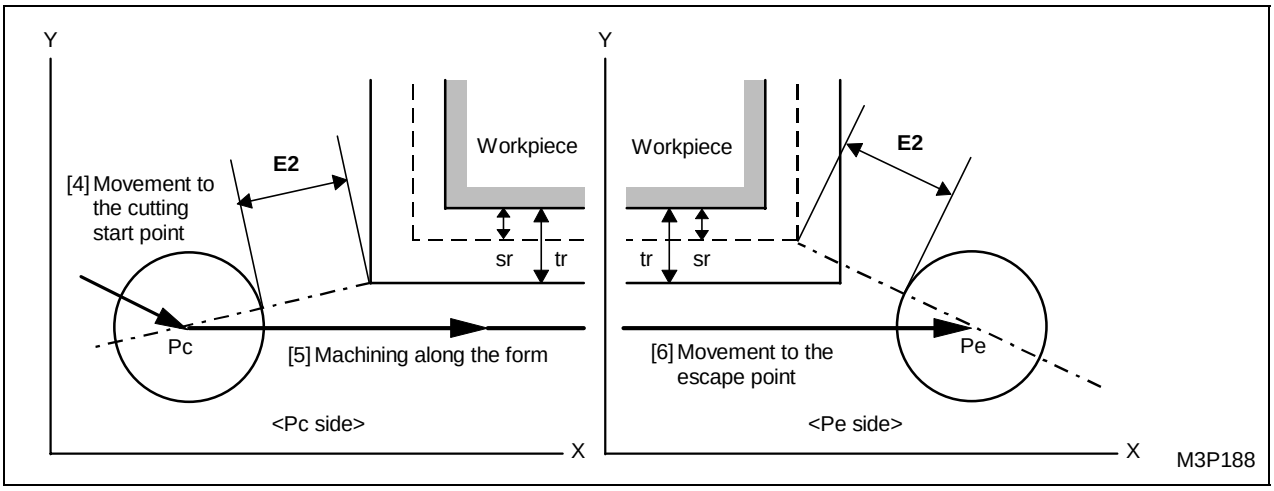
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

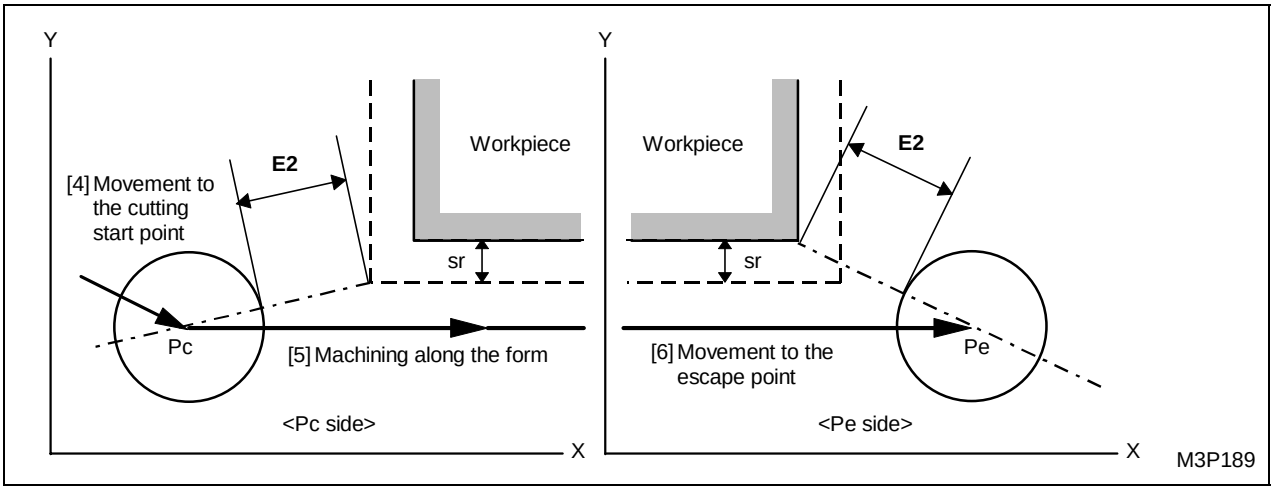
Note 1: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form

- In case of roughing



- In case of finishing



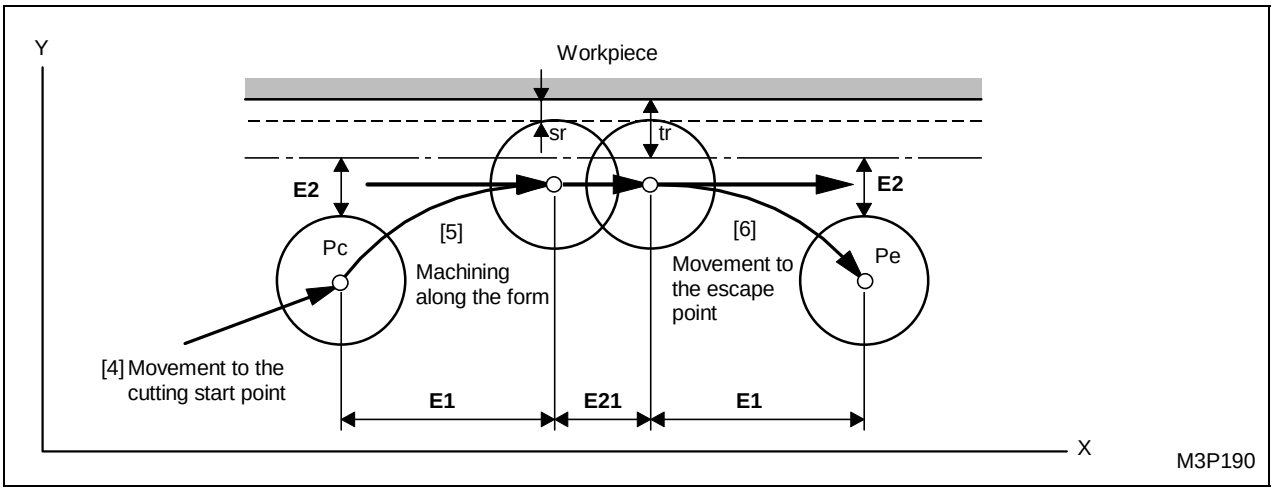
The bold codes represent parameter addresses.

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

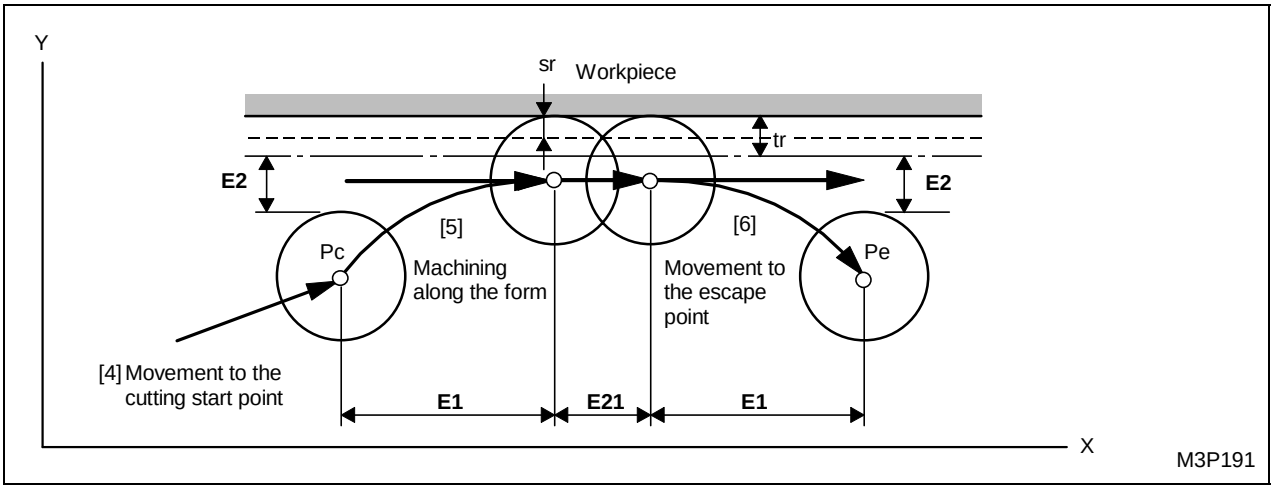
sr: Radial finishing allowance determined by the data FIN-R in the machining unit

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 4-7-6, “Precautions in line machining.”

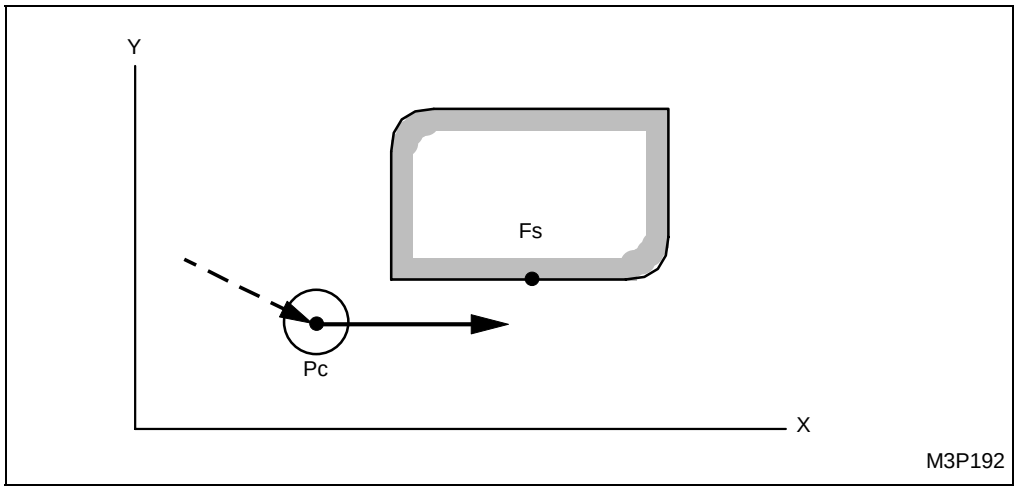
Note 3: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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).

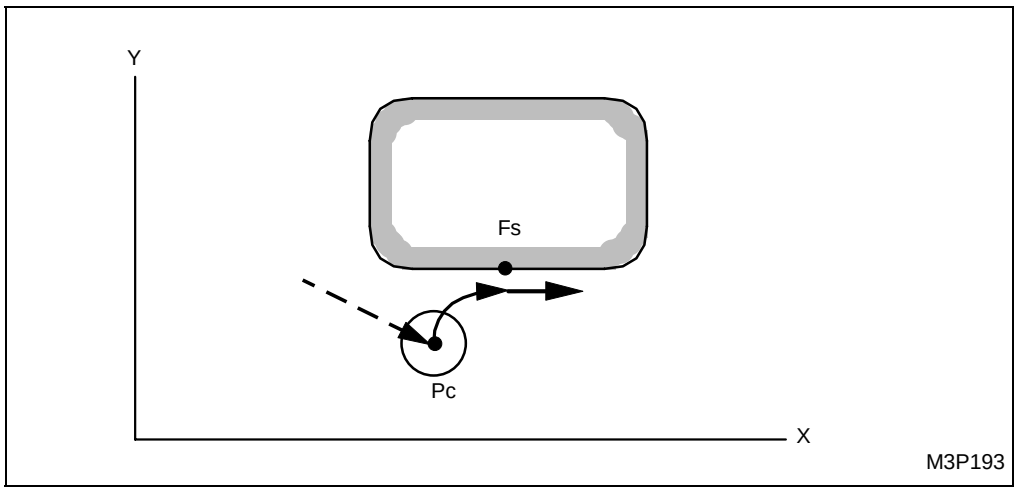
When ? is displayed in the articles APRCH-X, -Y

- Form having a convex point:



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

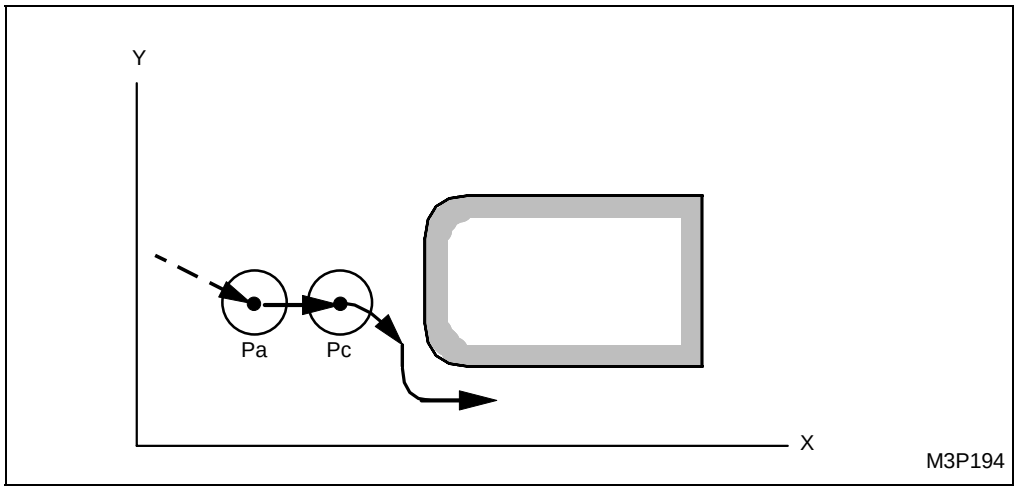
- Form having no convex point:



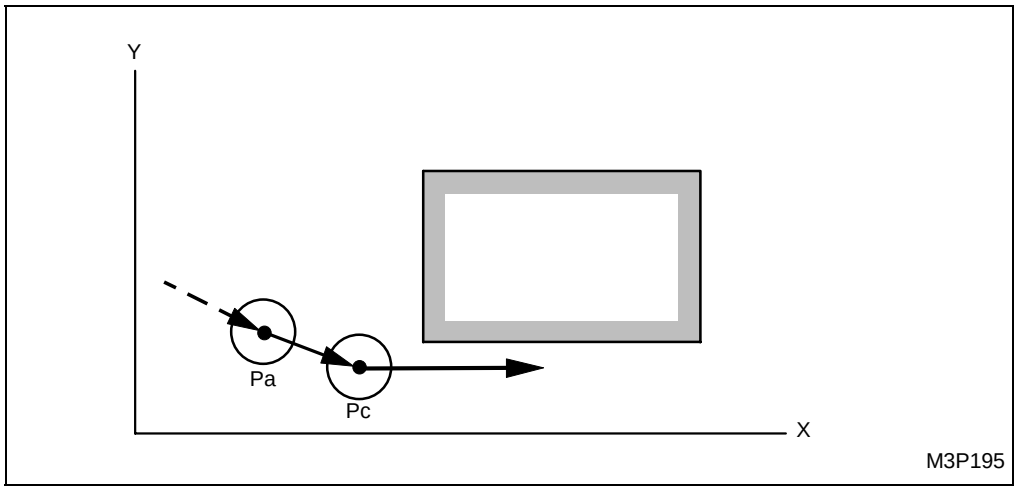
Cutting is started from the start point (Fs) entered in the shape sequence.

When the data is entered in the articles 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	99.999	99.999	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	E-MILL								◆			
F2	E-MILL							◆	◆			

Remark 1: Data in unit represent a maximum input value.

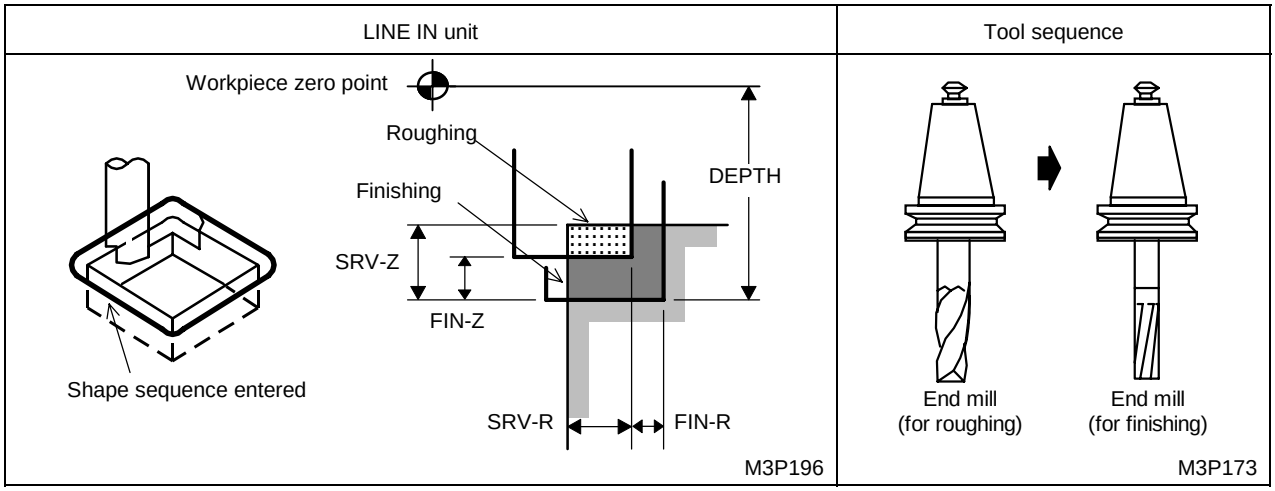
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z or SRV-R ≤ FIN-R
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-7-4.



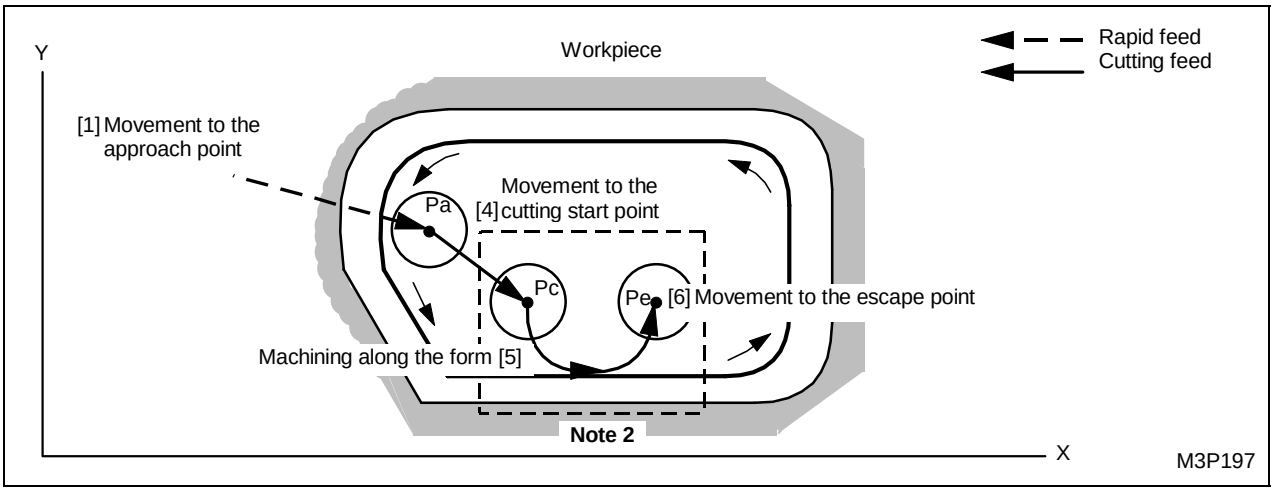
RGH: A roughness code should be selected out of the menu.

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

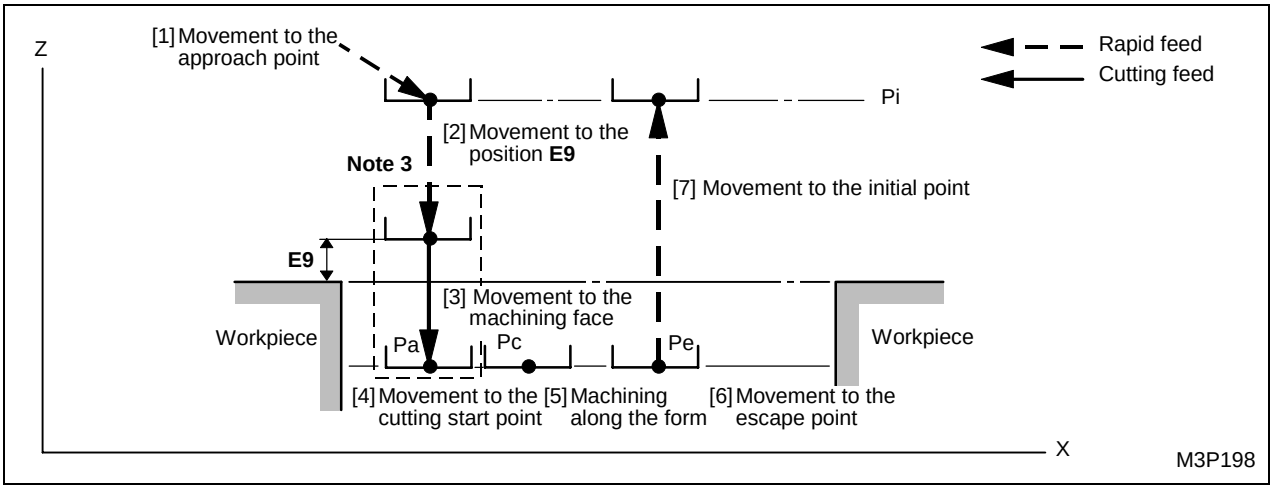
FIN-R: A radial finishing allowance is also automatically established once a roughness code has been selected.

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

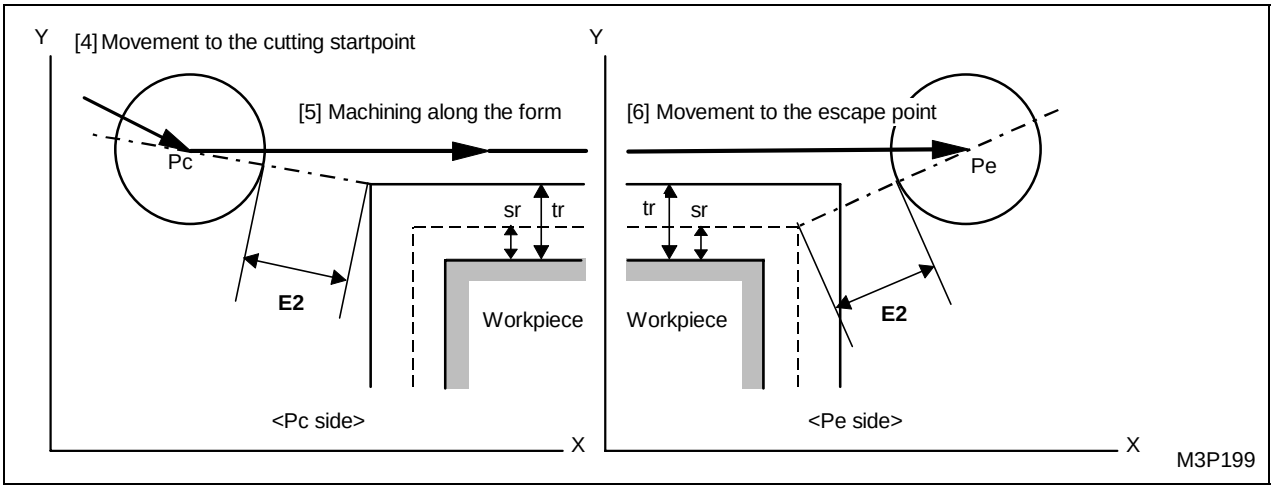
Pe: Escape point to be automatcially established

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

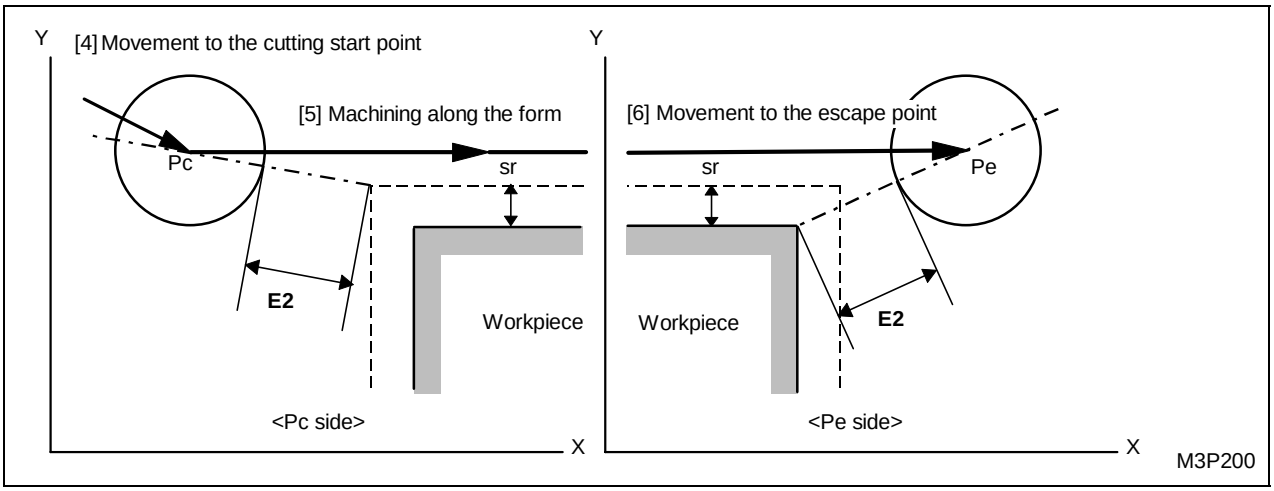
Note 2: Detail description of tool path near approach point and escape point

When the cutting begins near the convex form

- In case of roughing



- In case of finishing



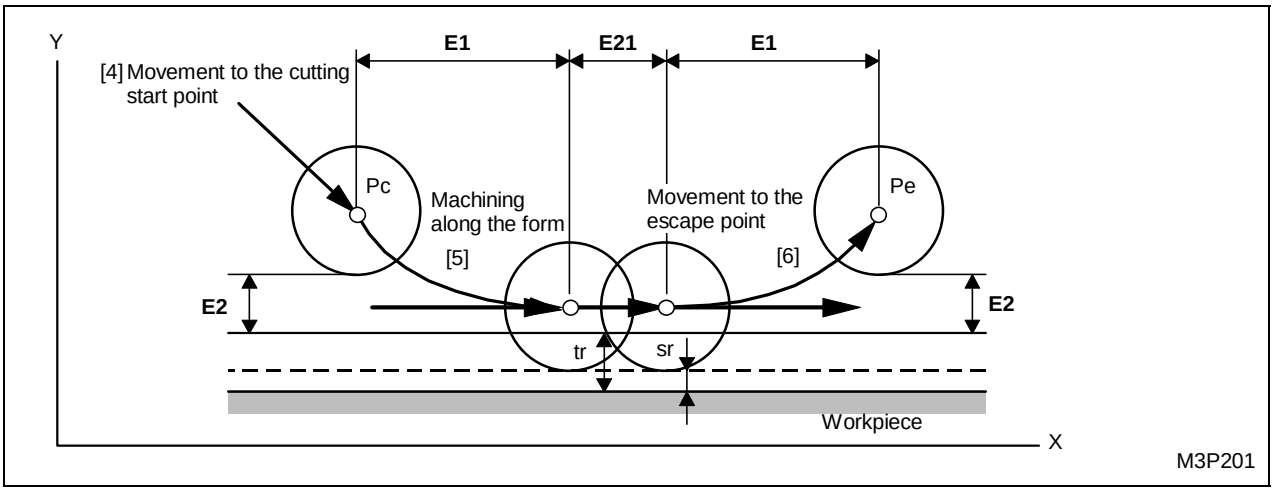
The bold codes represent parameter addresses.

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

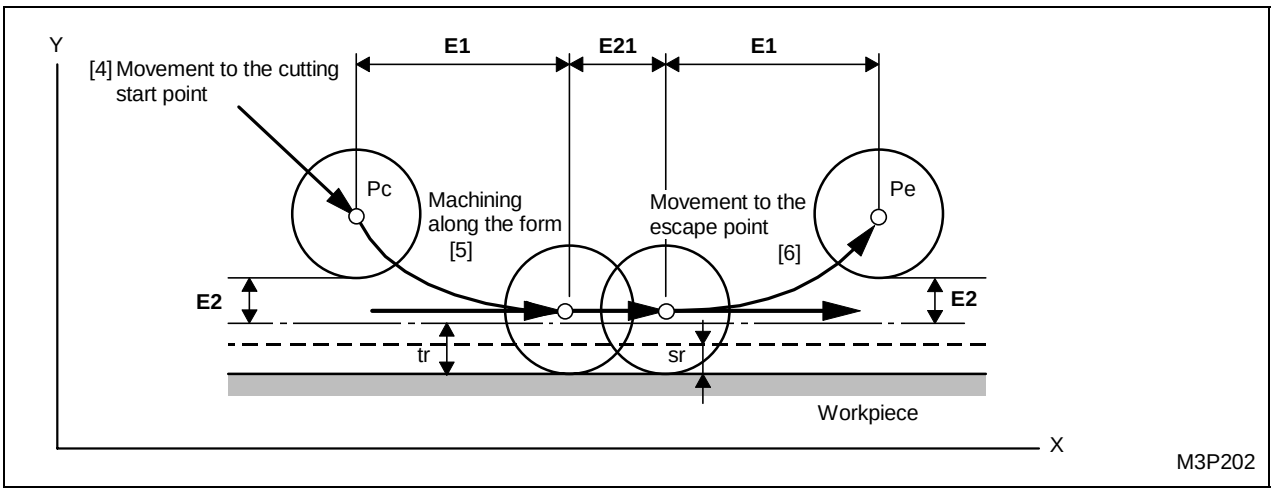
sr: Radial finishing allowance determined by the data FIN-R in the machining unit

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

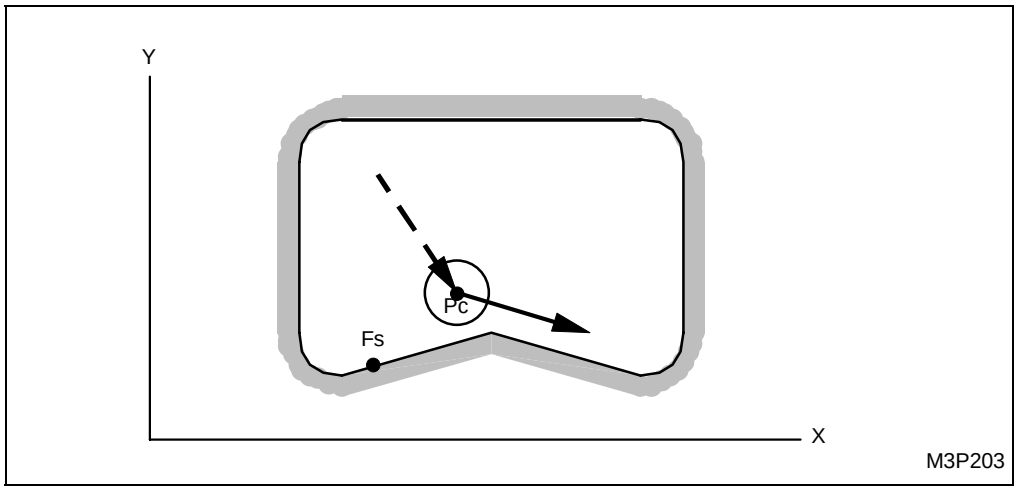
Note 3: See Subsection 4-7-6, “Precautions in line machining.”

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).

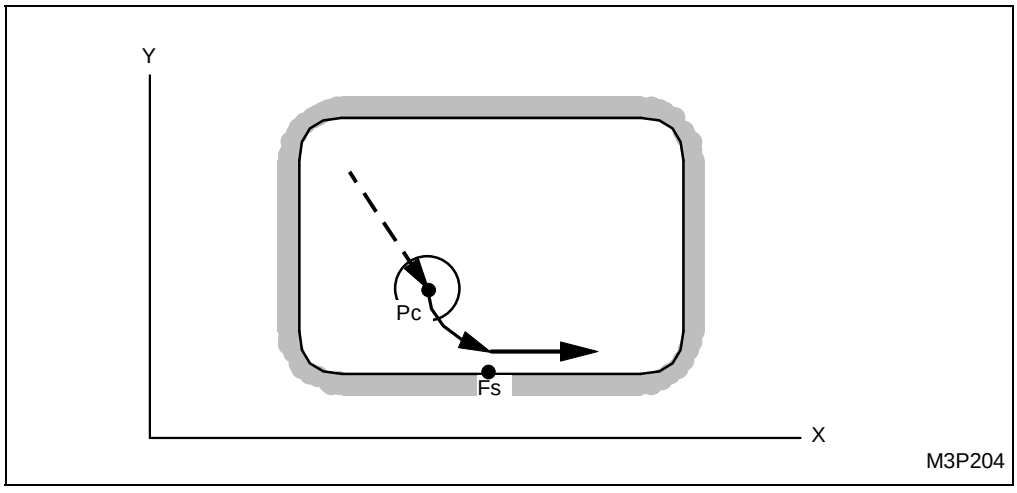
When ? is displayed in the articles APRCH-X, -Y

- Form having a convex point:



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

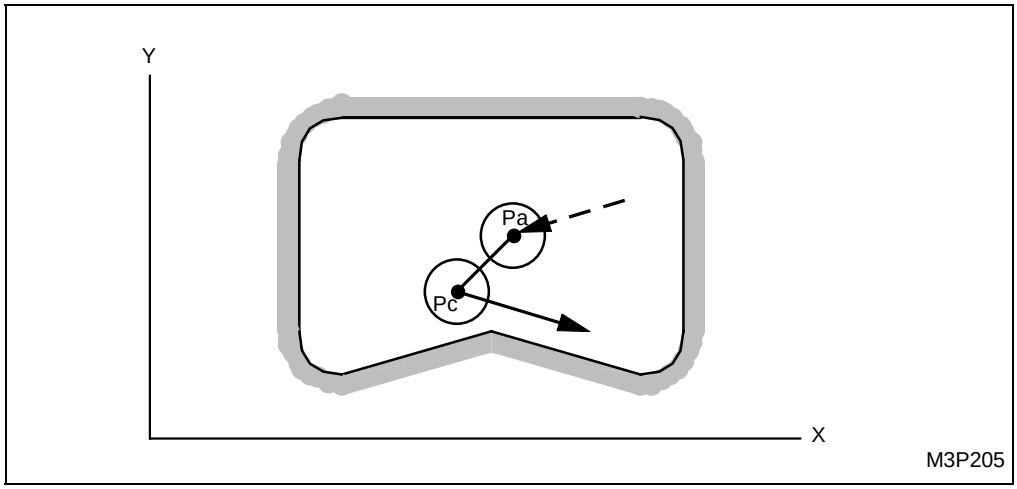
- Form having no convex point:



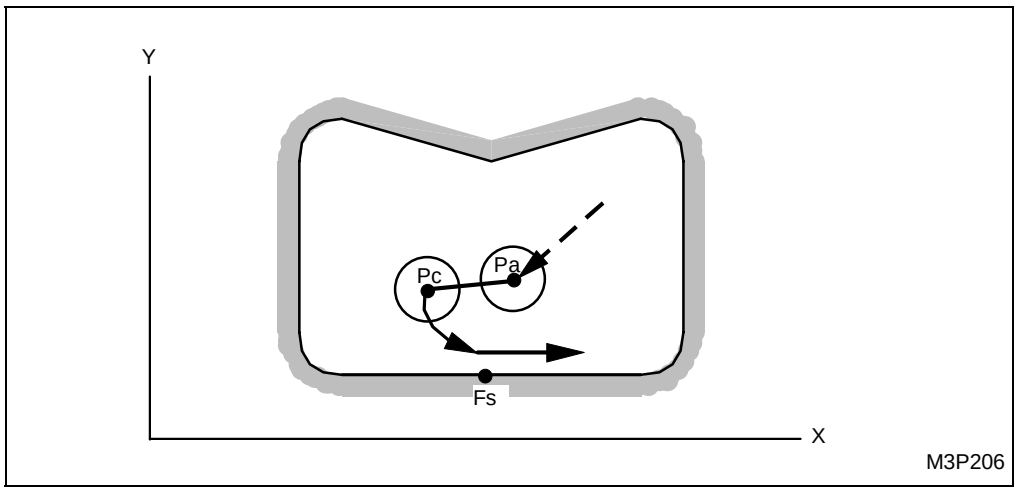
Cutting is started from the start point (Fs) entered in the shape sequence.

When the data is entered in the articles 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	CHF-C					◆		◆	◆				

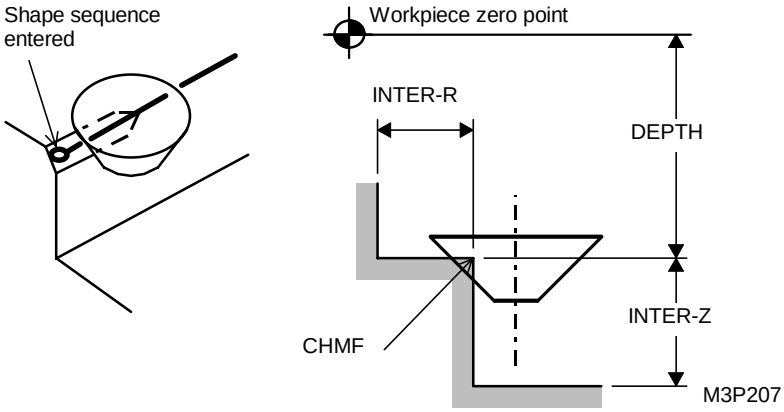
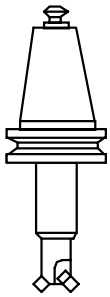
Remark 1: Data in unit represent a maximum input value.

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.

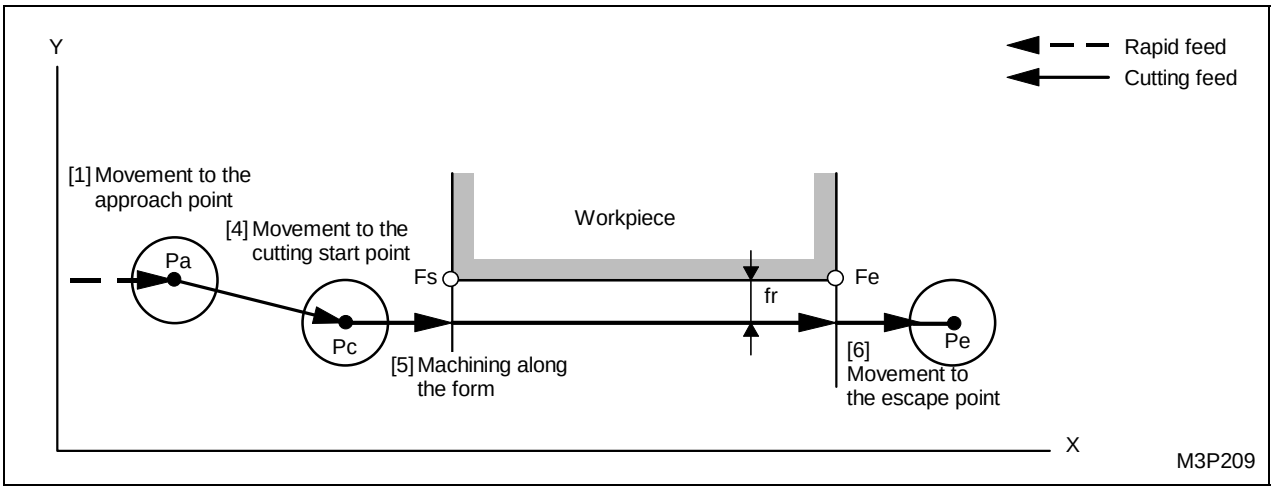
Remark 4: For the tool sequence data setting, refer to Subsection 4-7-4.

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

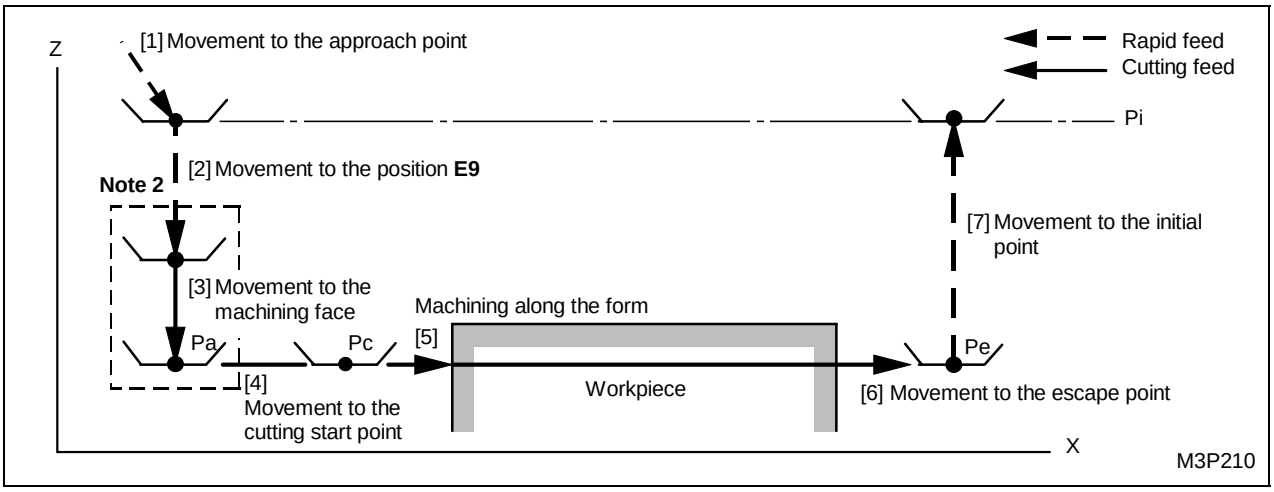
CHMF RGT unit	Tool sequence
 M3P207	 Chamfering cutter M3P208

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

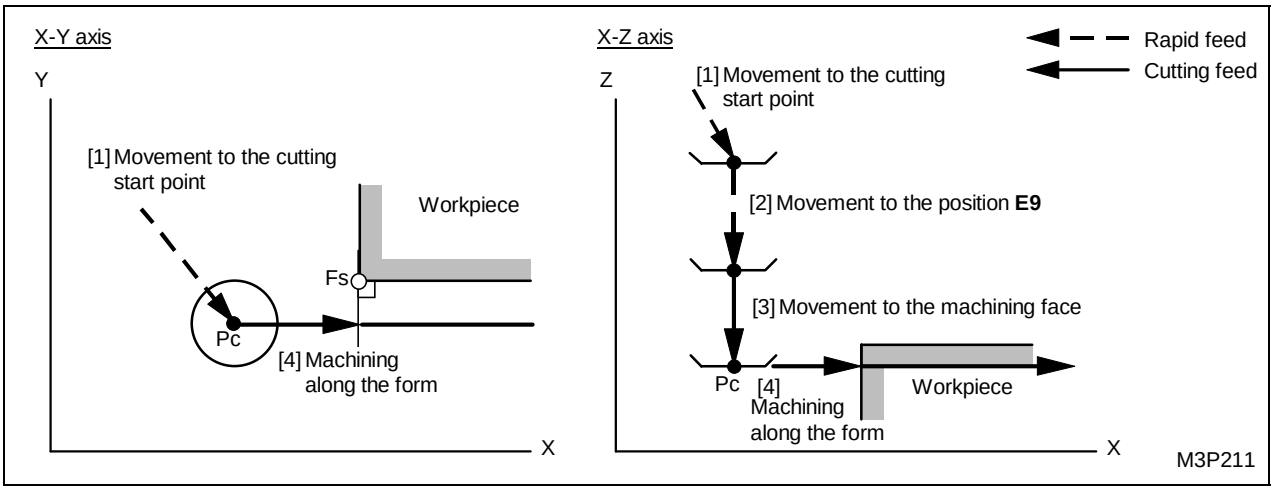
Fs: Start point of form to be entered in the shape sequence

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

Note 1: When ? is displayed in the articles 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 articles.



The bold codes represent parameter addresses.

Note 2: See Subsection 4-7-6, “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	CHF-C					◆		◆	◆				

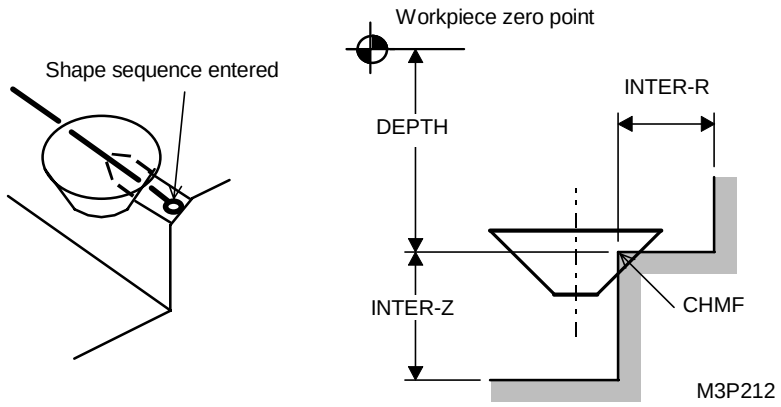
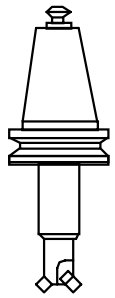
Remark 1: Data in unit represent a maximum input value.

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.

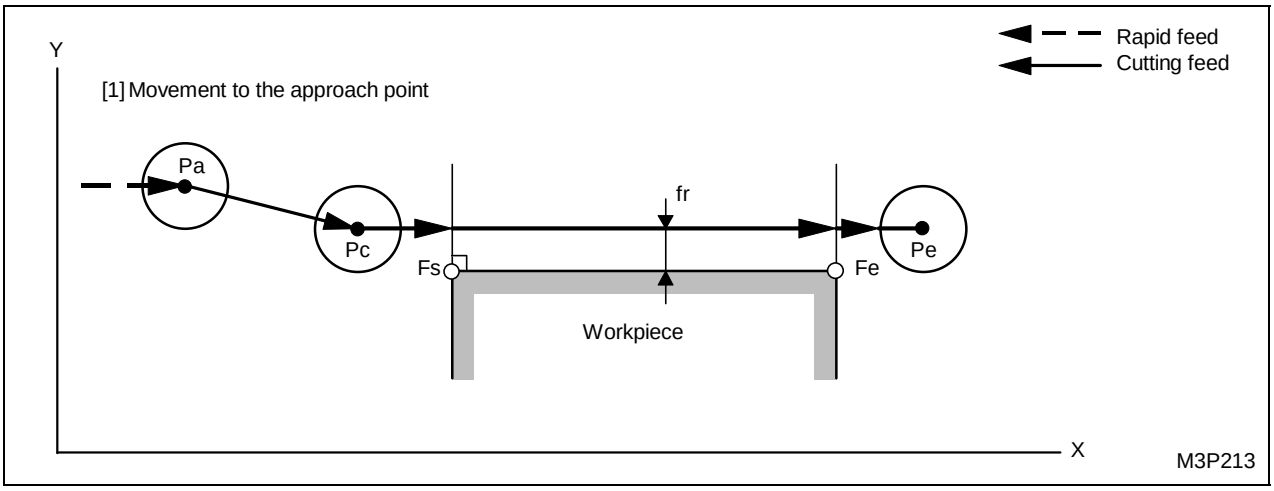
Remark 4: For the tool sequence data setting, refer to Subsection 4-7-4.

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

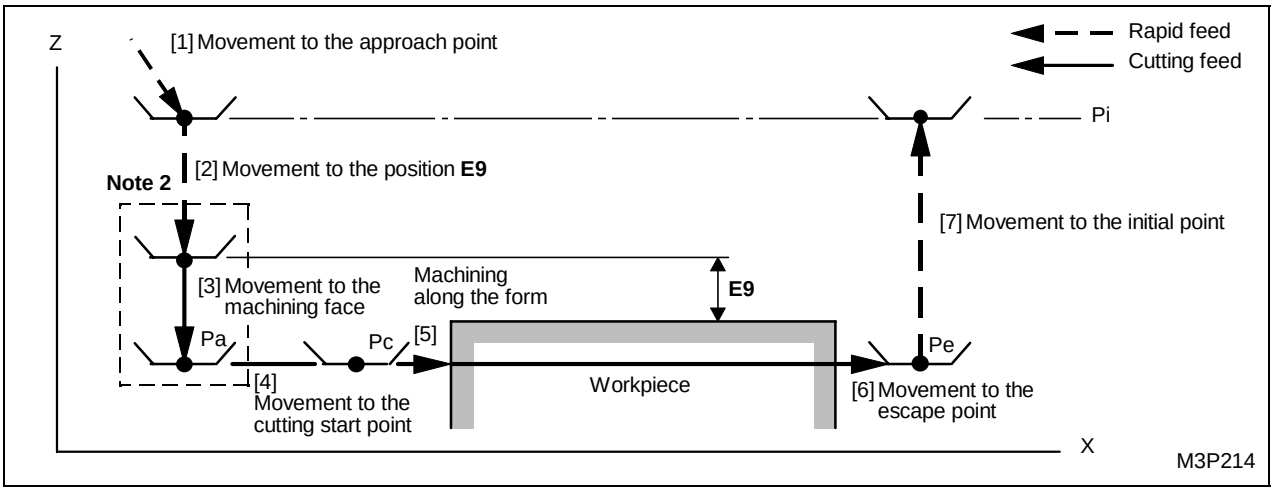
CHMT LFT unit	Tool sequence
	

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

Pc: Cutting start point to be automatically established

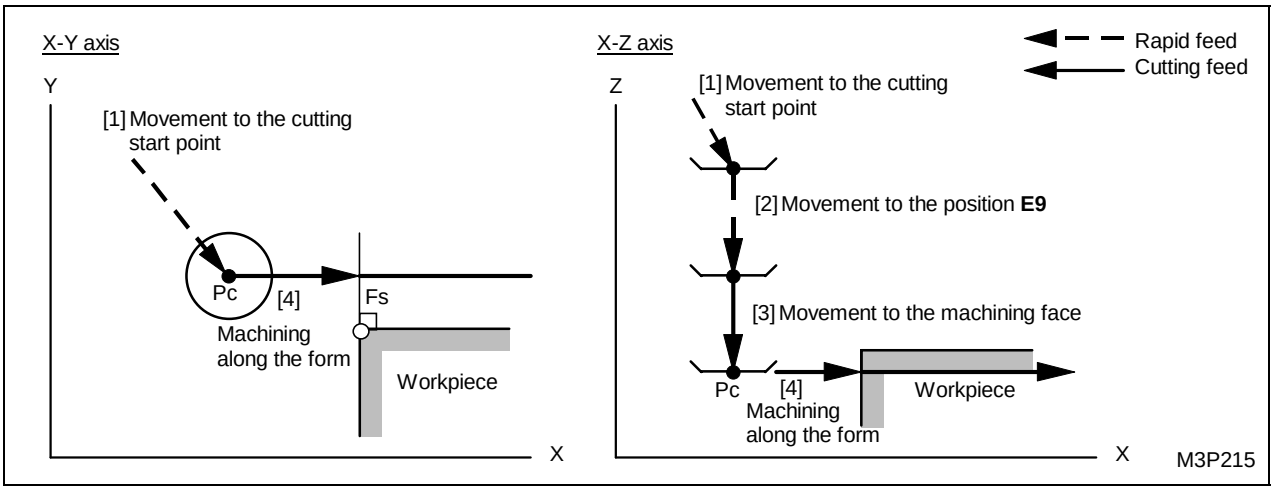
Fs: Start point of form to be entered in the shape sequence

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

Note 1: When ? is displayed in the articles 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 articles.



Note 2: See Subsection 4-7-6, “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	CHF-C							◆	◆				

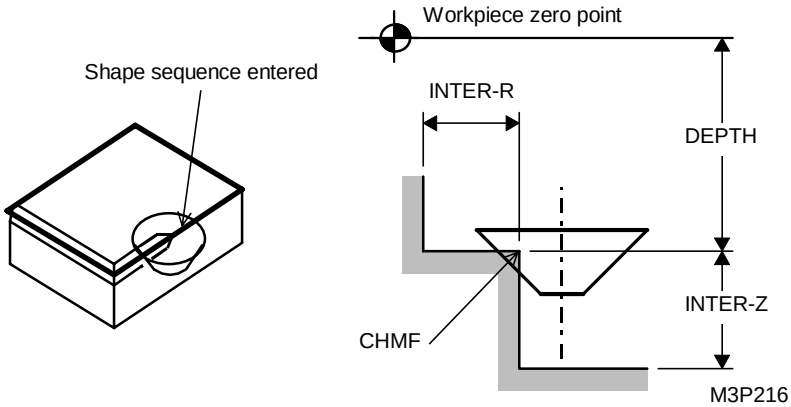
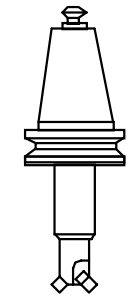
Remark 1: Data in unit represent a maximum input value.

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.

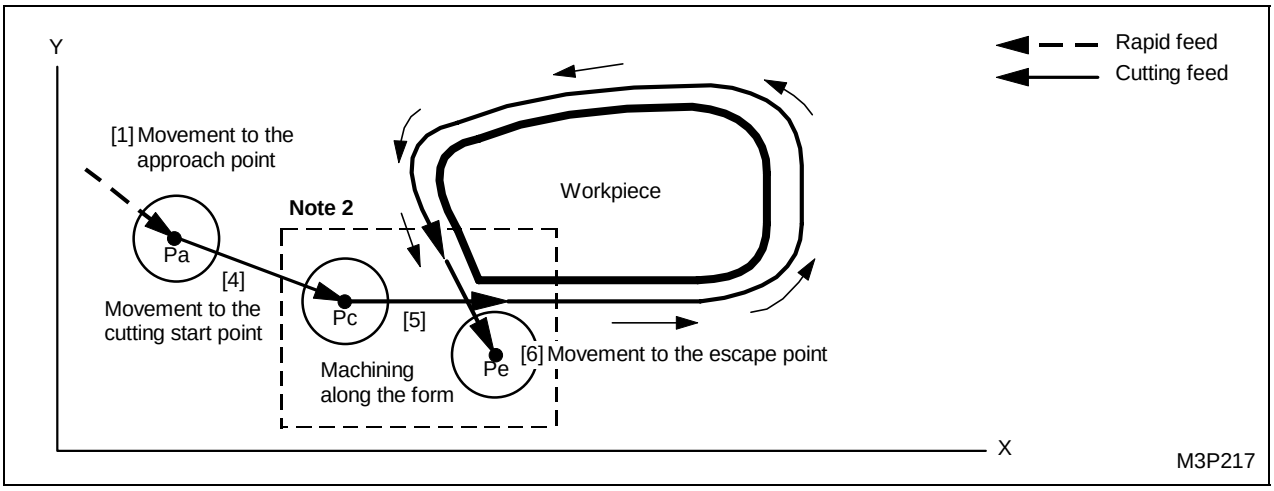
Remark 4: For tool sequence data setting, refer to Subsection 4-7-4.

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

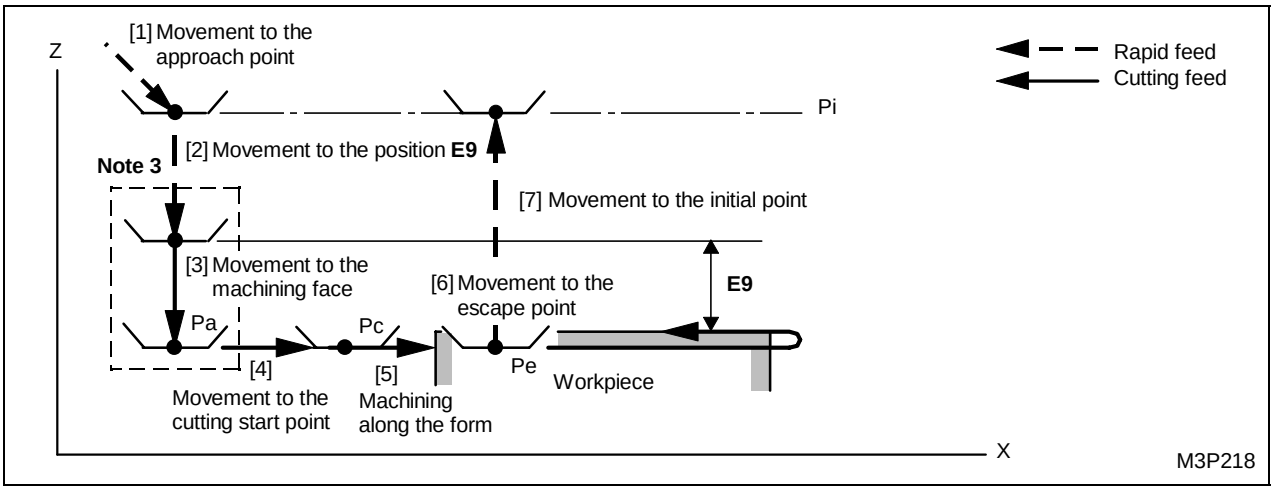
CHMF OUT unit	Tool sequence
	

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

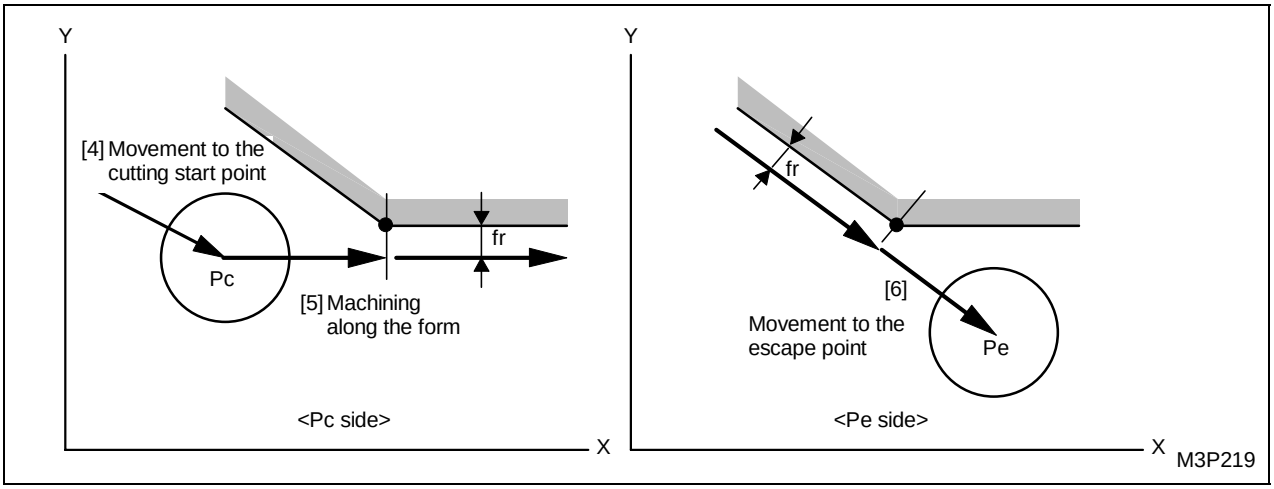
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

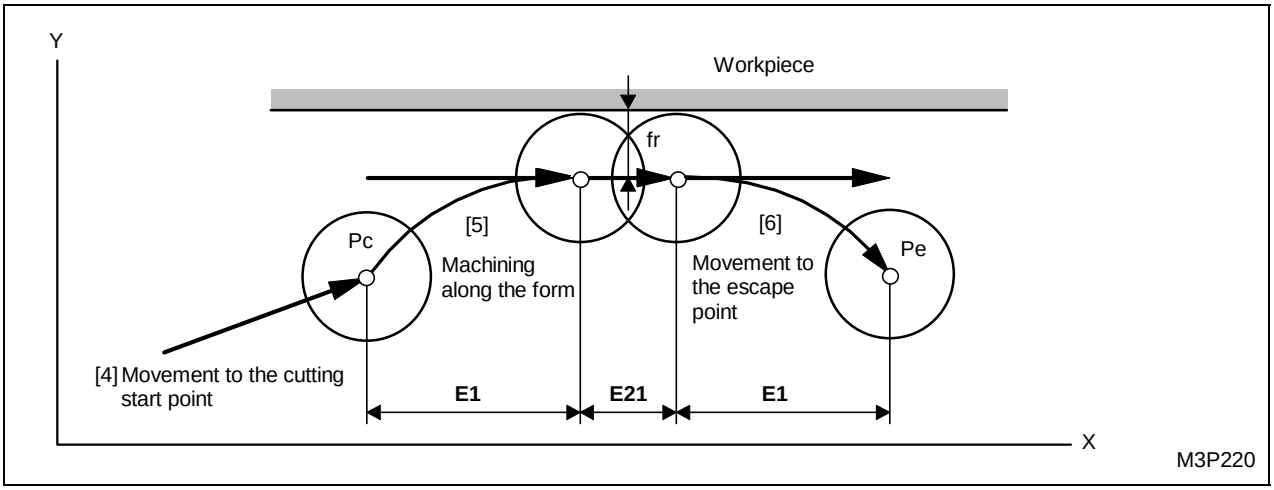
Note 1: The feedrate on tool path [3] is dependent upon the ZFD (Z-axis feed) in the tool sequence.

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

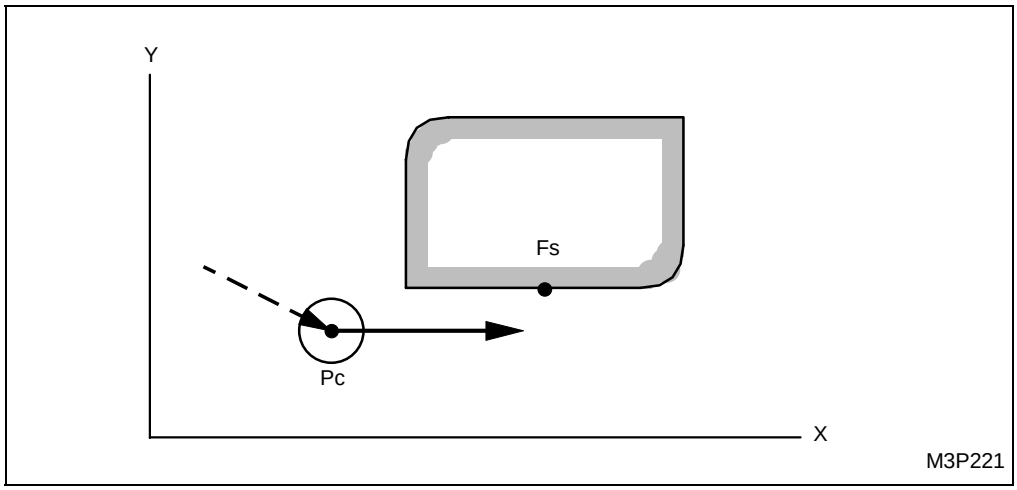
Note 3: See Subsection 4-7-6, “Precautions in line machining.”

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).

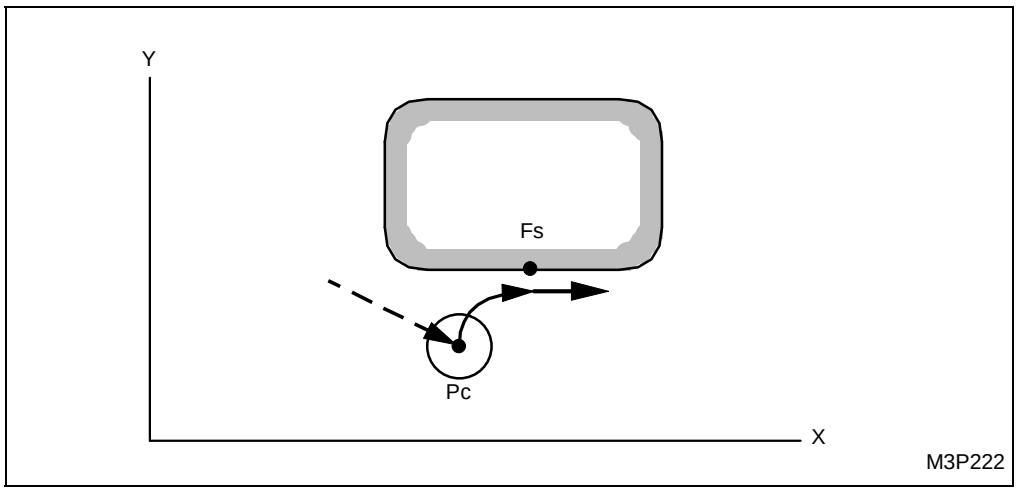
When ? is displayed in the articles APRCH-X, -Y

- Form having a convex point:



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

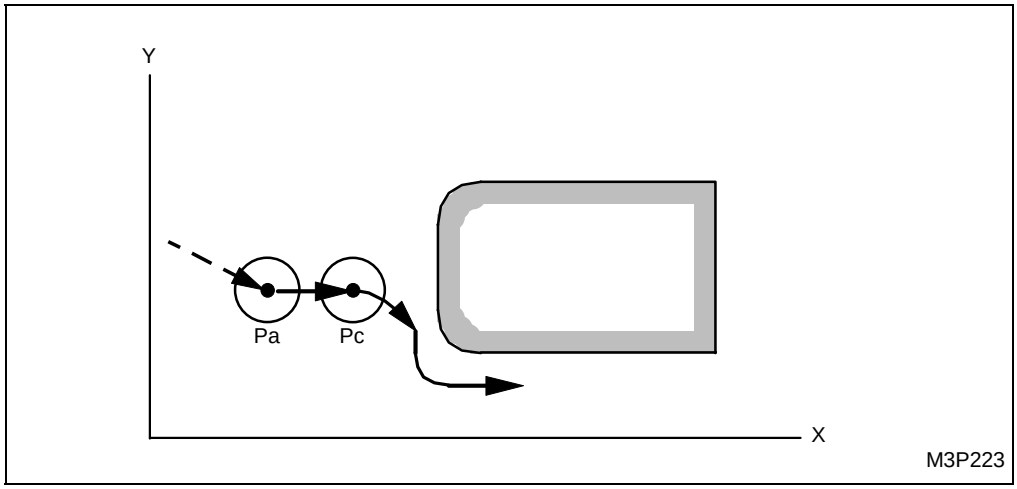
- Form having no convex point:



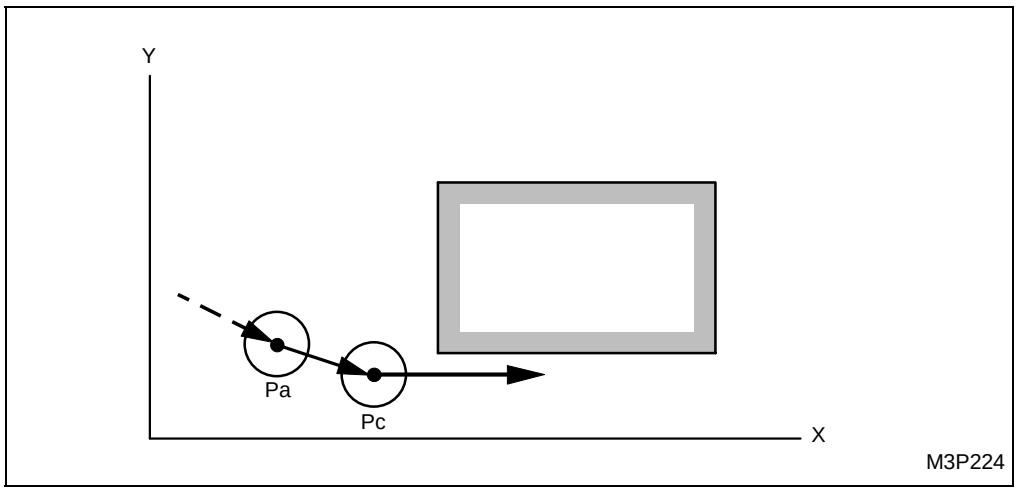
Cutting is started form the start point (Fs) entered in the shape sequence.

When the data is entered in the articles 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	CHF-C							◆	◆				

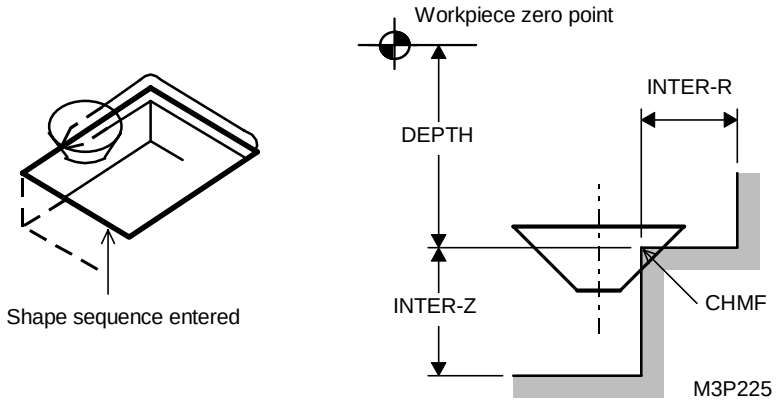
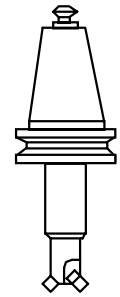
Remark 1: Data in unit represent a maximum input value.

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.

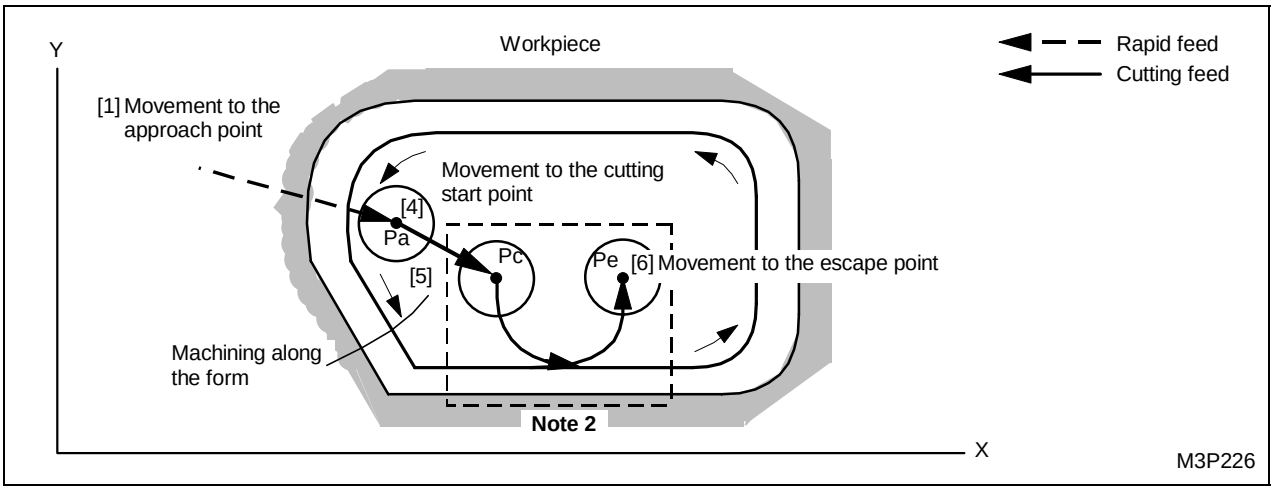
Remark 4: For sequence data setting, refer to Subsection 4-7-4.

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

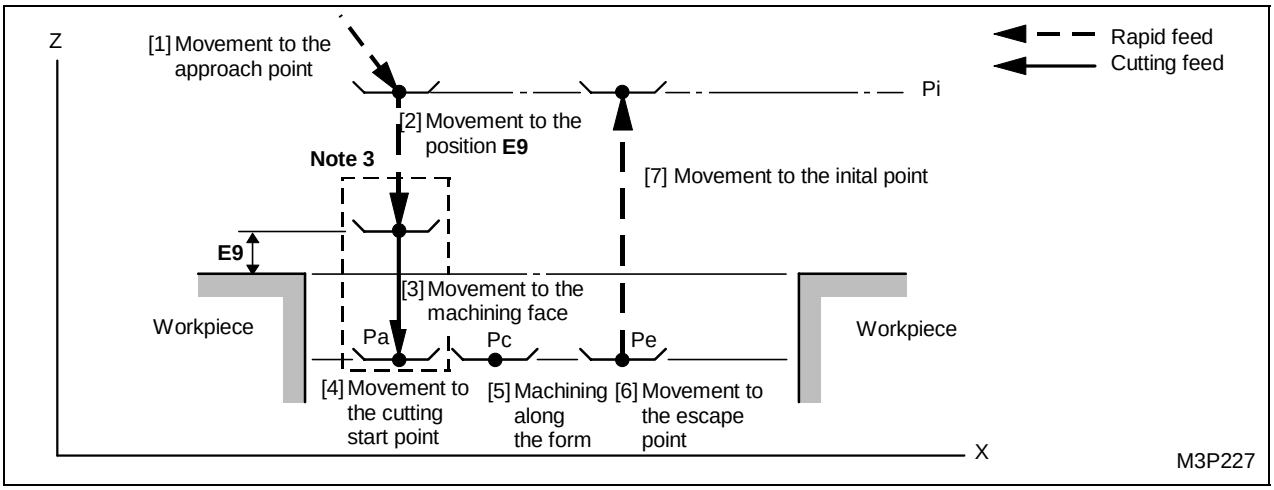
CHMF IN unit	Tool sequence
	

B. Tool path

X-Y axis



X-Z axis



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

Pa: Approach point to be determined by the data APRCH-X, -Y in the tool sequence

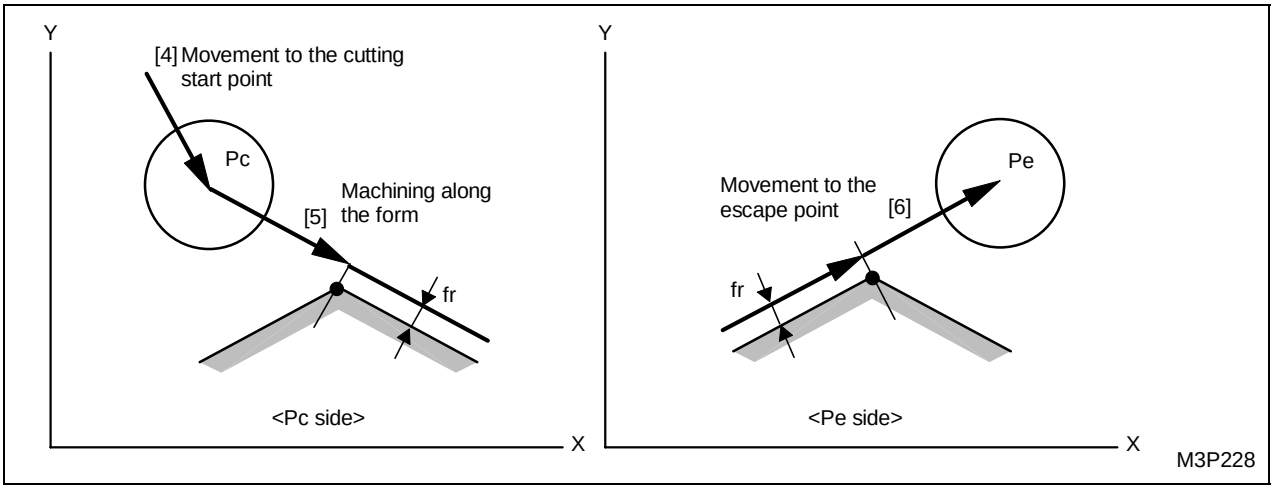
Pc: Cutting start point to be automatically established

Pe: Escape point to be automatically established

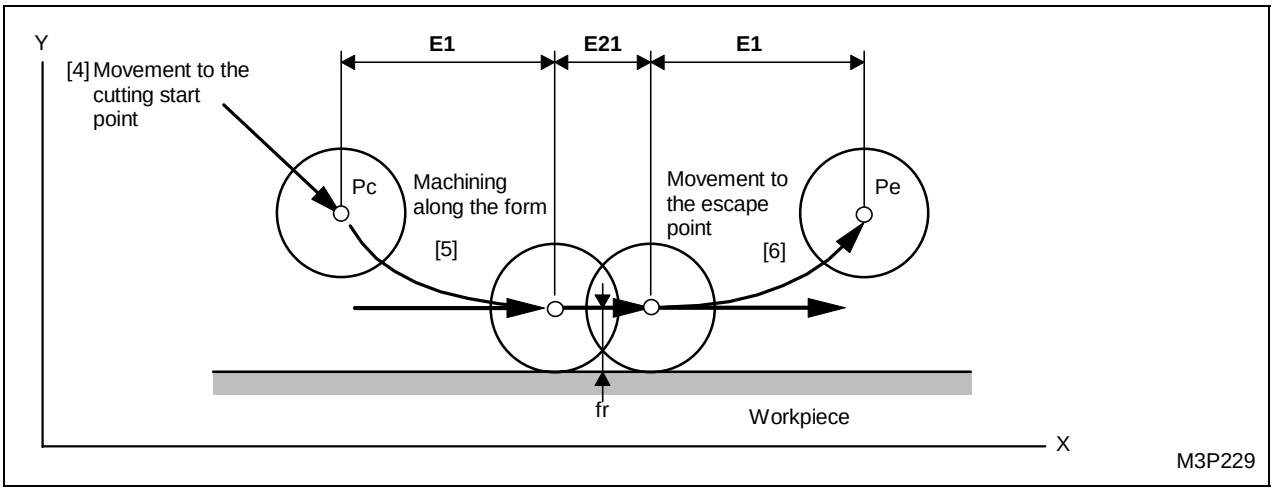
Note 1: The feedrate on tool path [3] is dependent upon the ZFD (Z-axis feed) in the tool sequence.

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

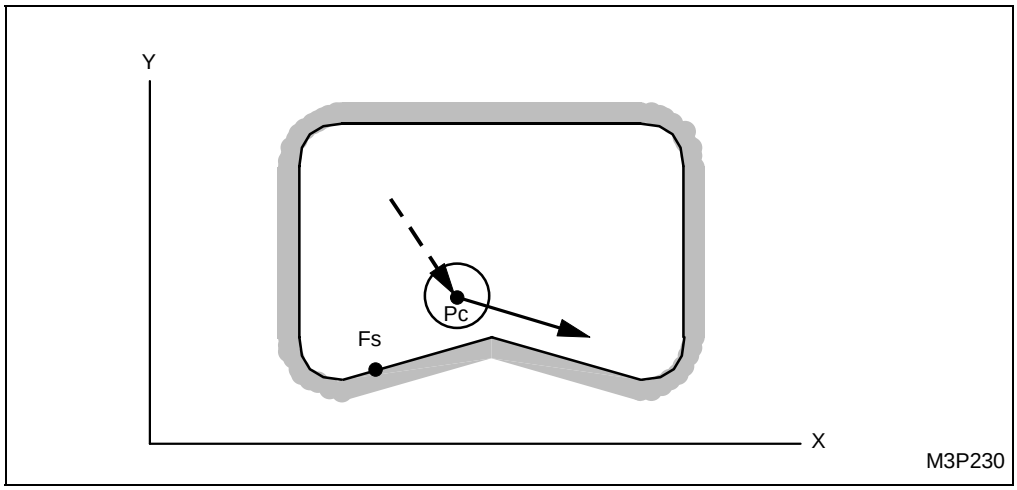
Note 3: See Subsection 4-7-6, “Precautions in line machining.”

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).

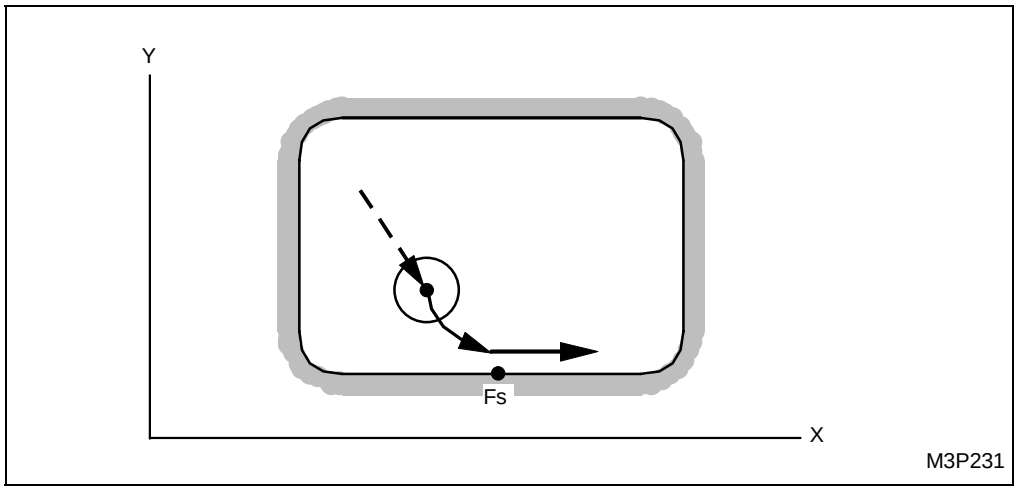
When ? is displayed in the articles APRCH-X, -Y

- Form having a convex point:



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

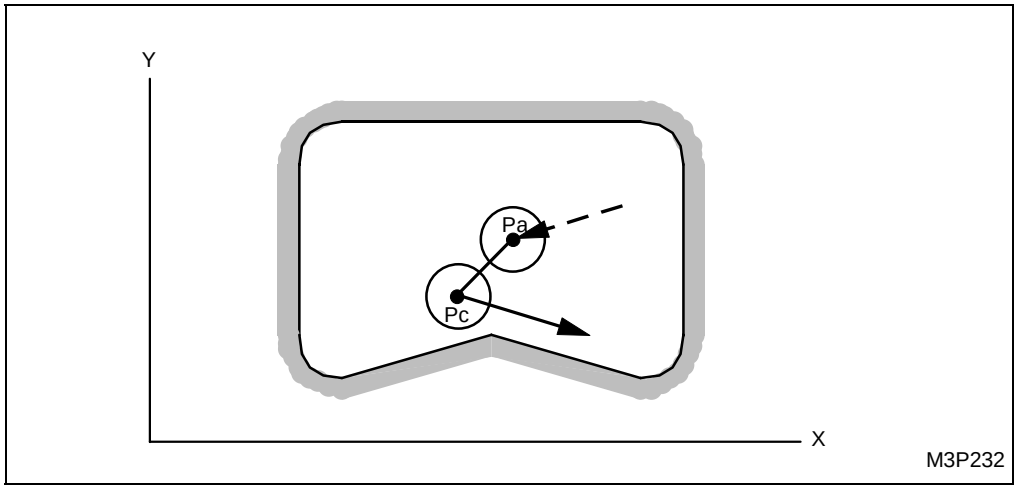
- Form having no convex point:



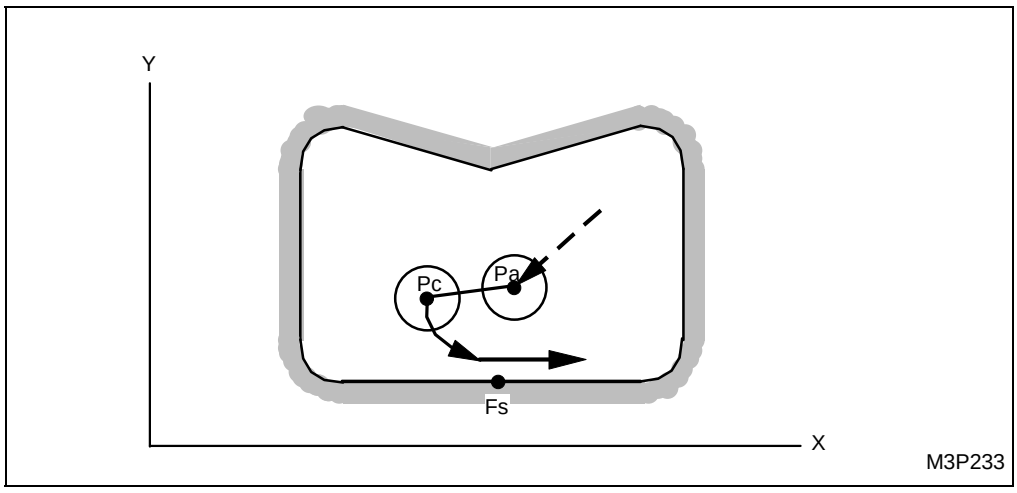
Cutting is started form the start point (Fs) entered in the shape sequence.

When the data is entered in the articles 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

4-7-4 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.

SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL								◆				
F2	E-MILL							◆	◆				
	↑	↑	↑	↑	↑	↑	↑	↑		↑	↑	↑	↑
	1	2	3	4	5	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 and 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

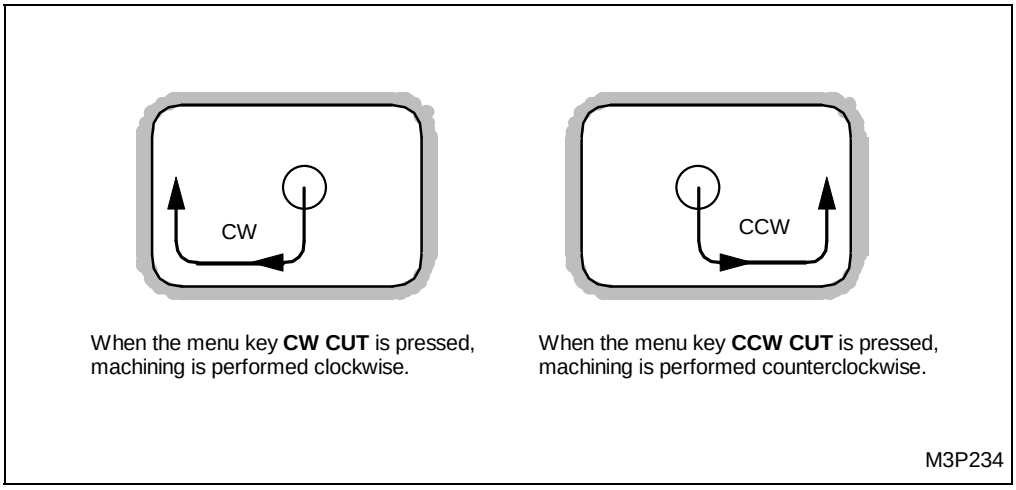
Enter an X, Y coordinates of the position at which a tool is to cut in axially.

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.

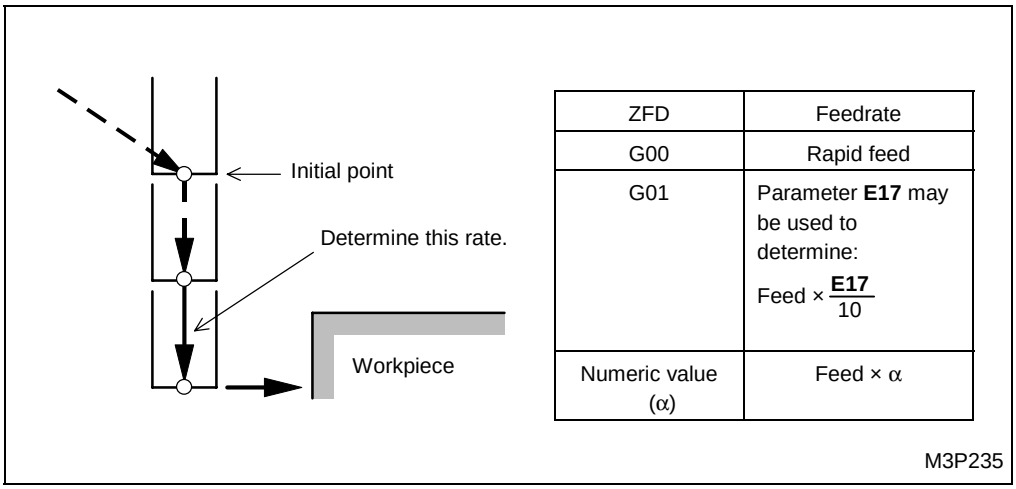
CW CUT	CCW CUT								



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 G01	RAPID G00								



ZFD	Feedrate
G00	Rapid feed
G01	Parameter **E17** may be used to determine: Feed × $\frac{\textbf{E17}}{10}$
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 4-7-6, “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 APPENDIX-1, “LIST OF M CODES.”)

4-7-5 Line machining shape sequence data

The data setting articles of shape sequence for the line machining units are the same as those for the face machining units. For the shape sequence data setting, see Input procedure in Subsection 4-8-7.

4-7-6 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]

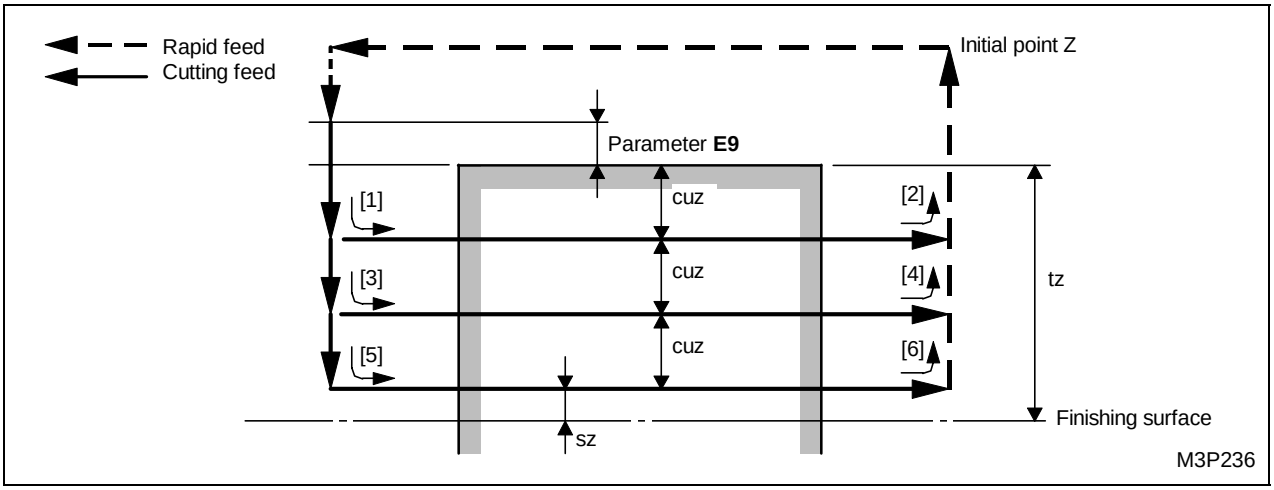


Fig. 4-18 Basic tool path

cuz: Cutting depth in the Z axis per pass

Calculation of cuz:

$$cuz = \frac{tz - sz}{n}$$

$$n = \frac{tz - sz}{cz}$$

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

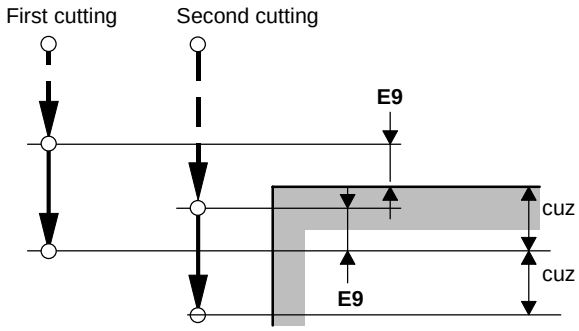
cz: Z-axial cutting depth DEP-Z to be entered in the tool sequence

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

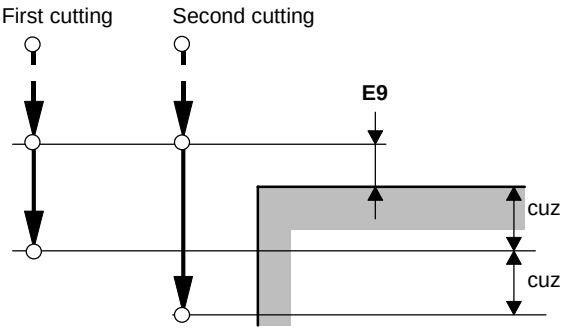
A. Cutting start position along the Z axis

Select one of the following two types:

(1) Rapid feed to the position **E9** above the machining surface



(2) Cutting start position fixed



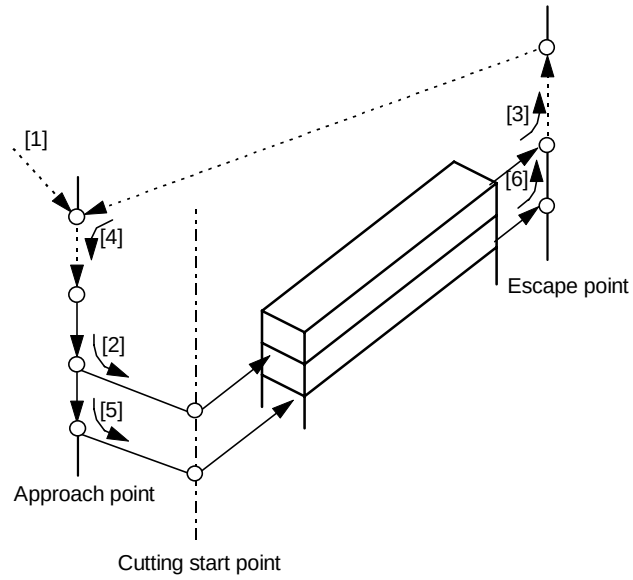
M3P237

The bold codes represent parameter addresses.

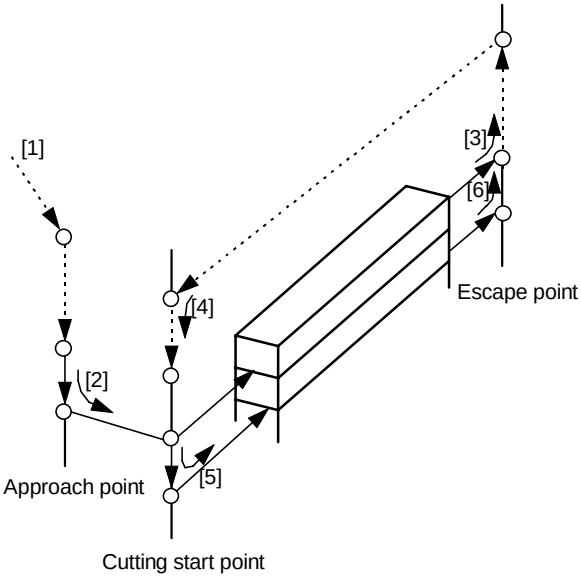
B. Type of routing via approach point

Select one of the following two types:

(1) Routing via approach point each time



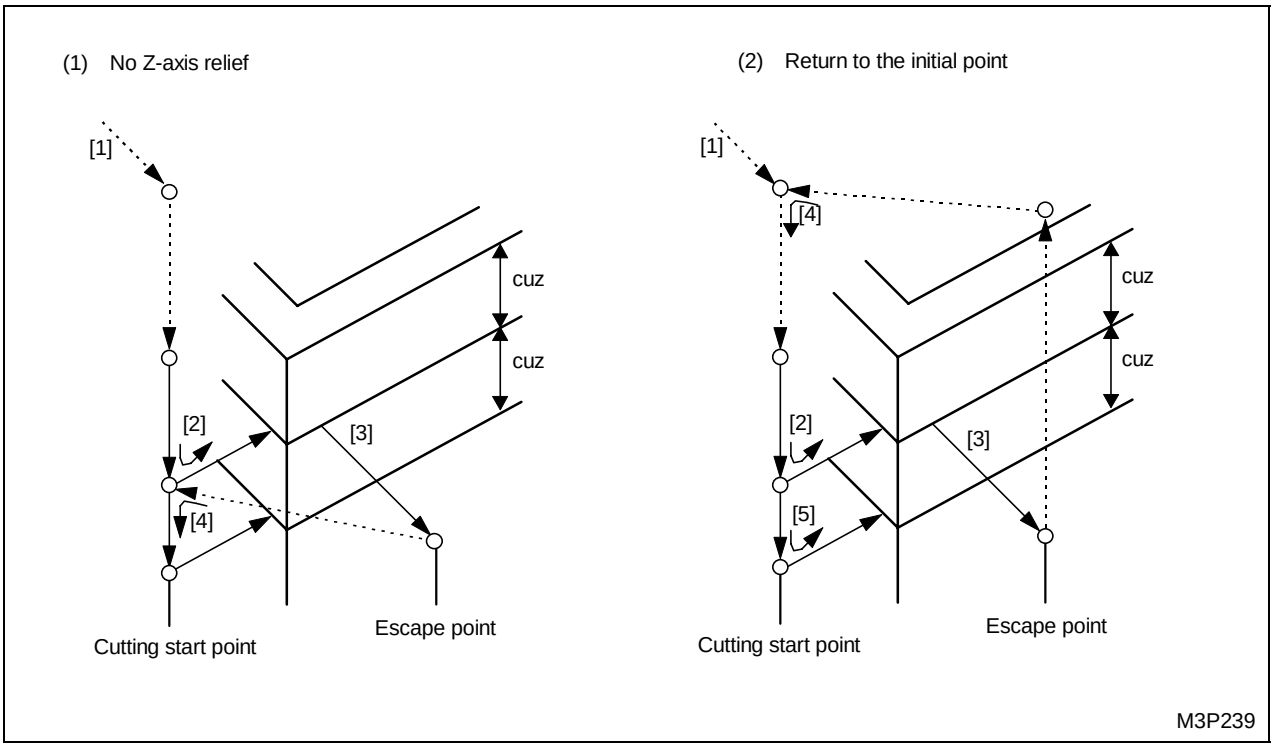
(2) Routing via approach point only in the first pass



M3P238

C. Type of escape along the Z axis after machining

Select one of the following two types:



Tool path setting parameter

Parameter **E95**

For A: bit 4 = 0: Cutting start position fixed -- (2)

1: Rapid feed to the position **E9** above the machining surface -- (1)

* As for pattern (1), the starting position of cutting feed is determined by the setting of parameter **E7** (instead of **E9**) from the second cutting when the following conditions are satisfied:

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

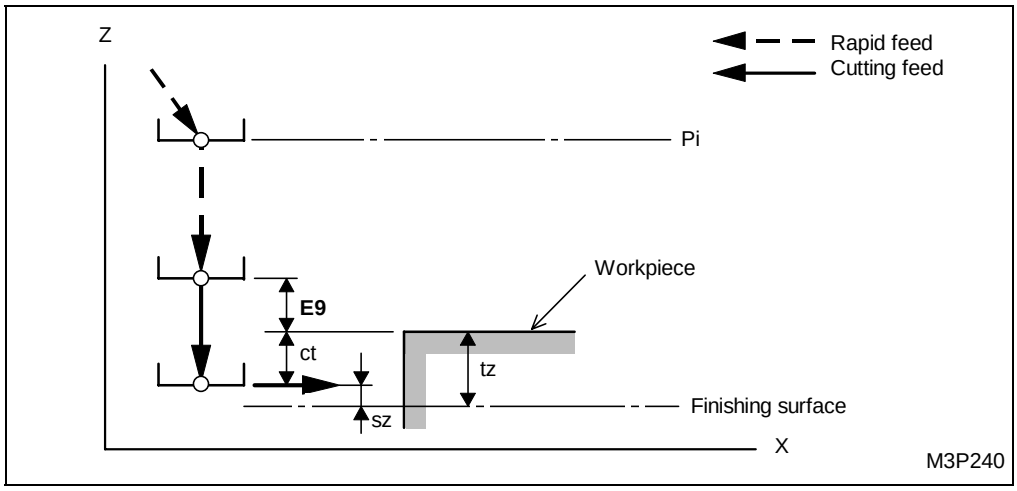
1: No escape along the Z axis -- (1)

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.

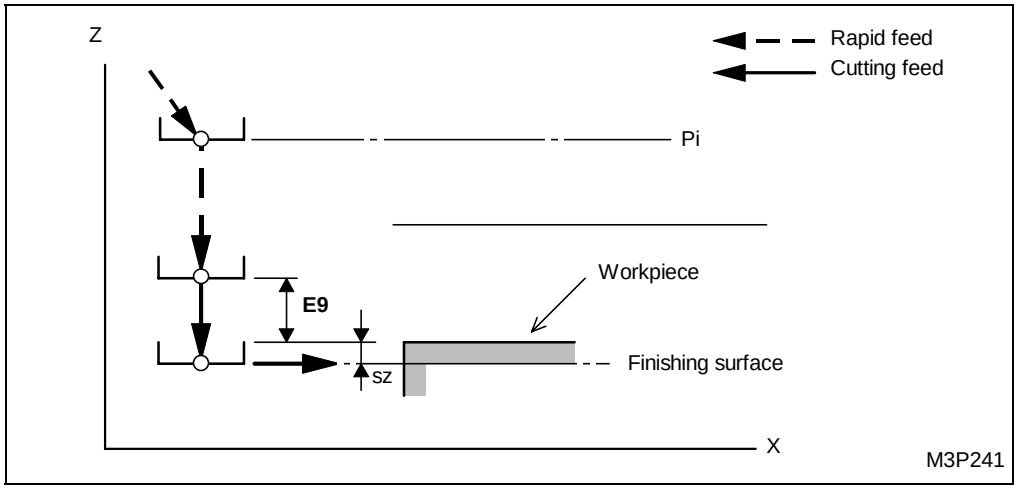
Note 2: The tool path shown at basic tool path above is selected automatically for machining units that are not subject to the selection of the parameter **E95.**

2. Detail tool path of a Z-axial cut-in

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

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: The starting allowance of axial cutting, specified by parameter **E9**, will become equal to parameter **E7** if the following three states occur at the same time:

- Bit 6 of parameter **E95** is set to 1.
- A pre-machining tool is included in that tool sequence.
- The machining unit is either central linear, right-hand linear, left-hand linear, outside linear or inside linear machining.

Note 2: The starting allowance of cutting in radial direction, specified by parameter **E2**, will become equal to parameter **E5** if the following three states occur at the same time:

- Bit 7 of parameter **E95** is set to 1.
- A pre-machining tool is included in that tool sequence.
- The machining unit is either outside linear or inside linear machining.

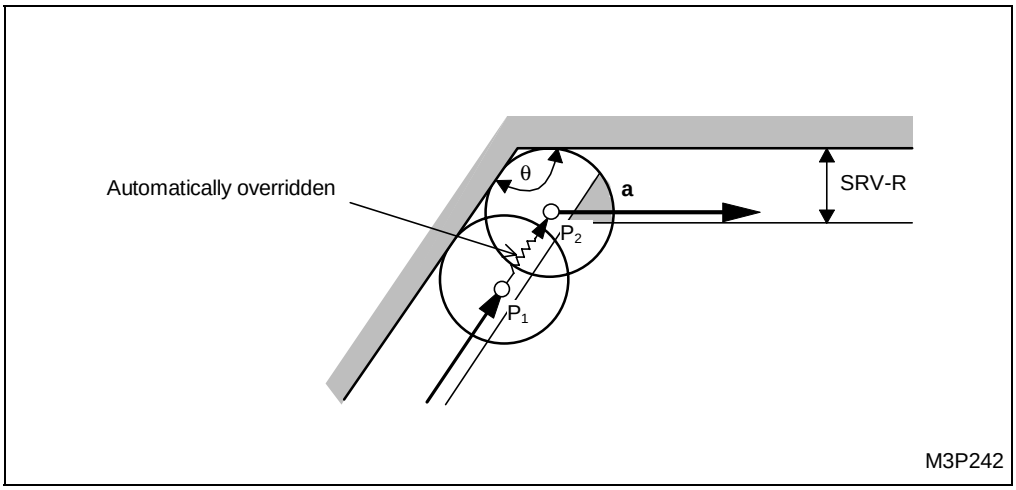
3. Other precaution on tool path

If shape data, tool data or parameter are modified after the automatic determination of 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.

4-7-7 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.

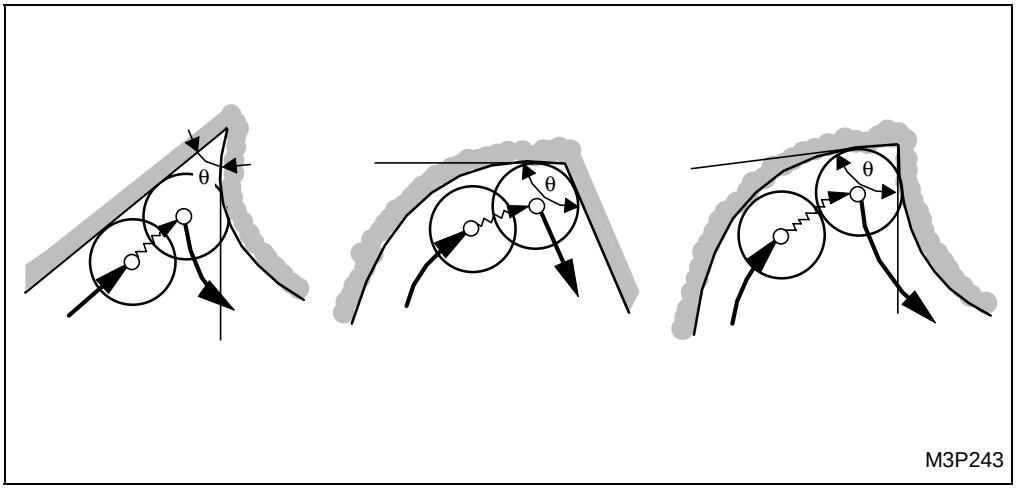
1. Operating conditions



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 overriden.

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

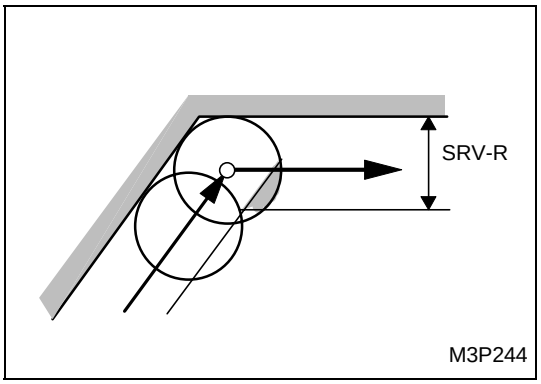
A. Inside corner angle θ is equal to or less than the value entered in the parameter **E25** (with $\theta \leq$ **E25**).



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

(SRV-R ≤ tool diameter × **E23**/100)

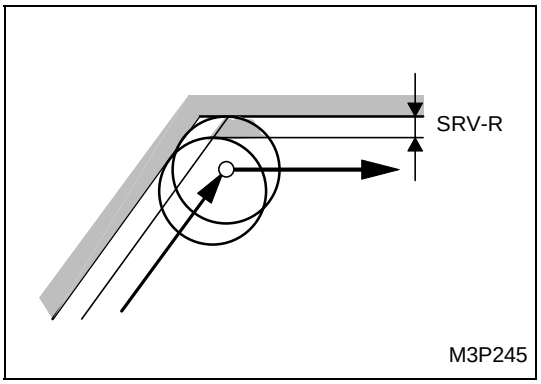
The load scarcely varies when SRV-R is near to the tool diameter.



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

(SRV-R ≤ tool diameter × **E24**/100)

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



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.

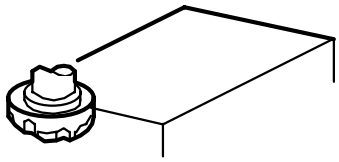
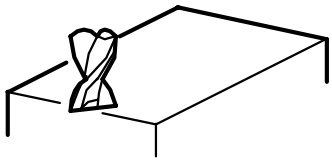
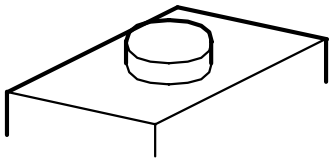
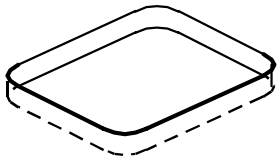
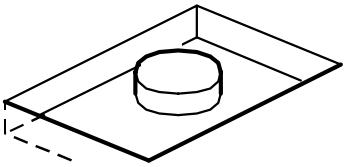
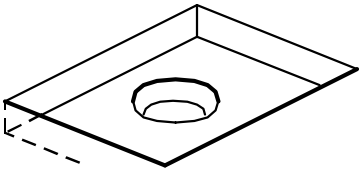
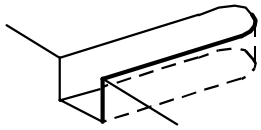
3. Override rate

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.

4-8 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.

4-8-1 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

4-8-2 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	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE CHECK	

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

FACE MIL	TOP EMIL	STEP	POCKET	PCKT MT	PCKT VLY	SLOT			

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

4-8-3 Unit data, automatic tool development and tool path

1. Face milling unit (FACE MIL)

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	FACE MIL	999.999	99.999	9	◆	99.999	◆

SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	F-MILL						◆						
F2	F-MILL						◆	◆					

Remark 1: Data in unit represent a maximum input value.

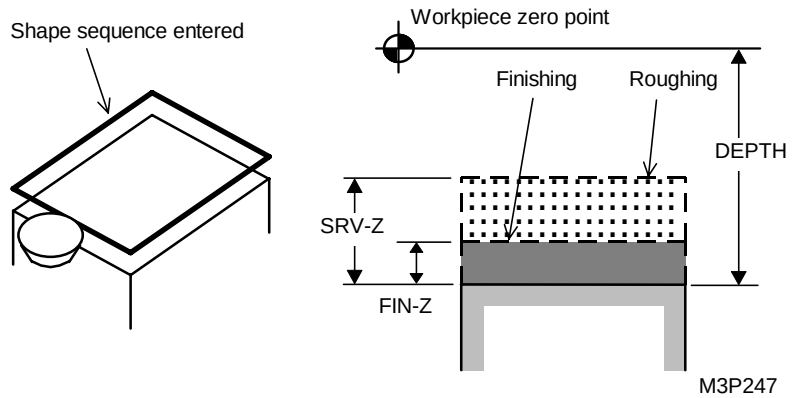
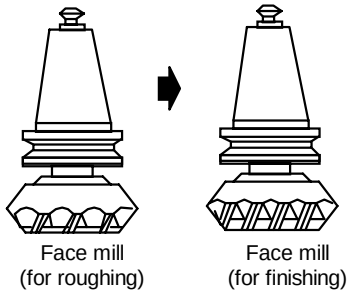
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.

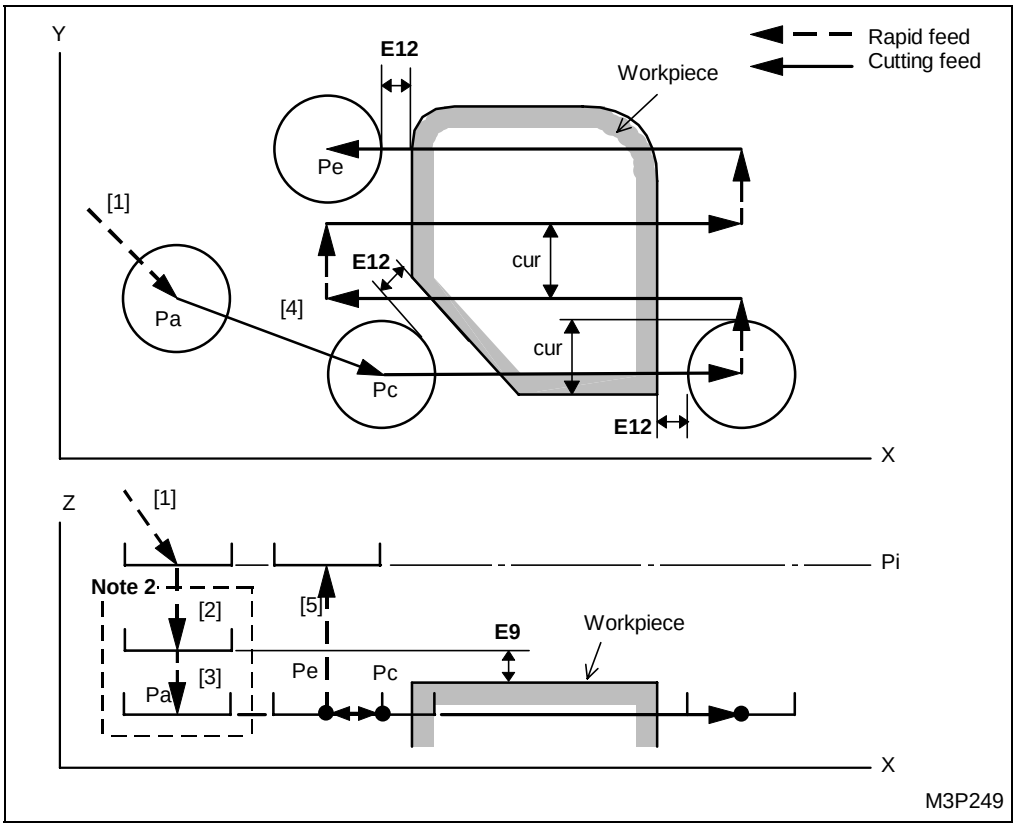
FACE MIL unit	Tool sequence
	

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 article 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 to be determined by the data INITIAL-Z in the common unit

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

<Route on which tool is to move>

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

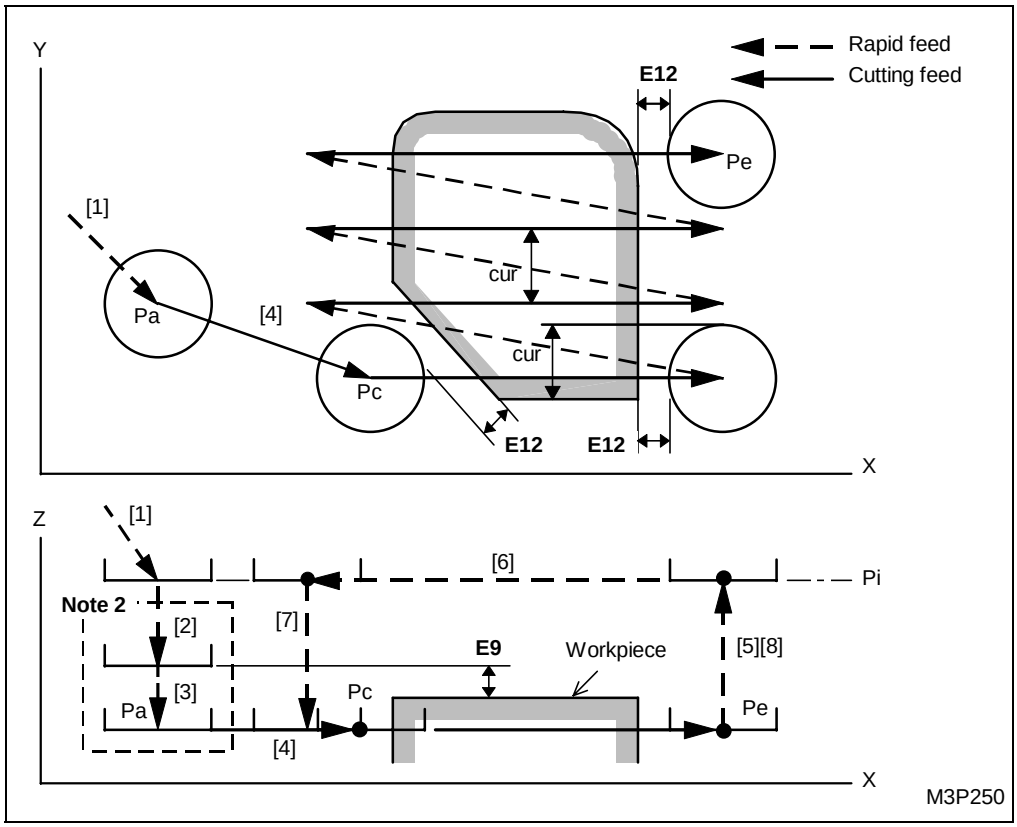
[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

[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 article 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 to be determined by the data INITIAL-Z in the common unit

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

<Route on which tool is to move>

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

[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

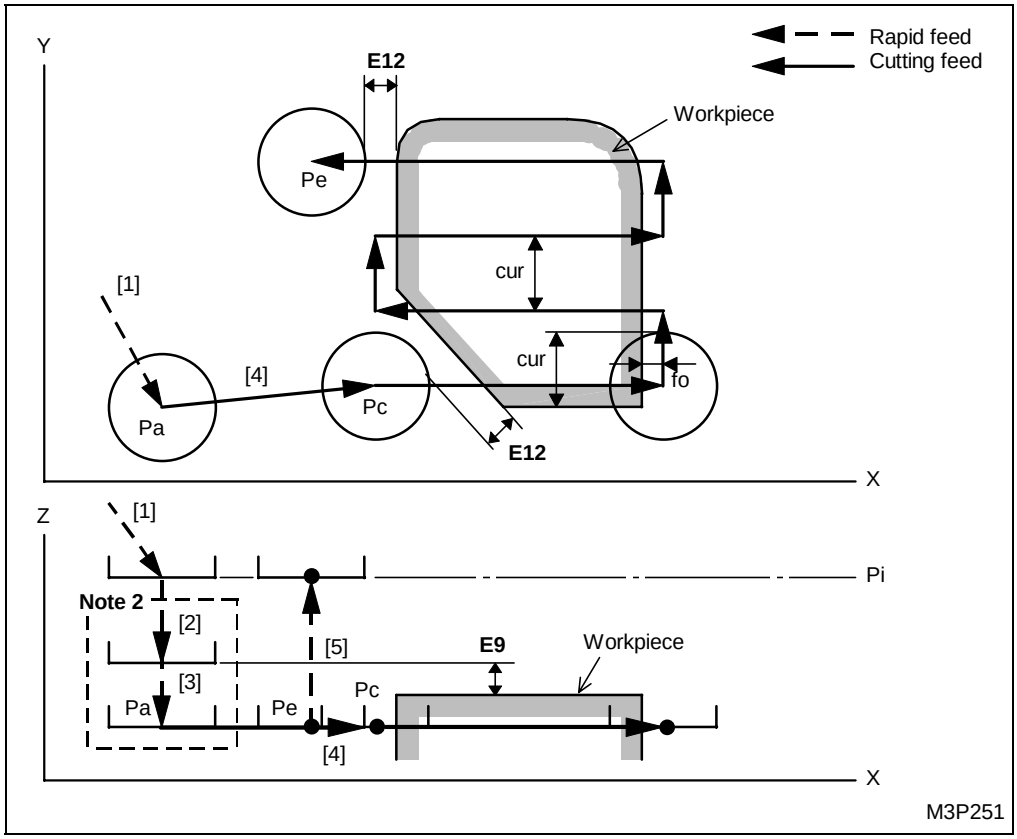
[3] The tool moves at a rapid feedrate to the face to be machined.

[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 article 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 to be determined by the data INITIAL-Z in the common unit

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

fo: Form-offsetting clearance

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

<Route on which tool is to move>

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

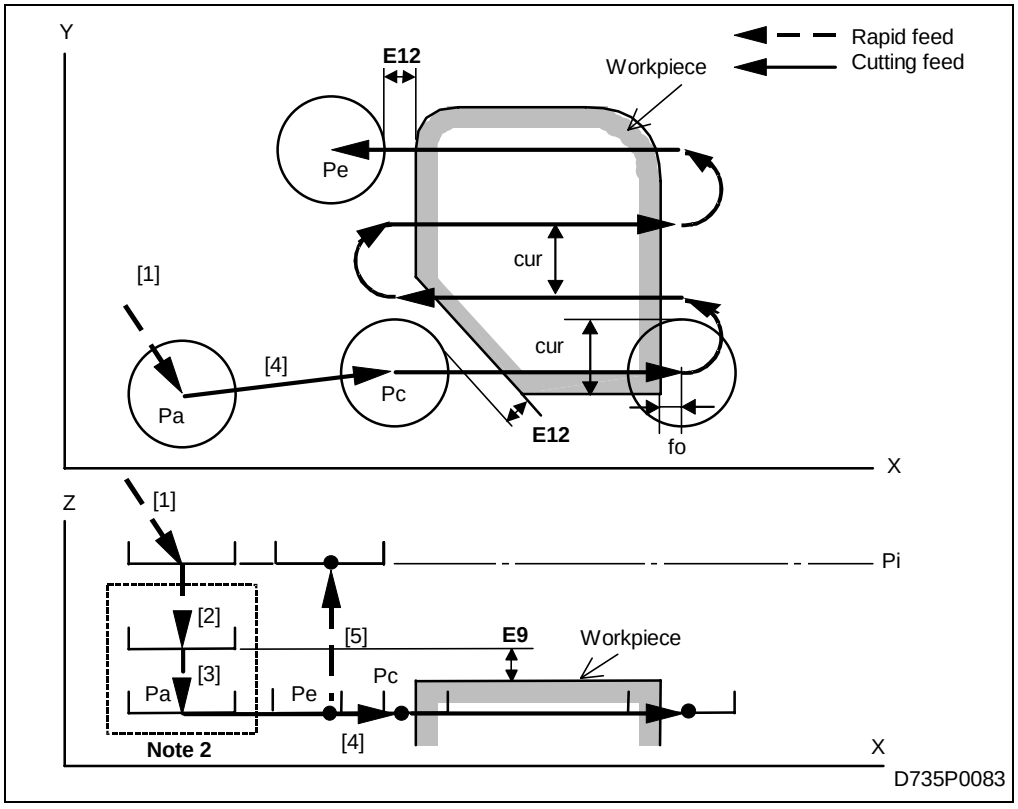
[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

[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 article 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 to be determined by the data INITIAL-Z in the common unit

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

fo: Form-offsetting clearance

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

<Route on which tool is to move>

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

[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

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

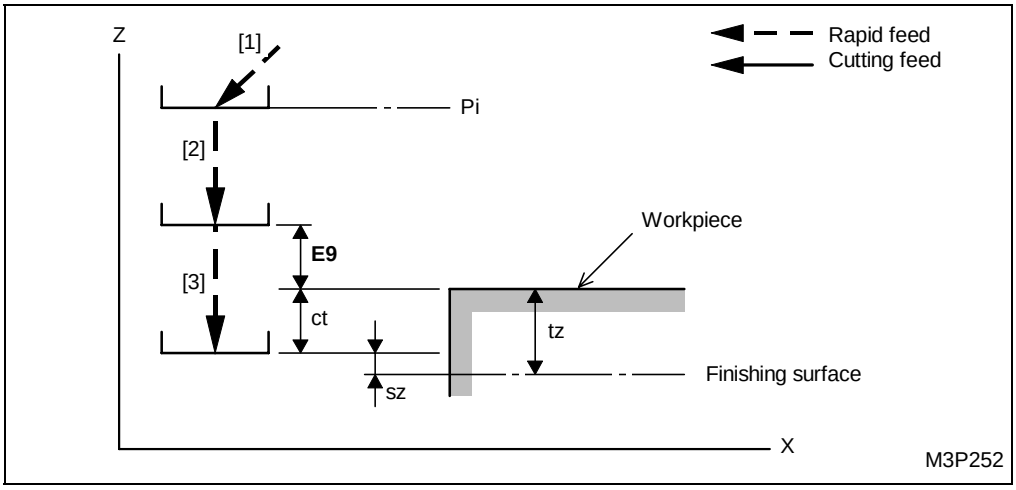
[4] The tool moves at a cutting feedrate to the cutting start point and carries out machining.

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

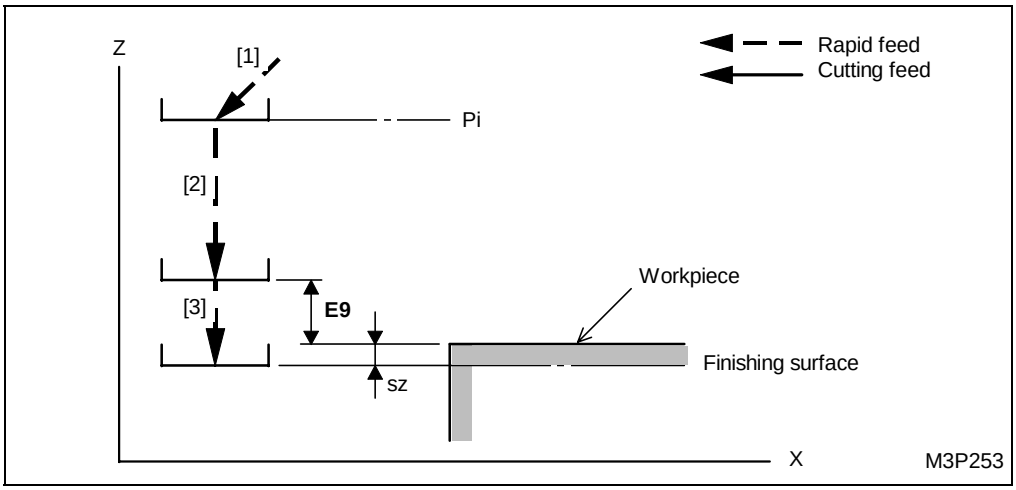
Note 1: When ? is displayed in the article APRCH-X, -Y by pressing the menu key **AUTO SET**, the tool is positioned directly at the cutting start point and operations [2] and [3] are performed. In this case, the coordinate of cutting start point will be entered in these articles.

Note 2: Detail of the Z-axial tool path. (See Subsection 4-8-5, “Precautions in face machining.”)

- Roughing



- Finishing



The bold codes represent parameter addresses.

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

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

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	99.999	9	◆	99.999	◆

SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL												
F2	E-MILL							◆					

Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

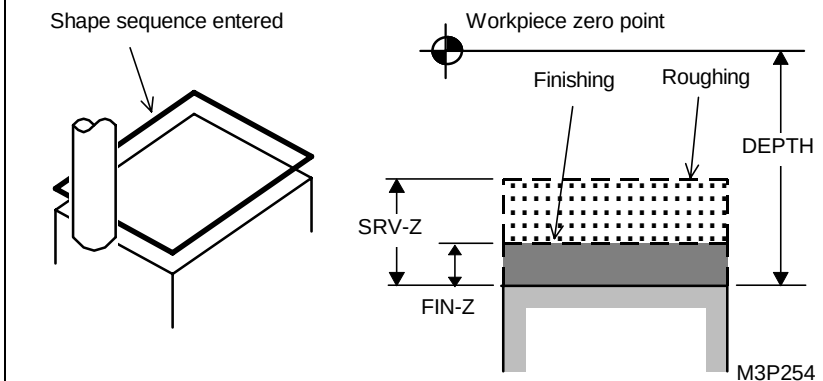
Remark 3: In this unit, end mills are automatically developed.

Remark 4: In the tool sequence, a maximum of up to two tools are automatically developed, based on SRV-Z and on FIN-Z.

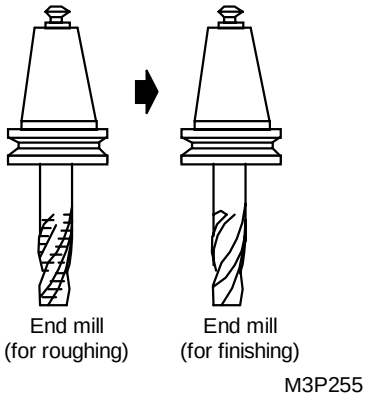
Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.

TOP EMIL unit



Tool sequence

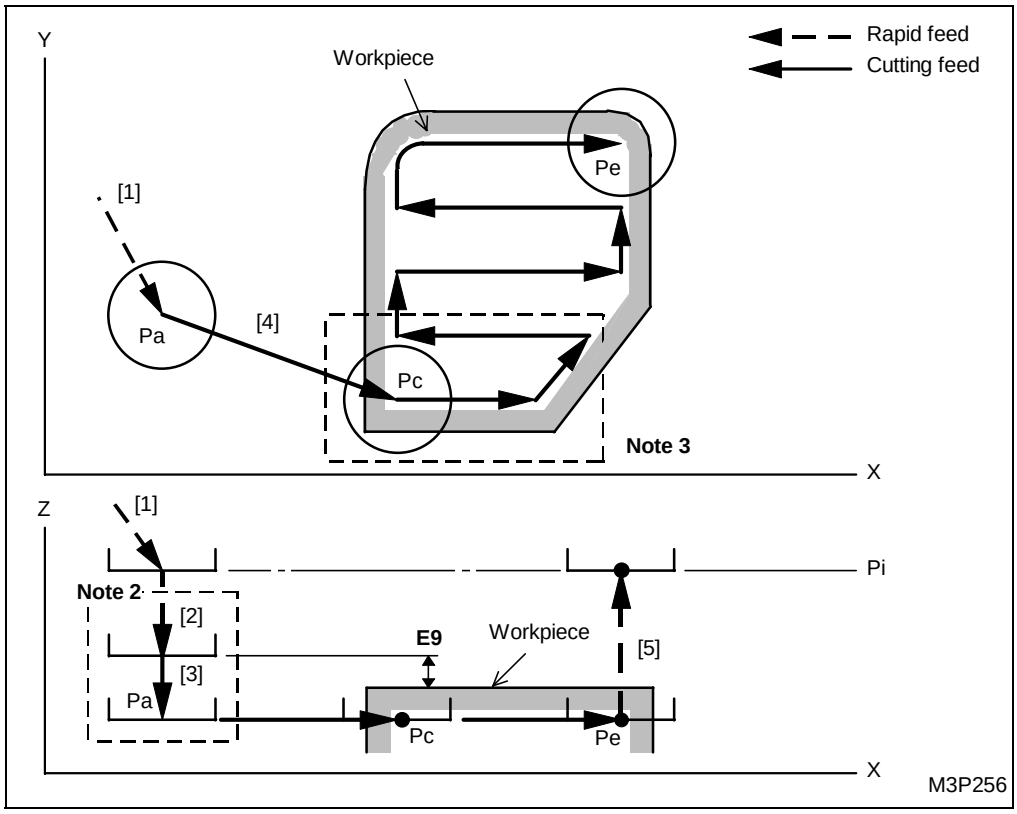


BTM: A bottom roughness code is selected out of the menu.

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 article 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 to be determined by the data INITIAL-Z in the common unit

<Route on which tool is to move>

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

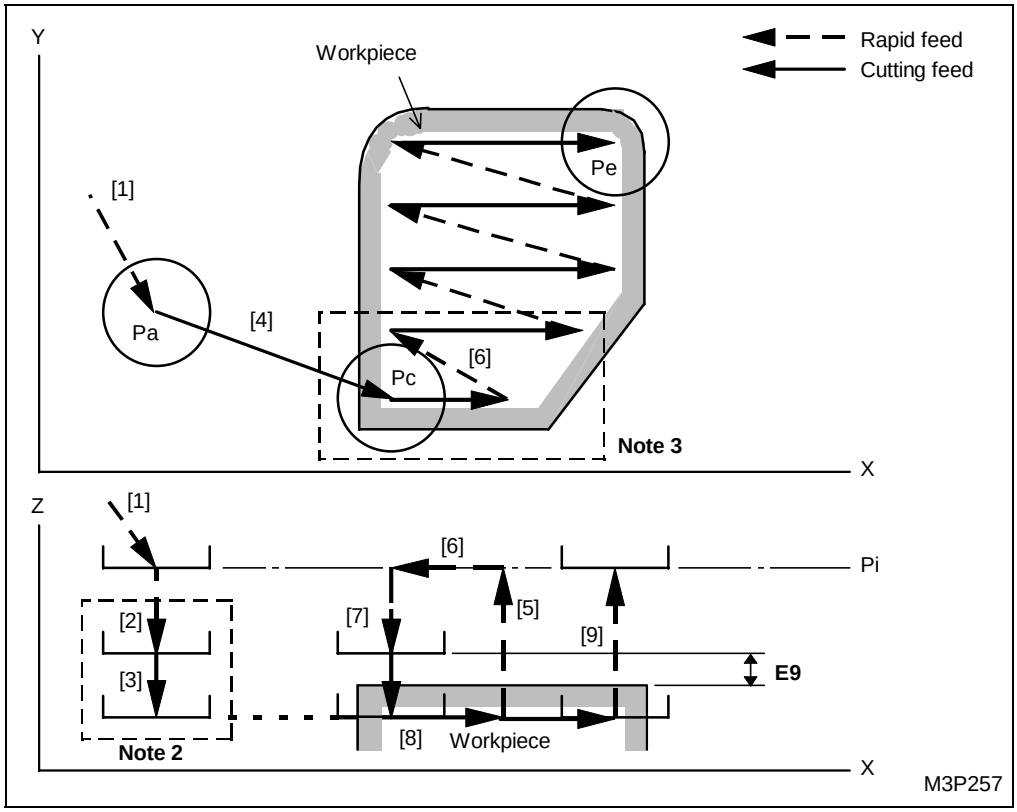
[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

[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 article 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 to be determined by the data INITIAL-Z to be entered in the common unit

<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 position entered by the parameter **E9**.

[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 parameter **E9** above the next cutting start point.

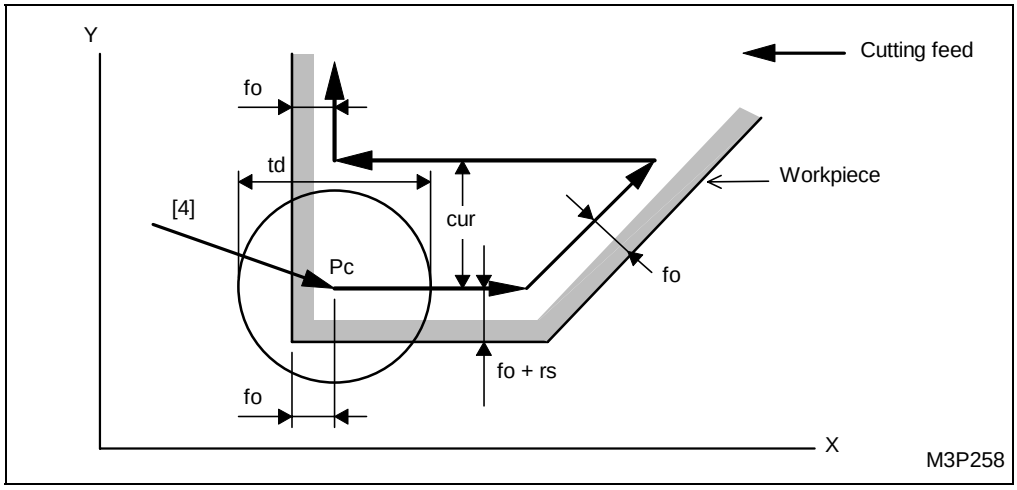
[8] The tool moves at a cutting feedrate to the face to be machined and starts machining.

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

Note 1: When ? is displayed in the articles 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 articles.

Note 2: See Subsection 4-8-5 “Precautions in face machining.”

Note 3: Detail description of tool path



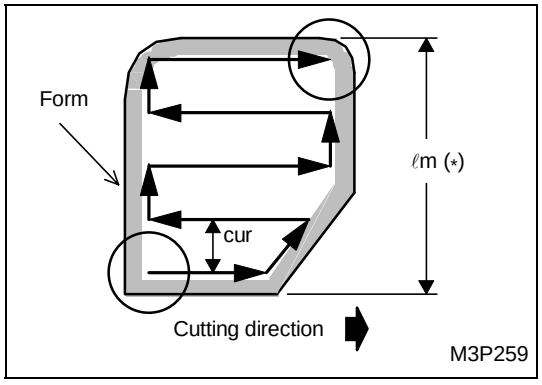
td: Diameter of a tool

fo: Form offset clearance dependent upon both td and parameter **E13**

$$fo = td \times \frac{\mathbf{E13}}{10}$$

rs: Form offset amount rectangular to the cutting direction

$$rs = \frac{td}{20}$$



cur: Radial cutting depth per cycle, obtainable as follows:

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

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

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

cr: Radial cutting depth (WID-R) to be entered in the tool sequence

n: Number of radial cutting pass (an integer with fractions below the decimal point rounded up)

3. End milling-step unit (STEP)

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

A. Data setting

UNo.	UNIT	DEPTH	SRV-Z	BTM	WAL	FIN-Z	FIN-R
1	STEP	999.999	99.999	9	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	E-MILL												
F2	E-MILL							◆					

Remark 1: Data in unit represent a maximum input value.

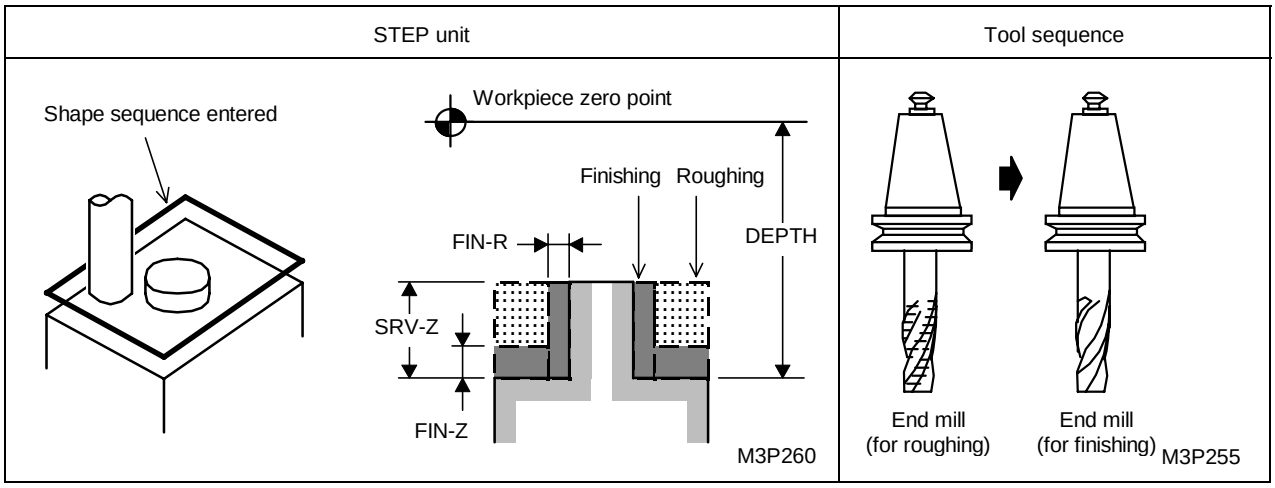
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.



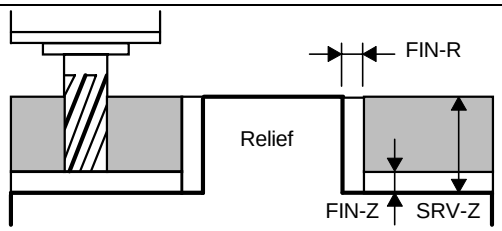
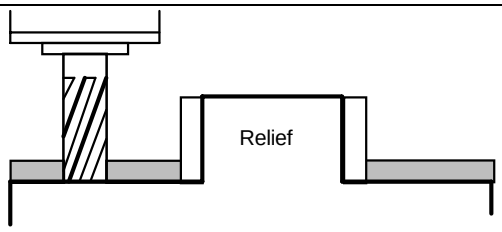
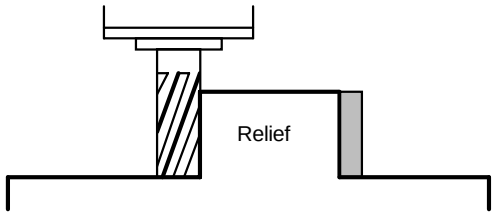
BTM: A buttom roughness code is selected out of the menu.

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

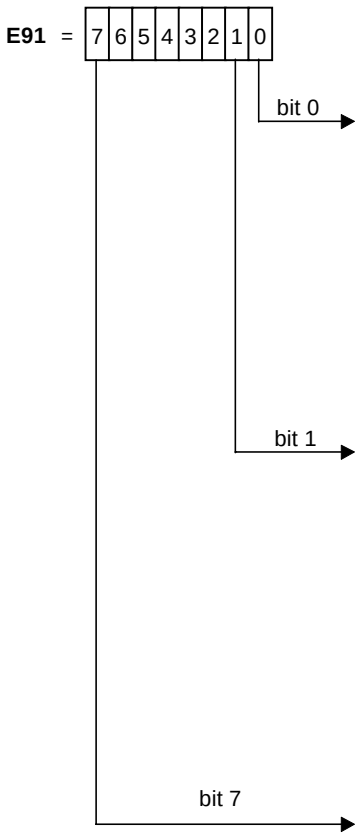
End milling-step is performed in the following order.

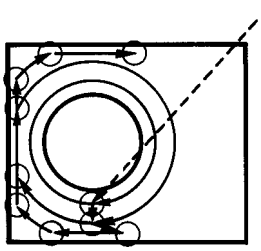
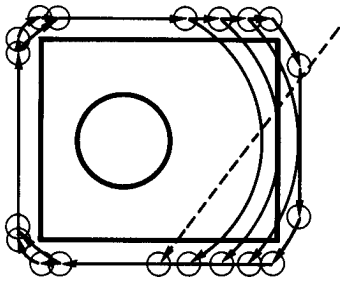
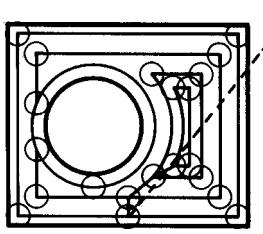
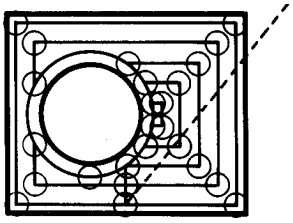
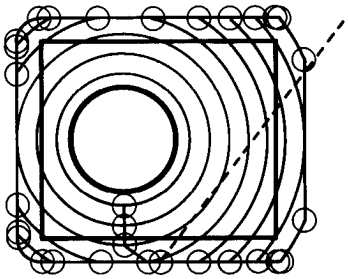
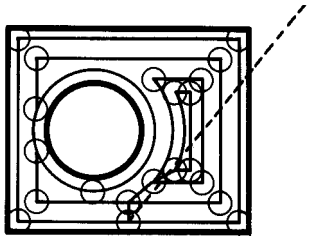
Roughing			Machining is performed with the end mill developed in the tool sequence R1. With SRV-Z = FIN-Z, this machining is not performed.
Finishing	Bottom		Machining is performed with the end mill developed in the tool sequence F1. With FIN-Z = 0, this machining is not performed.
	Wall	 M3P261	Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.

C. Machining pattern

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

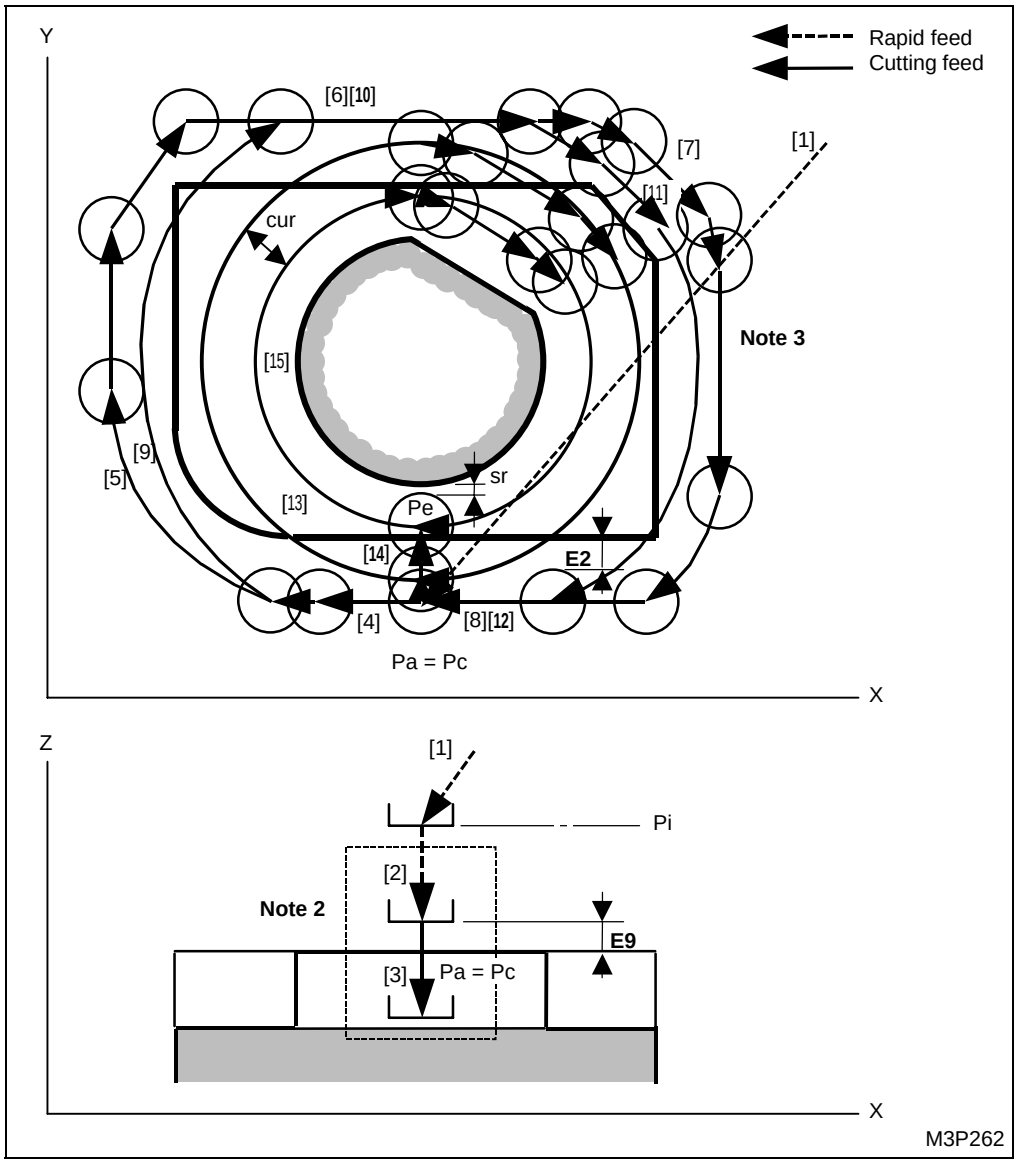
- Either 0 or 1 in the related bit accordingly.



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

D. Tool path

Roughing 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 to be determined by the data INITIAL-Z in the common unit

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

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

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

Note 2: See Subsection 4-8-5, “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 position entered by the parameter **E9**.

[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 [2] 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

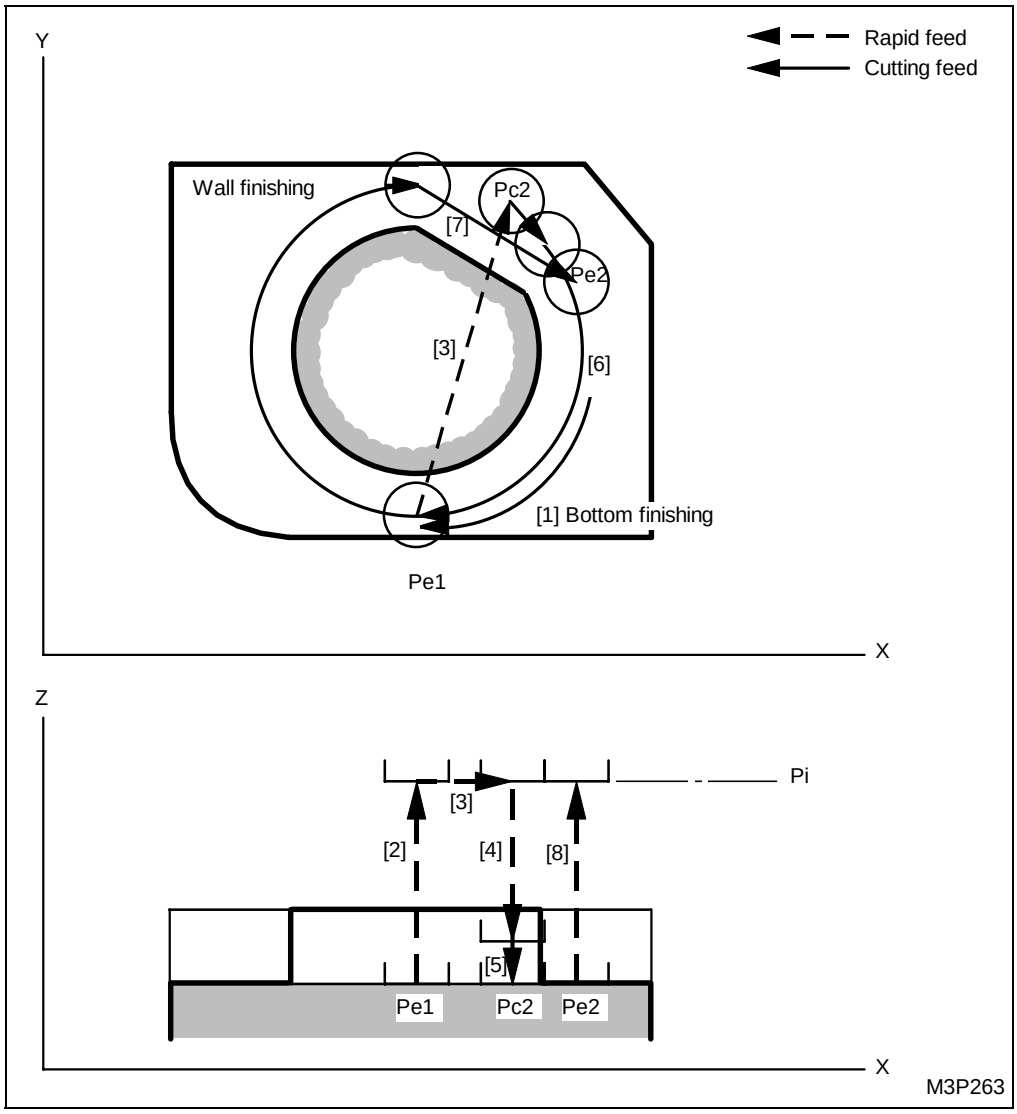
Finishing is performed, based on the entered data FIN-Z and FIN-R.

Bottom finishing is performed, with 0 < FIN-Z.

Wall finishing is performed, with 0 < FIN-R.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-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 to be determined by the data INITIAL-Z in the common unit

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

Note 2: See Subsection 4-8-5, “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	99.999	9	9	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
R1	E-MILL												
F2	E-MILL							◆					
3	CHF-C							◆	◆				

Remark 1: Data in unit represent a maximum input value.

Remark 2: ◆: Data are not necessary to be set here.

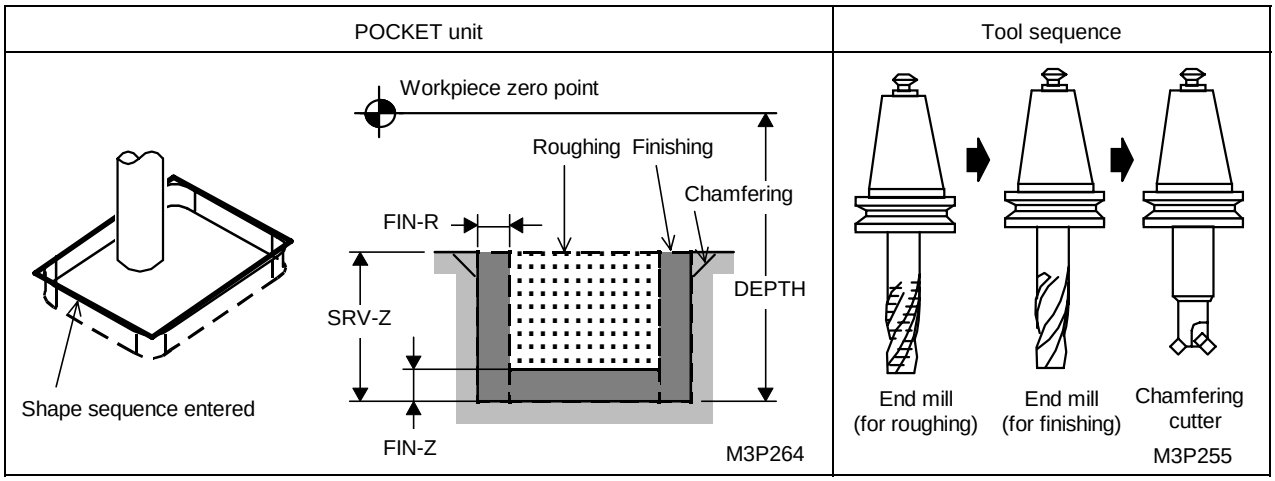
Remark 3: In this unit, end mills and a chamfering cutter are automatically developed. Instead of the chamfering cutter, a centering drill can be used for chamfering.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above
1	(Chamfering)	CHMF ≠ 0

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.

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.

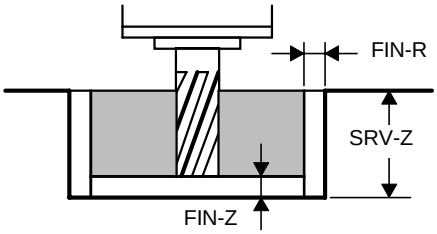
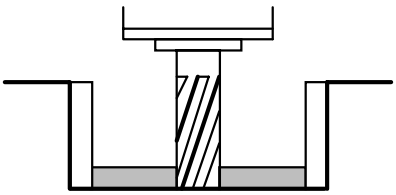
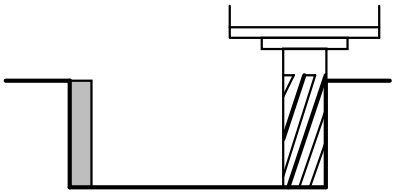
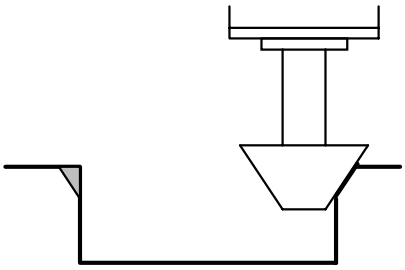
WAL: A wall roughness code is selected out of the menu.

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

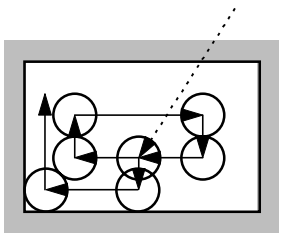
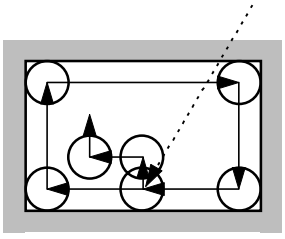
Pocket milling is performed in the following order.

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.
Chamfering			Machining is performed with the chamfering cutter or centering drill in the tool sequence. With CHMF = 0, this machining is not performed.

C. Machining pattern

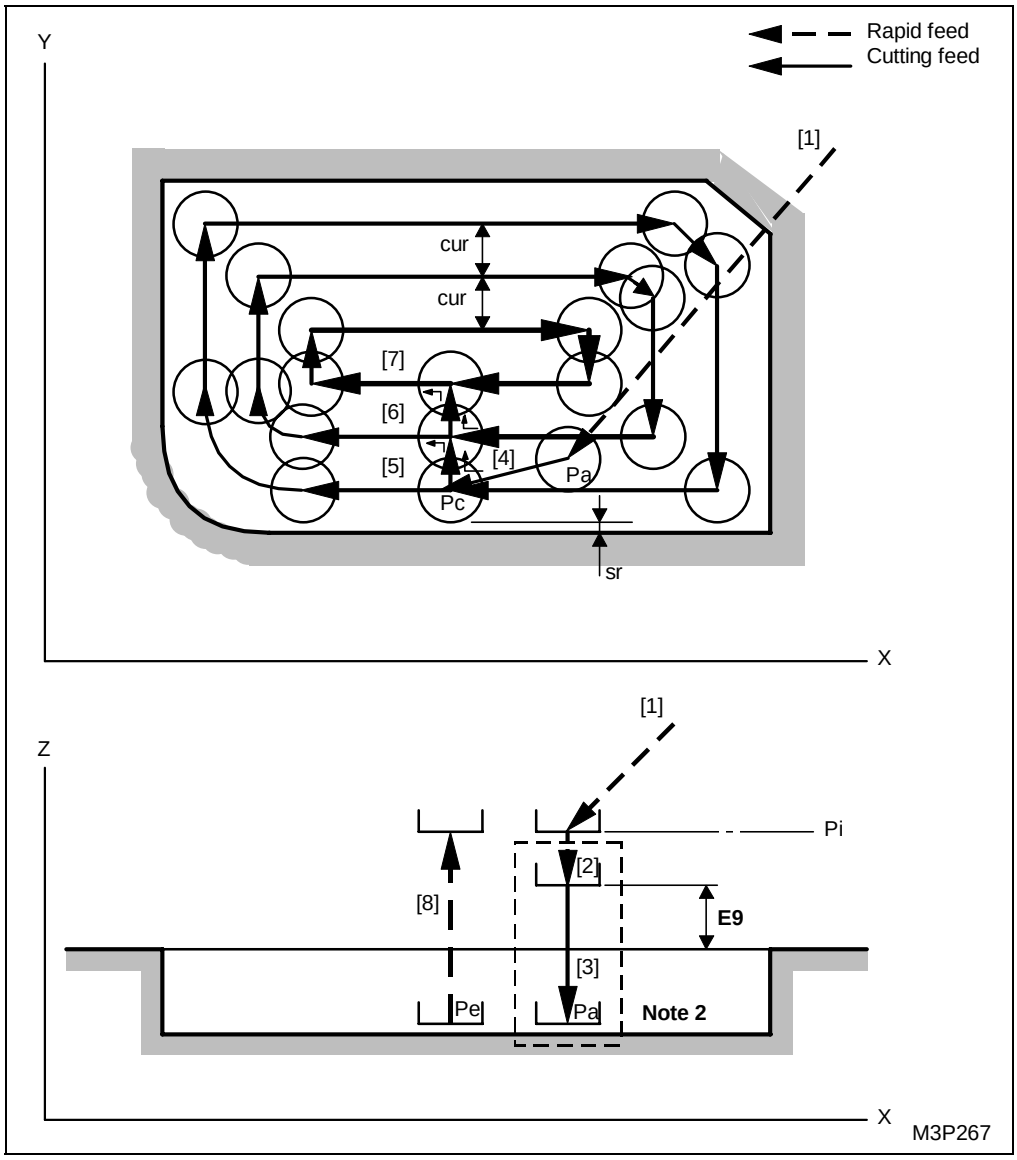
For roughing or bottom finishing, a machining pattern is selected by the parameter **E92**.

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

	0	1
E92 = 7 6 5 4 3 2 1 0 bit 0 M3P266		
	Machining is performed from inside to outside.	Machining is performed from outside to inside.

D. Tool path

Machining from outside (roughing or bottom finishing)



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 to be determined by the data INITIAL-Z in the common unit

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

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

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “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 position entered by the parameter **E9**.

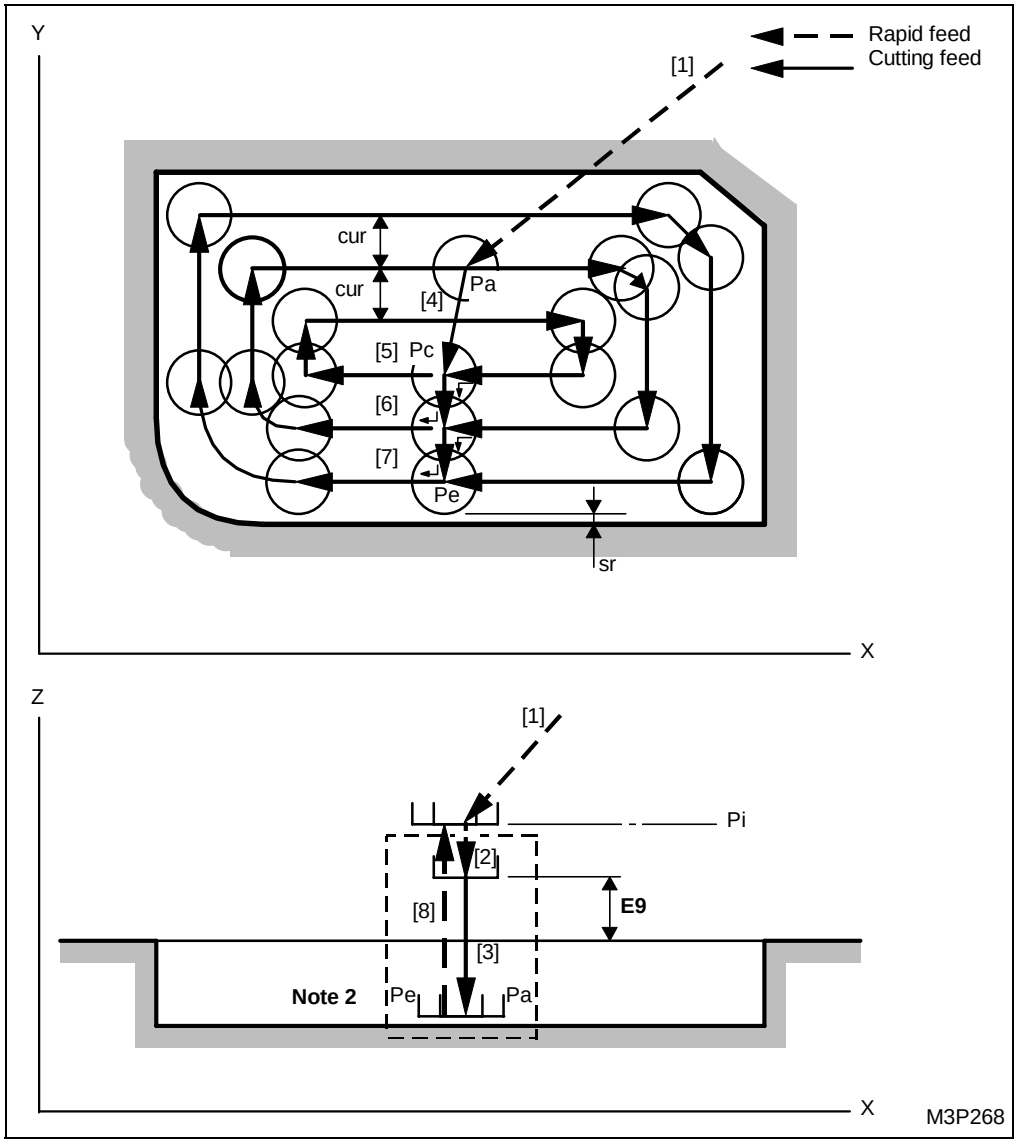
[3] The tool moves to the face to be machined. (The feedrate is dependent upon the data ZFD in the tool sequence.)

[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 to be determined by the data INITIAL-Z in the common unit

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

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

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “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 position entered by the parameter **E9**.

[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.

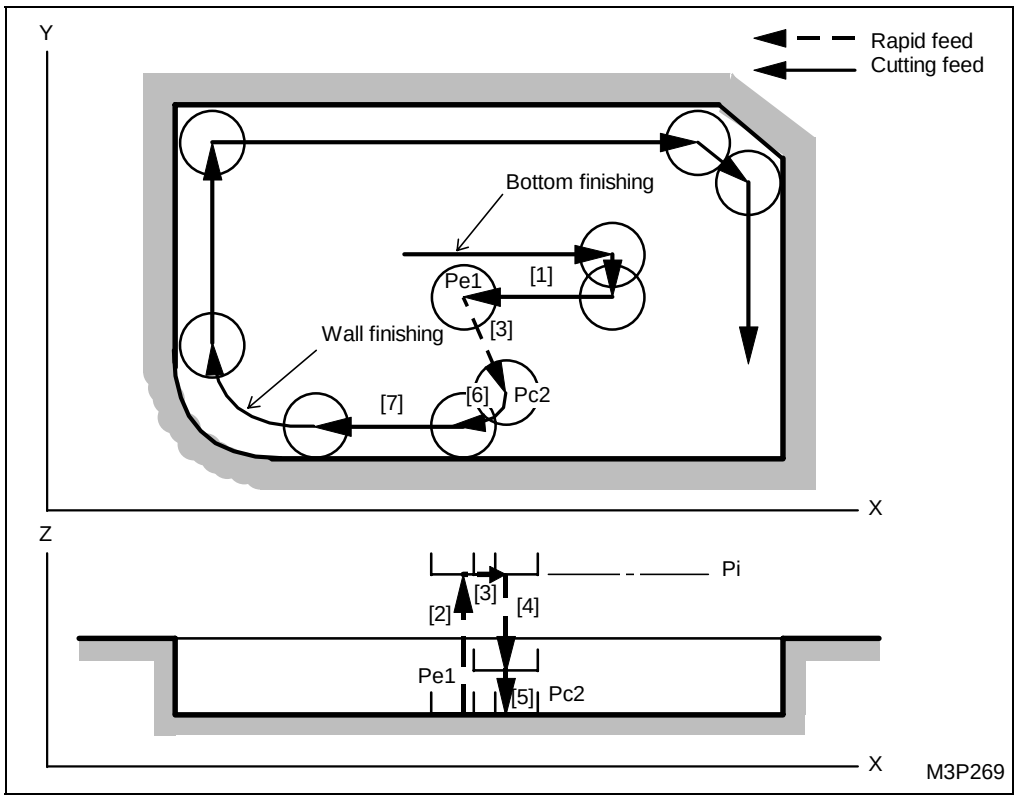
E. Finishing

Finishing is performed, based on the entered values of FIN-Z and FIN-R.

- Bottom finishing is performed, with 0 < FIN-Z.
- Wall finishing is performed, with 0 < FIN-R.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-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 to be determined by the data INITIAL-Z in the common unit

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “Precautions in face machining.”

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	99.999	9	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	E-MILL												
F2	E-MILL							◆					

Remark 1: Data in unit represent a maximum input value.

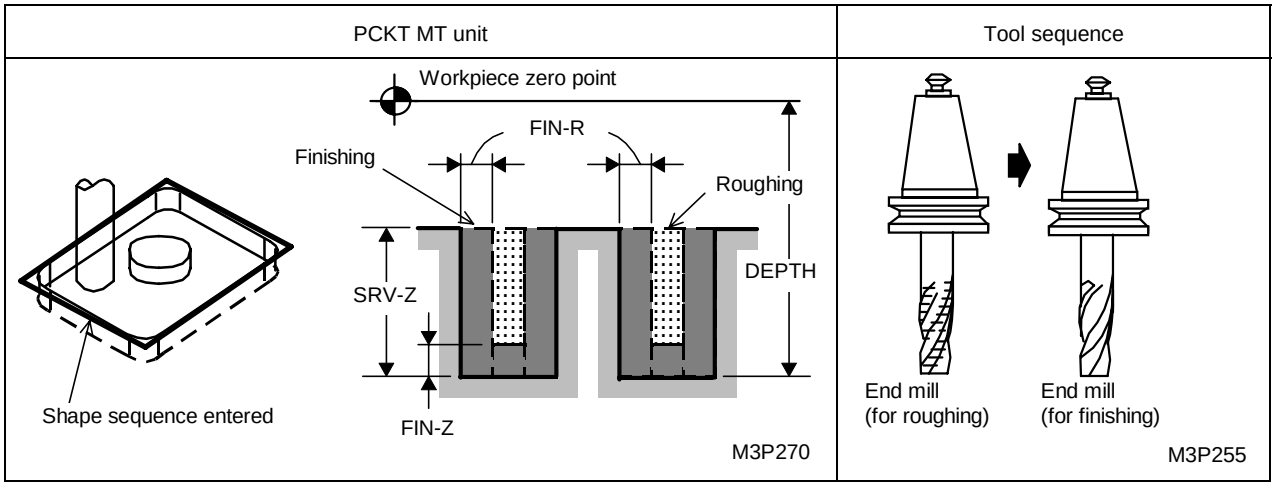
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.



BTM: A buttom roughness code is selected out of the menu.

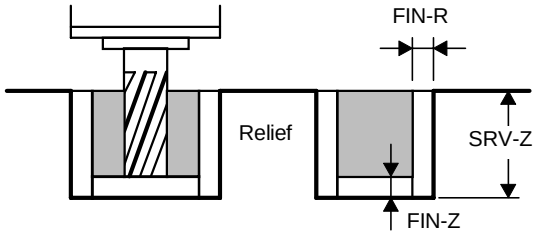
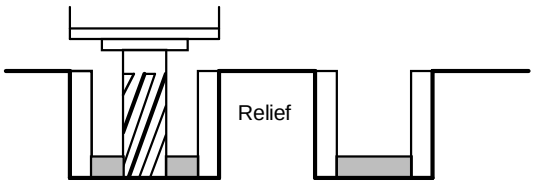
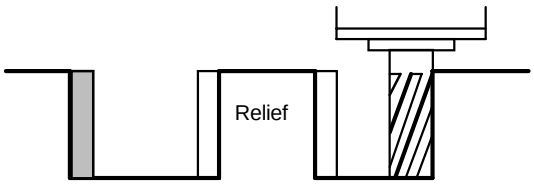
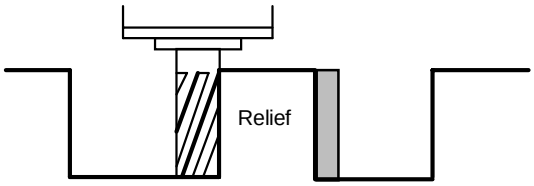
WAL: A wall roughness code is selected out of the menu.

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.

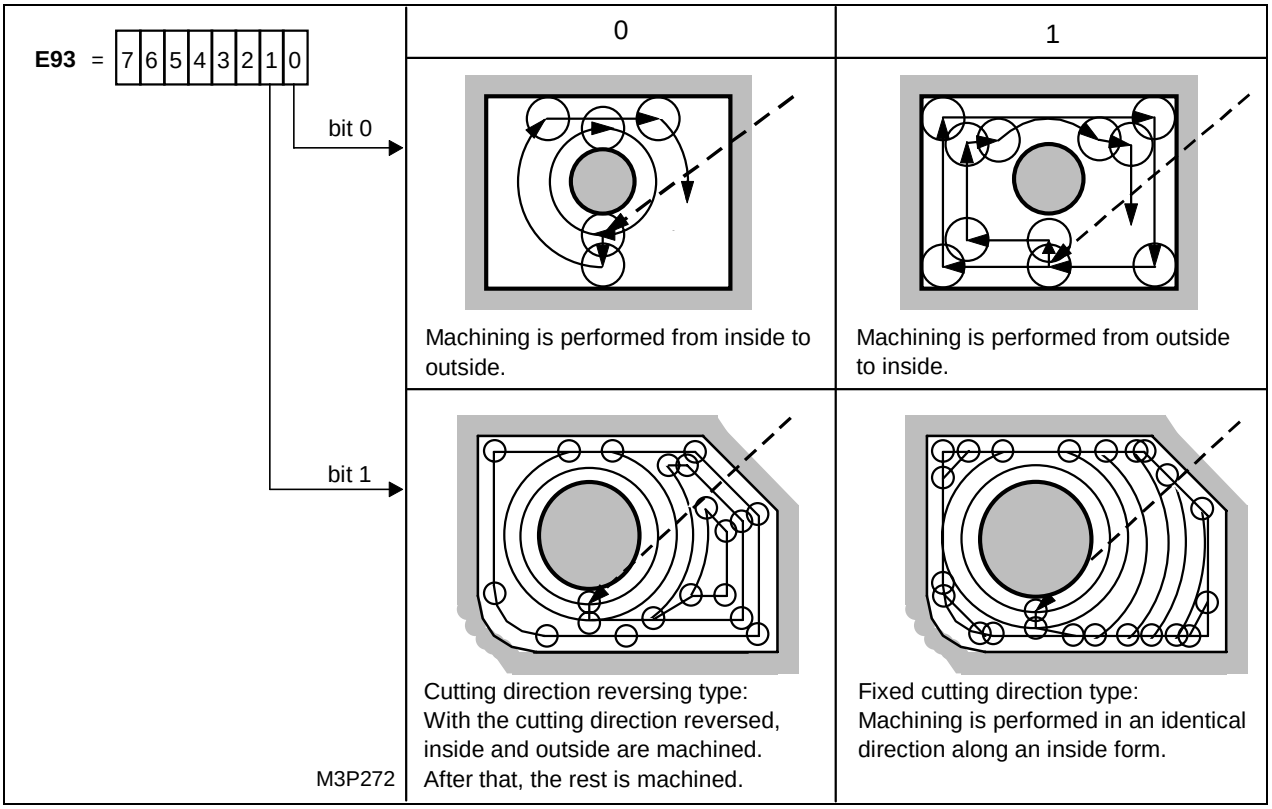
<table>
<tr><td colspan="2">Roughing</td><td> 
</td><td>Machining is performed with the end mill developed in the tool sequence R1. With SRV-Z = FIN-Z, this machining is not performed.</td></tr>
<tr><td rowspan="3">Finishing</td><td>Bottom</td><td> 
</td><td>Machining is performed with the end mill developed in the tool sequence F1. With FIN-Z = 0, this machining is not performed.</td></tr>
<tr><td>Outer wall</td><td> 
</td><td rowspan="2">Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.</td></tr>
<tr><td>Inner wall</td><td> 

M3P271</td></tr>
</table>

C. Machining pattern

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

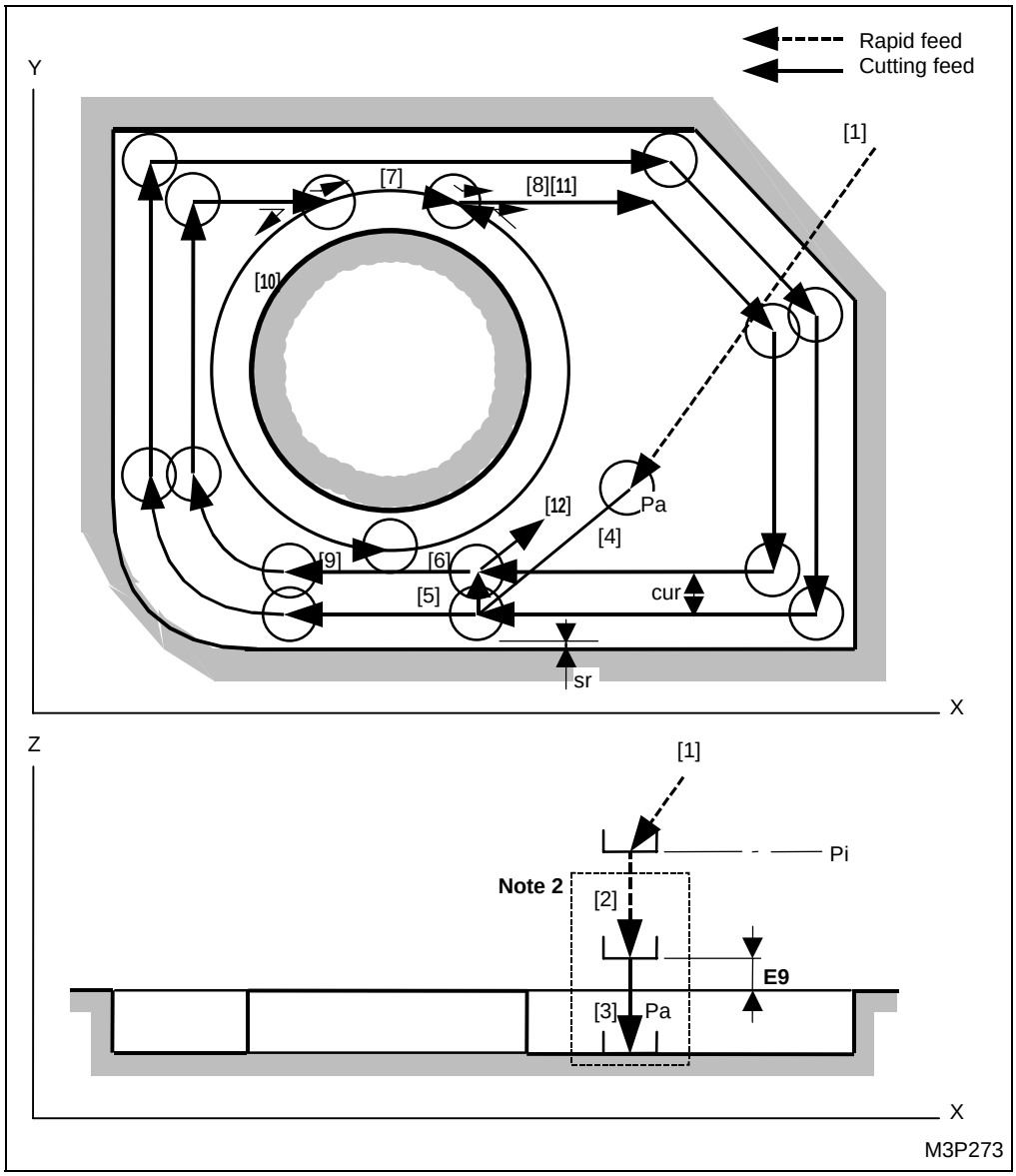
- Enter 0 or 1 in the related bit accordingly.



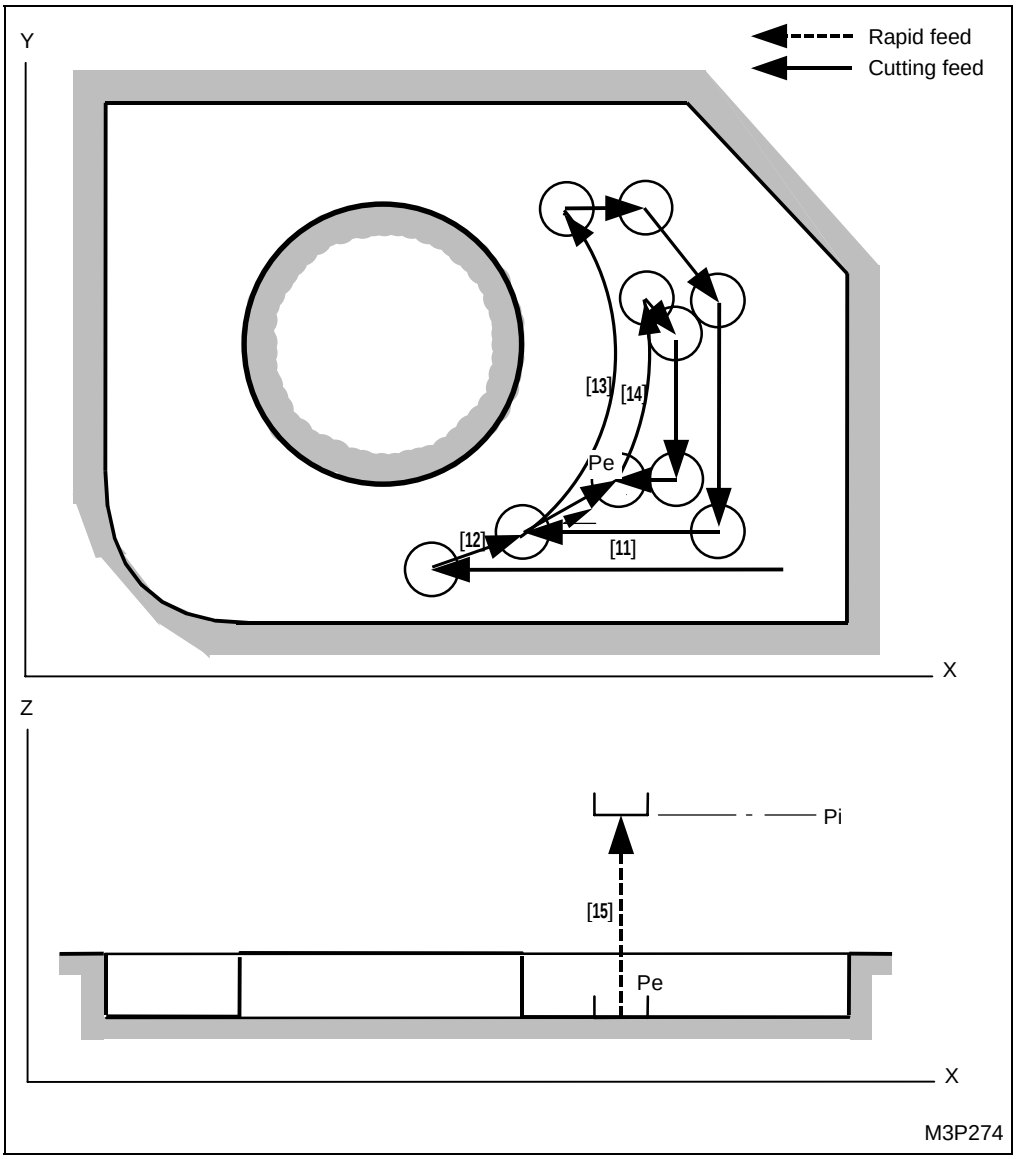
D. Tool path

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 to be determined by the data INITIAL-Z in the common unit

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

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

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “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 position entered by the parameter **E9**.

[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].

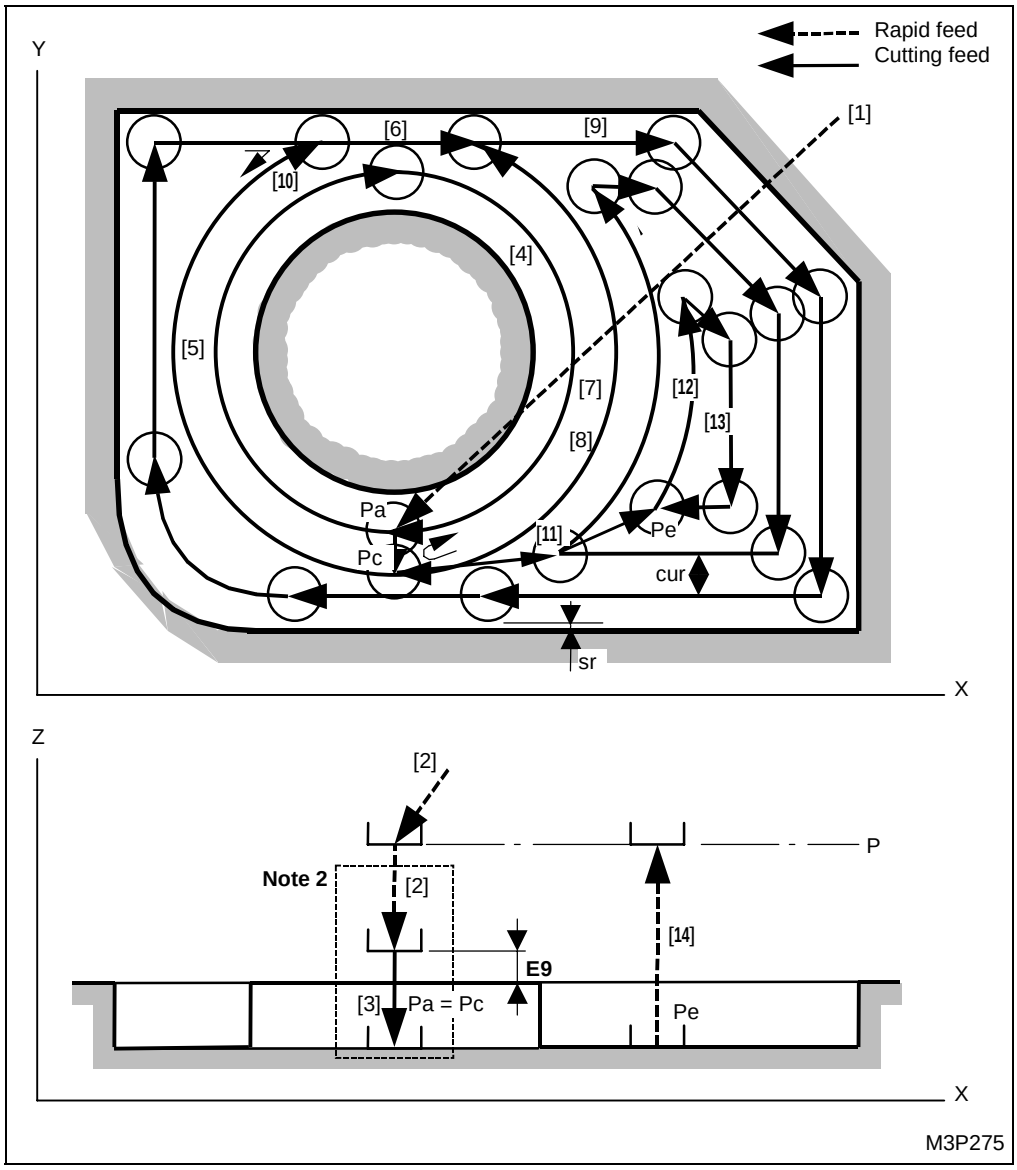
[10] Because of the path reversed, once it has interfered with the inside form, the tool machines along the inside form.

[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.

Machining from inside (roughing 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 to be automatically established

<Route on which tool is to move>

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

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “Precautions in face machining.”

Outer wall finishing

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

Inner wall finishing

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

E. Finishing

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

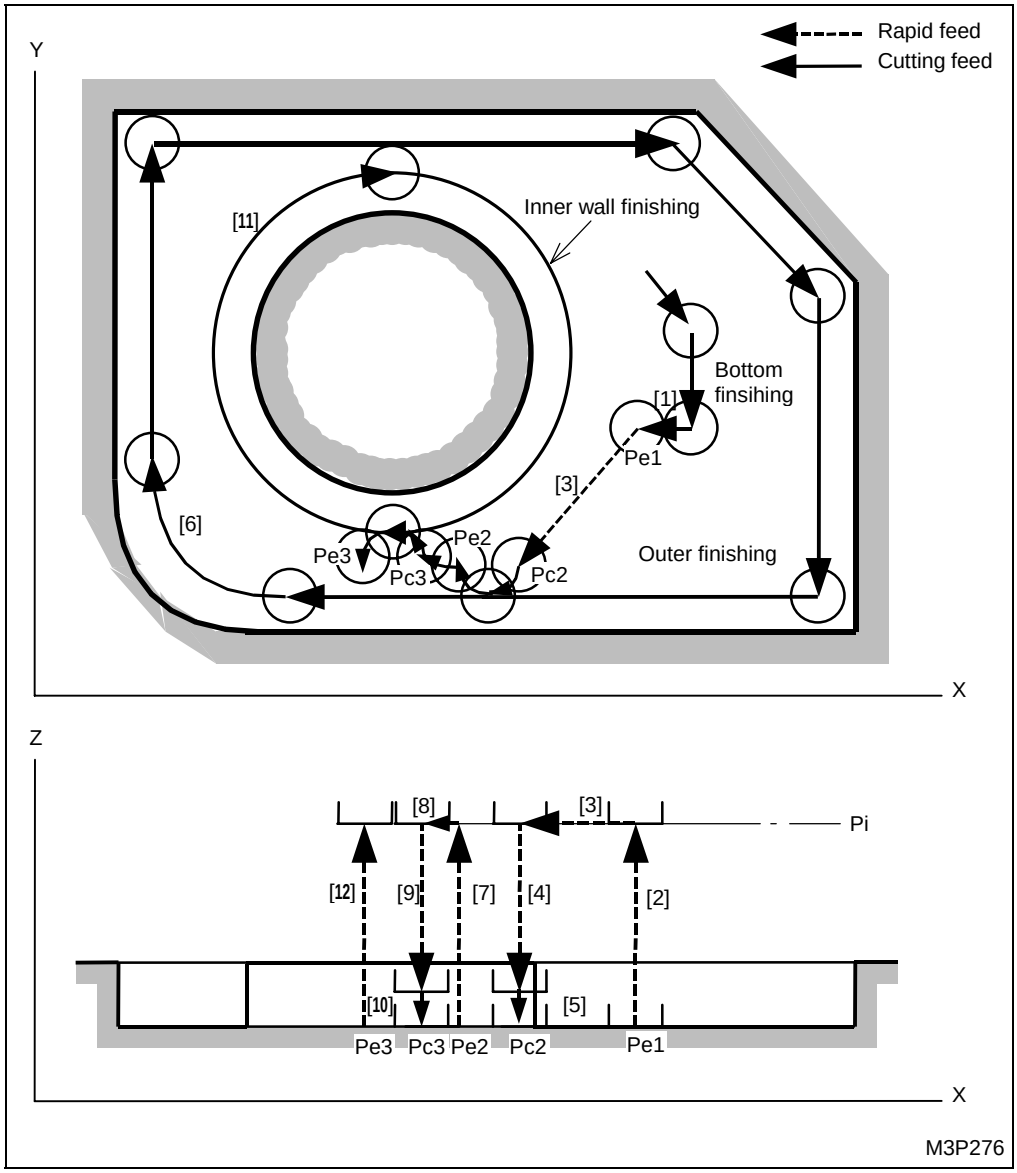
- Bottom finishing is performed, with 0 < FIN-Z.
- Outer and inner walls are finished, with 0 < FIN-R.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-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 to be determined by the data INITIAL-Z in the common unit

Note: When ? is displayed in the article 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 articles.

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	99.999	9	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	E-MILL												
F2	E-MILL							◆					

Remark 1: Data in unit represent a maximum input value.

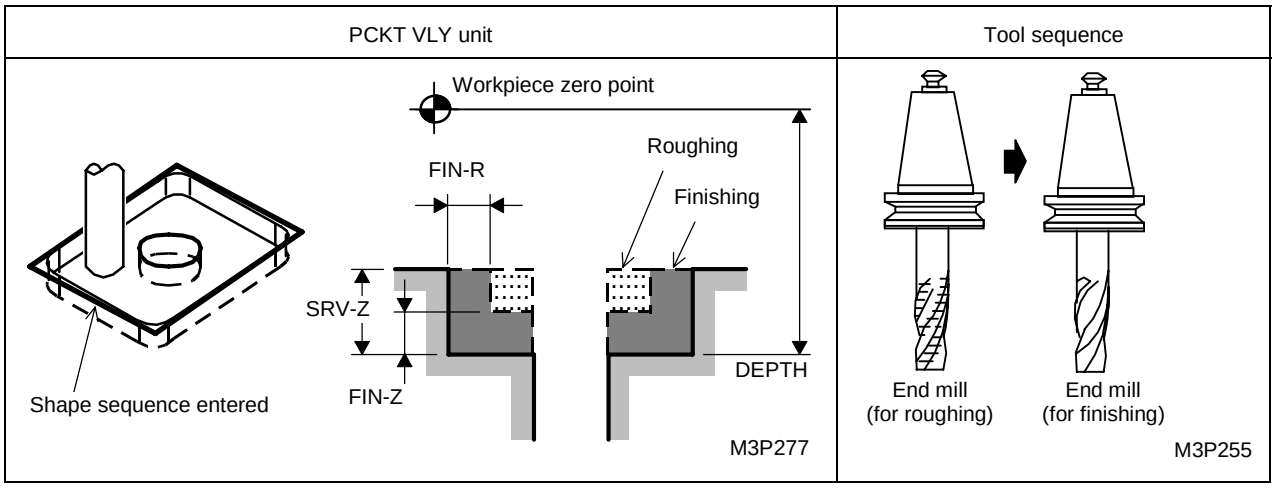
Remark 2: ◆: Data are not necessary to be here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.



BTM: A buttom roughness code is selected out of the menu.

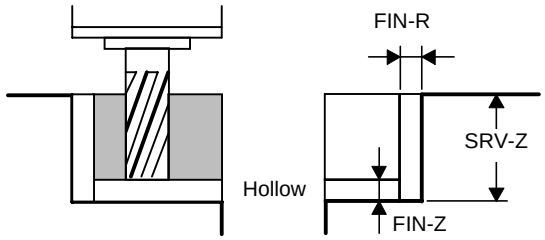
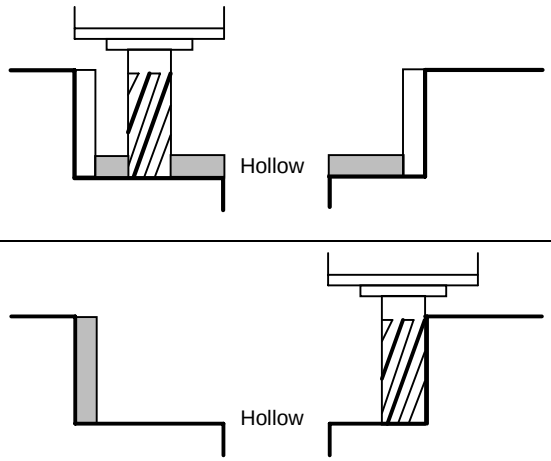
WAL: A wall roughness code is selected out of the menu.

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

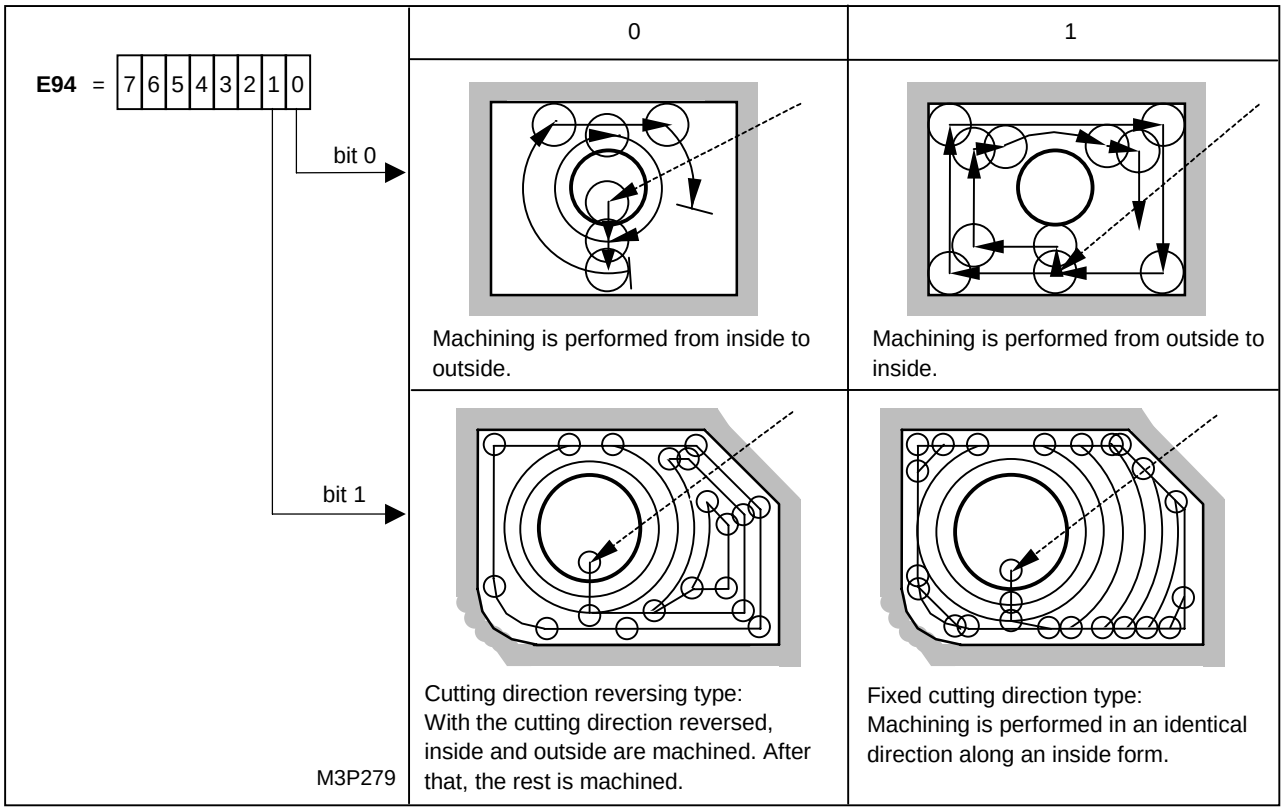
Pocket milling-valley is performed in the following order.

<table>
<tr><td colspan="2">Roughing</td><td> 
</td><td>Machining is performed with the end mill developed in the tool sequence R1. With SRV-Z = FIN-Z, this machining is not performed.</td></tr>
<tr><td rowspan="2">Finishing</td><td>Bottom</td><td rowspan="2"> 
</td><td>Machining is performed with the end mill developed in the tool sequence F1. With FIN-Z = 0, this machining is not performed.</td></tr>
<tr><td>Wall</td><td>Machining is performed with the end mill developed in the tool sequence F1. With FIN-R = 0, this machining is not performed.</td></tr>
</table>

C. Machining pattern

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

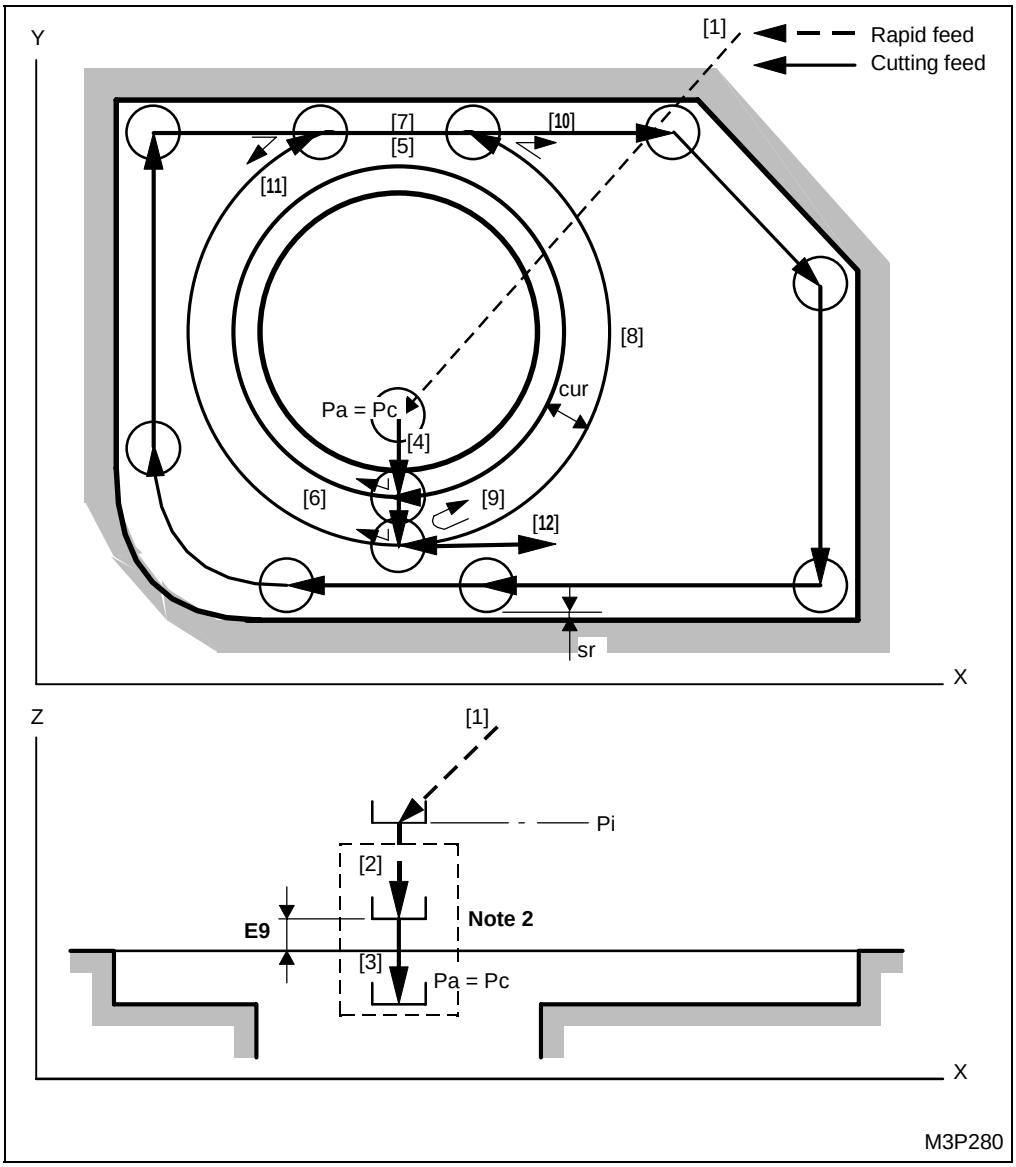
- Enter 0 or 1 in the related bit accordingly.



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.

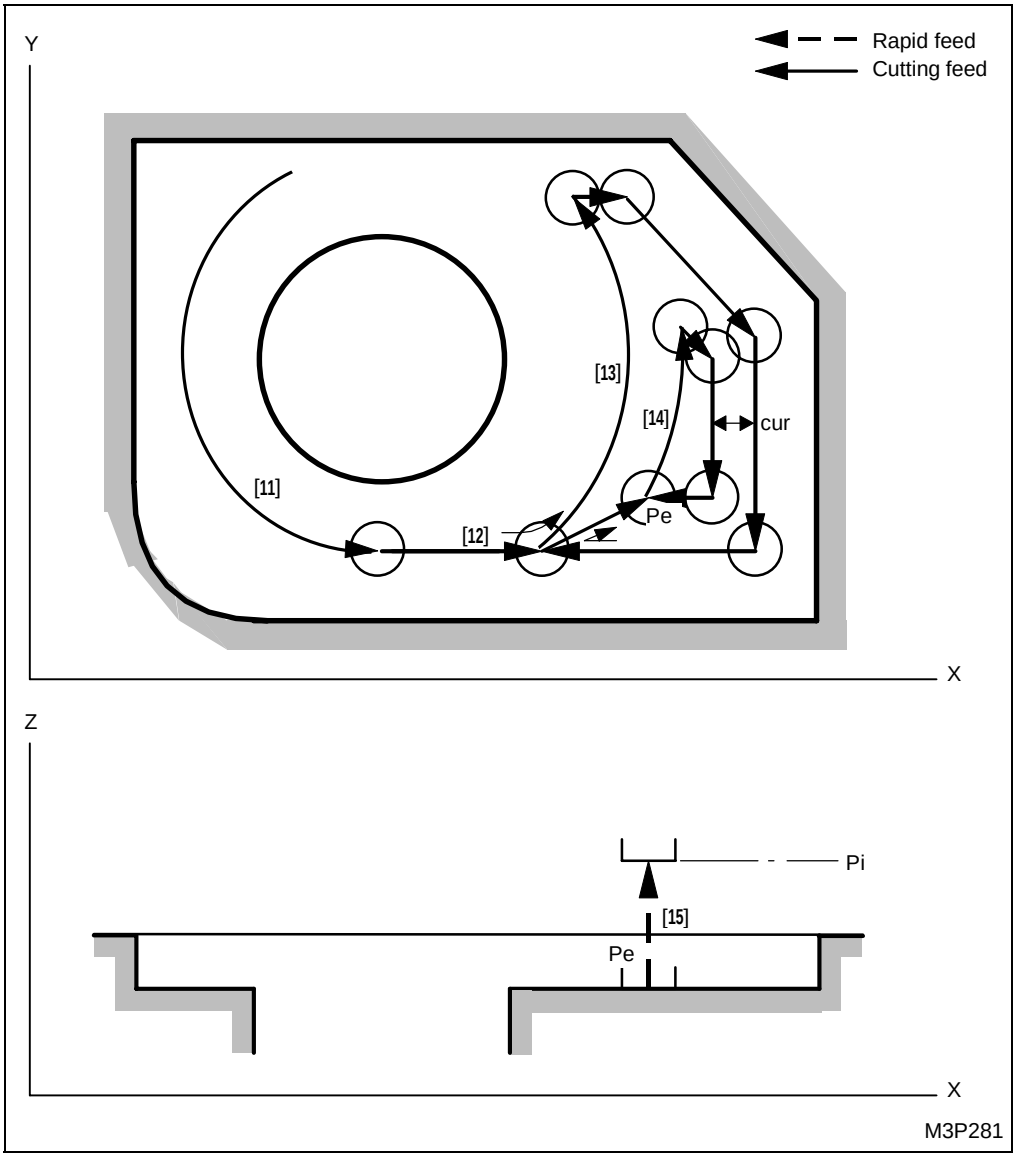
Pa, Pc: Approach point to be determined by the data APRCH-X, -Y in the tool sequence and cutting start point (In the illustration above, the cutting start point is the approach point.)

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

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

- Machining the rest:



Pe: Escape point to be automatically established

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

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

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “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 position entered by the parameter **E9**.

[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].

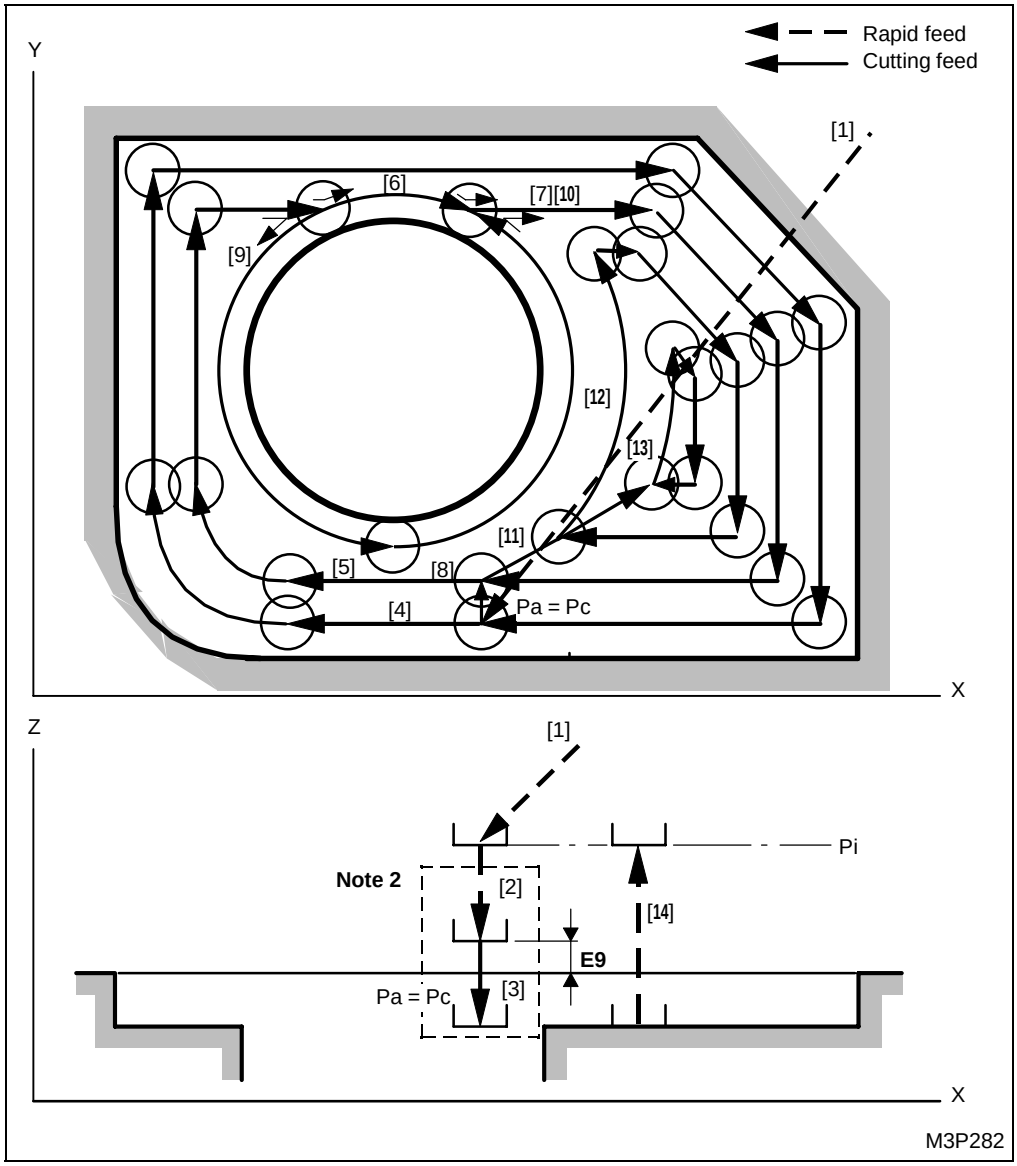
[10] Once it has interfered with the outside form, the tool machines along the outside form.

[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.

Note 1: When ? is displayed in the article 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 articles.

Note 2: See Subsection 4-8-5, “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

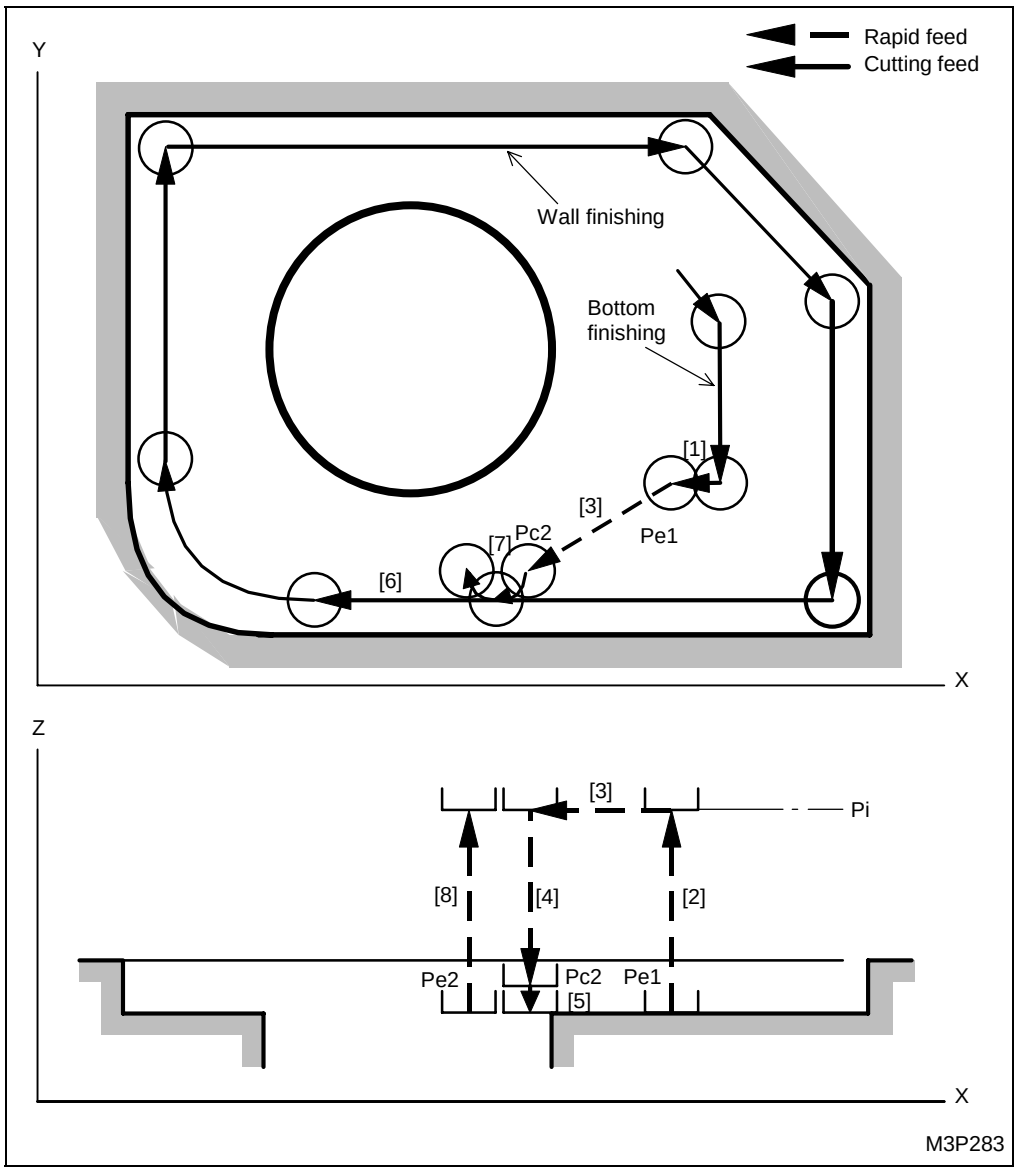
Finishing is performed, based on the entered data FIN-Z and FIN-R.

Bottom finishing is performed, with 0 < FIN-Z.

Wall finsihing is performed, with 0 < FIN-R.

<To perform both bottom- and wall-finishing>

When both bottom and wall are finished in finishing, the point determined by the data APRCH-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 to be determined by the data INITIAL-Z in the common unit

Note: When ? is displayed in the article 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 articles.

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	99.999	99.999	9	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	E-MILL								◆			
F2	E-MILL							◆	◆			

Remark 1: Data in unit represent a maximum input value.

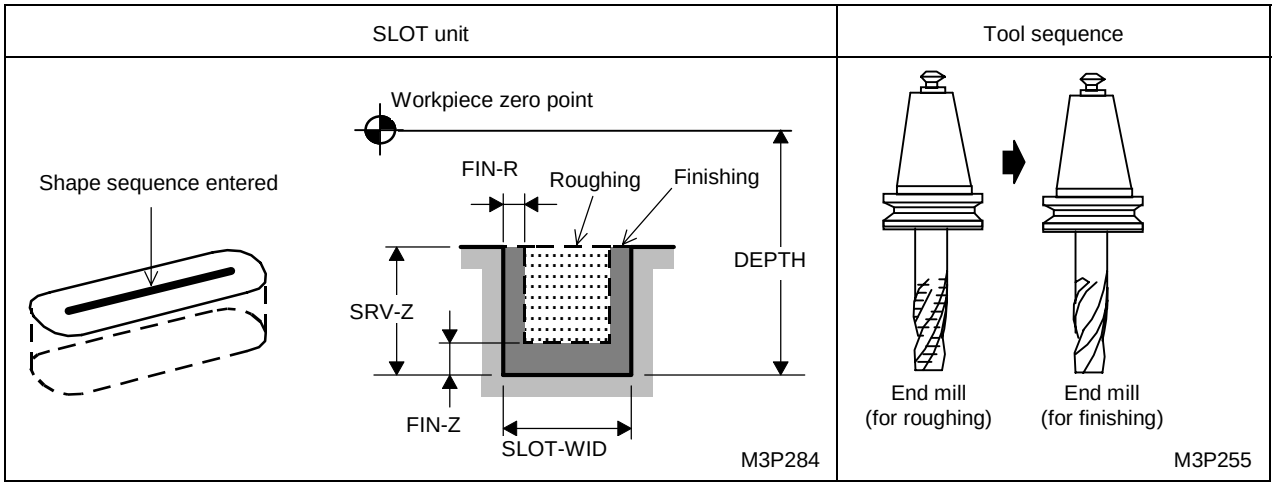
Remark 2: ◆: Data are not necessary to be set here.

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.

Q'ty	Machining	Pattern
1	R1 (Roughing)	FIN-Z = 0 and FIN-R = 0
1	F1 (Finishing)	SRV-Z ≤ FIN-Z or SLOT-WID ≤ (2 × FIN-R)
2	R1, F1 (Roughing/Finishing)	Other than those specified above

Remark 5: For the tool sequence data setting, see Subsection 4-8-4.



BTM: A buttom roughness code is selected out of the menu.

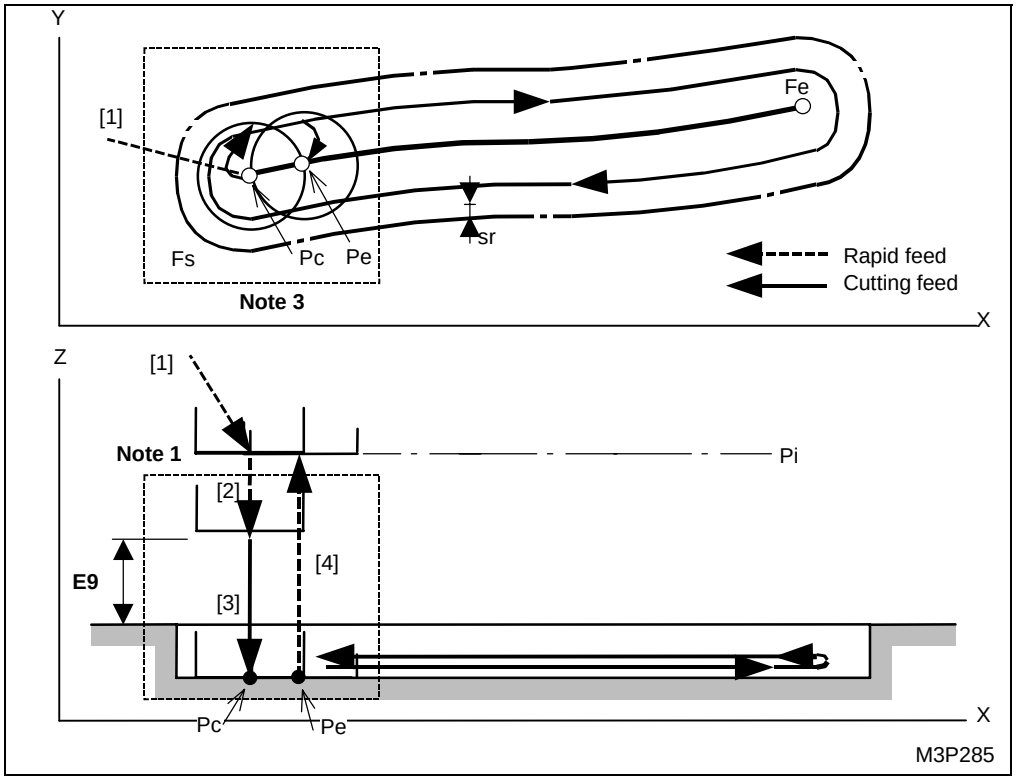
WAL: A wall roughness code is selected out of the menu.

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

With tool radius + FIN-R < $\frac{\text{SLOT-WID}}{2}$ < tool diameter + FIN-R



The bold codes represent parameter addresses.

Pc: Cutting start point to be automatically established

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

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

Pe: Escape point automatically established

Pi: Initial point to be determined by the data INITIAL-Z in the common unit

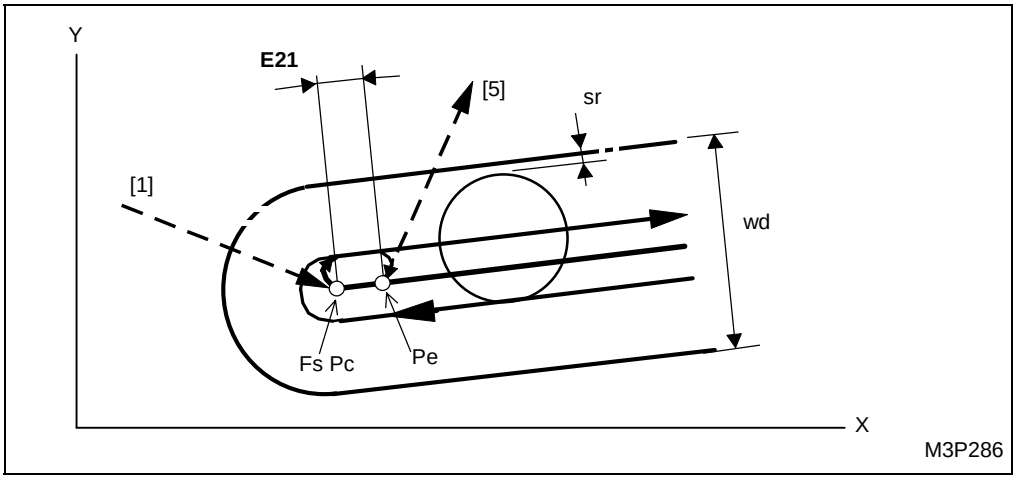
sr: Radial finishing allowance (FIN-R) to be entered in a machining unit

Note 1: See Subsection 4-8-5, “Precautions in face machining.”

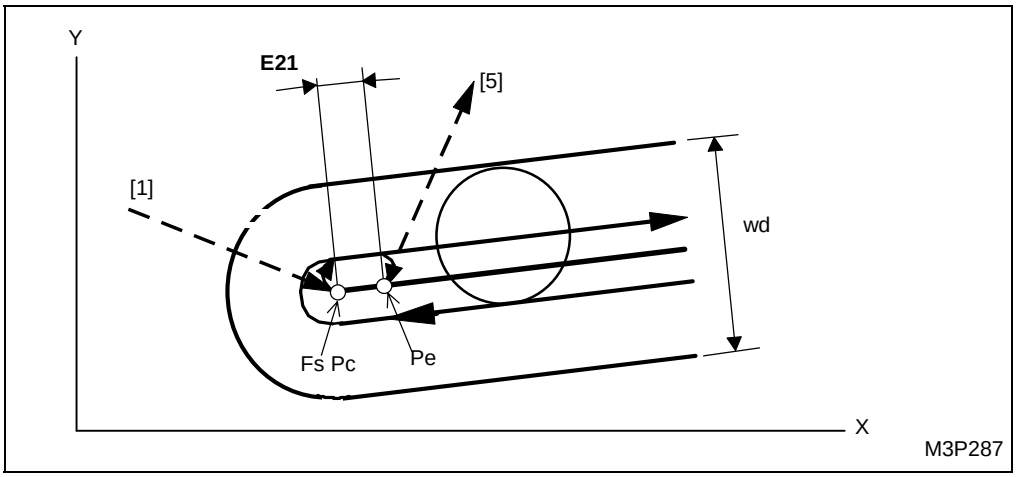
Note 2: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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>

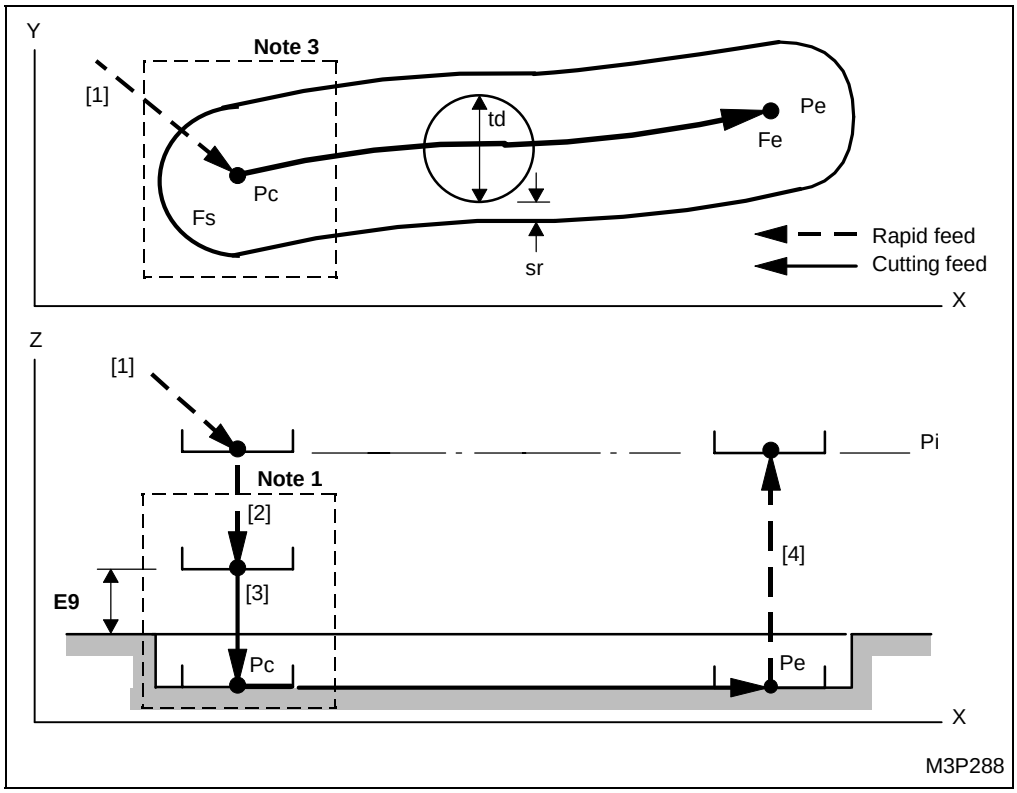
[1] The tool moves at a rapid feedrate to approach point (cutting start point).

[2] The tool moves at a rapid feedrate to the position entered by the parameter **E9**.

[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.

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 to be determined by the data INITIAL-Z in the common unit

td: Tool diameter to be registered in the **TOOL DATA** display

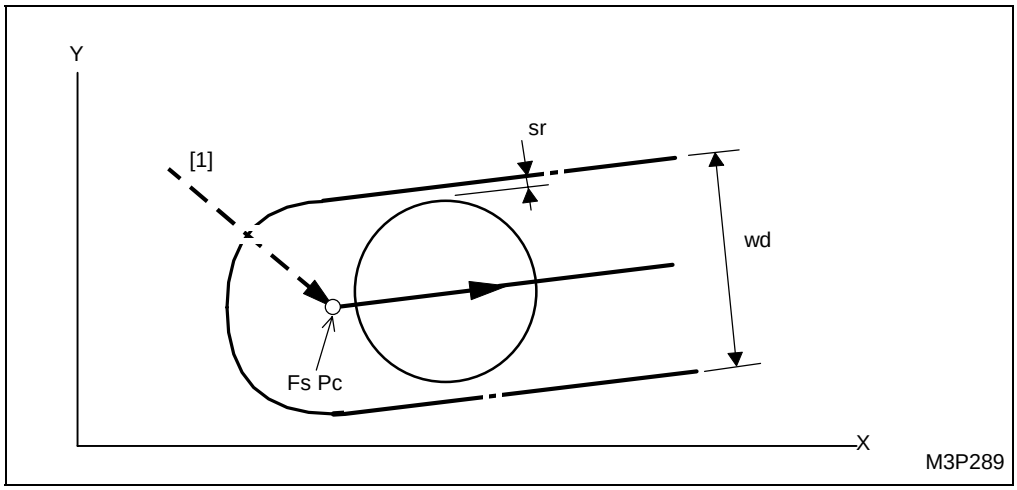
sr: Radial finishing allowance (FIN-R) to be entered in a machining unit

Note 1: See Subsection 4-8-5, “Precautions in face machining.”

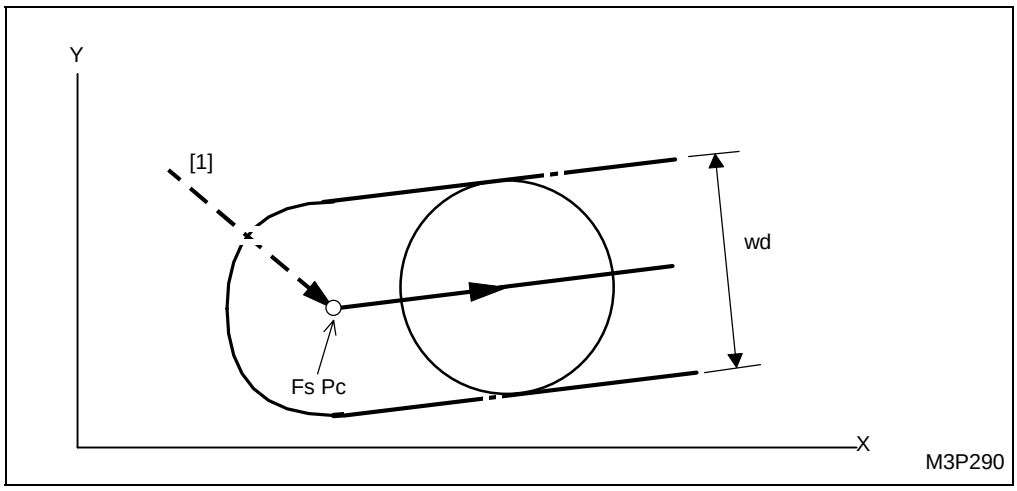
Note 2: The feedrate on tool path [3] is dependent upon the data ZFD (Z-axis feed) in the tool sequence.

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 position entered by the parameter **E9**.

[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.

4-8-4 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.

SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL												
F2	E-MILL							◆					
	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑	↑
	1	2	3	4	5	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.

X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR	X BI-DIR SHORT	Y BI-DIR SHORT	X BI-DIR ARCSHORT	Y BI-DIR ARCSHORT		

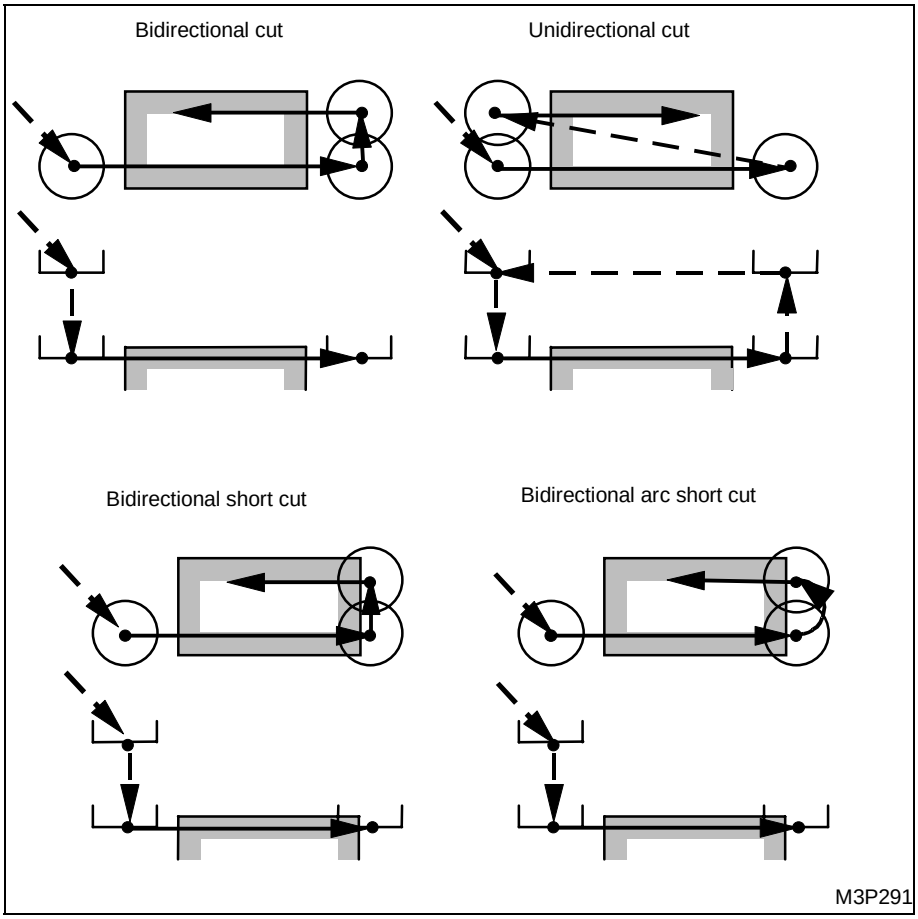
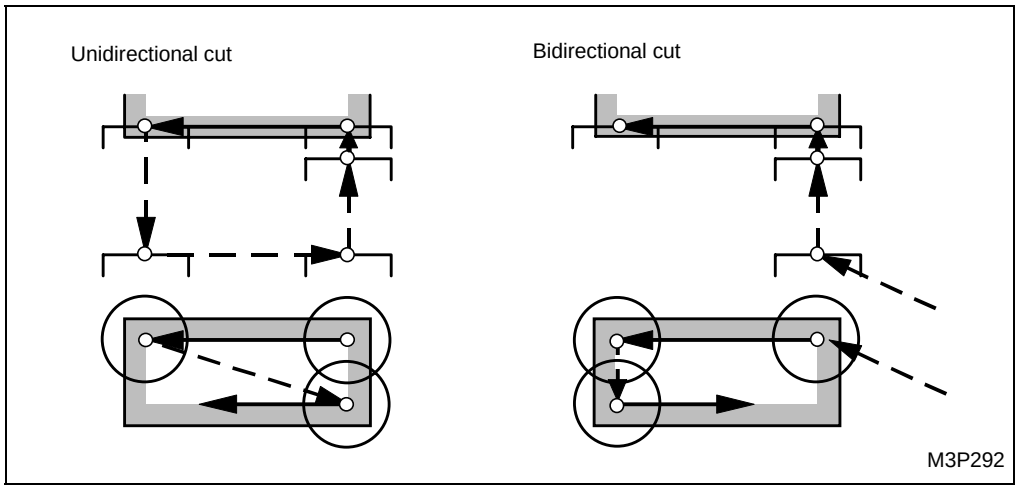


Fig. 4-20 Tool path patterns

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.

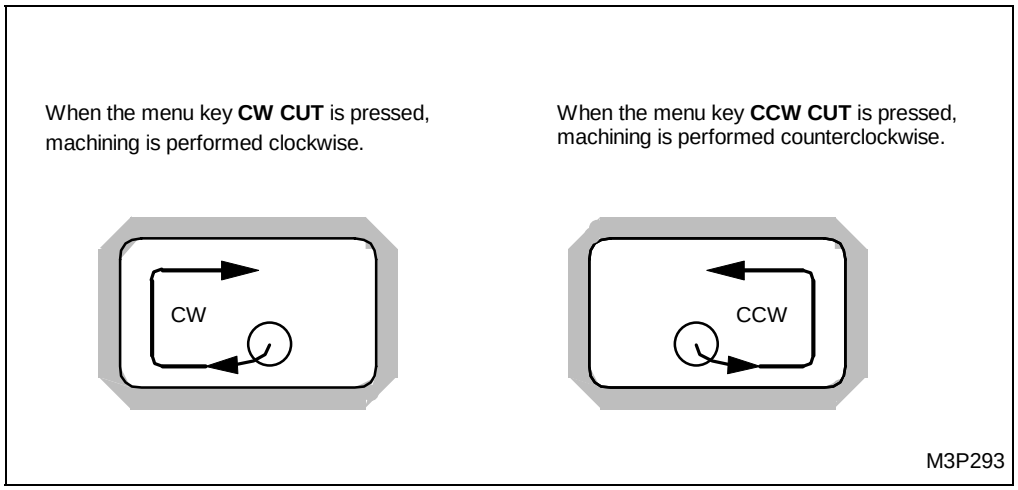
X BI-DIR	Y BI-DIR	X UNI-DIR	Y UNI-DIR						



C. Other units

A machining (turning) direction is selected by the use of menu keys.

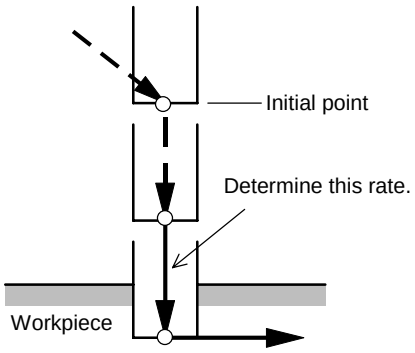
CW CUT	CCW CUT								



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: Feed × $\frac{\textbf{E17}}{10}$
Numeric value (α)	Feed × α

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 4-8-5, “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.

$$\text{WID-R} = \text{Norminal tool diameter} \times \frac{\textbf{E10}}{10} : \text{FACE MIL, TOP EMIL and STEP}$$

$$\text{WID-R} = \text{Norminal tool diameter} \times \frac{\textbf{E14}}{10} : \text{POCKET, PCKT MT and PCKT VLY}$$

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 cutitng 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 APPENDIX-1, “LIST OF M CODES.”)

4-8-5 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 path]

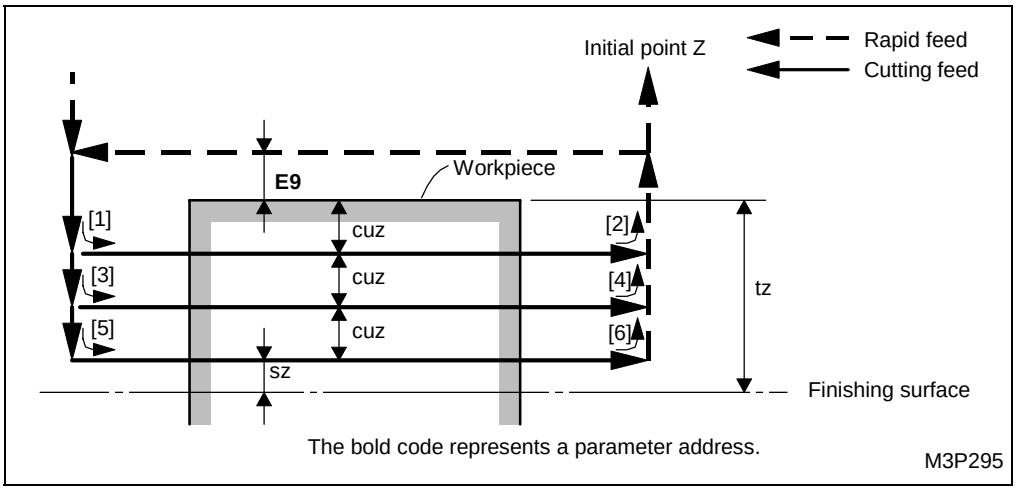


Fig. 4-21 Basic tool path

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

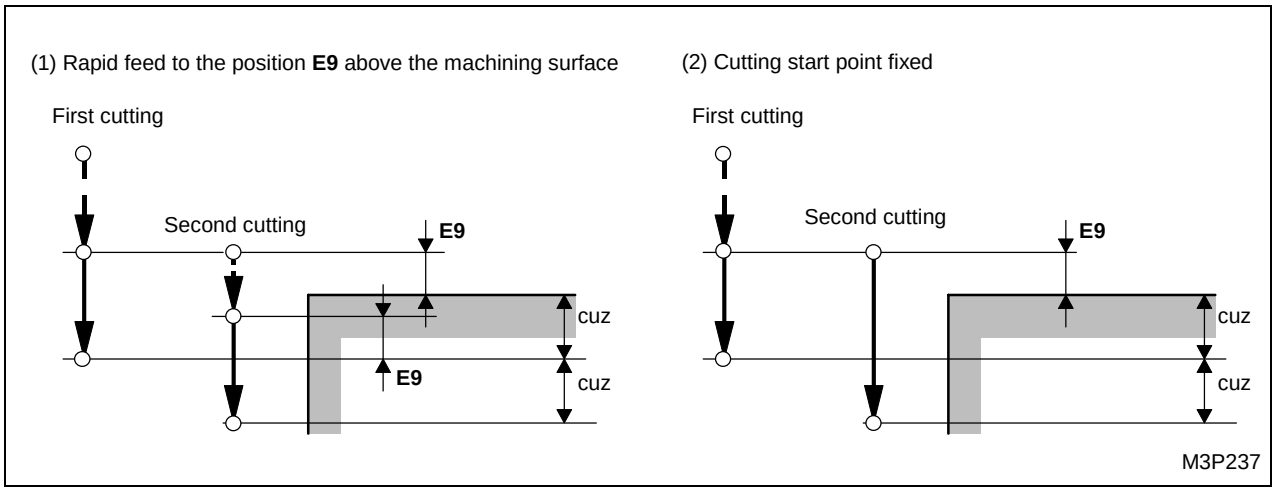
sz: Z-axial finishing allowance FIN-Z to be entered in the machining unit

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)

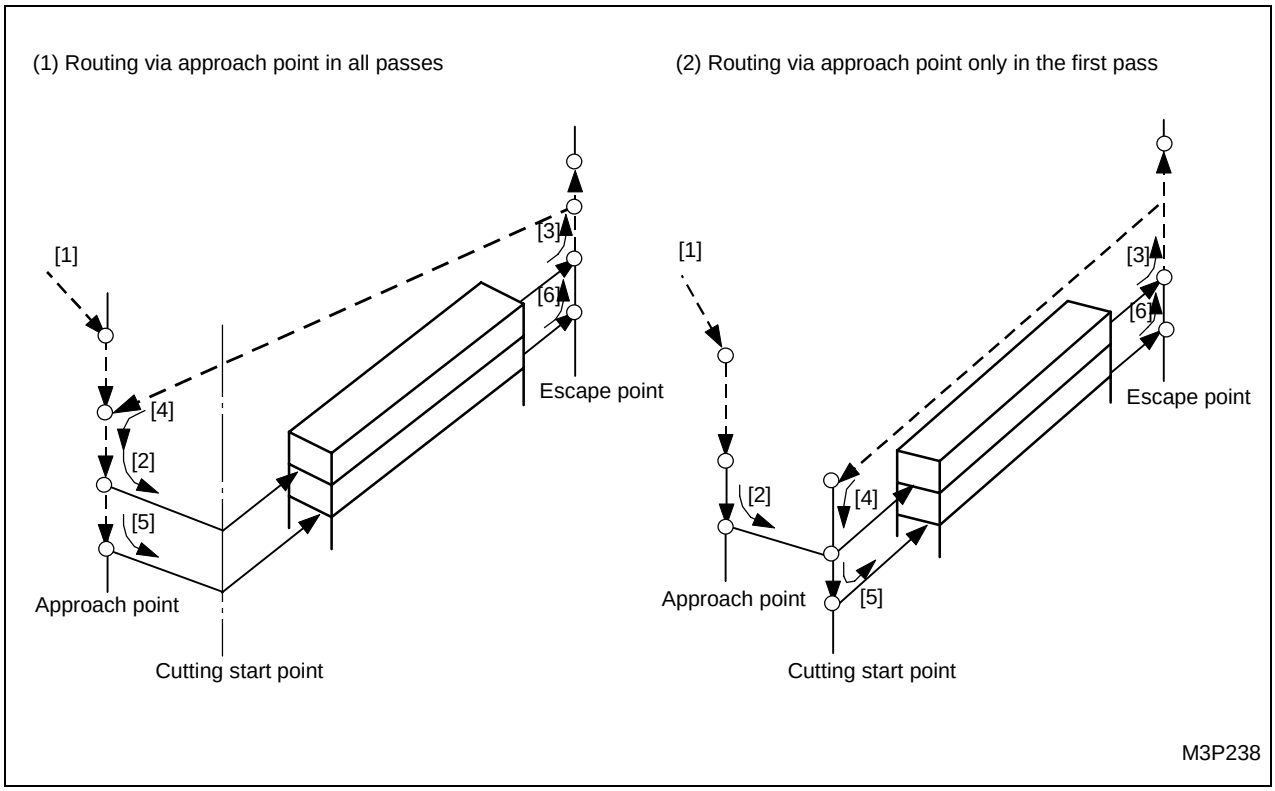
A. Setting the cutting start position along the Z axis

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 position **E9** above the machining surface, (1)

* As for pattern (1), the starting position of cutting feed is determined by the setting of parameter **E7** (instead of **E9**) from the second cutting when the following conditions are satisfied:

- Bit 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)

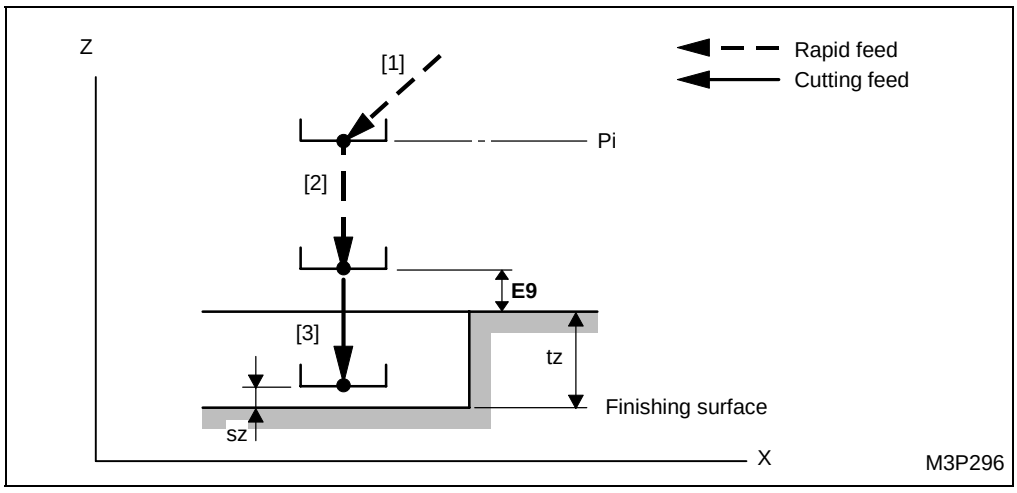
1: Routing via approach point in all passes, (1)

Note 1: B can only be used in the end milling-slot (SLOT) unit.

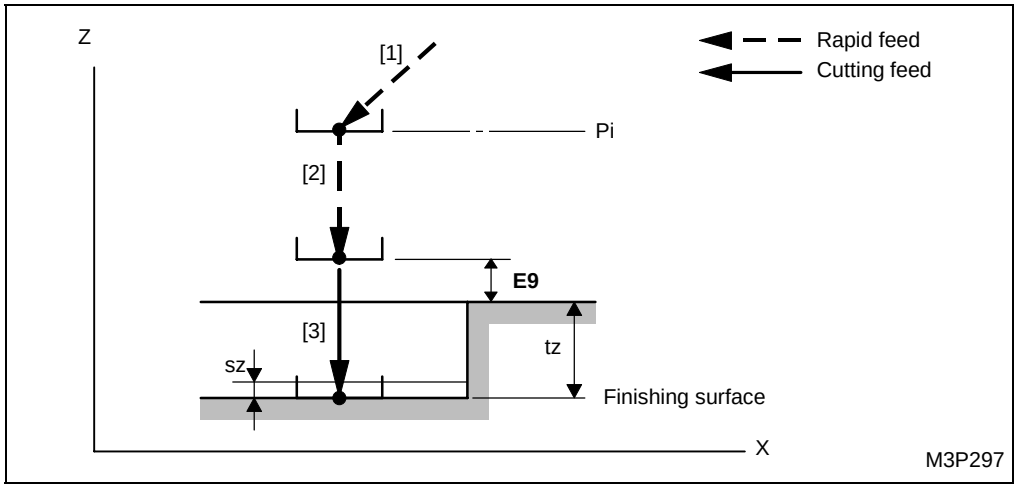
Note 2: The tool path shown at basic tool path above is selected automatically for face machining unit that is not assigned by these parameters.

2. Detail tool path of a Z-axial cutting

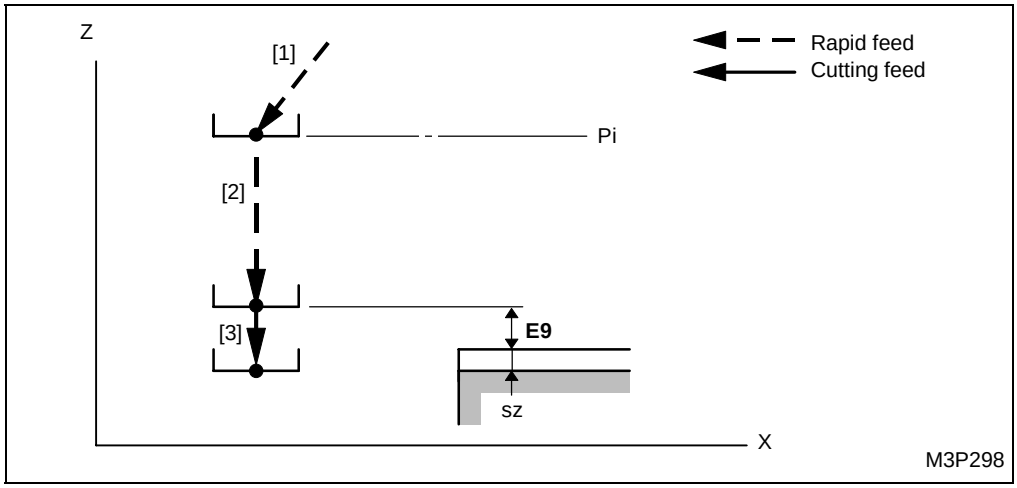
- Roughing



- 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 to be determined by the data INITIAL-Z in the common unit

tz: Z-axial machining 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: The starting allowance of axial cutting, specified by parameter **E9**, 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.

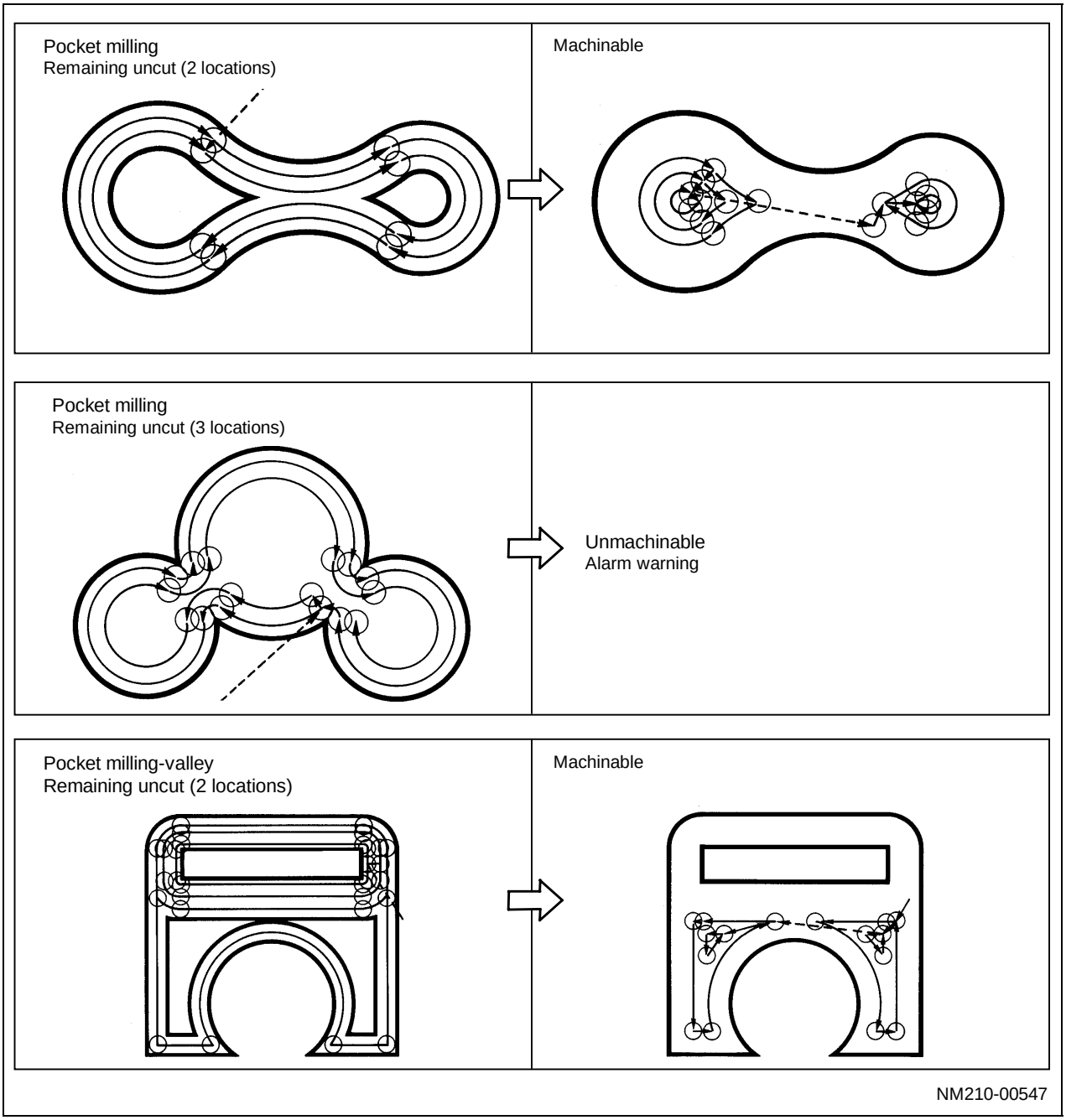
Unit	Parameter	Unit	Parameter
End milling-step	**E91**, bit 3	Pocket milling-mountain	**E93**, bit 3
Pocket milling	**E92**, bit 3	Pocket milling-valley	**E94**, bit 3

- 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, if will result in an alarm.

Example:


Pocket milling
Remaining uncut (2 locations)
Machinable
Pocket milling
Remaining uncut (3 locations)
Unmachinable
Alarm warning
Pocket milling-valley
Remaining uncut (2 locations)
Machinable
NM210-00547

4-8-6 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

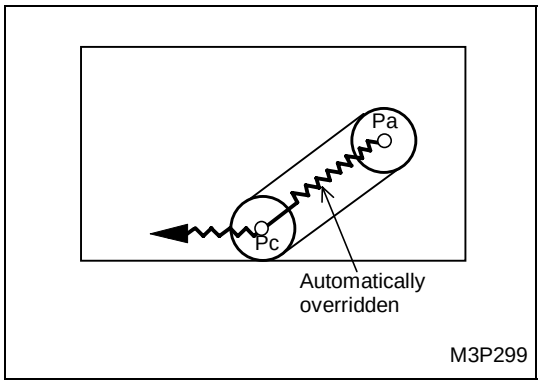
The overall width cutting override is valid in pocket milling on the following four cases on which the cutting width equals the tool diameter:

A. To move from the approach point to the cutting start point:

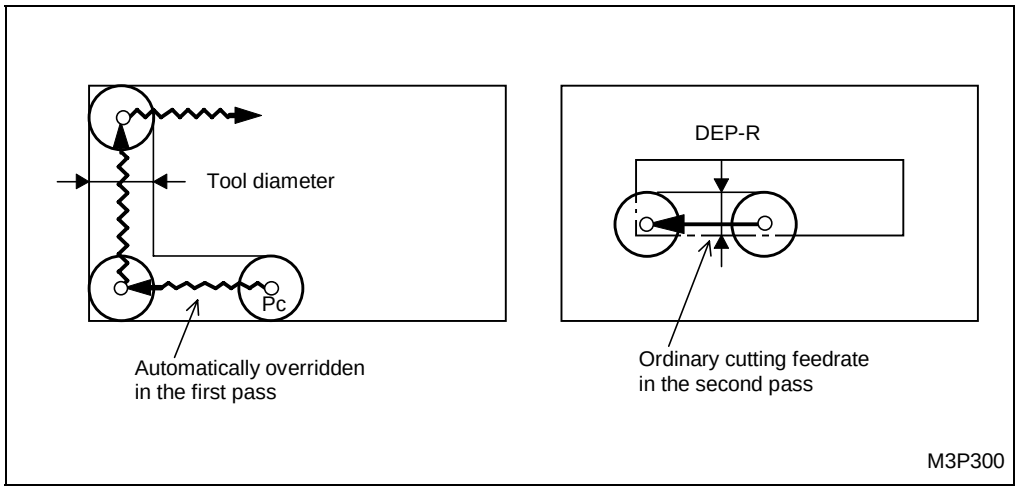
Illustration below is an example of the pocket milling unit.

Pa: Approach point

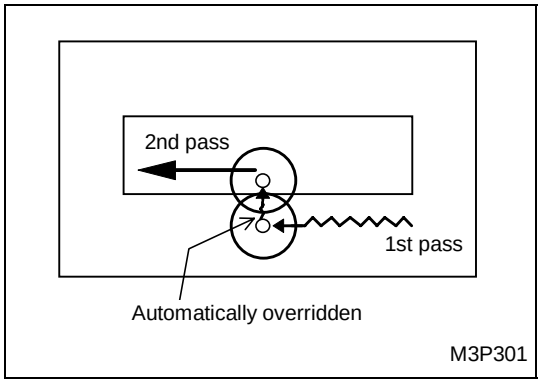
Pc: Cutting start point



B. First pass

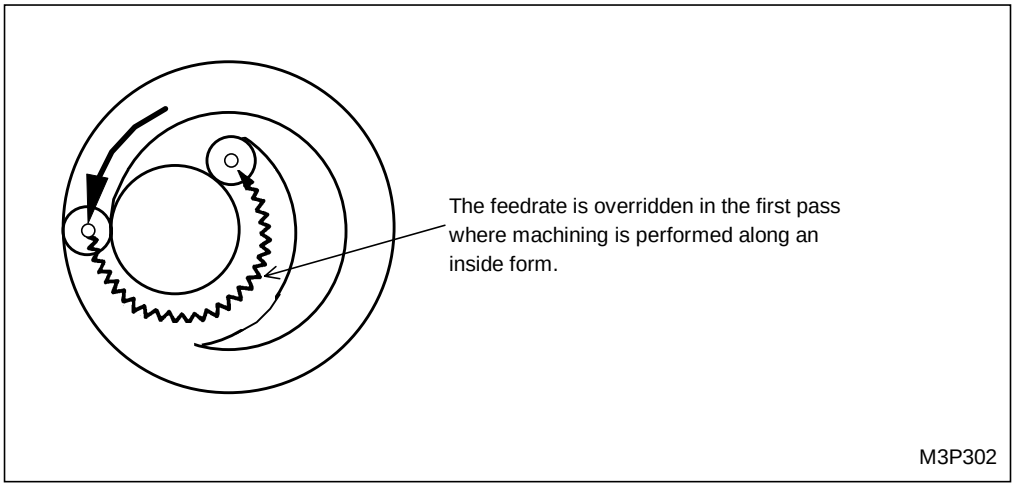


C. To move to the next cutting start point



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**.

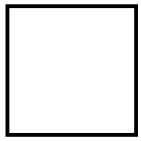
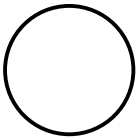
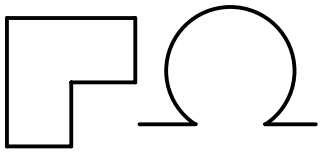
With the parameter set to 0, the overall width cutting override is invalid.

4-8-7 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.

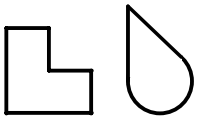
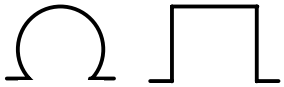
1. Definitions of forms

In line machining and face machining units one of the following three patterns can be selected.

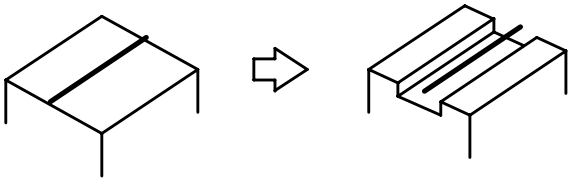
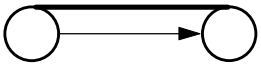
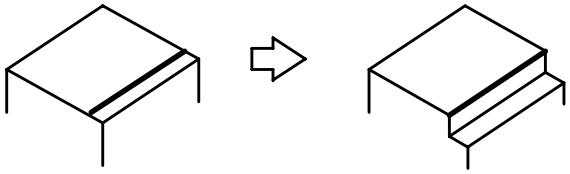
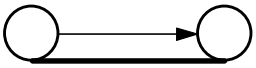
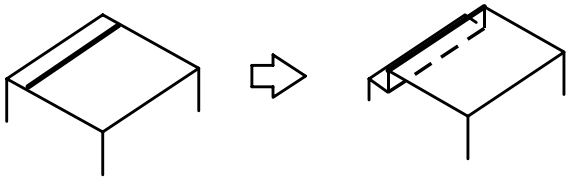
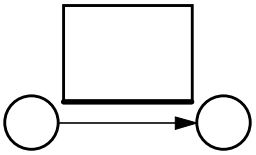
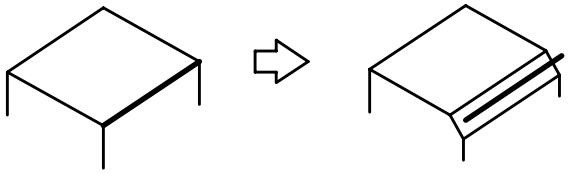
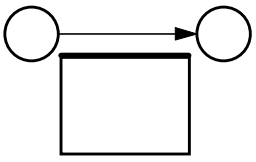
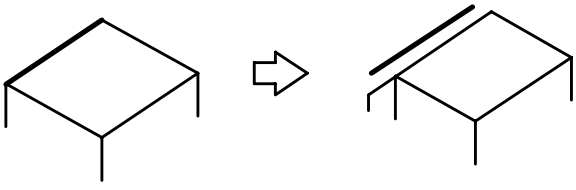
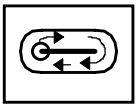
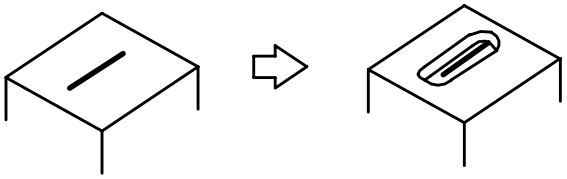
Fixed form		Arbitrary form
SQUARE	CIRCLE	ARBITRY
		 M3P303

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	FACE MIL, TOP EMIL, POCKET	SLOT M3P304
	Machining with at least two defined shapes required	STEP, PCKT MT, PCKT VLY	

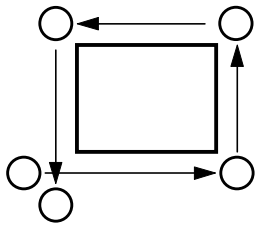
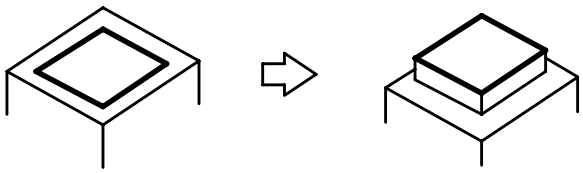
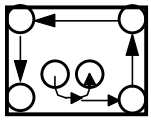
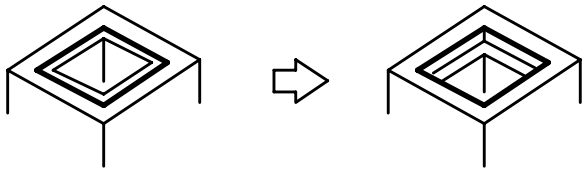
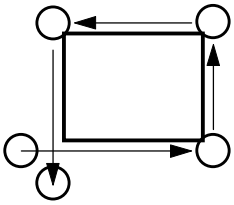
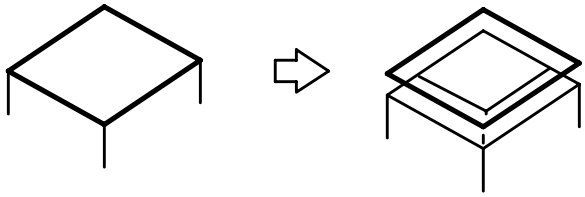
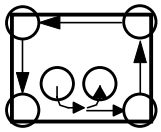
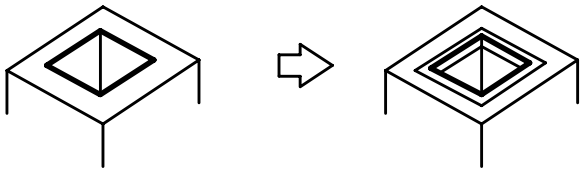
B. Machining units related to the open form

Machining unit		Tool path	Before machining → After machining
Line machining	LINE CTR		
	LINE RGT		
	LINE LFT		
	CHMF RGT		
	CHMF LFT		
Face machining	SLOT		

M3P305

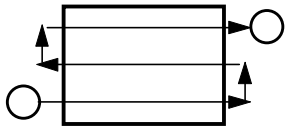
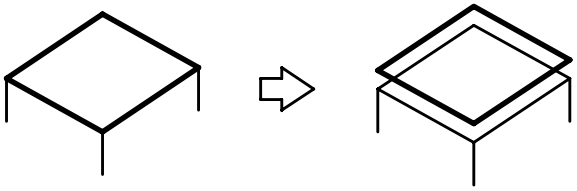
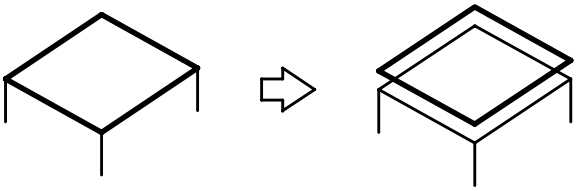
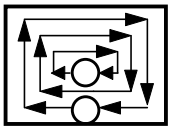
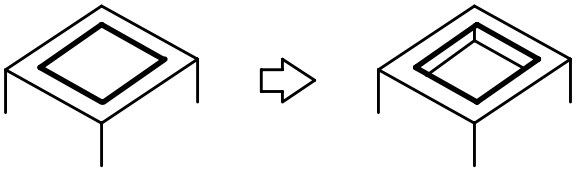
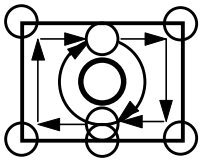
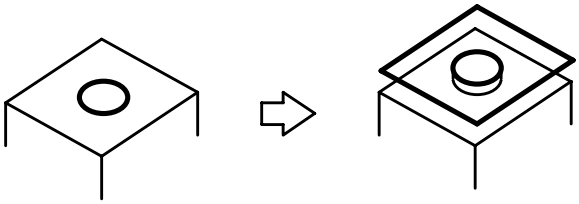
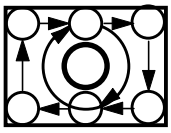
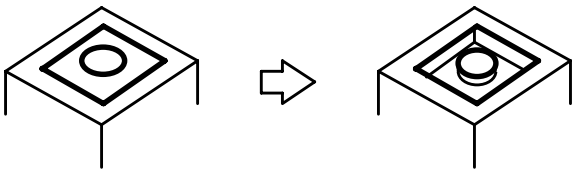
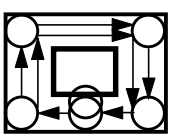
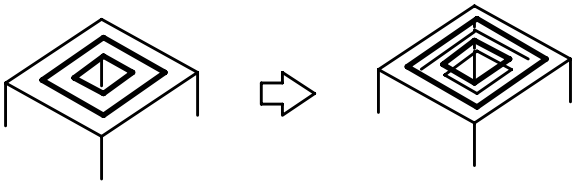
C. Machining units related to the closed form

- 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	FACE 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:

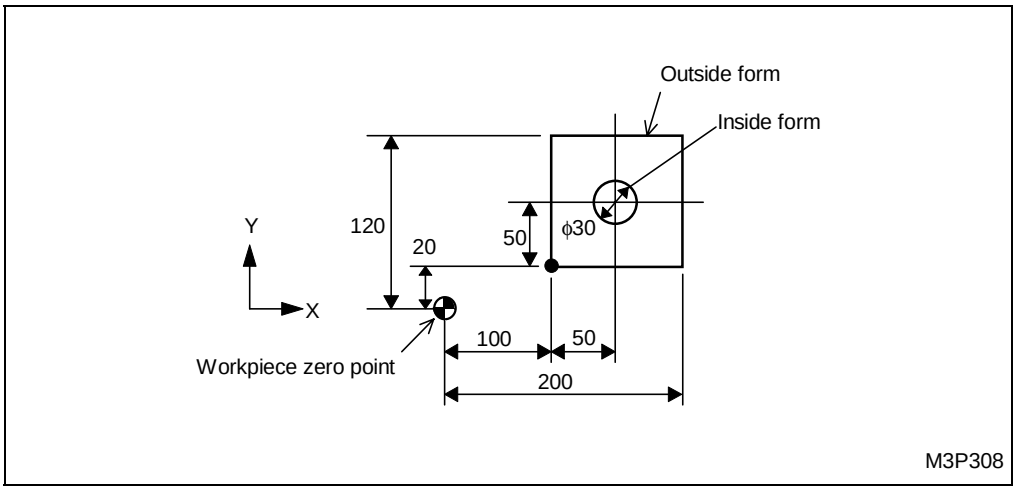


FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	100.	20.	200.	120.				
2	CIR	150.	70.	15.	◆	◆	◆	◆	◆

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**.

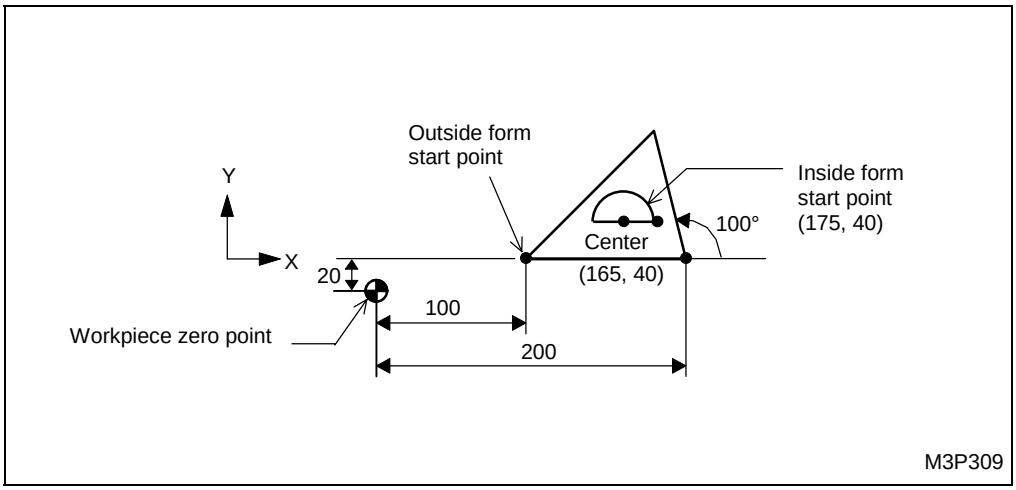


FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	100.	20.					
2	LINE	200.	20.					
3	LINE		80.	100.				
4	LINE	175.	40.					
5	CCW	155.	40.		165.	40.		

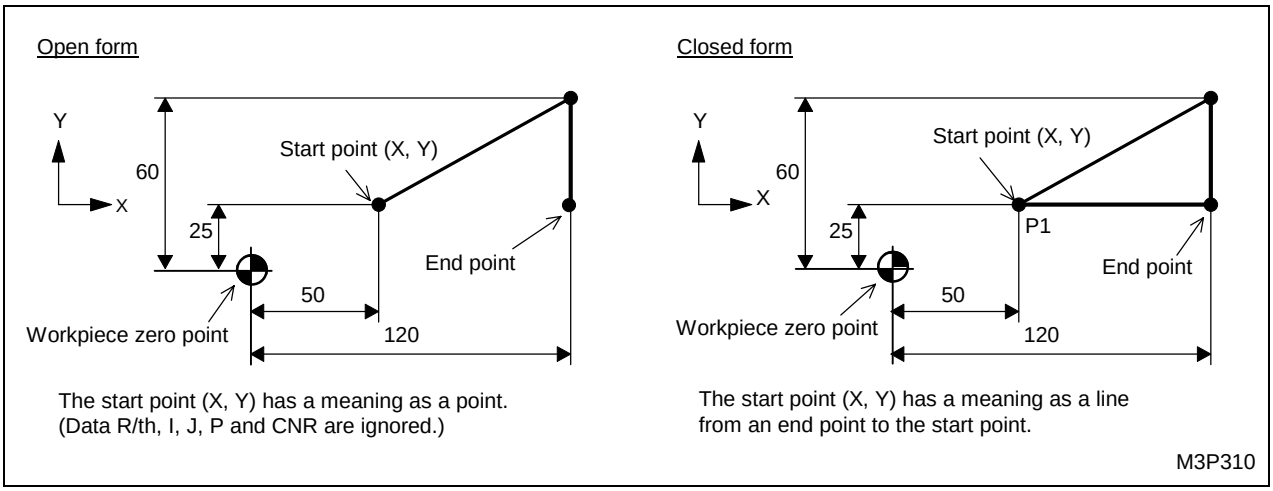
STARTING POINT → LINE

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:

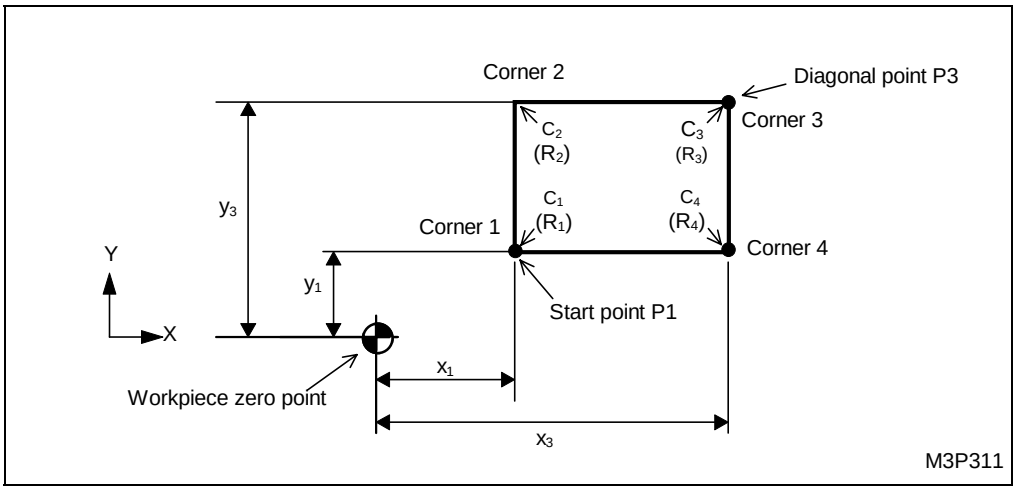


3. Entry of shape sequence data

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		ARBITRY				SHAPE END		

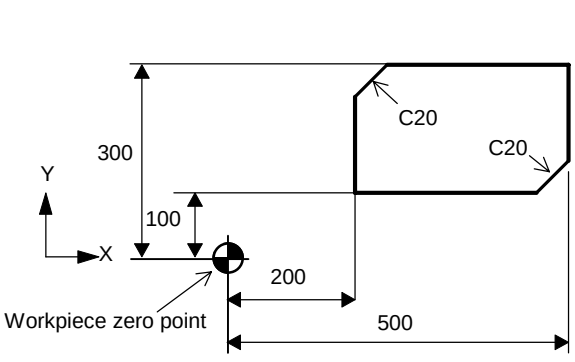
Press the menu key **SQUARE**.

(b) Data setting in shape sequence SQUARE (see figure above)

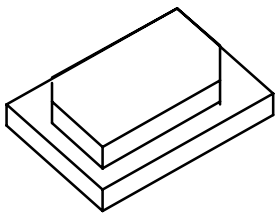
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	x_1	y_1	x_3	y_3	C_1 (R_1)	C_2 (R_2)	C_3 (R_3)	C_4 (R_4)

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 CN3 CN4	Enter the same data as those for corner 1.

Example 1:



A workpiece machined in a line machining unit (LINE OUT)



M3P312

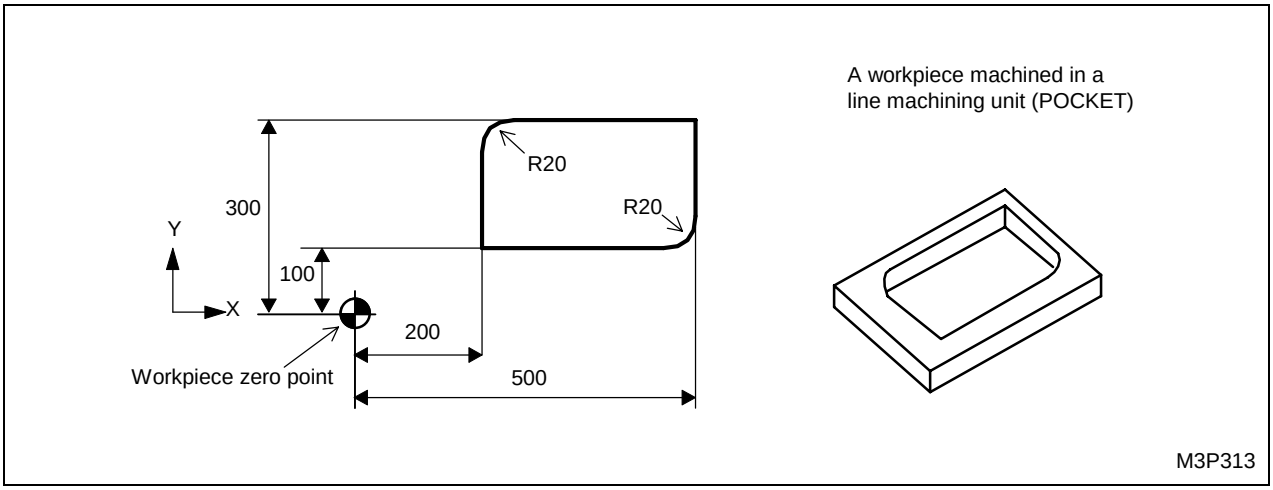
UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH	FIN-Z	FIN-R
1	LINE OUT	5.	5.	5.	1	0.	0.

SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MIL	10.A		?	?	CW	G01	5.	◆	15	0.046		

FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4
1	SQR	200.	100.	500.	300.	0.	C20.	0.	C20.

← Shape sequence data

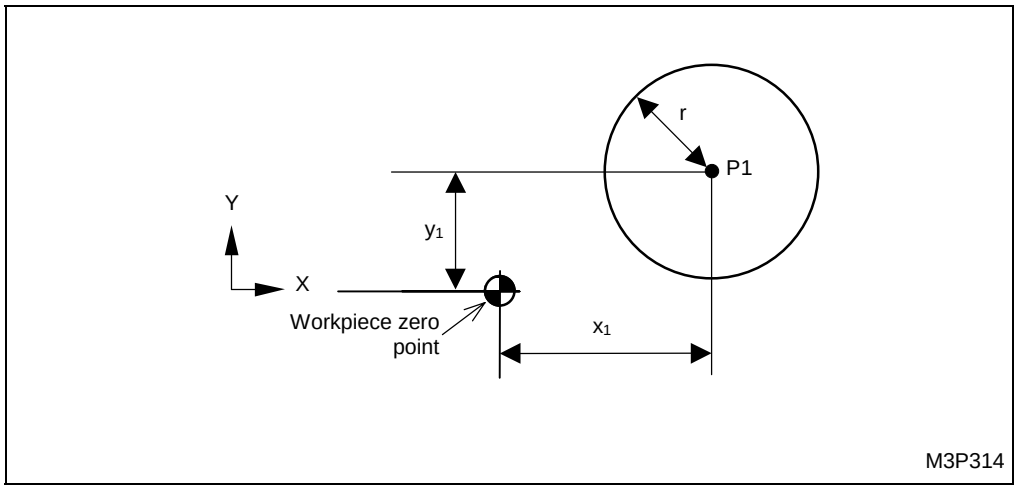
Example 2:



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	E-MIL	10.A		?	?	CW	G01	5.	5.			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4			
1	SQR	200.	100.	500.	300.	0	R20.	0.	R20.			

← Shape sequence data

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		ARBITRY				SHAPE END	CHECK	
□	○								

Press the menu key **CIRCLE**.

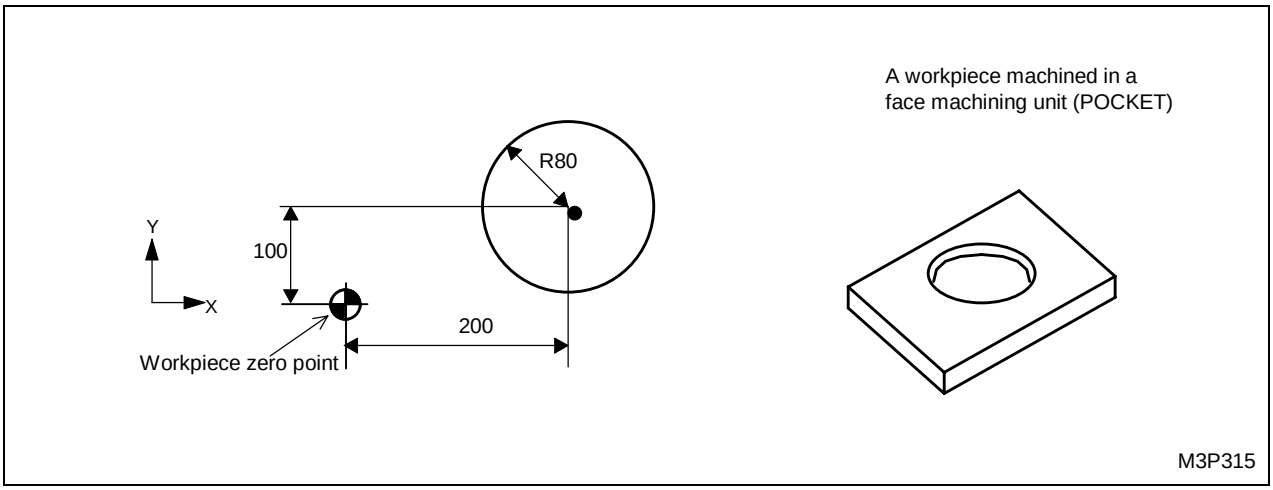
(b) Data setting in shape sequence CIRCLE (see figure above)

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	E-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>?

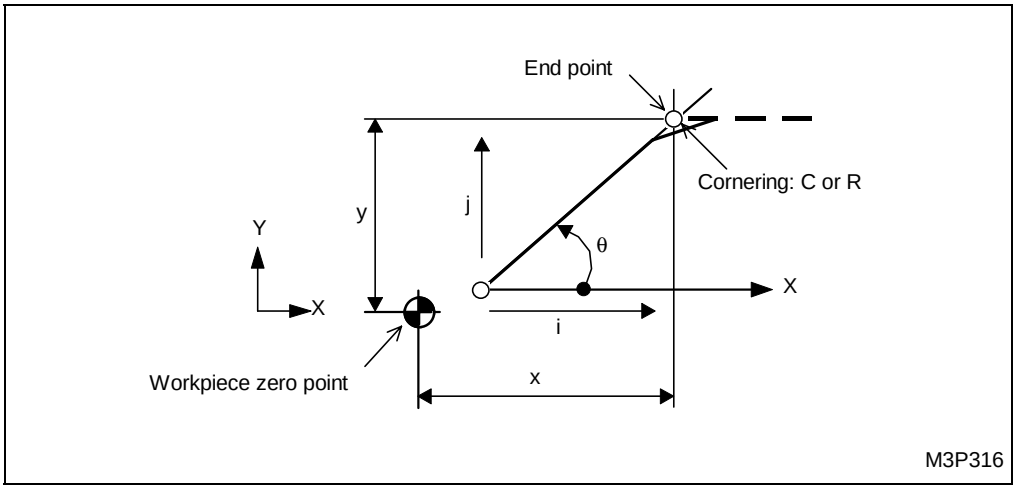
SQUARE	CIRCLE		ARBITRY				SHAPE END	CHECK	
□	○		⟆						

Press the menu key **ARBITRY**.

➔ The following menu will be displayed.

LINE	CW ARC	CCW ARC	SHAPE ROTATE	SHAPE SHIFT	REPEAT END	STARTING POINT	SHAPE END	CHECK	
	↻	↺							

2. LINE

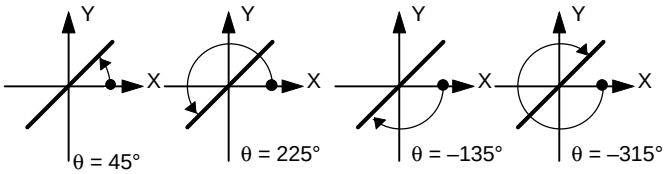


(a) Menu selection

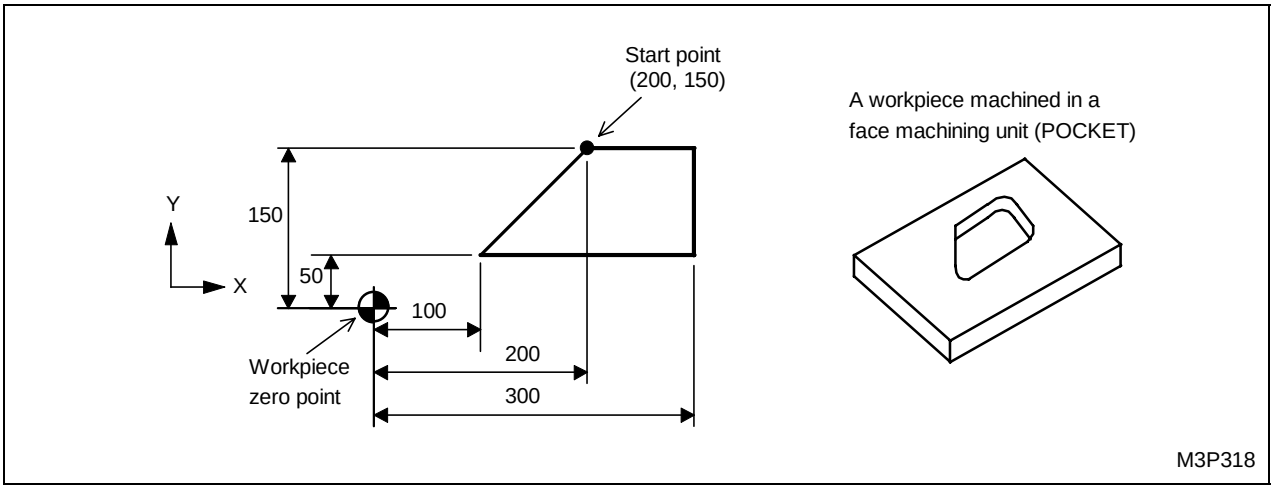
Press the menu key **LINE**.

(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.  M3P317
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): UP | DOWN | LEFT | RIGHT Four types given to the left are available. **Note:** For details refer to "C. Function of automatically calculating a point of intersection."
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:

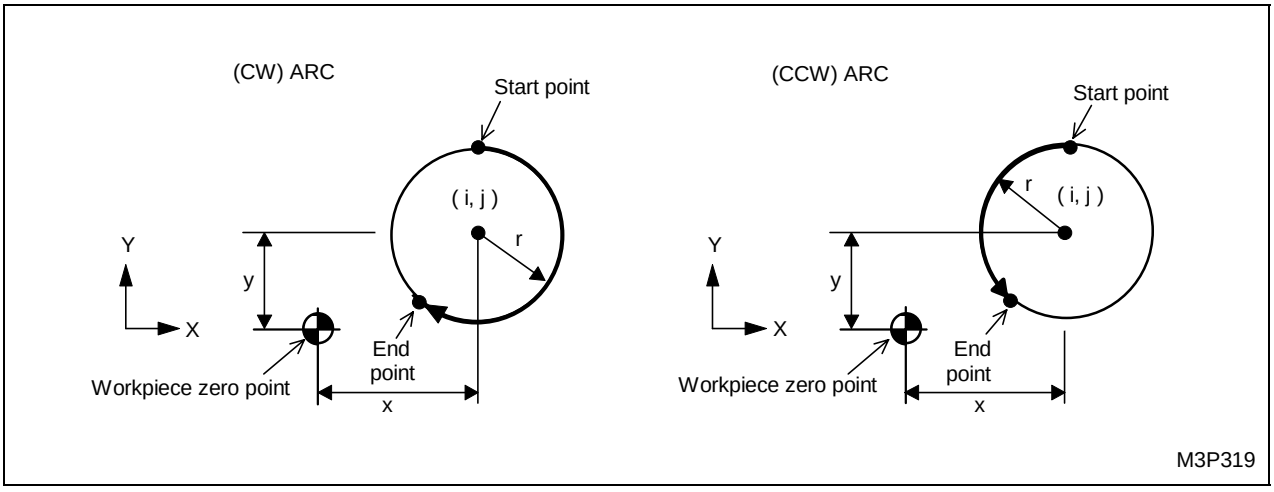


UNo.	UNIT	DEPTH	SRV-Z		BTM	WAL	FIN-Z	FIN-R				
1	POCKET	5.	5.		1	1	0.	0.				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M M
R1	E-MILL	10.A		?	?	CW	G01	5.	6.	12	0.03	
FIG	PTN	X	Y	R/th	I	J	P	CNR				
1	LINE	200.	150.									
2	LINE	300.	?		1.	0.						
3	LINE	?	50.	90.								
4	LINE	100.	?	↑	-1.	0.						

← Shape sequence data

(θ = –90°, 270° or –270° is applicable.)

3. ARC (CW and CCW)



(a) Menu selection

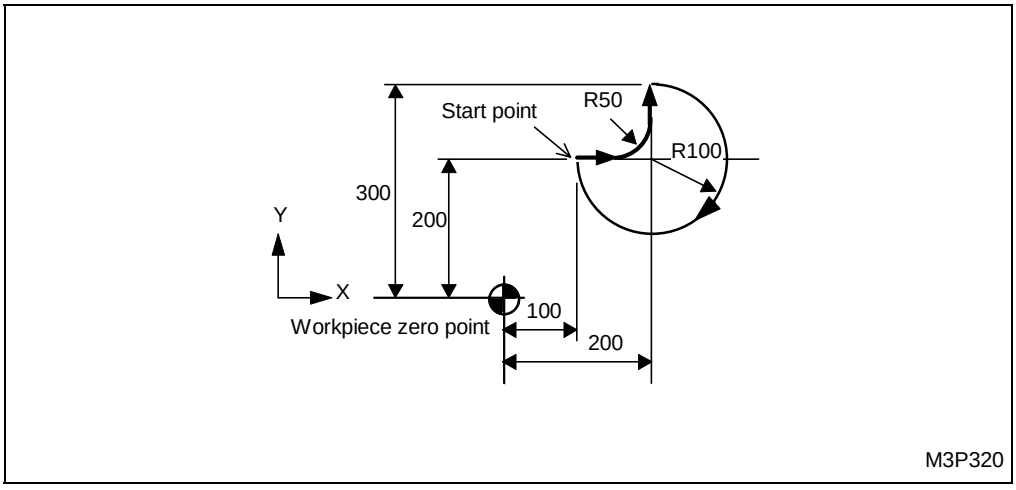
Press the menu keys **CW ARC** or **CCW ARC**.

(b) Data setting in shape sequence CW/CCW ARC. (see figure above)

FIG	PTN	X	Y	R/th	I	J	P	CNR
4	CW	x	y	r	i	j	p	C
	(CCW)							(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): UP | DOWN | LEFT | RIGHT Four types given to the left are available. **Note:** For details refer to "C. Function of automatically calculating a point of intersection.
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:

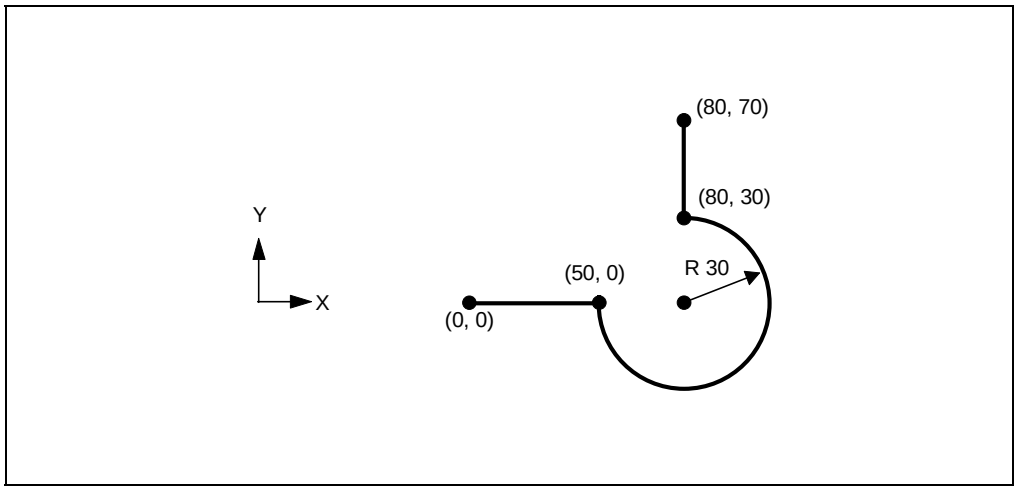


UNo.	UNIT	DEPTH		SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE OUT	5.		5.	5.	1		0.	0.				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	10.A		?	?	CCW	G01	5.	◆	17	0.006		

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:

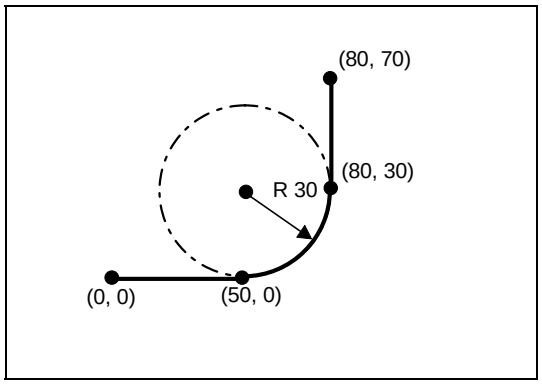


UNo.	UNIT	DEPTH		SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
2	LINE LFT	5.		5.	10.	3		0.1	0.1				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	◆		4.9	◆	76	0.123		
F2	E-MILL	20.A		?	?	◆		◆	◆	76	0.313		

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.					

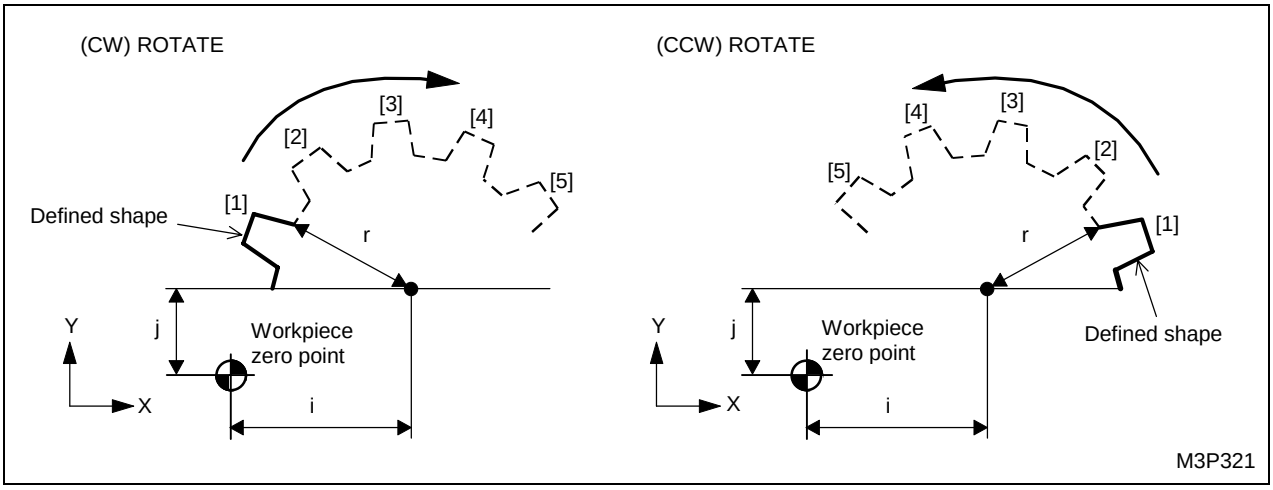
← Shape sequence data

- 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/th	I	J	P	CNR
5	CW-SH	◆	◆	r	i	j	p	◆
	(CCW-SH)							
	Use LINE, CW ARC or CCW ARC to enter a defined shape.							
	REP-EN	◆	◆	◆	◆	◆	◆	◆

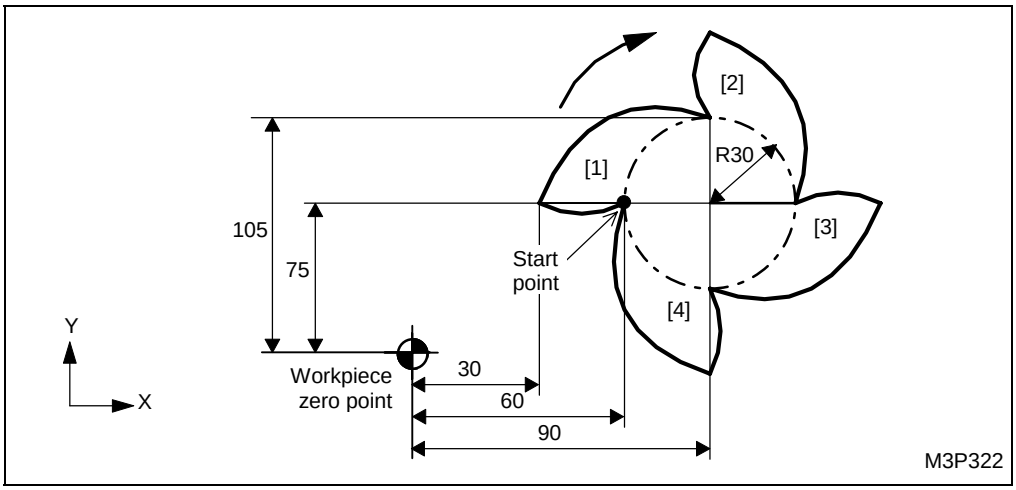
◆: Data are not necessary to be set here.

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



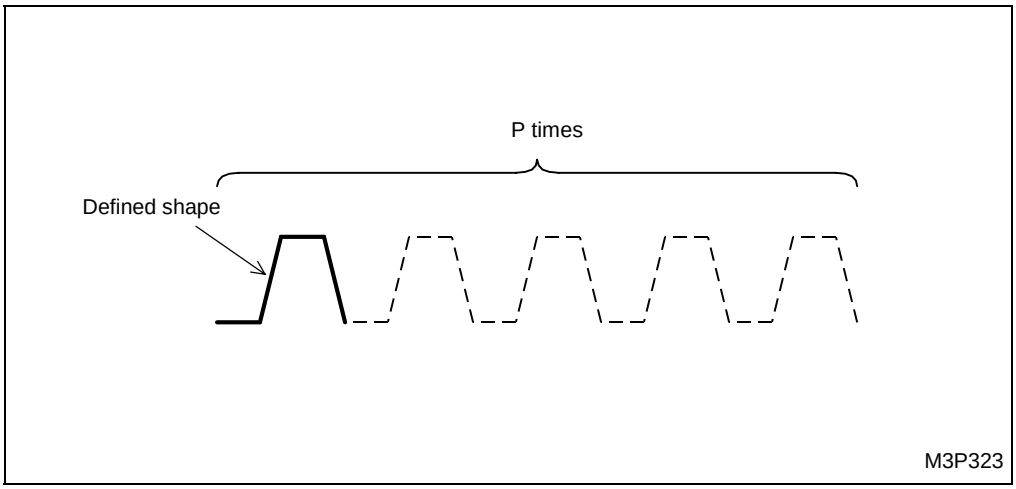
UNo.	UNIT	DEPTH		SRV-Z	SRV-R	RGH		FIN-Z	FIN-R				
1	LINE OUT	10.		10.	10.	4		0.6	0.6				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	20.A		?	?	CW	G01	9.4	◆	14	0.043		
F2	E-MILL	10.A		?	?	CW	G01	◆	◆	20	0.224		

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	◆	◆	◆	◆	◆	◆	◆

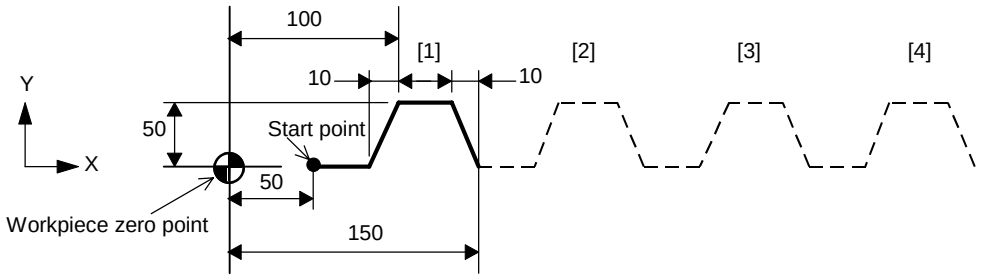
◆ : Data are not necessary to be set here.

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:



M3P324

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	E-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.

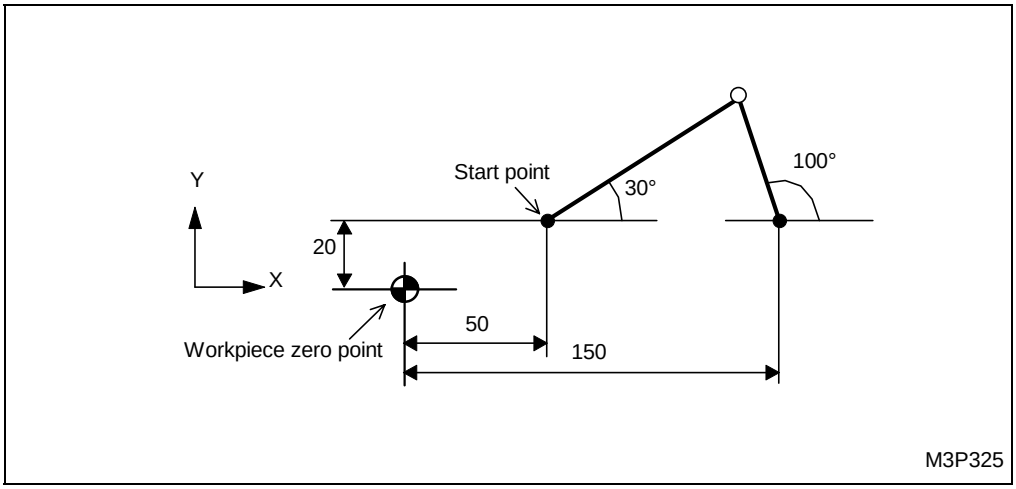


Fig. 4-22 Function of automatically calculating a point of intersection

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.)

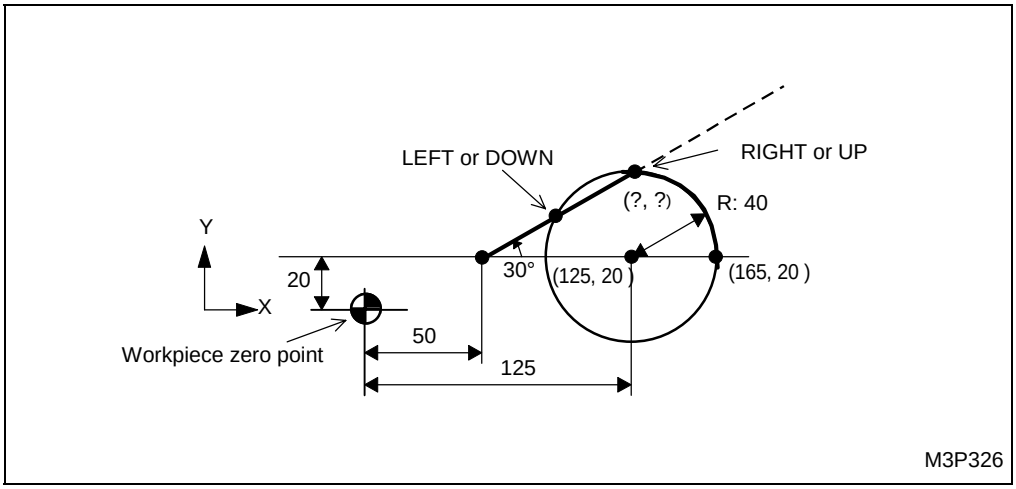
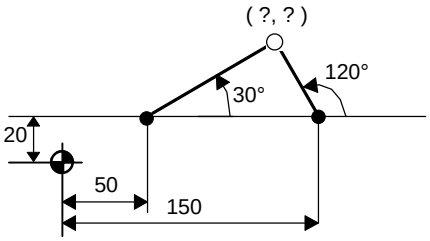
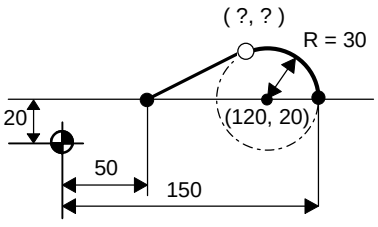
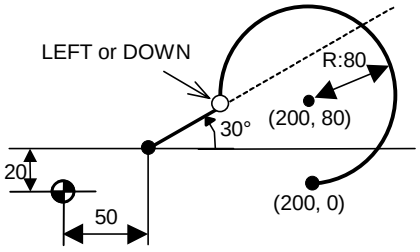
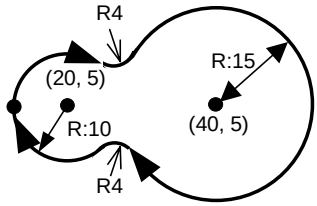
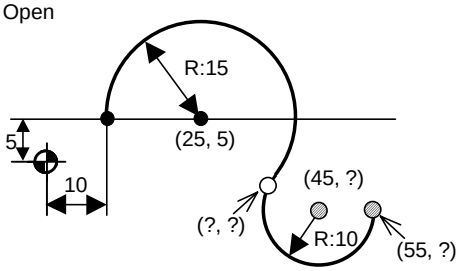
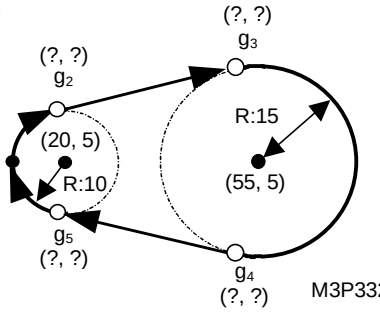
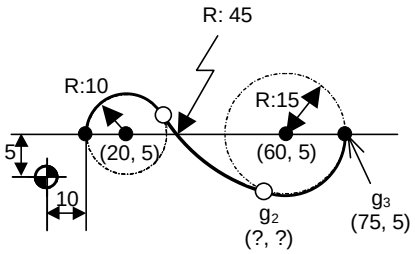


FIG	PTN	X	Y	R/th	I	J	P	CNR
1	LINE	50.	20.					
2	LINE	?	?	30.				
3	CW	165.	20.	40.	125.	20.		

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	 M3P327	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)	 M3P328	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)	 M3P329	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	Closed  M3P330	FIG PTN X Y R/th I J P CNR 1 CW ? ? 10. 20. 5. U R4 2 CW ? ? 15. 40. 5. D R4
	Open  M3P331	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	Closed  M3P332	FIG PTN X Y R/th I J P CNR 1 LINE ? ? 2 CW ? ? 10. 20. 5. 3 LINE ? ? 4 CW ? ? 15. 55. 5.
ARC | ARC | ARC	 M3P333	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.

● : 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).

4-9 End Unit

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	MANUAL PROGRAM	OTHER	WPC	OFFSET	END	SHAPE CHECK	

2. Data setting in end unit

UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	*4*	ANGLE (*5*)
	END	☐	☐	☐	☐	☐	☐	☐	☐

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

Cursor position	Description
CONTI.	Specify if the program is repeated or not. - No repetition ..... 0 ⏎ - Repetition ..... 1 ⏎ **Note :** Set 1 for the case of sub-program.
NUMBER	Specify if the number of executions of the program is displayed or not by the counter on the **POSITION** display. - No display ..... 0 ⏎ - Display ..... 1 ⏎ **Note :** The number of target times is entered in the **POSITION** display. Moreover, the cumulative value can be reset to zero.
ATC	Specify the ATC movement at the end of machining. The tool is not returned. ..... 0 ⏎ After returning the tool, the axes are moved. ..... 1 ⏎ After the axes movement, the tool is returned. ..... 2 ⏎ **Note 1:** If no data is set, it is regarded that 1 has been set. **Note 2:** If 1 is set in the article CONTI, ◆ is displayed and no data can be entered.

Cursor position	Description
X/Y/Z/*4*/*5*	The axes are moved in rapid feed to the position entered in the articles 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 article ATC, X, Y and Z axes are moved at the same time. **Note 1:** If the coordinates are not specified, the axes movement are not done. **Note 2:** If 1 is set in the article CONTI., ◆ are displayed in the articles X, Y, Z, 4, 5 and ANGLE and no data can be set.
X	Enter the X coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999)
Y	Enter the Y coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999)
Z	Enter the Z coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999) Permissible input range for the rotational axis (0 to ±999.999 degrees)
4	Enter the 4th axis coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999) Permissible input range for the rotational axis (0 to ±999.999 degrees)
ANGLE	Specify the angle of the index table at the end of machining. Permissible input range (0 to 360 degrees) **Note :** This article is valid only for the model provided with index table.
5	Enter the 5th axis coordinate of the machining end position in the machine coordinates system. Permissible input range (0 to ±99999.999) Permissible input range for the rotational axis (0 to ±999.999 degrees)

4-10 Special Mode Unit

This unit serves to control a special movement other than machining.

4-10-1 Procedure for calling up the special mode unit

(1) Press the menu selector key (key located at the right of the menu keys). The following menu will be displayed.

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

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

➔ The following menu will be displayed.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

(3) Press the menu key for the intended special mode unit.

4-10-2 M code unit

This is for making the machine execute a special movement other than machining.

The respective movement (operations) are numbered. For details, refer to APPENDIX-1, “LIST OF M CODES.”

1. Menu selection

Press the menu key **M CODE**.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

2. Data setting in M code unit

UNo.	UNIT	No.	M1	M2	M3	M4	M5	M6	M7	M8
	M-CODE	☐	☐	☐	☐	☐	☐	☐	☐	☐

Cursor position	Description
No.	Specify the priority machining number (prior machining, subsequent machining). There are the following three entry methods: (1) Prior machining number: To be entered by means of numeric keys. Permissible input range (0 to 99) (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) MACHINING PRIORITY No.? ( ) DELAY PRIORITY | PRI. No. CHANGE | PRI. No. ASSIGN | PRI. No. ALL ERAS | SUB PROG PROC END (3) Without entry: ordinary machining For details, refer to Chapter 5, “PRIORITY FUNCTION FOR THE SAME TOOL.”
M1 to M8	Specify the desired operation during the machining of a workpiece. Pressing the menu key >>> changes the menu **a** → **b** → **c** → **a** in this order. **a**: 01 OPT. STOP | 03 SPNDL FWD | 04 SPNDL REV | 05 SPNDL STOP | 07 MIST COOLANT | 08 FLOOD COOLANT | 09 OFF COOLANT | 50 AIR BLAST | >>> **b**: 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 | >>> **c**: 16 OPEN ATC CVR | 15 CLOSE ATC CVR | 11 TOOL UNCLAMP | 10 TOOL CLAMP | 53 CHIP VAC ON | 09 CHIP VAC OFF | >>> For the M codes other than those of the displayed menu, enter the number of required M code by referring to APPENDIX-1, “LIST OF M CODES” using the numeric key. For the M codes entered, firstly M1 to M4, and then M5 to M8 are output simultaneously. **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. **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.

4-10-3 Sub-program unit

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**.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

2. Data setting in sub-program unit

UNo.	UNIT	WORK No.	REPEAT			
	SUB PRO	☐	☐			
SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
1	☐	☐	☐	☐	☐	☐

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 basic 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 basic coordinates system is entered again in the sub-program, the auxiliary coordinates system and basic 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 basic coordinates system of the MAZATROL program

4-10-4 Basic coordinate shift unit (option)

To calculate the coordinates of the plane-by-plane program origins for machining multi-plane workpieces using a machine equipped with a tilt table, programming is a very time-consuming task since each plane must be indexed to the top face for measurement. However, automatic calculation of the program origin for the plane indexed to the top face is possible by assigning the amount of shift from the reference origin (WPC) to the program origin of that plane, to this basic coordinate shift unit.

1. Menu selection

Press the menu key **WPC SHIFT**.

M CODE	SUB PROGRAM	MMS	WPC SHIFT	PALLET CHANGE	INDEX	PROCESS END			

2. Data setting in basic coordinate shift unit

UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	SHIFT-A	ROT.-th
1	WPCSHIFT	☐	☐	☐	☐	☐	☐

Cursor position	Description
SHIFT-X	Enter the amount of shift from the reference origin (WPC) in the X-axial direction to the program origin. Permissible input range (0 to ±99999.999)
SHIFT-Y	Enter the amount of shift from the reference origin (WPC) in the Y-axial direction to the program origin. Permissible input range (0 to ±99999.999)
SHIFT-Z	Enter the amount of shift from the reference origin (WPC) in the Z-axial direction to the program origin. Permissible input range (0 to ±99999.999)
SHIFT-C	Specify the rotational angle for the table rotational axis (C axis) in the workpiece coordinates system.. Permissible input range (0 to ±9999.999)
SHIFT-A	Specify the rotational angle for the tilt axis (A axis) in the workpiece coordinates system Permissible input range (0 to ±9999.999)
ROT.-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. Two methods are available to enter the angle: 1. Set the angle using the numeric keys. Permissible input range (0 to ±999.999) 2. Select the **AUTO SET** menu function. ROT.-th will be auto-set according to the particular value of SHIFT-A, as shown in the table below.

SHIFT-A	Auto-set value of ROT.-th
SHIFT-A < 0	180°
0 ≤ SHIFT-A	0°

Note: An alarm will be displayed if a value is not assigned to SHIFT-A.

3. Automatic program origin coordinate calculation

Automatic calculation of the program origin for the plane indexed to the top face is possible by assigning the amount of shift from the reference origin (WPC) to the program origin of that plane, to the basic coordinate shift unit.

This makes it unnecessary to index the intended plane to the top face for measuring the coordinates of the program origin, and thus simplifies programming. The reference origin (WPC) always refers to the basic coordinates system unit that was last registered before the basic coordinate shift unit was set.

The procedure for creating a sample program that automatically calculates the coordinates of the program origin is shown below.

1. Measure the coordinates of the reference origin (WPC) and set the basic coordinates system unit. See “UNo. 1 WPC-1” in Fig. 4-23 and “Reference origin A” in Fig. 4-24.
2. Assign to the basic coordinate shift unit the amount of shift (i.e., X-, Y-, and Z-axial coordinates) from the reference origin (WPC) to the program origin of the plane to be machined. See “UNo. 2 WPCSHIFT (SHIFT-X, SHIFT-Y, SHIFT-Z)” in Fig. 4-23 and “Point B” in Fig. 4-24.
3. Specify in the basic coordinate shift unit the angle through which the table is to be rotated for indexing the intended plane to the top face, in terms of the amount of shift (A- and C-axial coordinates). See “UNo. 2 WPCSHIFT (SHIFT-A, SHIFT-C)” in Fig. 4-23 and “(b) and (c)” in Fig. 4-24.
4. Set “ROT.-th” (rotational angle) in the basic coordinate shift unit. See Item 4, Coordinate rotating function, for details.
5. The coordinates of the program origin are calculated automatically. See “Point B” in Fig. 4-24.

Note 1: After the basic coordinate shift unit has been set, operation proceeds on the coordinate system that has been determined by the above automatic calculation. The coordinates in this coordinate system are valid until the next basic coordinates system unit and/or basic coordinate shift unit has been set.

Note 2: The plane to be machined is not indexed by a basic coordinate shift unit. To perform the index operation, use the indexing unit as before.

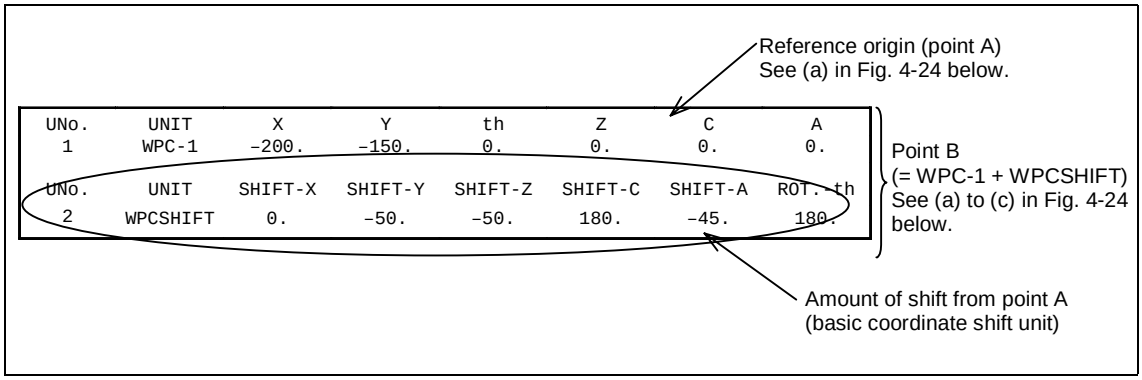


Fig. 4-23 Sample program that uses basic coordinate shift unit

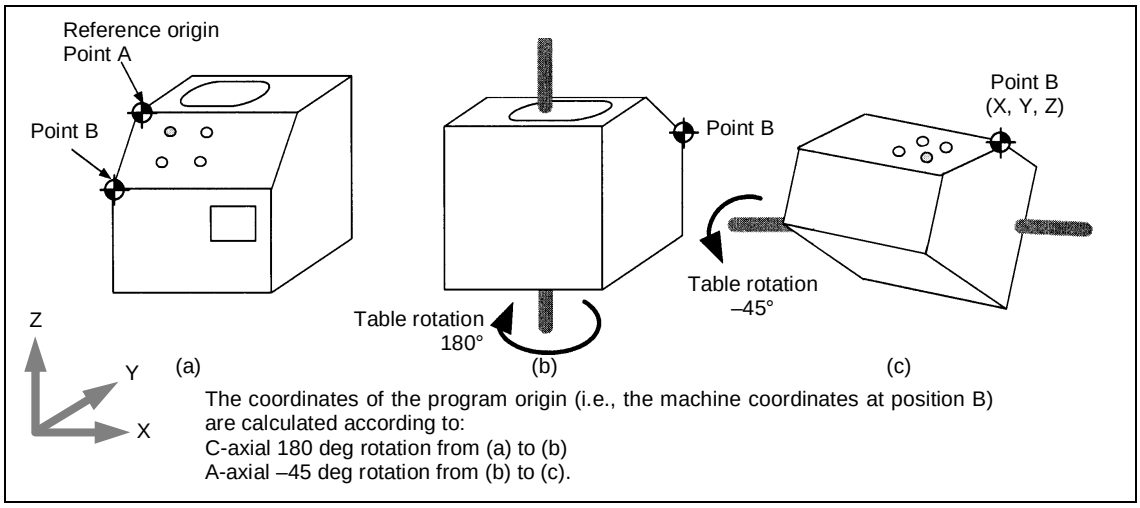


Fig. 4-24 Machining coordinate index process

4. Coordinate rotating function

When the program is created on one side, the indexing of that side to the top face will orient the program coordinate system in the direction opposite to that of the coordinate system actually used during machining. However, setting the basic coordinate shift unit (“ROT.-th”) makes the coordinate rotating function rotate the program coordinate system to ensure that this coordinate system matches to the direction of the coordinate system actually used during machining.

A sample program for performing the coordinate rotating function is shown below.

Set the basic coordinate shift (“ROT.-th”) so as to ensure that the direction of the coordinate system existing after the plane to be machined has been indexed to the top face, i.e., the direction of coordinate system F in Fig. 4-26 (b) is matched to the direction of the coordinate system actually used during machining, i.e., the direction of coordinate system E’ in Fig. 4-26 (b).

UNo.	UNIT	X	Y	th	Z	C	A
1	WPC-1	–200.	–200.	0.	0.	0.	0.

UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	SHIFT-A	ROT.-th
2	WPCSHIFT	0.	0.	–200.	180.	–90.	180.

Coordinate rotational angle:
The program coordinates system is rotated about the Z axis of the actual coordinate system with the program origin as its center.

Fig. 4-25 Basic coordinate shift unit

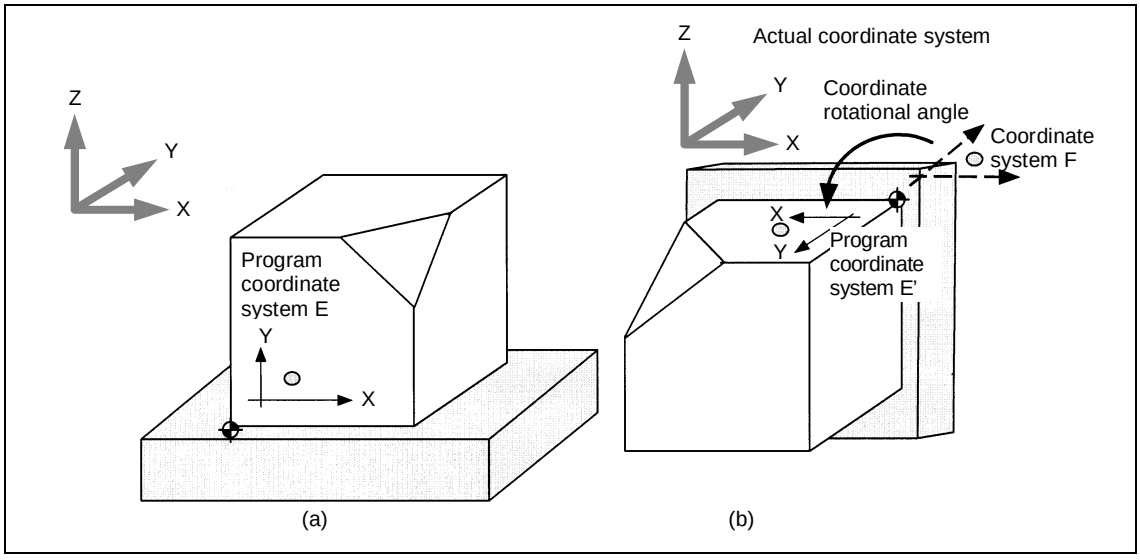


Fig. 4-26 Coordinate rotational view

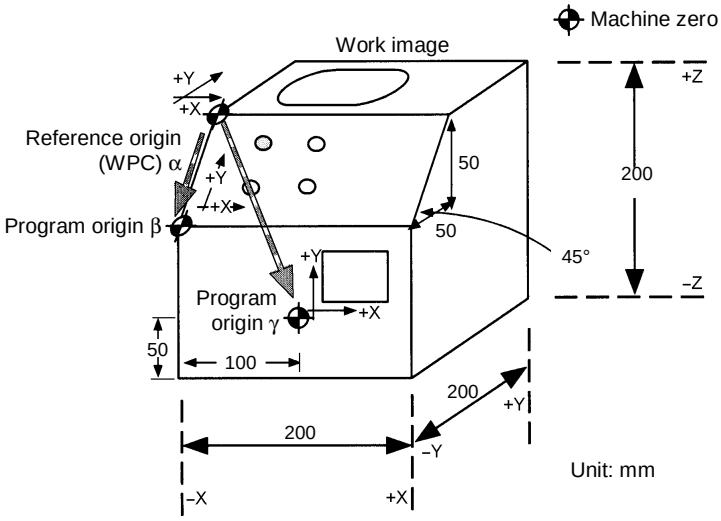
Coordinate system E in Fig. 4-26 (a) is the program coordinate system for one side.

When this side is indexed to the top face, the program coordinate system will change to the state shown in Fig. 4-26 (b), and if the plane is machined under this state, the machining operation will take place in the same coordinate system F as the actual coordinate system.

Coordinate system E’ obtained by rotating the above coordinate system through 180 degrees can be used as the program coordinate system by specifying the coordinate rotational angle value shown in Fig. 4-25 above.

5. Sample program

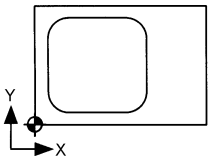
The following shows a sample machining program that uses the basic coordinate shift unit:



UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	S45C	50.	0	OFF	◆	◆	◆

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	TURN DIR.C	ANGLE A
1	INDEX	0.	0.	0.	0.	0.

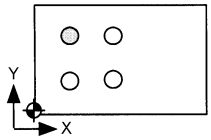
UNo.	UNIT	ADD.WPC	X	Y	th	Z	C	A	
2	WPC-1		–200.123	–165.771	0.	–30.555	0.	0.	Reference origin α



UNo.	UNIT	DEPTH	SRV-Z		BTM	WAL	FIN-Z	FIN-R	INTER-R	CHMF
3	POCKET	5.	5.		1	1	0.	0.	9.	0.
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	WID-R	C-SP	FR	M	M
R1	E-MILL	25.	1			CW	G01	5.	17.5	58	1.		
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	CN1	CN2	CN3	CN4				
1	SQR	10.	10.	150.	150.	R5.	R5.	R5.	R5.				

UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	SHIFT-A	ROT.-th	
4	WPCSHIFT	0.	–50.	–50.	180.	–45.	180.	Amount of shift (α → program origin β)

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	TURN DIR.C	ANGLE A
5	INDEX	0.	0.	0.	180.	–45.

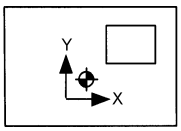


UNo.	UNIT	DIA		DEPTH	CHMF									
6	DRILLING	10.		20.	0.									
SNo.	TOOL	NOM-ϕ	No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M	
1	CTR-DR	10.		10.	◆	◆	◆	90°	CTR-D	20	0.			
2	DRILL	10.		10.	20.	0.	100	DRILL	T9.	25	0.132			
FIG	PTN	Z	X	Y	AN1	AN2	T1	T2	F	M	N	P	Q	R
1	SQR	0.	50.	10.	0.	90.	50.	50.	0	2	2	0	0	0

UNo.	UNIT	SHIFT-X	SHIFT-Y	SHIFT-Z	SHIFT-C	SHIFT-A	ROT.-th	
7	WPCSHIFT	100.	–50.	–150.	180.	–90.	180.	Amount of shift (α → program origin γ)

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	TURN DIR.C	ANGLE A
8	INDEX	0.	0.	0.	–90.	180.

UNo.	UNIT	DEPTH	SRV-Z	SRV-R	RGH		FIN-Z	FIN-R						
9	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	E-MILL	25.	1			◆	G01	5.	◆		58	1.		
FIG	PTN	X		Y		R/th	I	J	P		CNR			
1	LINE	0.		20.										
2	LINE	90.		20.										
3	LINE	90.		90.										
4	LINE	0.		90.										
5	LINE	0.		20.										



UNo.	UNIT	CONT.	NUMBER	ATC	X	Y	Z	C	A
10	END	0	0	0	0.	0.	0.	0.	0.

6. Precautions

When using the basic coordinate shifting function, observe the following four precautions:

1. When the basic coordinates system unit (WPC) is not assigned, the amount of shift from the machine zero point will be used.
2. If the basic coordinate shift unit alone is specified in the MAZATROL subprogram, the basic 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 basic coordinates system unit in the subprogram as well.
3. In an EIA/ISO subprogram, the rotational angle value “ROT.-th” on the basic coordinate shift unit specified in the main program is invalid, as with “th” of the basic 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.
4. The correct program zero point cannot be calculated if basic coordinates-A (WPC, A) and basic coordinate shift-A (WPCSHIFT, A) are both assigned a value other than “0”. (The correct program zero point cannot be calculated since the data in basic coordinates-A will be allowed for during the “automatic program origin coordinate calculation” process of the basic coordinate shifting function.)
   There is no problem, however, if “0” is assigned to basic coordinates-A and any other value is assigned to basic coordinate shift-A so as to shift the A-axis using the shifting function or if a value other than “0” is assigned to basic coordinates-A and a value is not assigned to basic coordinate shift-A so as not to shift the A-axis.

4-10-5 Pallet changing unit

This is the unit for changing the pallet.

1. Menu selection

Press the menu key **PALLET CHANGE**.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

2. Data setting in pallet changing unit

UNo.	UNIT	PALLET No.
	PALL CHG	□ (□)

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.

4-10-6 Indexing unit

This unit serves to control the angle of the surface to be machined on the machine with an indexing table, an NC (numerically controlled) rotating table or a tilting table.

When the function of priority for the same tool is used on each surface, use this unit for controlling the angle.

1. Menu selection

Press the menu key **INDEX**.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

2. Data setting in indexing unit

UNo.	UNIT	TURN POS X	TURN POS Y	TURN POS Z	ANGLE *Angle 1*	TURN DIR. *Angle 2*
	INDEX	□	□	□	□	□

Note: Instead of “Angle 1” or “Angle 2”, the axis name is indicated on the NC display.

Cursor position	Description
TURN POS X	Enter the X coordinate of the table rotation 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 Y	Enter the Y coordinate of the table rotation 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 table rotation 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)
ANGLE	Enter the angle of the indexing table by means of numeric keys. **Note 1:** In the case of the indexing table by B code or by M code, the entering of a value which cannot be divided by the minimum indexing angle of the table (parameter **L37**) will cause the alarm **640 ILLEGAL INDEX ANGLE INPUT** to be displayed. **Note 2:** In the case of the NC rotating table, enter the angle as an absolute value on the basis of the workpiece coordinates system.
Angle 1	Enter the indexing angle of the 4th axis. Without this entry, axis movement does not take place. Two methods are available to enter the angle: 1. Set the angle using the numeric keys. Permissible input range (0 to ±999.999) 2. When the **AUTO SET** menu key is pressed, the value of SHIFT-C in the basic coordinate shift unit located at the immediately previous position will be assigned. **Note 1:** Enter the angle as an absolute value on the basis of the workpiece coordinates system. **Note 2:** Alarm **414 AUTO CALCULATION IMPOSSIBLE** will occur if the unit located at the immediately previous position is not the basic coordinate shift unit or if data is missing in SHIFT-C of the basic coordinate shift unit.
TURN DIR.	Select the table rotation direction from the menu only for the indexing table which can pivot in both directions by the B code method. SHORTEST (AUTO) | CW | CCW **Note:** The non-entry of data will cause the alarm **641 MISSING INPUT DATA** to be displayed.
Angle 2	Enter the indexing angle of the 5th axis. Without this entry, axis movement does not take place. Two methods are available to enter the angle: 1. Set the angle using the numeric keys. Permissible input range (0 to ±999.999) 2. When the **AUTO SET** menu key is pressed, the value of SHIFT-A in the basic coordinate shift unit located at the immediately previous position will be assigned. **Note 1:** Enter the angle as an absolute value on the basis of the workpiece coordinates system. **Note 2:** Alarm **414 AUTO CALCULATION IMPOSSIBLE** will occur if the unit located at the immediately previous position is not the basic coordinate shift unit or if data is missing in SHIFT-A of the basic coordinate shift unit.

Note 1: If the machining unit, which will use the tool, or the end unit is to follow, then indexing unit will be executed as appropriate according to the particular operation of that unit.
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.

Note 2: In the case of the M code indexing table, the execution of the indexing unit outputs the M code (entered by the parameter **L38**) as many times as necessary until the table comes to the specified position. The number of outputs is determined by the minimum indexing angle (entered by the parameter **L37**).

Note 3: In the case of the M code indexing table in which one indexing movement requires more than one M code or for which the program is executed step by step by M code coming from the Numerical Control through a programmable controller located on the indexing side, it is impossible to control the table by this indexing unit. In this case, program with the M code unit.

Note 4: In order to change the pallet on a machine equipped with an indexing table, enter the indexing unit before the pallet changing unit and specify the indexing angle which permits changing of the pallet.

Example:

```
UNo.   UNIT  ..........    ANGLE
       INDEX                 0
UNo.   UNIT              PALLET No.
       PALL CHG              1
```

Enter the value 0 if the changing of the pallet is possible, the indexing table being at 0^{o}.

Note 5: Since rotary direction is unavailable for the machine with an NC rotating table or a tilting table, set 0 at parameter **L38**.

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 indexing unit.

Note 2: With parameter **L41** = 1, the entering of data is only possible in the article TURN POS X of the indexing unit.
The entering of data for the Y axis and the Z axis will cause an alarm.

4-10-7 Process end unit

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		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 special mode unit 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
                CTR-DR
                   ⋮
   UNo.          UNIT
               PROC END
   UNo.          UNIT
               DRILLING
   SNo.          TOOL
                CTR-DR
                   ⋮
   UNo.          UNIT   <- When this line is to be the lower line of the useful scope,
                           place the cursor on the next line.
```

(6) Insert the **PROCESS END** unit by the same operaiton as (2) to (4).

➔ The lower limit of the useful scope is set as follows.

```
UNo.      UNIT
        DRILLING
SNo.      TOOL
         CTR-DR
            ⋮
            ⋮
UNo.      UNIT
        PROC END
UNo.      UNIT
        DRILLING
SNo.      TOOL
         CTR-DR
            ⋮
            ⋮
UNo.      UNIT
        PROC END
```

Scope of priority number for the same tool

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 FUNCITON FOR THE SAME TOOL.”

4-11 Manual Program Mode Unit

This unit serves for programming in detail each of the movement of the machine.

4-11-1 Input procedure

Press the menu key **MANUAL PROGRAM**.

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

4-11-2 Structure of the unit

1. Data setting in manual program mode unit

UNo.	UNIT	TOOL	NOM-ϕ	No.
	MANU PRO	☐	☐	☐

Cursor position	Description
TOOL	Select menu key corresponding to the tool used. Pressing the menu key >>> changes the menu **a** → **b** → **a** in this order. **a**: ENDMILL | FACEMILL | CHAMFER CUTTER | BALL ENDMILL | OTHER TOOL | TOUCH SENSOR | | | >>> | **b**: CENTER DRILL | DRILL | BACKSPOT FACER | REAMER | TAP | BORING BAR | BACK BOR.BAR | CHIP VACUUM | >>> | **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: 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**, **FACE MILL**, **CHAMFER CUTTER** and **BALL ENDMILL** is selected in the article TOOL, the alarm **434 NO ASSIGNED TOOL IN TOOL FILE** is displayed if the selected tool has not been previously recorded in the **TOOL FILE** display.
No.	Determine the priority machining number (prior machining, subsequent machining). The input of data is done in accordance with the following 3 methods; - Priority No. for prior machining: Enter the number by means of numeric keys. Input range (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) MACHINING PRIORITY No.? ( ) DELAY PRIORITY | PRI. No. CHANGE | PRI. No. ASSIGN | PRI. No. ALL ERAS | SUB PROG PROC END | - Without input: Normal machining. The machining order is not specified. For details, refer to Chapter 5, “PRIORITY FUNCTION FOR THE SAME TOOL.”

4-11-3 Composition of the sequence

In the manual program mode unit, one sequence data line corresponds to one EIA/ISO program block.

For the details of each of the commands, refer to the Programming Manual for EIA/ISO Programming.

1. Setting sequence data

SNo.	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
1	☐	☐	☐	☐	☐	☐	☐	☐	☐	☐

Cursor position	Description
G1, 2	Enter the G codes (preparetory function) by means of menu keys or numeric keys. G00 | G01 | G02 | G03 | G40 CANCEL | G41 LEFT | G42 RIGHT | MANUAL END | | **Note:** Up to two G code commands can be designated in one sequence data line.
DATA 1 to DATA 6	Select the address of the data to be entered from the following menus. Pressing the menu key >>> changes the menu **a** → **b** → **a** in this order. X | Y | Z | 4 | F | R | 5 | | >>> | **a** I | J | K | P | D | | | | >>> | **b** After the address has been set, enter and set data using the numeric keys.
S	Enter the rotational speed of the spindle by means of numeric keys. Input range (0 to 99999)
M/B	Enter the M code (auxiliary function) or the B code (second auxiliary function, 3 digits) by means of numeric keys. For B code setting, press the menu key **B CODE INPUT** and enter a numeric value. For M code, refer to APPENDIX-1, “LIST OF M CODES.” If the sequence data contains G65 in “G1” or “G2” item, the data set here will be handled as an argument.

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 article ACT-ϕ on the **TOOL DATA** display (on the nominal diameter in case of the data absence) when the tool diameter correction command (G41 or G42) is given.
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.
7. For the manual program unit, specify address U to control the C-axis of turning spindle 2.

4-12 MMS Unit

The MMS unit automatically compensates the workpiece coordinates system by performing the centering measurement in automatic operation mode.

The measuring movement is performed by specifying the feeler (touch sensor) as a tool and the type of measurement.

4-12-1 Procedure for calling up the MMS unit

(1) Press the menu selector key (key located at the right of the menu keys). The following menu will be displayed.

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

(2) Press the menu key **OTHER**. The following menu will be displayed.

M CODE	SUB PROGRAM	MMS		PALLET CHANGE	INDEX	PROCESS END			

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

4-12-2 Composition of the unit

1. Data setting in MMS unit

UNo.	UNIT	TOOL	NOM-ϕ	U. SKIP	$
	MMS	T. SENS	☐	☐	

Cursor position	Description
NOM-ϕ	Specify the nominal diameter of the feeler. Enters the aproximate diameter of the point of feeler by means of numeric keys.
U. SKIP	Specify whether the MMS unit is executed or not. Execution ..... 0 ⏎ Non-execution ..... 1 ⏎

4-12-3 Composition of the MMS sequence

1. Setting MMS sequence data

SNo.	PTN	X	Y	Z	*4*	*5*	R	D/L	K
1	☐	☐	☐	☐	☐	☐	☐	☐	☐

Cursor position	Description
PTN	Select the type of measurement from the following menus. Pressing the menu key >>> changes the menu **a** → **b** → **a** in this order. **a**: X FACE | Y FACE | Z FACE | X GRV | Y GRV | X STP | Y STP | PTN END | >>>| **b**: X-Y BORE | X-Y BOSS | X-Y-th CNR | | | | CALIBR. | PTN END | >>>| For details, refer to Subsection 4-12-6, “Type of measurement.”
X, Y, Z	Specify the measurment starting position by means of numeric keys.
4	Specify the positioning coordinate of the 4th axis for measurement movement 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. **Note:** 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. **Note:** The contents of the entry differ according to the type of measurement. For details, see Subsection 4-12-6, “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.

Note 1: The measurement movement is executed after the movement, if any, on the 4th or 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 basic coordinates system until the completion of all the types of measurement specified in the unit.

Example:

[1]
[2]
[3]

UNo.	UNIT	X	Y	th	Z	4
1	WPC-0	-300.	-400.	0.	-300.	0.

UNo.	UNIT	TOOL	NOM-ϕ No.	U. SKIP	$
2	MMS	T. SENS	5.	0.	

SNo.	PTN	X	Y	Z	4	R
1	X-FACE	-20.	0.	-10.	0.	0.
2	Y-FACE	0.	-20.	-10.	0.	0.

UNo.	UNIT	DIA	DEPTH	CHMF
3	DRILLING	10.	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 basic coordinates unit [1].

From unit [3], the machining is performed according to the new system of basic coordinates (program origin) corrected in the MMS unit [2].

But the data of the basic 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 basic 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 article th of the basic 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.

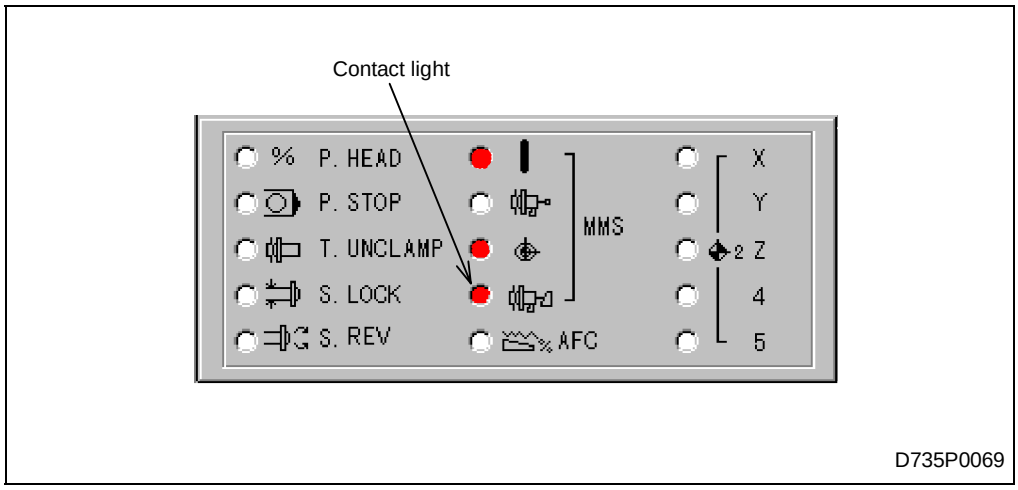
4-12-4 Measurement of length of feeler

It is necessary to measure the length of the feeler in order to enter the data into the article 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.
In the case of horizontal type machine an edge locator is used for setting the reference surface.

(3) In manual handle speed mode, slowly move the Z axis until the contact light (MMS) in the CNC signal window goes on.



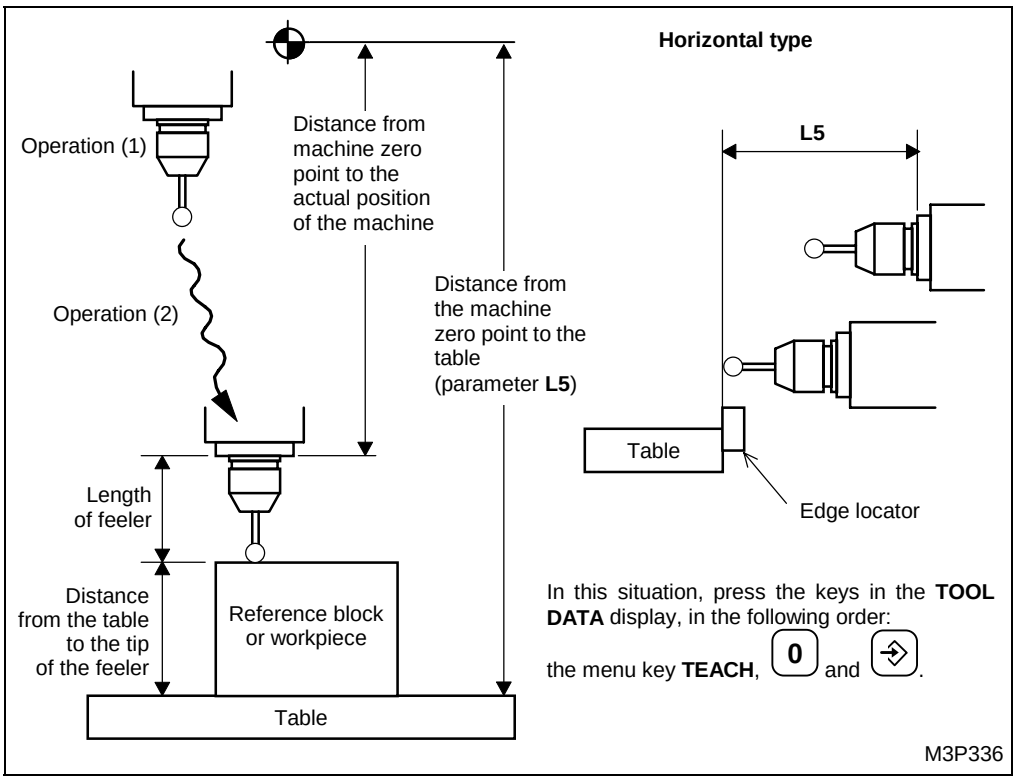
(4) Call up the **TOOL DATA** display.

(5) Press the menu key **TEACH**.

➔ The cursor appears in the article 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 article 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.

4-12-5 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.

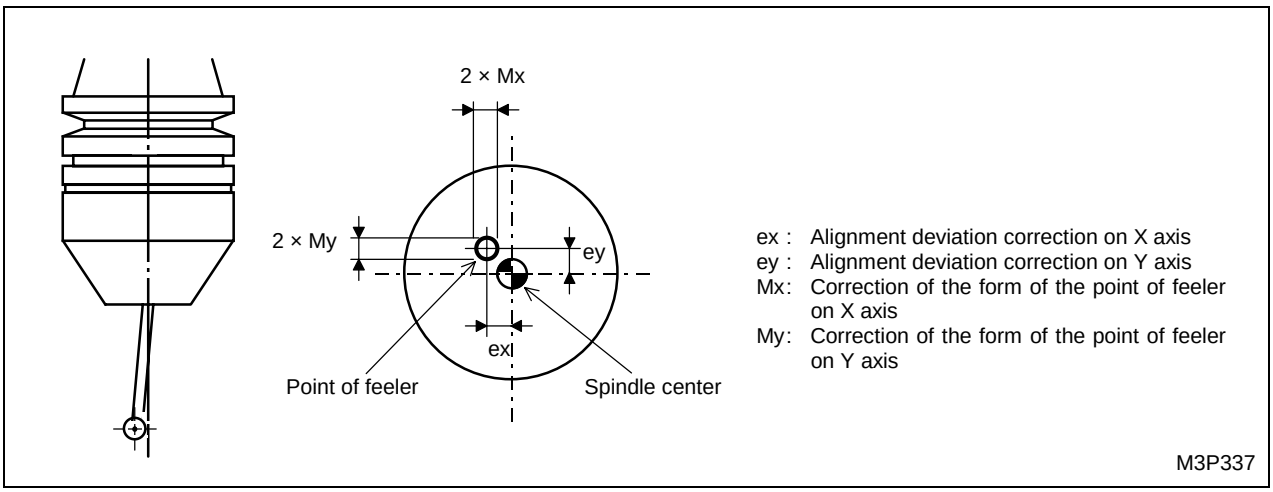
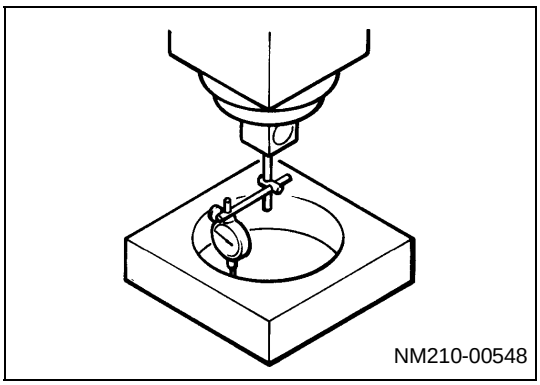


Fig. 4-27 Correction of calibration measurement

(1) Mount a reference block having a reference hole (ϕ50 to ϕ100).

(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.

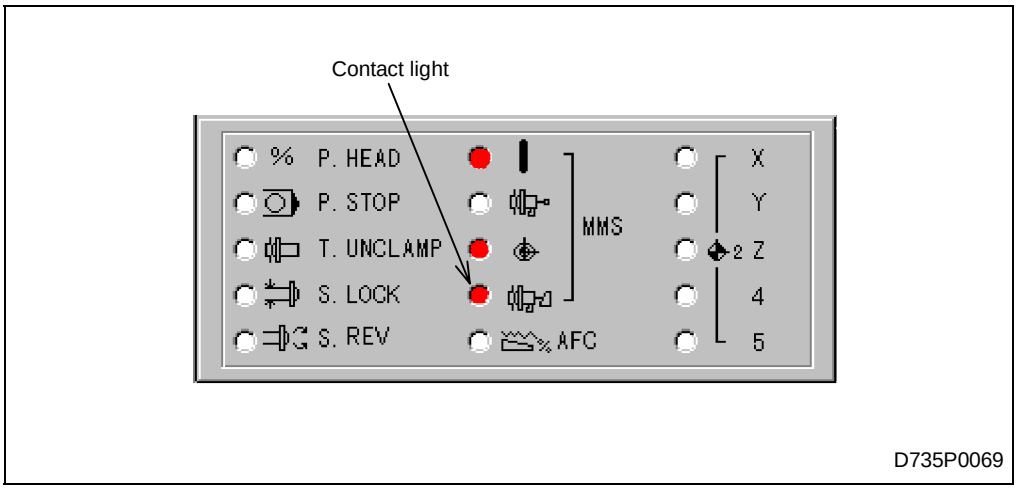
- The same as for the general measurement program, press the menu key **CALIBR.**, then prepare the program.

(5) Enter the X, Y coordinates of the spindle axis in the X and Y articles 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.    MAT              INITIAL-Z   ATC MODE   MULTI-MODE   MULTI-FLAG   PITCH-X PITCH-Y
 0      FC                  50.         0          OFF
UNo.    UNIT    ADD. WPC     X          Y          th           Z            4
 1      WPC-0      ◆        [1]        [1]         0           [2]          0
UNo.    UNIT      TOOL      NOM-ϕ       No.       U.SKIP     $
 2      MMS     T. SENS       5
SNo.    PTN        X     Y     Z        4     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 (Basic 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 (Basic 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 (Basic 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.

4-12-6 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:

X FACE	Y FACE	Z FACE	X GRV	Y GRV	X STP	Y STP	PTN END	>>>	
X-Y BORE	X-Y BOSS	X-Y-th CNR				CALIBR.	PTN END	>>>	

1. Measurement of the reference surface

The measurement of the reference surface is of three types: X, Y and Z.

- X reference face (X-FACE)
- Y reference face (Y-FACE)
- Z reference face (Z-FACE)

A. X reference face

SNo.	PTN	X	Y	Z	*4*	*5*	R	D/L	K
1	X-FACE	x1	y1	z1	0	0	rx	◆	◆

◆: Data are not necessary to set here.

Note: Instead of “4” or “5”, the axis name is indicated on the NC display. This also applies to the other measurement patterns.

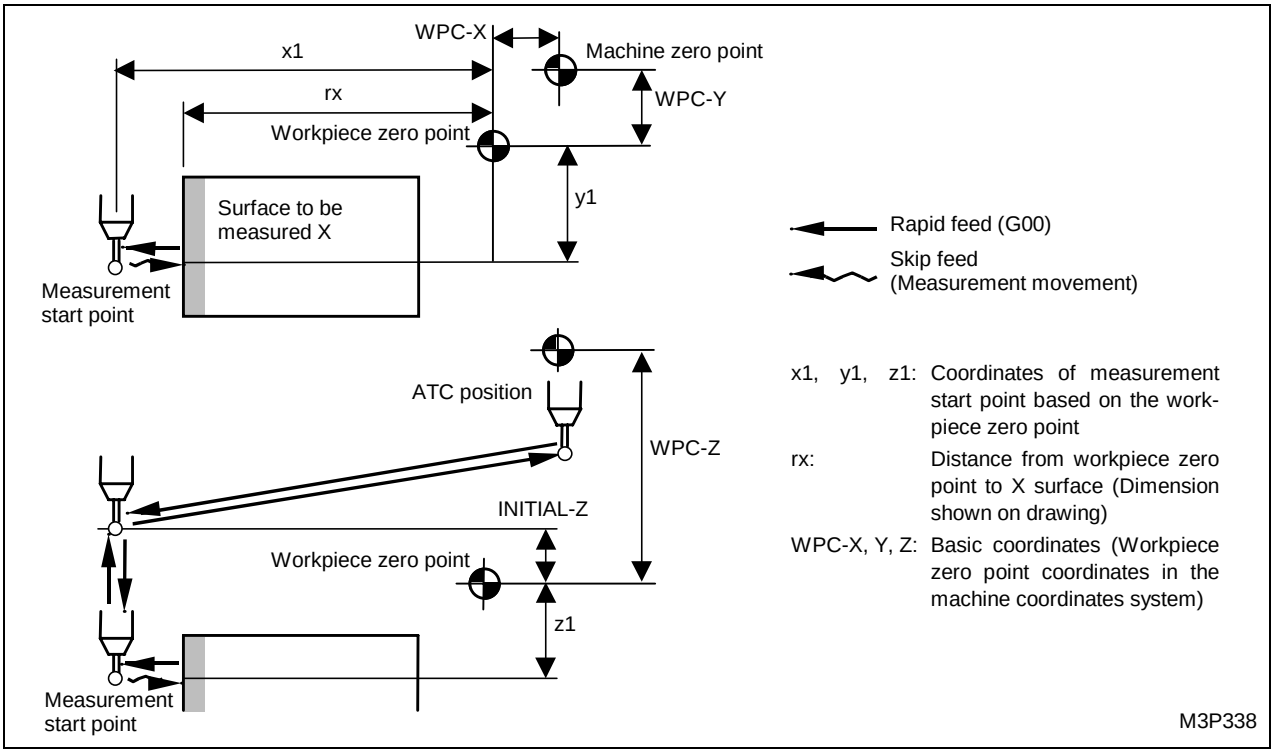
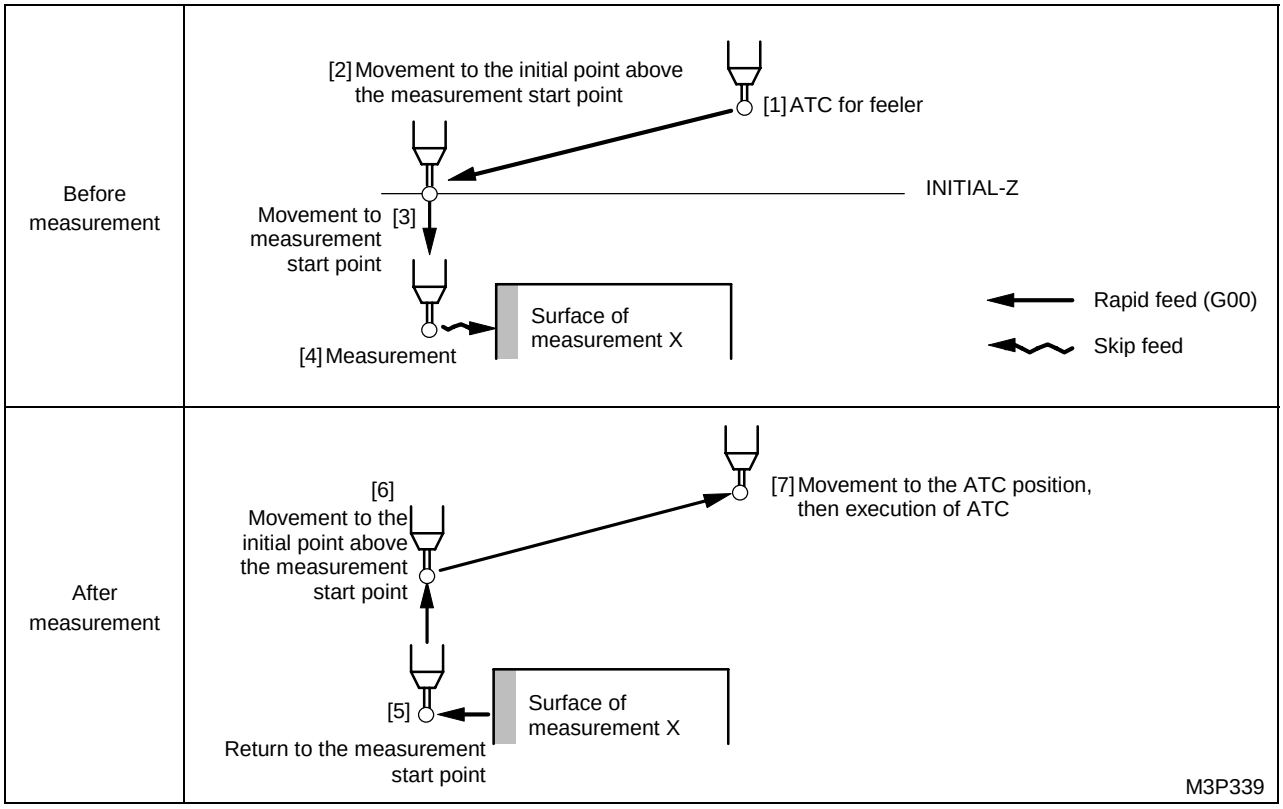


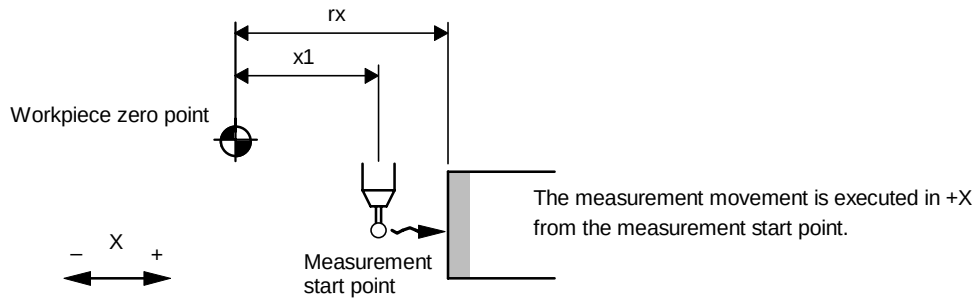
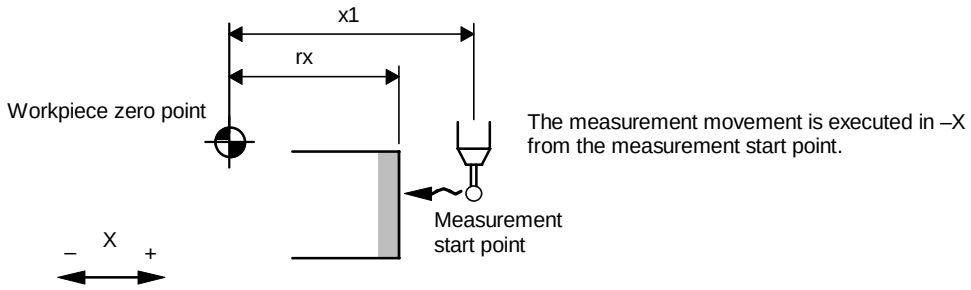
Fig. 4-28 X reference face

[Measurement movement]



Note 1: The X basic 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 x1 and rx.

x1 < rx	
x1 > rx	

M3P340

B. Y reference face

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
1	Y-FACE	x1	y1	z1	0	0	ry	◆	◆

◆: Data are not necessary to set here.

The movement is, in principle, identical to that for the X reference face. ry signifies the distance of the workpiece zero point on the Y surface (dimension shown on the drawing), in other words, the distance of the deviation of the Y basic coordinate.

C. Z reference face

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
1	Z-FACE	x1	y1	z1	0	0	rz	◆	◆

◆: Data are not necessary to set here.

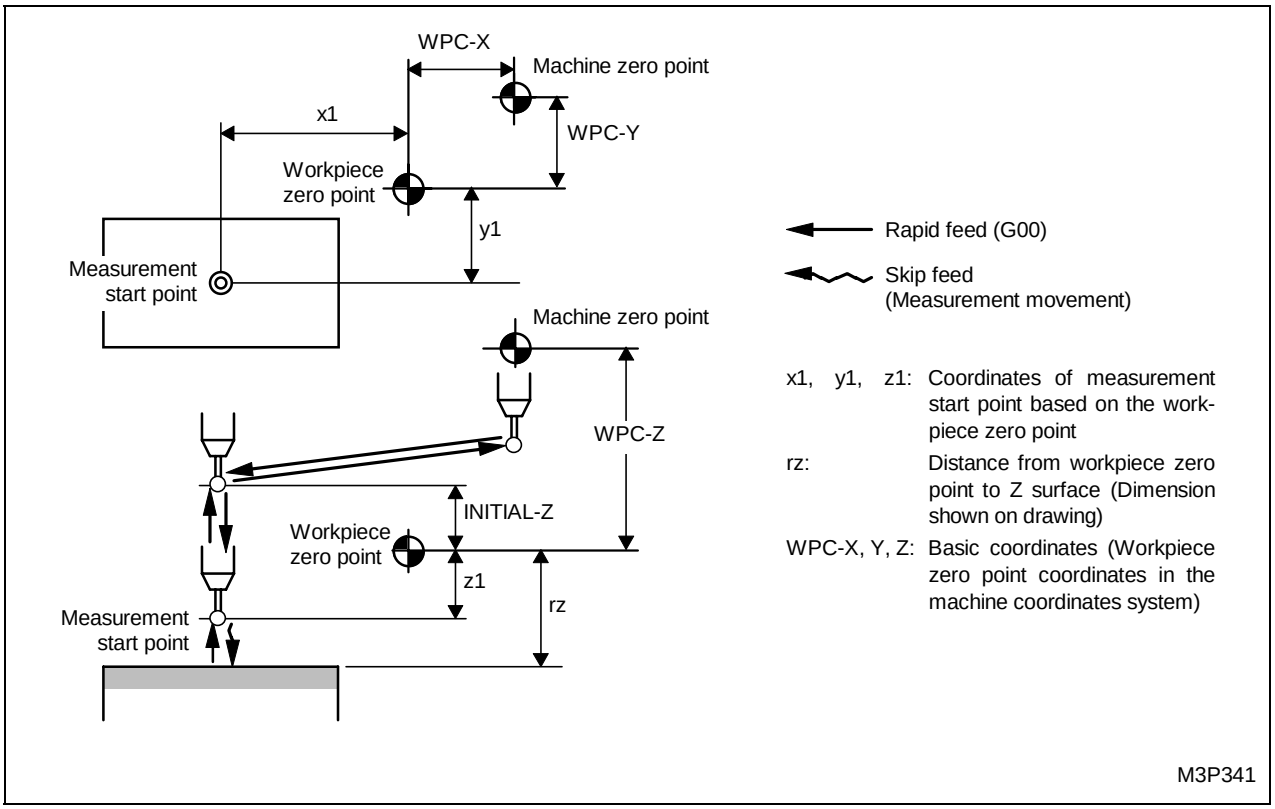
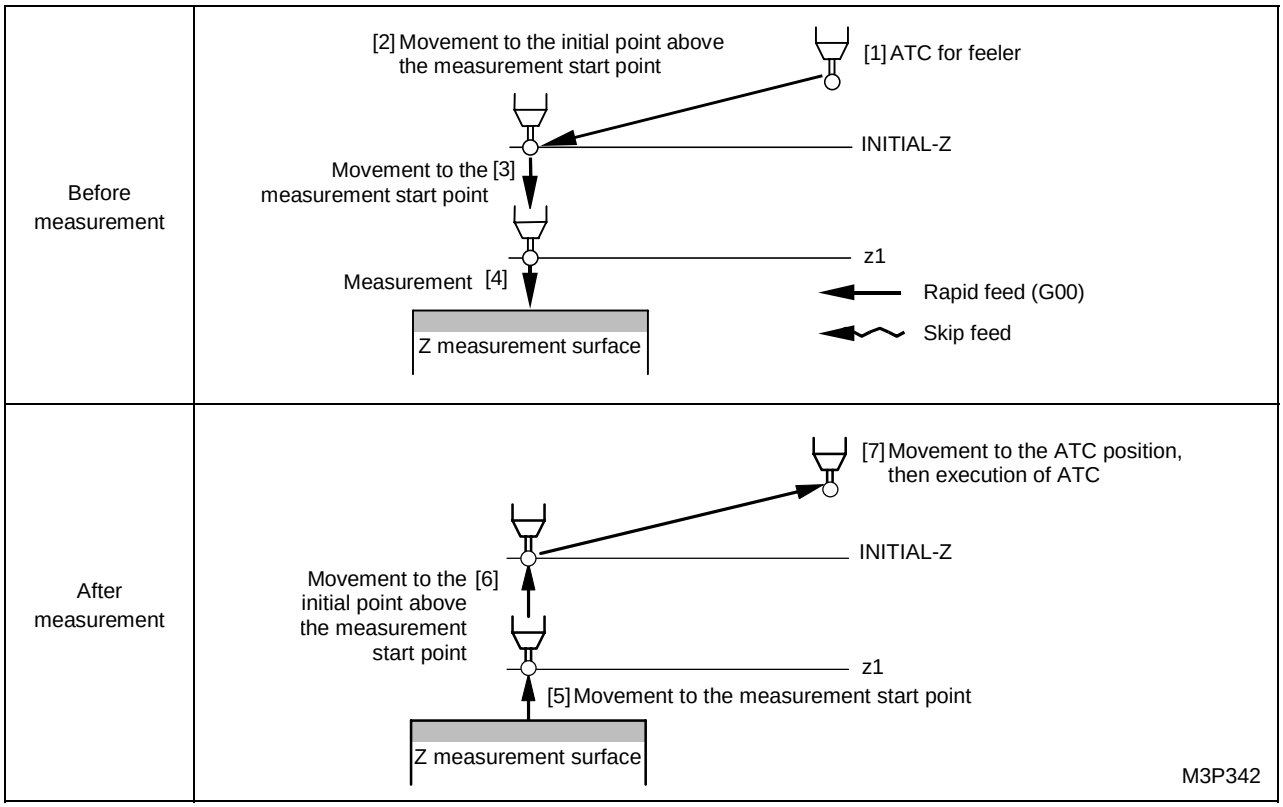


Fig. 4-29 Z reference face

[Measurement movement]



Note: The Z basic coordinate is corrected so that the Z coordinate of the measurement surface is equal to the coordinate entered in rz, when the feeler is moved to the initial point [6].

2. Measurement of groove center

The measurement from groove center is available in two types according to whether the direction of the groove width is on the X axis or the Y axis.

- Groove center-X (X-GRV)
- Groove center-Y (Y-GRV)

A. Groove center-X

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
1	X-GRV	x1	y1	z1	0	0	◆	ℓ	k

◆: Data are not necessary to set here.

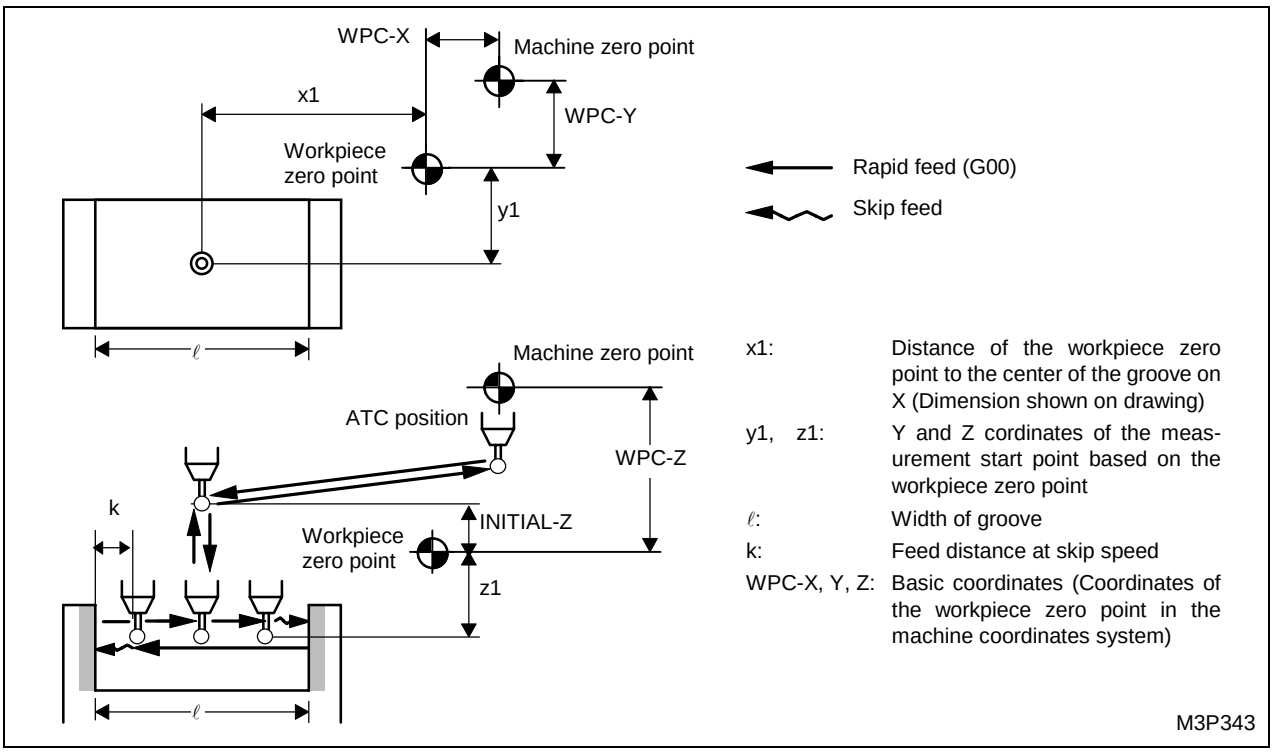
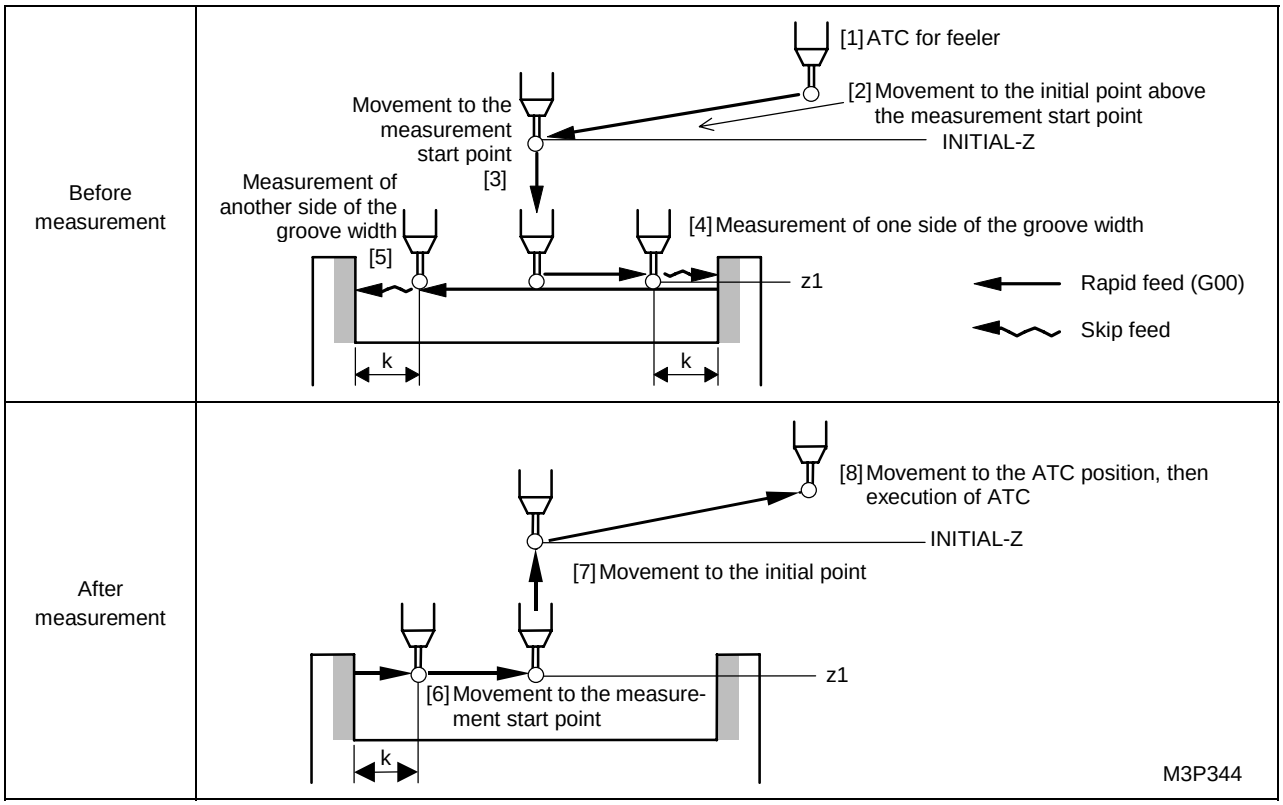


Fig. 4-30 Groove center-X

[Measurement movement]



Note: The X basic 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

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
1	Y-GRV	x1	y1	z1	0	0	◆	ℓ	k

◆: 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), in other words, the distance of the deviation of the Y basic coordinate.

3. Measurement of center of projection width

The measurement of the center of the projection width is available in two types according to whether the direction of the width of the projection is on the X axis or on the Y axis.

Projection width on X axis (X-STP)

Projection width on Y axis (Y-STP)

A. Projection width on X axis

SNo.	PTN	X	Y	Z	*4*	*5*	R	D/L	K
1	X-STP	x1	y1	z1	0	0	◆	ℓ	k

◆: Data are not necessary to set here.

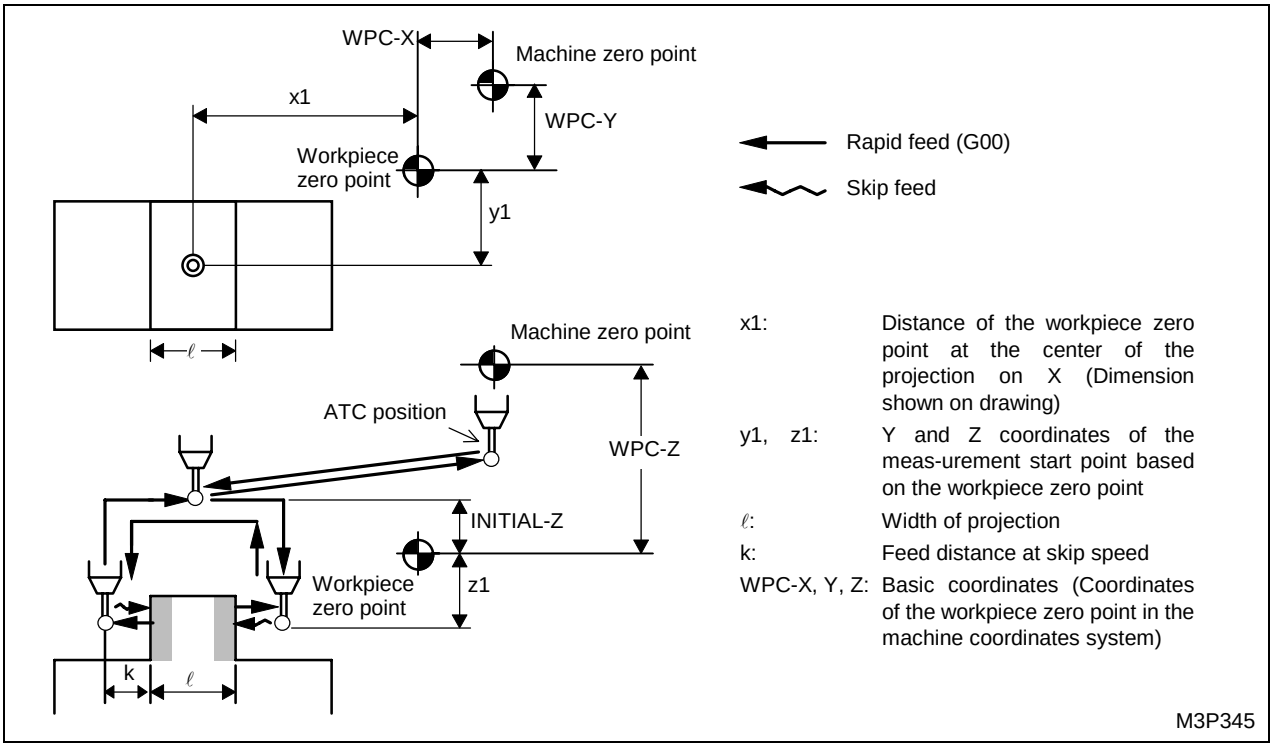
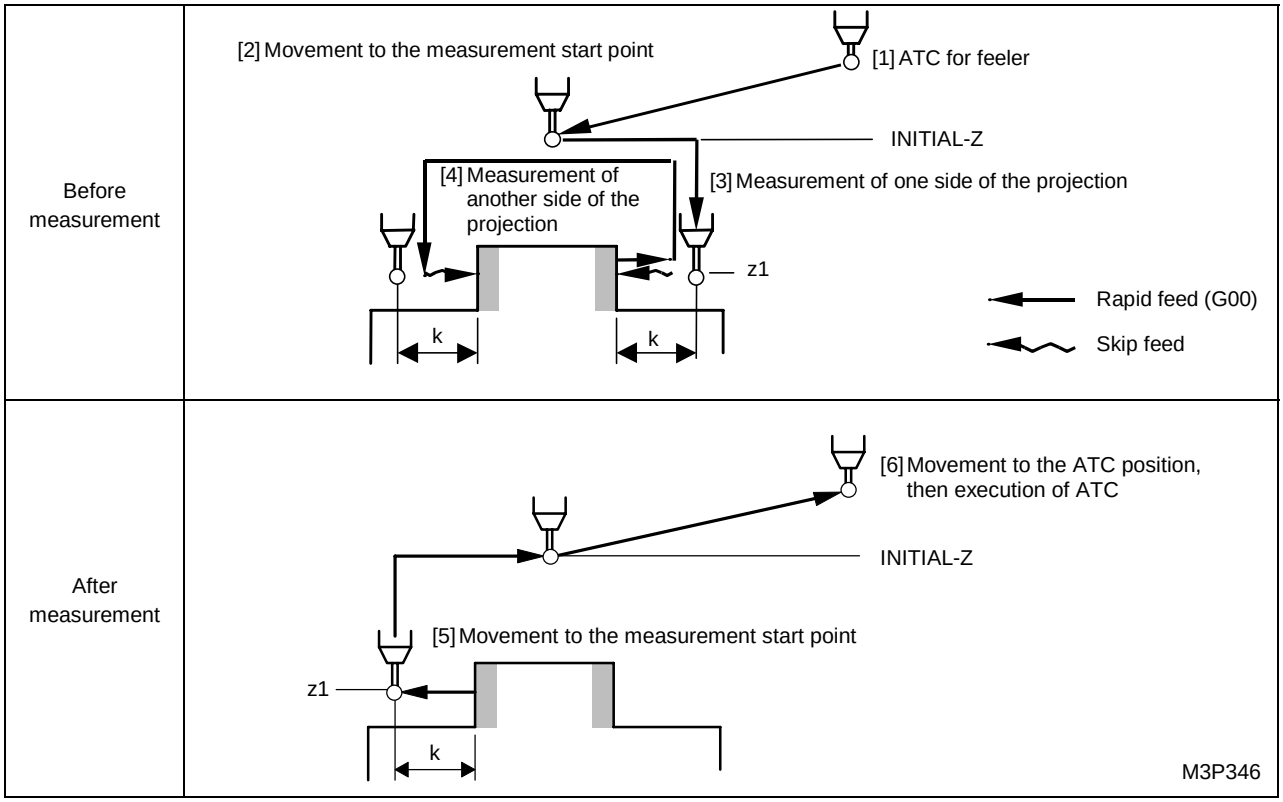


Fig. 4-31 Projection width on X axis

[Measurement movement]



Note: The X basic 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

SNo.	PTN	X	Y	Z	*4*	*5*	R	D/L	K
1	Y-STP	x1	y1	z1	0	0	◆	ℓ	k

◆: 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) which serves the distance of deviation of the Y basic coordinate.

4. Measurement of boring center (XY-HOL)

The X and Y basic coordinates are corrected by entering the coordinates of the boring center in the workpiece coordinate system and also the boring diameter.

SNo.	PTN	X	Y	Z	*4*	*5*	R	D/L	K
1	XY-HOL	x1	y1	z1	0	0	◆	d	k

◆: Data are not necessary to set here.

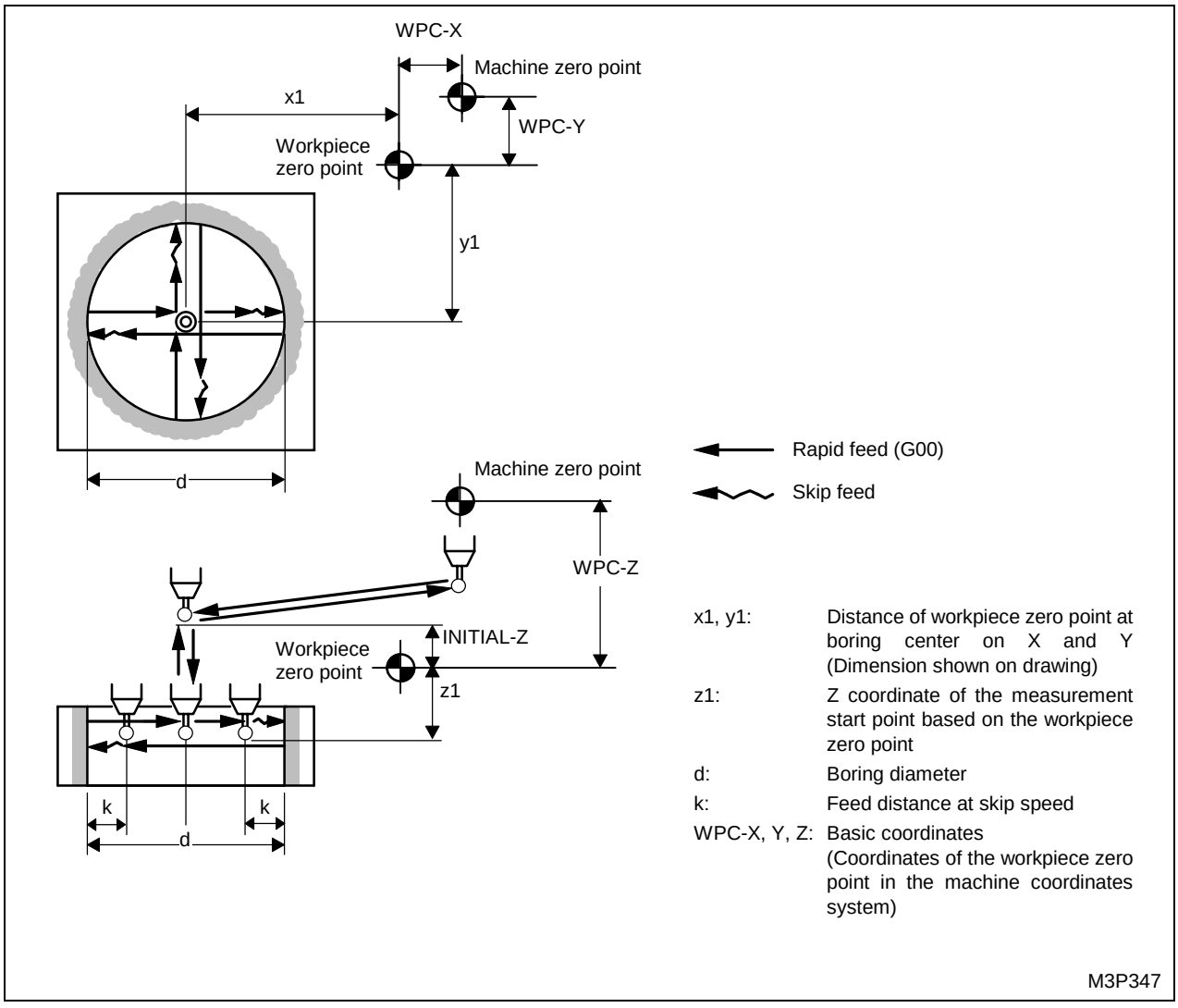
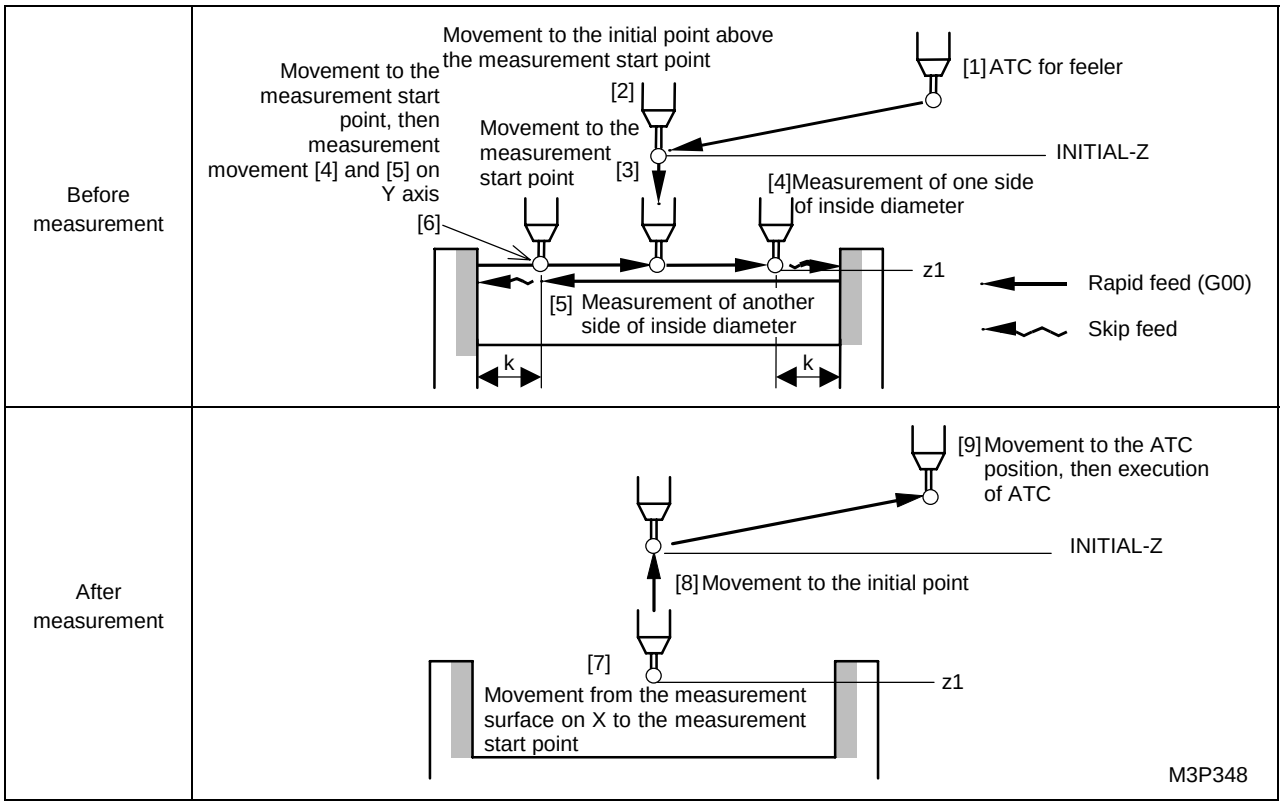


Fig. 4-32 Measurement of boring center

[Measurement movement]



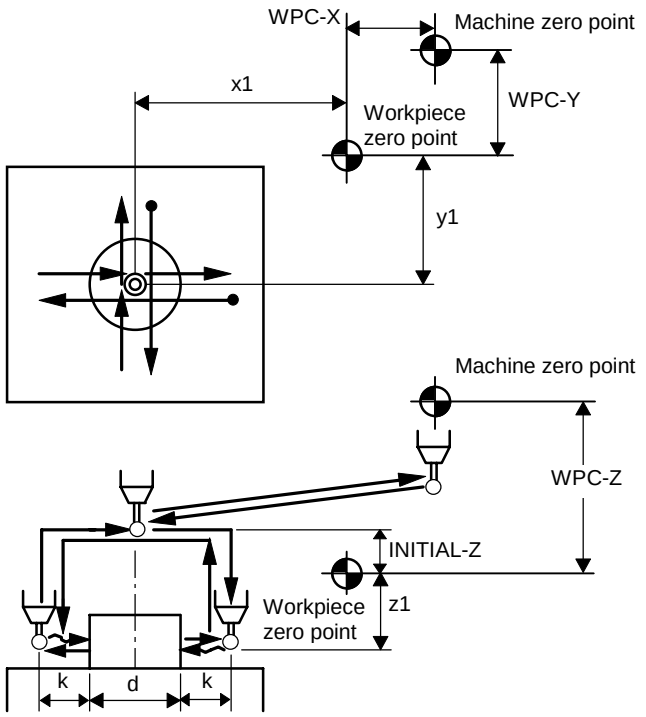
Note: The X and Y basic coordinates are corrected so that the X and Y coordinates of the boring center to be measured are equal to the coordinates x1 and y1 respectively when the feeler is moved to the initial point [8].

5. Measurement of boss center (XY-BOS)

The correction of the basic coordinates X and Y are done by entering the distance of the workpiece zero point at the boss center on X and Y and also the diameter of the boss.

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
1	XY-BOS	x1	y1	z1	0	0	◆	d	k

◆: Data are not necessary to set here.



Rapid feed (G00)

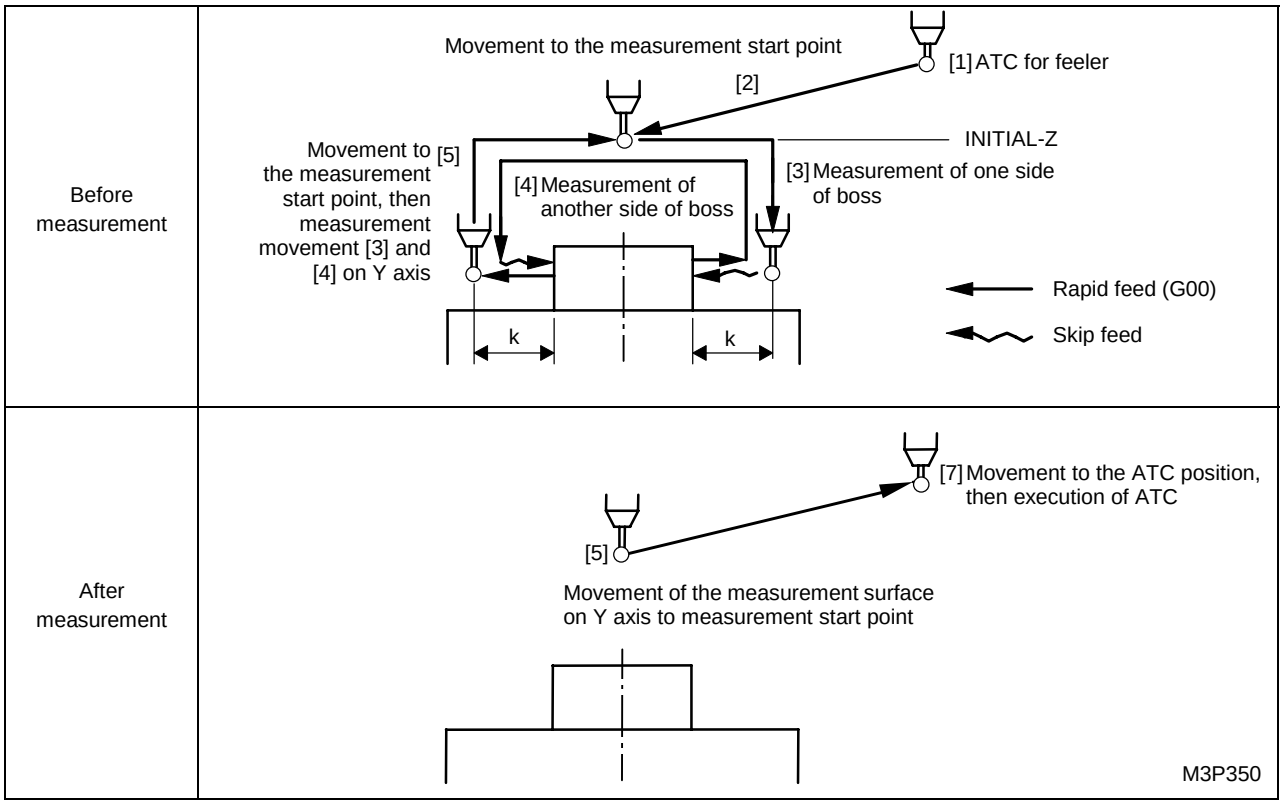
Skip feed

x1, y1:	Distance of workpiece zero point at boss center on X and Y (Dimension shown on drawing)
z1:	Z coordinate of the measurement start point based on the workpiece zero point
d:	Boss diameter
k:	Feed distance at skip speed
WPC-X, Y, Z:	Basic coordinates (Coordinates of the workpiece zero point in the machine coordinates system)

M3P349

Fig. 4-33 Mesurement of boss center

[Measurement movement]



Note: The X and Y basic coordinates are corrected so that the X and Y coordinates of the boss center to be measured are equal to the coordinates x1 and y1 respectively at the time of the movement of the feeler to the initial point [6] .

6. Measurement of workpiece inclination (XYthCNR)

The X and Y basic coordinates 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	*4*	*5*	R	D/L	K
1	XYthCNR	x1	y1	z1	0	0	r	◆	◆

◆: Data are not necessary to set here.

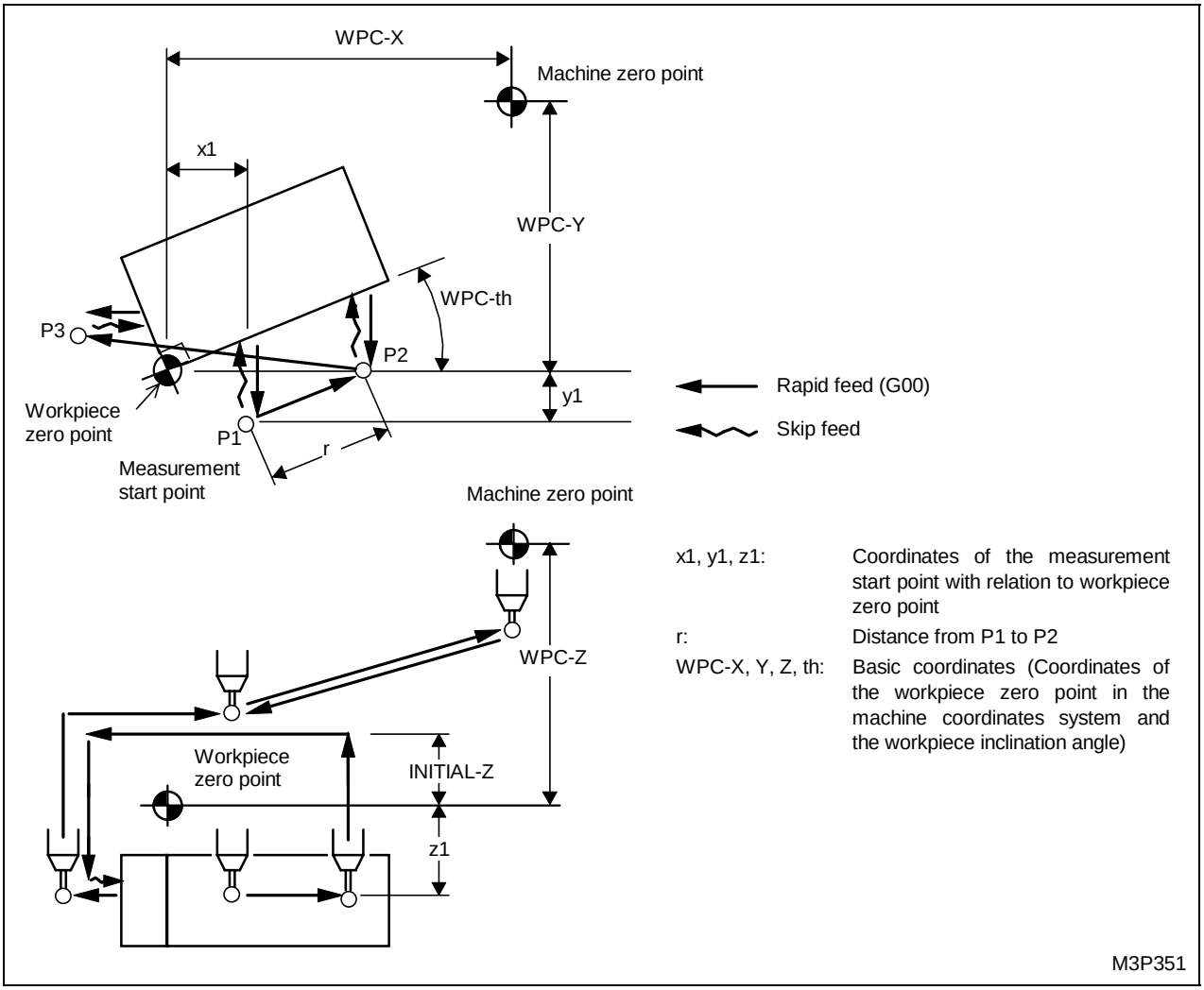


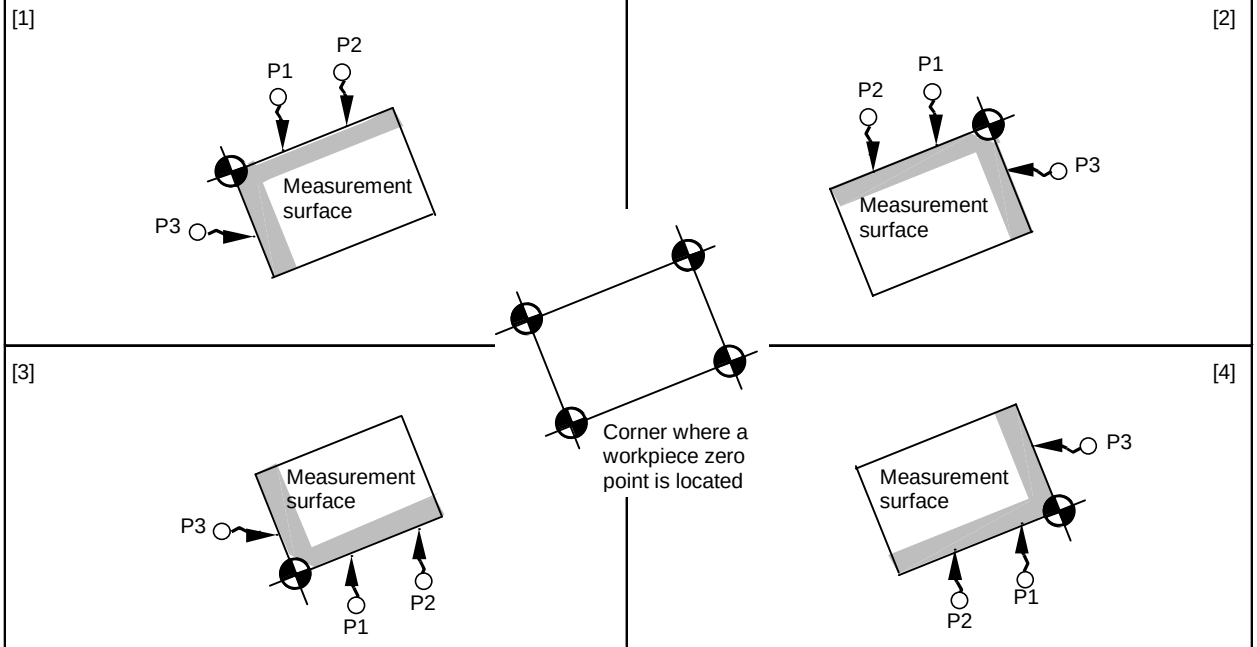
Fig. 4-34 Measurement of workpiece inclination

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.

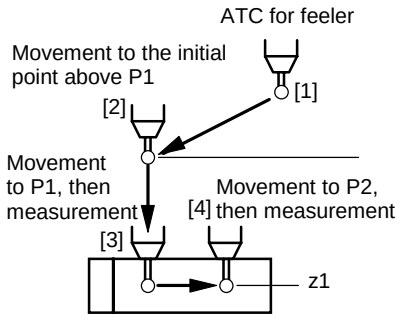
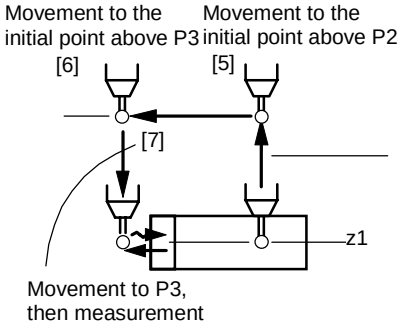
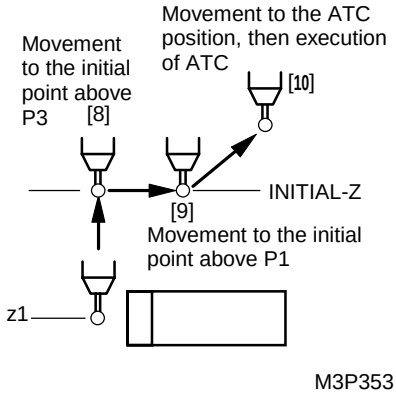
Article	X	Y	Z	4	R	X	Y	Z	4	R
Data	x1 > 0	y1 > 0	—	—	r > 0	x1 < 0	y1 > 0	—	—	r < 0



Article	X	Y	Z	4	R	X	Y	Z	4	R
Data	x1 > 0	y1 < 0	—	—	r > 0	x1 < 0	y1 < 0	—	—	r < 0

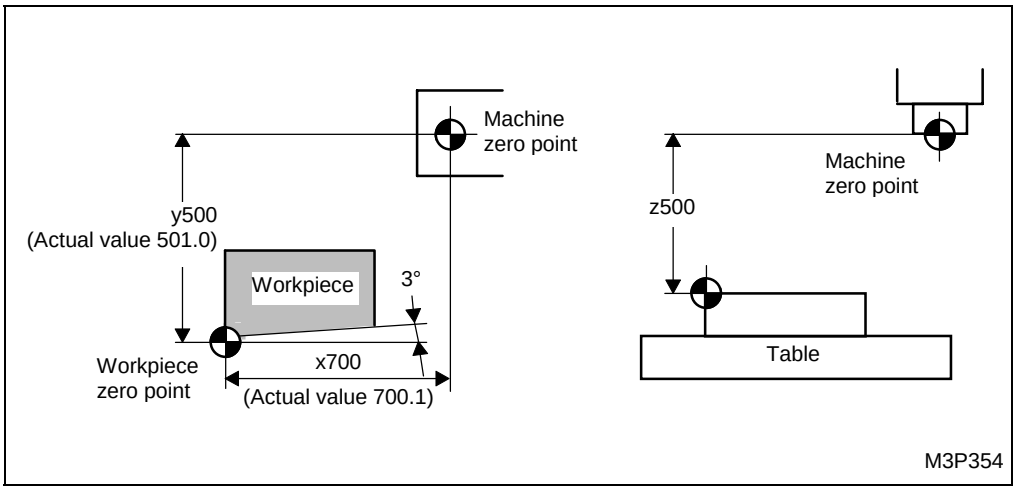
Note: In types [1] to [4] above, the positions related to the points P1 and P2 depend on the sign of r.

B. Measurement movement according to type [3]

Measurement at P1, P2	Measurement at P3	After measurement
		

Note: The X, Y and th basic 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        4
 1     WPC-0                -700.    -500.     0.      -500.     0.
UNo.   UNIT       TOOL      NOM-ϕ     No     U.SKIP    $
 2     MMS       T.SENS      5.               0.       ◆
SNo.   PTN        X     Y     Z     4     R    D/L     K
 1   XYth CNR   10.   -10.  -10.    0    100    ◆      ◆
```

The execution of the MMS unit has the effect of correcting the basic coordinates as follows.

```
UNo.   UNIT    ADD. WPC      X        Y        th       Z        4
 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 starting point P1

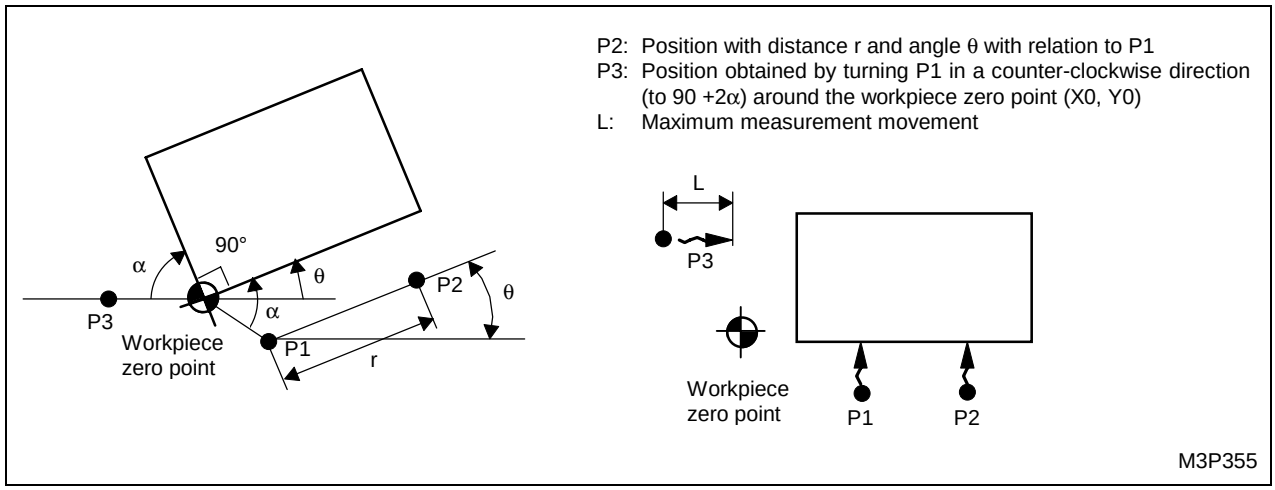


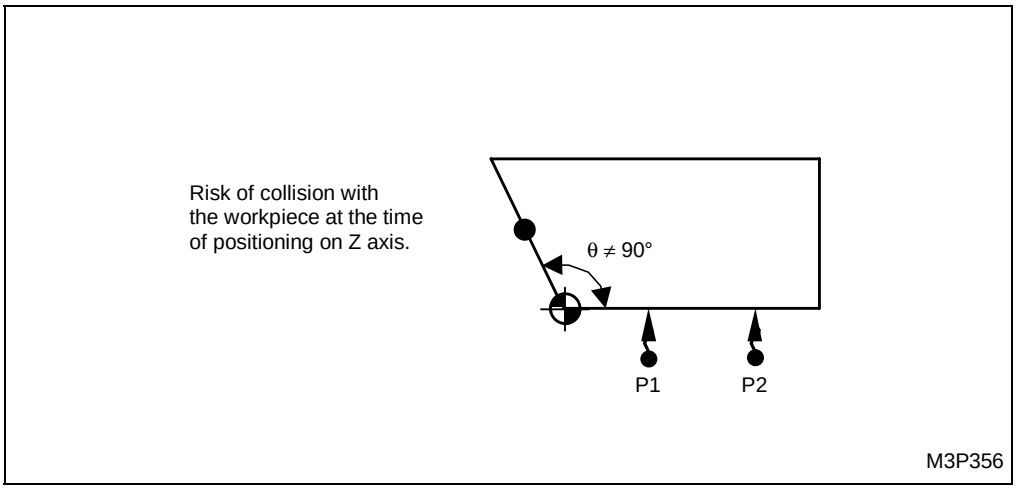
Fig. 4-35 Positioning relation of measuring points (P2, P3) and measurement start point (p1)

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.



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).

Program without priority number

UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE					
0	S45C	50.	1	OFF					
UNo.	UNIT	ADD. WPC	X	Y	th				
1	WPC-1		-300.	-300.	0.				
UNo.	UNIT	DEPTH	SRV-Z		BTM				
2	FACE MIL	0.	5.		1				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD		
1	F-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	CHF-C	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	CHF-C	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

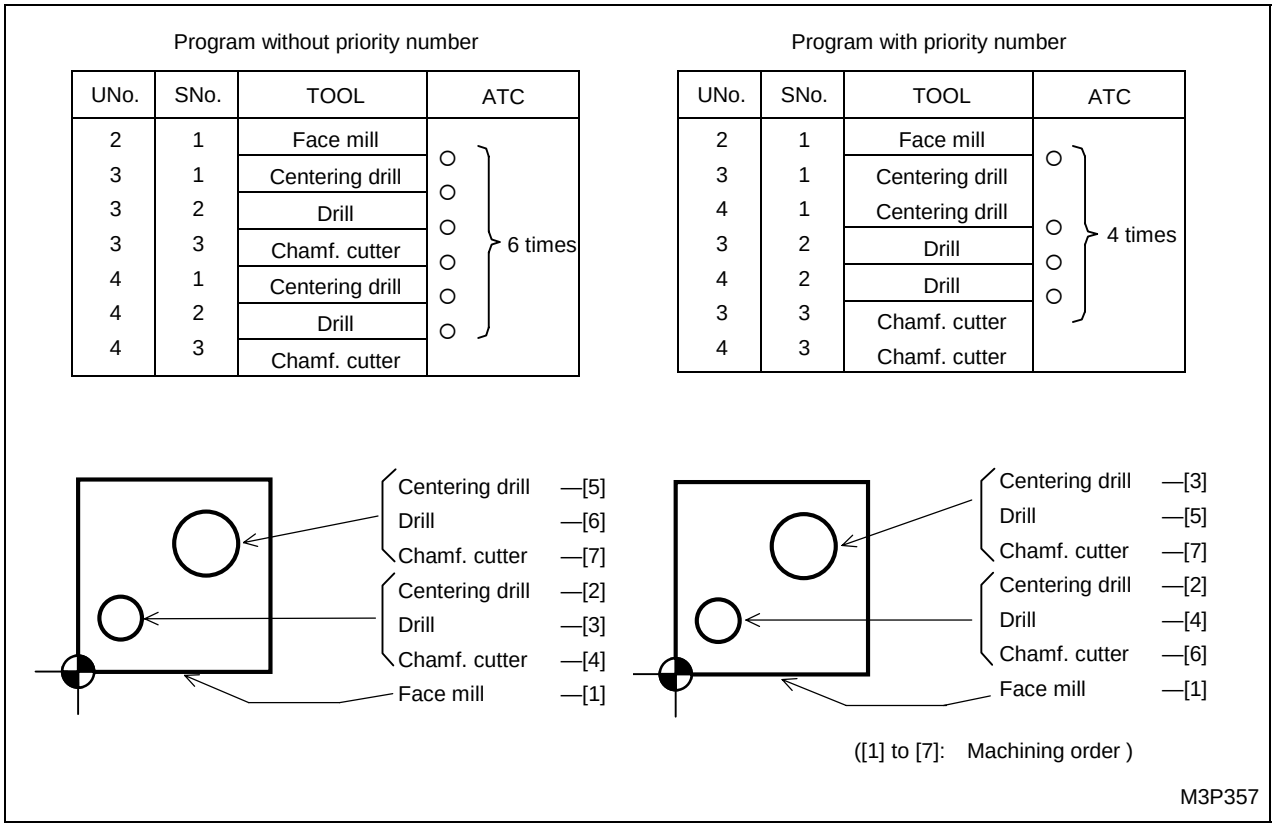
UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE					
0	S45C	50.	1	OFF					
UNo.	UNIT	ADD. WPC	X	Y	th				
1	WPC-1		-300.	-300.	0.				
UNo.	UNIT	DEPTH	SRV-Z		BTM				
2	FACE MIL	0.	5.		1				
SNo.	TOOL	NOM-ϕ	No.	APRCH-X	APRCH-Y	TYPE	ZFD		
1	F-MILL	100.	A [1]	?	?	XBI			
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y				
1	SQR	0.	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	CHF-C	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	CHF-C	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

Subsequent machining priority number

If one reclassifies these two programs by machining order, the following tables are obtained.



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.

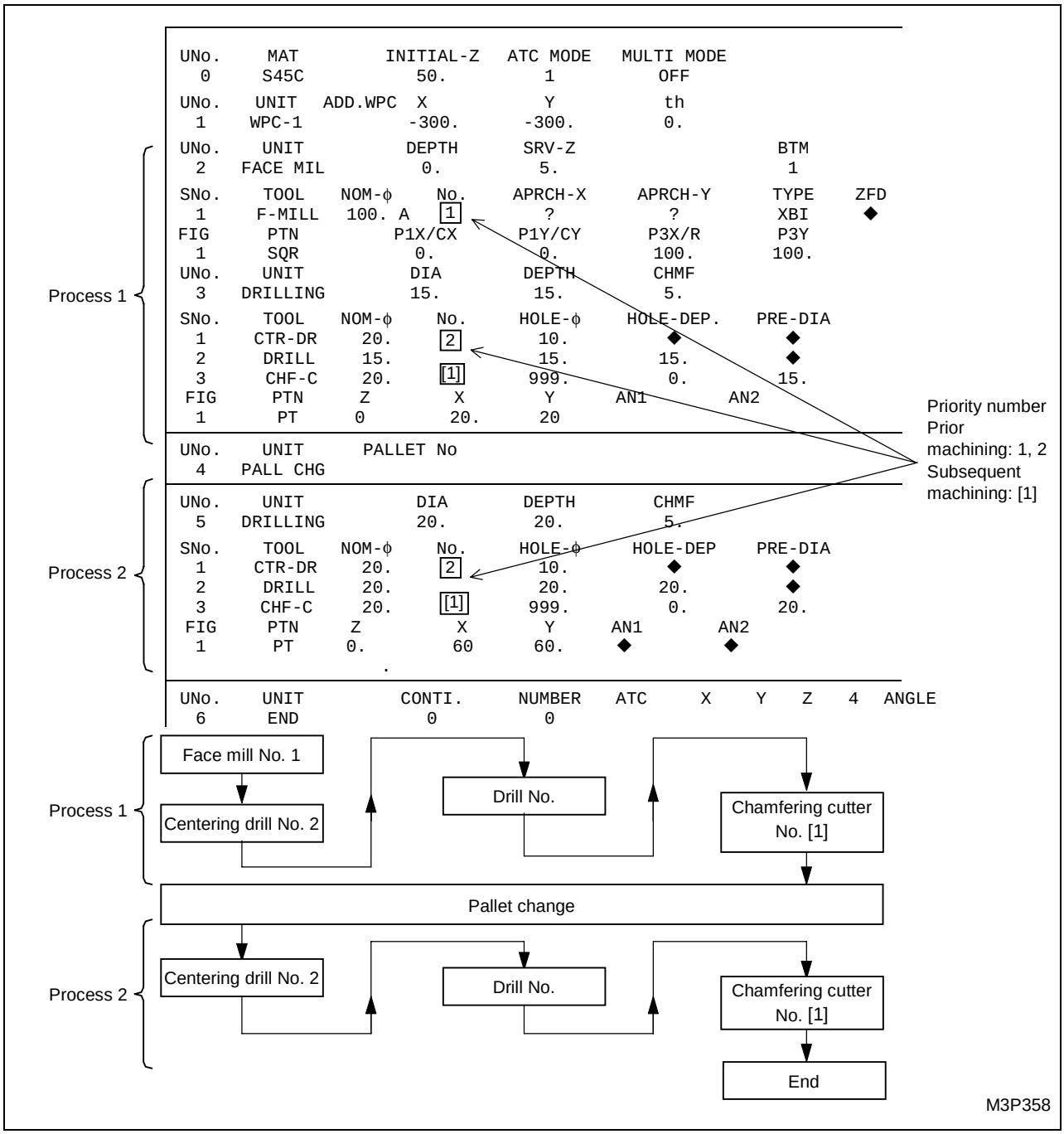
Note 2: When the priority number is assigned to all the tools of the same process, the M code unit without a priority number is executed once for extra between the prior machining and the subsequent machining.

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

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 charmfering cutter, the machining is done in the following order.



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 article “No.”

UNo.	UNIT	DEPTH	SRV-Z	
2	FACE MIL	0.	5.	
SNo.	TOOL	NOM-ϕ	No.	APRCH-X
1	F-MILL	100. A	▌	

➔ 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	F-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 CANCEL.

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 article “No.”

```
UNo.      UNIT      DEPTH      SRV-Z
  2     FACE MIL      0.         5.
SNo.      TOOL      NOM-ϕ      No.      APRCH-X
  1     F-MILL    100. A       ▌
```

(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: [2] [⇨]

- 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     F-MILL    100. A       2          ▌
UNo.      UNIT      DEPTH      SRV-Z
  3     FACE MIL      0.         10.
SNo.      TOOL      NOM-ϕ      No.      APRCH-X
  1     F-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 [CANCEL] 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 article “No.”

```
UNo.      UNIT      DEPTH      SRV-Z
  2     FACE MIL      0.         5.
SNo.      TOOL      NOM-ϕ      No.      APRCH-X
  1     F-MILL    100. A       ▌
```

(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	F-MILL	100. A	5	
UNo.	UNIT	DEPTH	SRV-Z	
3	FACE MIL	0.	10.	
SNo.	TOOL	NOM-ϕ	No.	APRCH-X
1	F-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 CANCEL 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 article “No.”

UNo.	UNIT	DEPTH	SRV-Z	
2	FACE MIL	0.	5.	
SNo.	TOOL	NOM-ϕ	No.	APRCH-X
1	F-MILL	100. A		

(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 [1] [⇨].

➔ All the priority numbers in the specified zone are deleted.

SNo.	TOOL	NOM-ϕ	No.	APRCH-X
1	F-MILL	100. A		
UNo.	UNIT	DEPTH	SRV-Z	
3	FACE MIL	0.	10.	
SNo.	TOOL	NOM-ϕ	No.	APRCH-X
1	F-MILL	100. A		

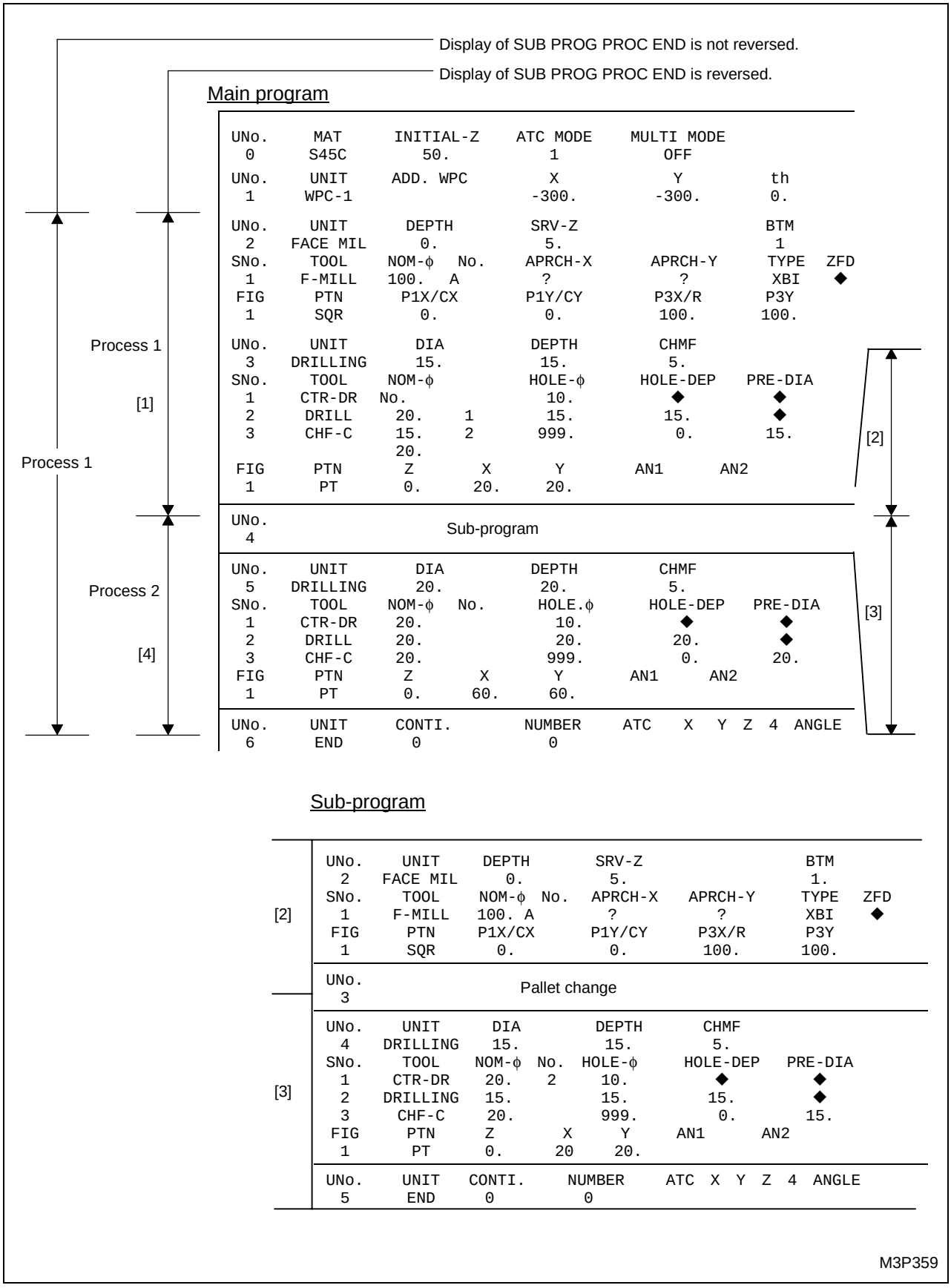
Note: This function is only useful for a program in the process of editing.

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.)



Sub-program

[2]

UNo.	UNIT	DEPTH	SRV-Z		BTM	
2	FACE MIL	0.	5.		1.	
SNo.	TOOL	NOM-ϕ No.	APRCH-X	APRCH-Y	TYPE	ZFD
1	F-MILL	100. A	?	?	XBI	◆
FIG	PTN	P1X/CX	P1Y/CY	P3X/R	P3Y	
1	SQR	0.	0.	100.	100.	

UNo. 3 — Pallet change

[3]

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	DRILLING	15.	15.	15.	◆	
3	CHF-C	20.	999.	0.	15.	
FIG	PTN	Z X	Y	AN1	AN2	
1	PT	0. 20	20.			

UNo.	UNIT	CONTI.	NUMBER	ATC	X	Y	Z	4	ANGLE
5	END	0	0						

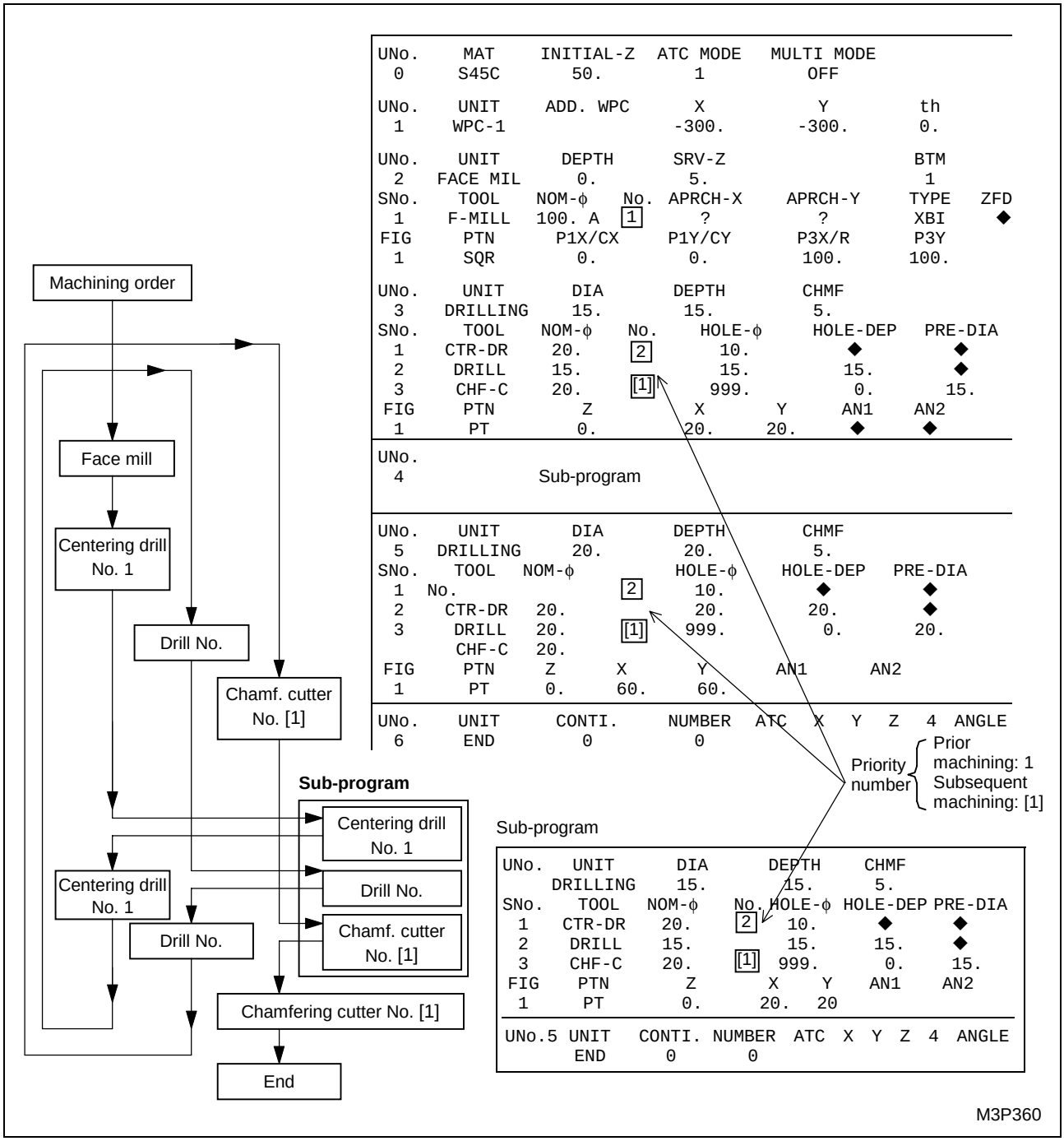
M3P359

Fig. 5-1 Sub-program unit = process end unit

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



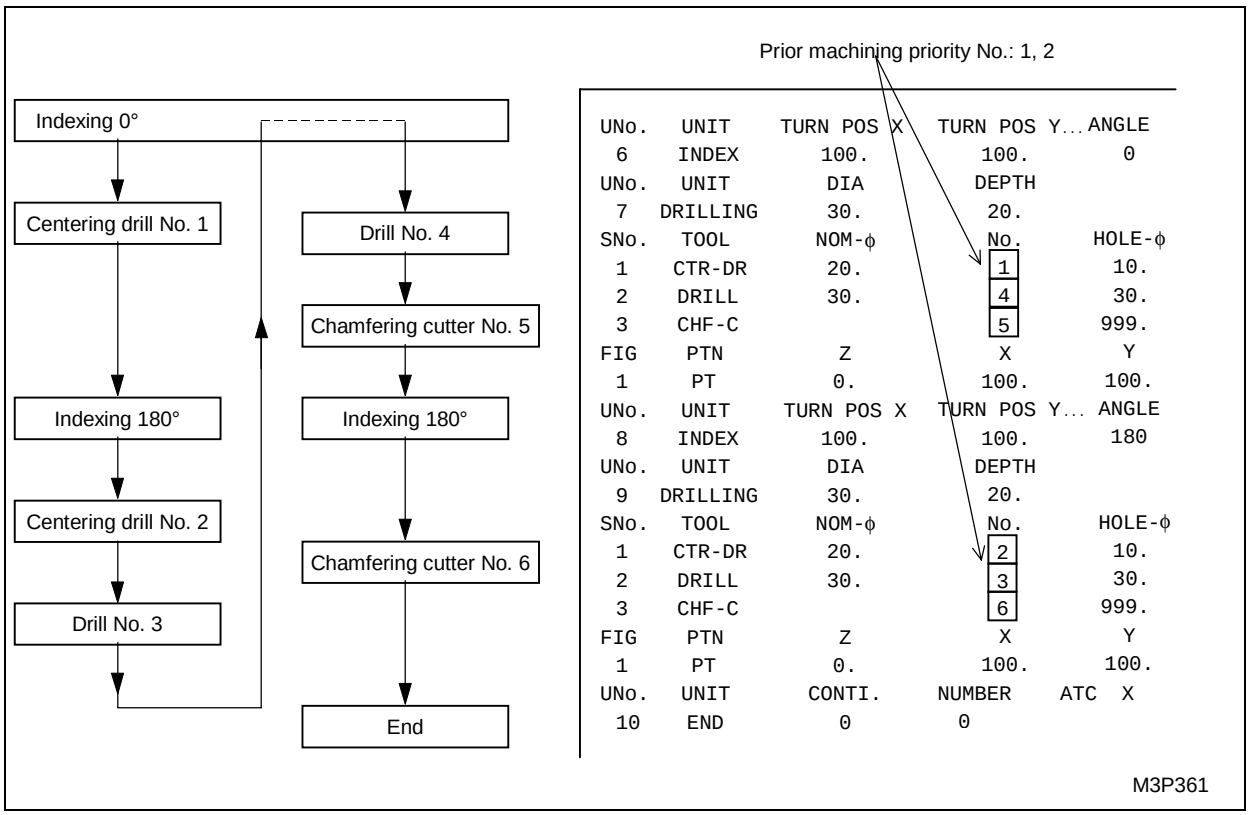
In the process of priority machining search, the sub-program unit is executed as follows:

- 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.

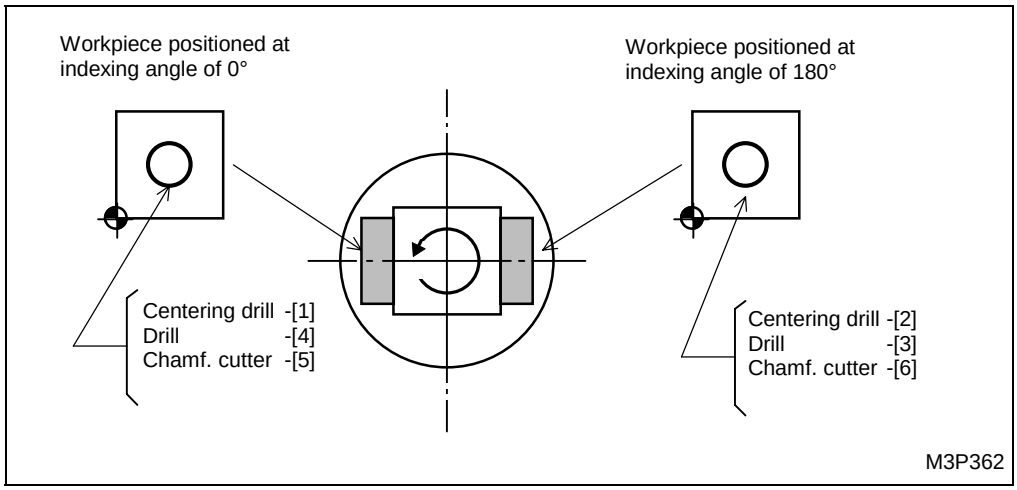
5-5 Relation between the Indexing Unit and Priority Machining Function

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 indexing unit, which contributes to the reduction of machining time.

Example: Priority number assigned to centering drill, drill and chamfering cutter

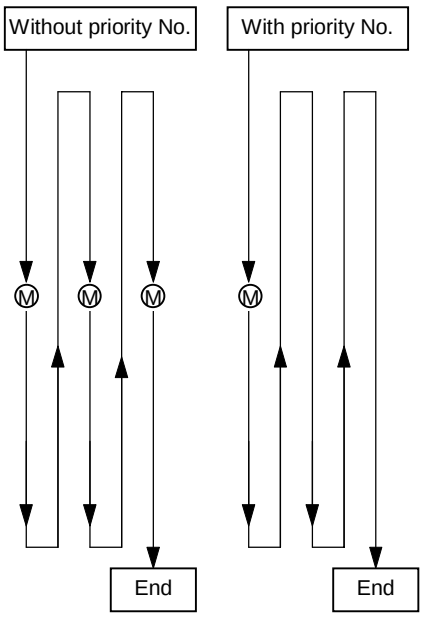


At the time of the execution of this program, the machining order is as follows:



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.




```
UNo.    MAT     INITIAL-Z  ATC MODE     MULTI MODE
 0      S45C       50.         1            OFF

UNo.    UNIT    ADD. WPC       X             Y          th
 1      WPC-1                -300.         -300.       0.

UNo.    UNIT      DEPTH      SRV-Z                     BTM
 2    FACE MIL     0.          5.                      1.
SNo.    TOOL      NOM-ϕ    No.     APRCH-X  APRCH-Y    TYPE   ZFD
 1    F-MILL     100. A     1          ?        ?        XBI    ◆
FIG     PTN      P1X/CX      P1Y/CY         P3X/R       P3Y
 1      SQR        0.          0.            100.       100.

UNo.
 3                          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    CHF-C       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    CHF-C       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

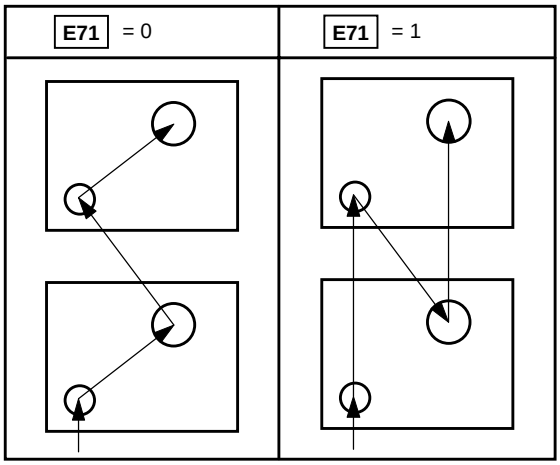
5-7 Relation between Multi-workpiece Machining and the Priority Machining Function

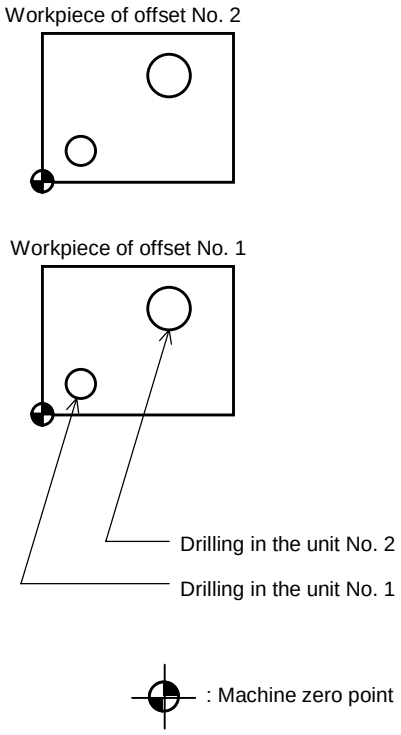
If the multi-workpiece machining process and same-priority-numbered tool data are set both in one program, the order of machining is assigned by the parameter **F71**.

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    INITIAL-Z     ATC
 0   CST IRN       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
```



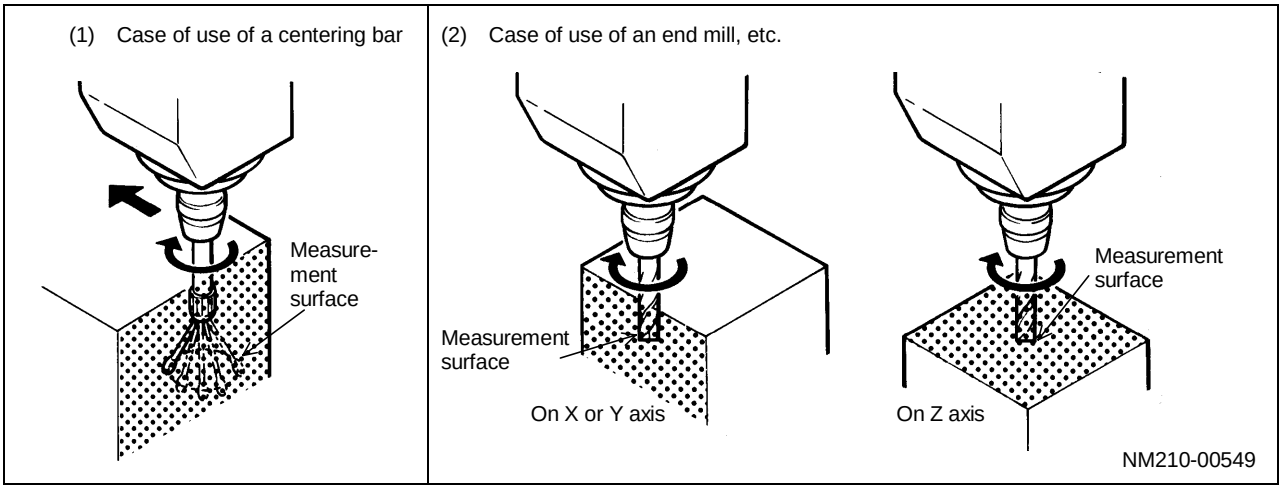


M3P364

- NOTE -

6 COORDINATES MEASUREMENT FUNCTION

The coordinate measurement function is intended for measuring the basic 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 basic coordinates can be calculated and recorded in the basic 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 SEARCH		TEACH		+X SENSOR	-X SENSOR	+Y SENSOR	-Y SENSOR	-Z SENSOR

(5) Pressing the menu key **WPC SEARCH** selects the WPC unit in which the measured basic 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
2	WPC-0		▮	

(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.

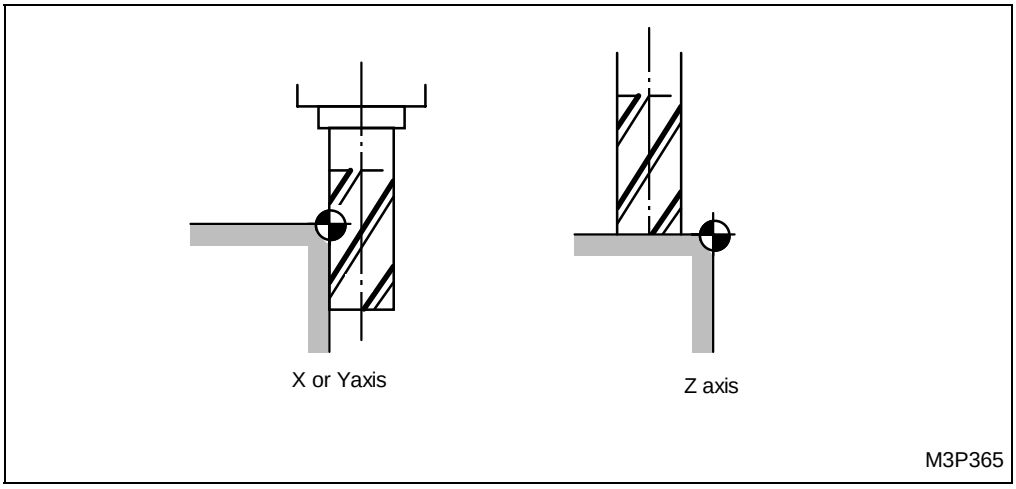


Fig. 6-1 Tool in contact with the surface

(9) Press the menu key **TEACH**.

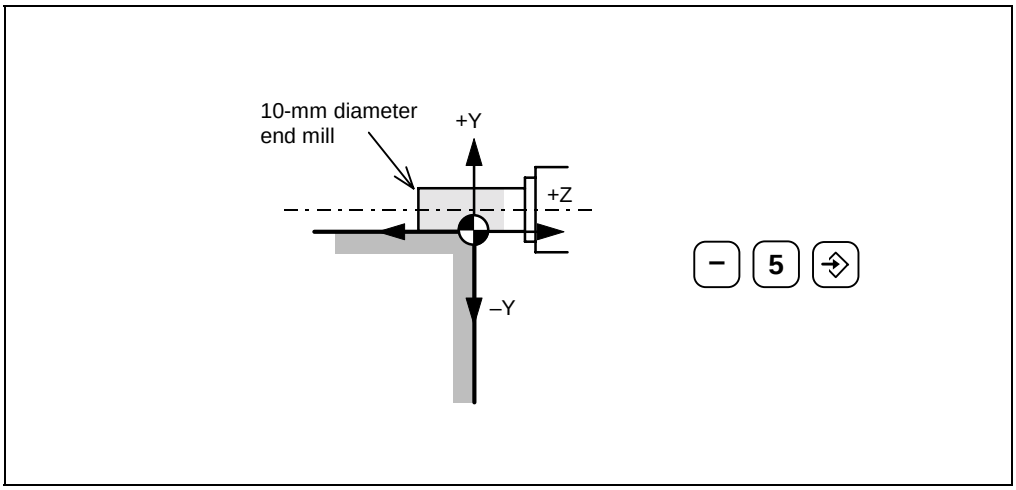
WPC SEARCH		TEACH		+X SENSOR	-X SENSOR	+Y SENSOR	-Y SENSOR	-Z SENSOR

➔ The display of TEACH is reversed and the message **WORKPIECE COORDINATE, WPC-X?** is displayed in the message zone on the display.

(10) Enter the coordinate of the workpiece zero point viewed from the center of the tool by means of numeric keys.

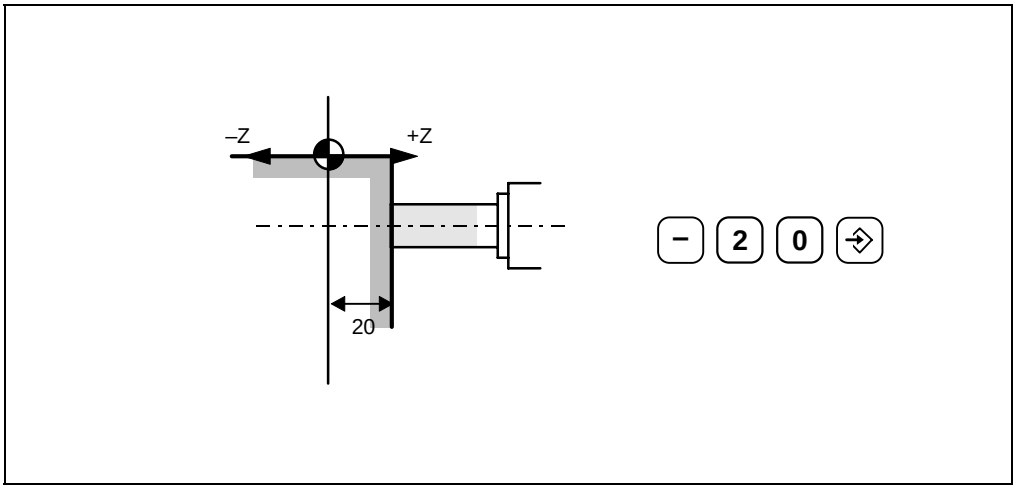
<For a radial direction>

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 orgin when seen from the tool nose. The system, after making reference to the tool length data included in the tool data, automatically calculates the basic coordinates.



(11) The basic coordinate is thus automatically calculated and recorded in the WPC unit.

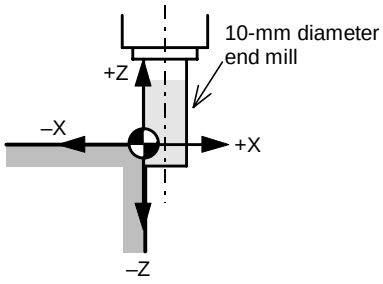
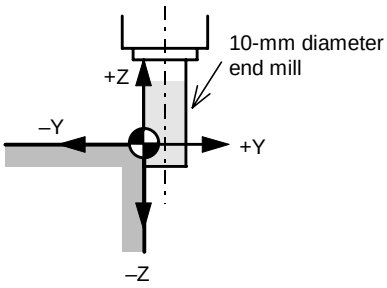
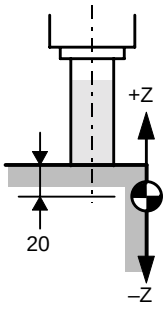
UNo.	UNIT	ADD. WPC	X	Y
2	WPC-0		-210.	

(12) Repeat the above-indicated operations to record the basic 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)

X (FRM-X)	 (–) (5) (→)
Y (FRM-Y)	 (–) (5) (→)
Z (FRM-Z)	 (–) (2) (0) (→)

6-2 Method of Measurement of Coordinates by MDI-MMS

The use of the feeler permits automatic calculation of the basic 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		+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 basic coordinates.

(6) Press the input key (→).

➔ The cursor is moved to the article 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.

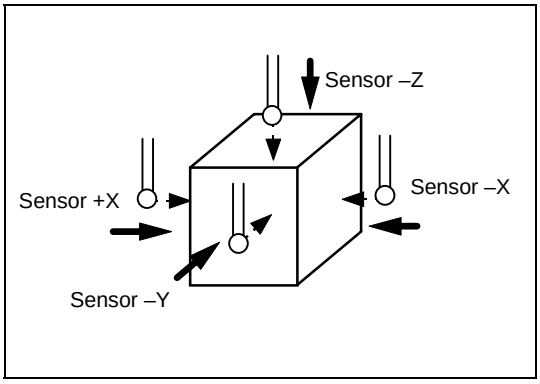
Note: Do not put the feeler point in contact with the surface to be measured.
- The position in which the feeler point is stopped is the measurement start point.

(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		+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 basic coordinate is automatically calculated and displayed.

```
UNo.    UNIT   ADD. WPC    X      Y     th     Z     4
  2     WPC-0              220.    ▌
```

Note: The basic 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.

- NOTE -

7 TPC DATA CREATION

7-1 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.

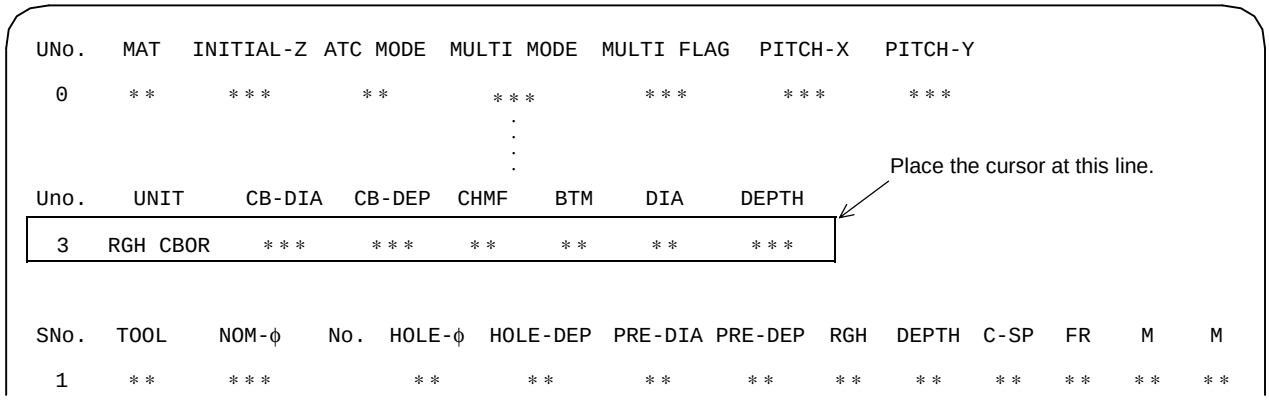
The TPC data, therefore, does not always need to be set to perform machining operations.

The TPC data cannot be set for the following units:

- Common unit
- M code unit
- Basic coordinates system unit (WPC)
- Auxiliary coordinates system unit (OFFSET)
- End unit
- Sub-program unit
- Pallet changing unit
- Indexing unit
- Process end unit
- Three-dimensional surface machining unit (3-D)

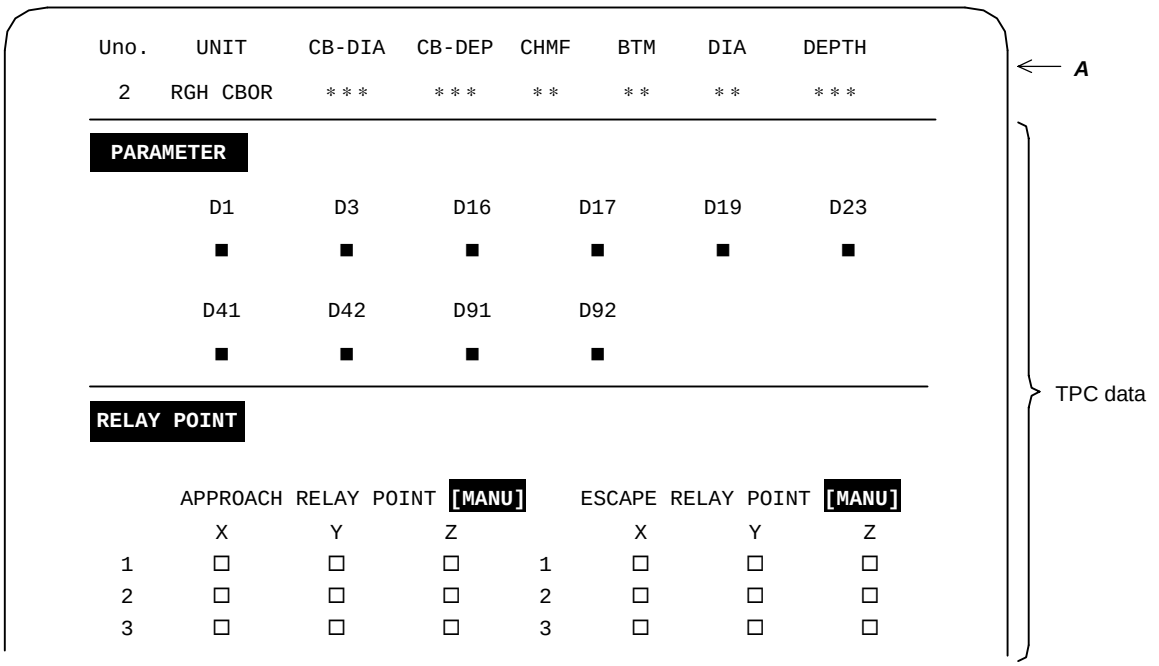
(1) Set the cursor on the unit data where the TPC data is to be set.

Example: If the TPC data is to be set for the RGH CBOR unit



(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 at items marked with ■. 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 overriden with the new data.
  The parameter settings will not change even if the displayed data is changed on the **TPC** display.
- Items marked with □ can be filled with data as required.
  To specify relay points for the tool approach or escape (return) path, first move the cursor to the item [**AUTO**] of the required data section, and then press the 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 INITIAL-Z   ATC MODE
 0    **    ***          **
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 INITIAL-Z  ATC MODE
 0    **    ***       **
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    **   **   **   **
```

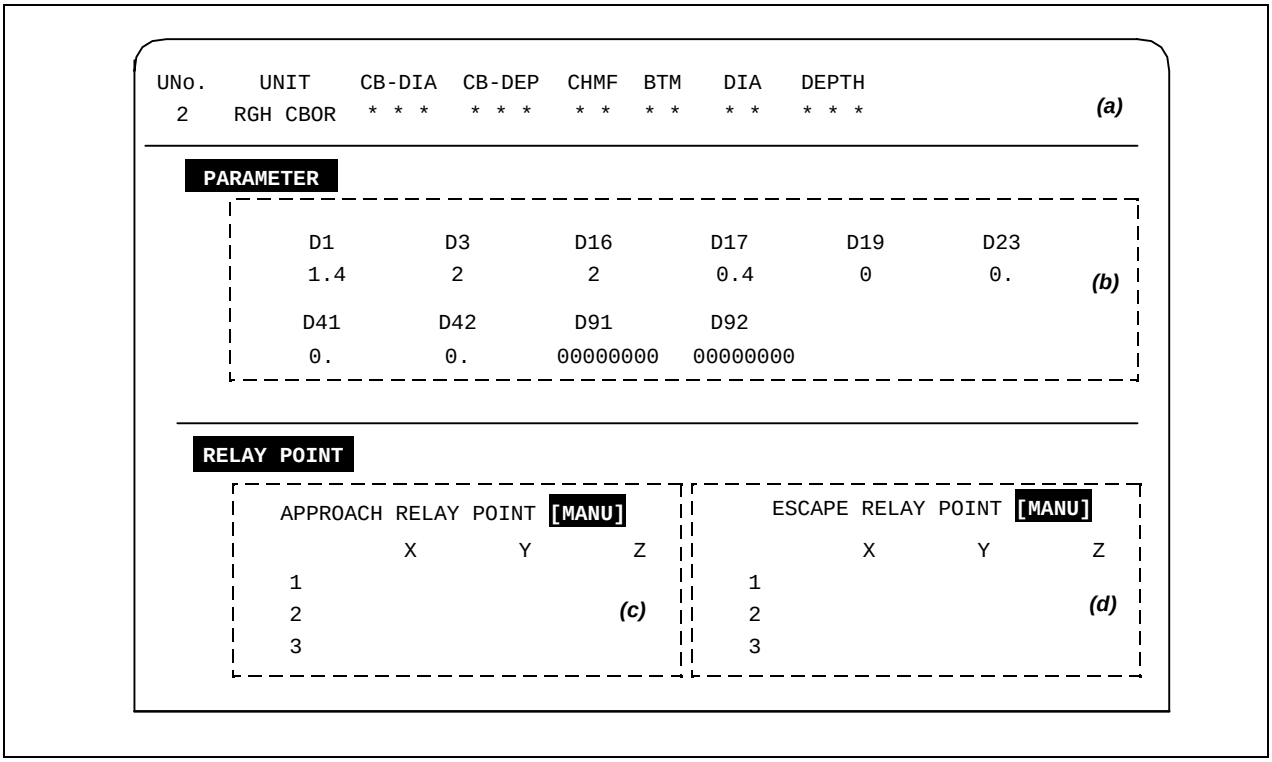
+ mark

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.

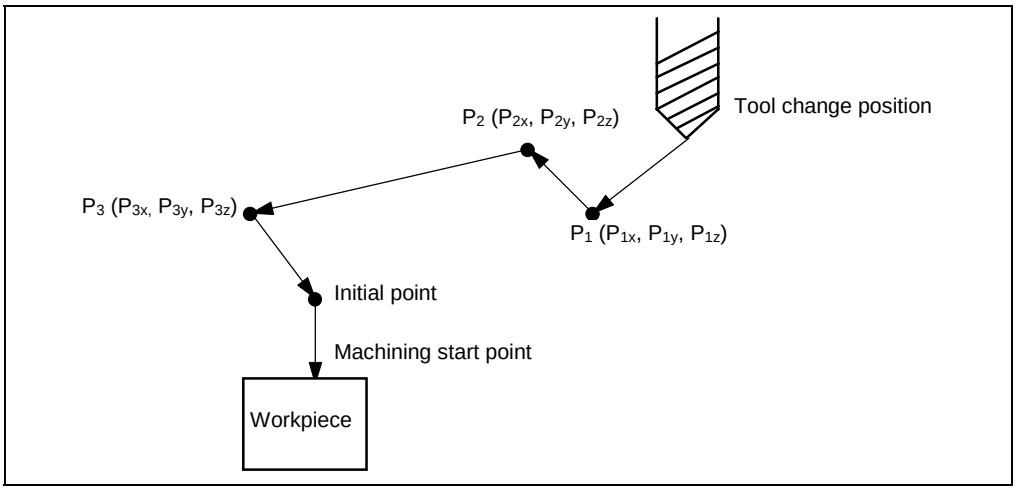
7-2 Description of Each TPC Data Item



(*a*) Unit data for which the **TPC** display is called up. The data cannot be changed with the TPC data displayed on the screen.

(*b*) The addresses of related parameters and the data that have been set on the **PARAMETER** display are displayed according to the particular type of unit. Modification of the data allows the machine to be correspondingly operated only during that unit. The data settings on the **PARAMETER** display, however, will not change by their modification on the **TPC** display. Refer to the separate Parameter List for details of parameter data.
Parameters denoting the distance are usually set in 0.1 mm (or 0.01 in.) steps, but they will be displayed here in 1 mm (or 1 in.) steps.

(*c*) Use this section to modify the approach path so that interference does not occur.
To modify the path, first set the cursor at [**AUTO**] of the required section and then press the menu key **MANUAL**, to display [**MANU**]. Finally enter the coordinates of the relay points on three lines (1, 2 and 3) in the desired order of relaying.



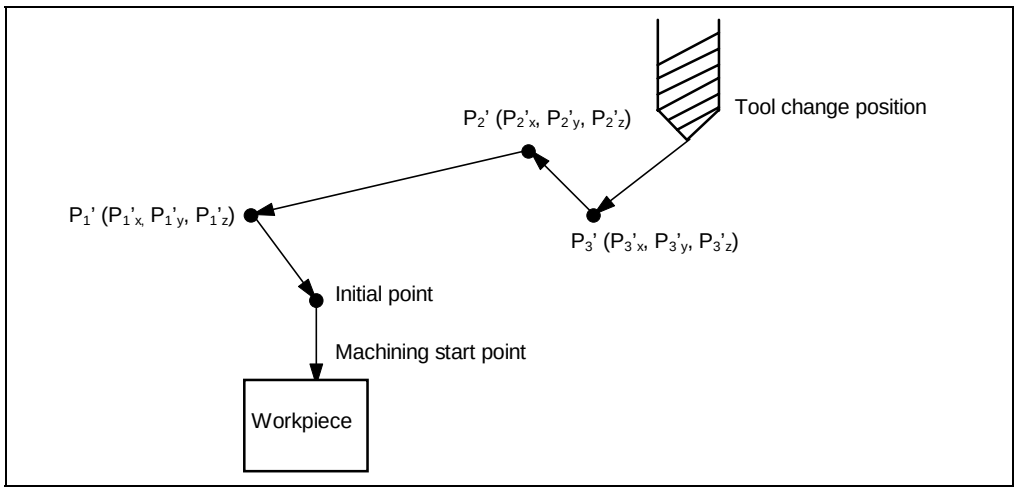
For approaching path from the tool change position through relay points P_1, P_2 and P_3 to the machining start point, as shown above, set data as follows:

APPROACH RELAY POINT **[MANU]**

	X	Y	Z	
1	P_{1X}	P_{1y}	P_{1Z}	Path through P_1
2	P_{2X}	P_{2y}	P_{2Z}	Path through P_2
3	P_{3X}	P_{3y}	P_{3Z}	Path through P_3

Specify the position using the program coordinates system.

(*d*) Use this section to modify the escape path so that interference does not occur. Refer to the description in **(*c*)** for details on data setting.



ESCAPE RELAY POINT **[MANU]**

	X	Y	Z	
1	$P_1'_X$	$P_1'_y$	$P_1'_Z$	Path through P_1'
2	$P_2'_X$	$P_2'_y$	$P_2'_Z$	Path through P_2'
3	$P_3'_X$	$P_3'_y$	$P_3'_Z$	Path through P_3'

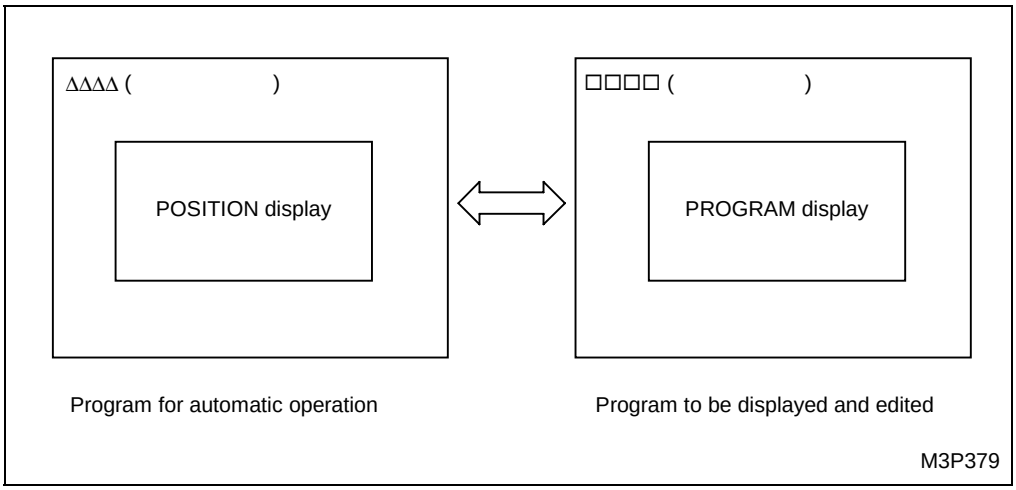
Specify the position using the program coordinates system.

- NOTE -

8 BACKGROUND PROGRAMMING

For the purpose of more efficient programming, M640M 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 -

9 CASE OF APPEARANCE OF ALARM

1. Alarm display

Alarm is displayed either in red or blue.

Red .....Lights 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 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 .

Red .....Clear the display with the reset key.

Blue .....Clear 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 -

10 THREE-DIGIT G-FORMAT

10-1 Outline

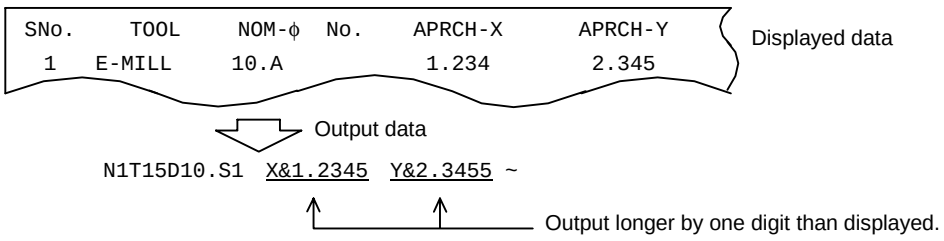
The three-digit G-format is a format of expressing MAZATROL program data and other NC data. The various types of data within the NC unit are each assigned to a specific “three-digit G + address + data” set. Use of the data input/output functions based on the three-digit G-format allows the NC-stored data to be managed under the same environment as those of EIA/ISO programs.
Data that have been output to external units in the three-digit G-format can be edited using a personal computer, and if the edited data are restored into the NC unit, the corresponding original data within the NC unit will be auto-modified according to the required edition.

10-2 Detailed Description

1. MAZATROL program data

- Unit data have an assigned specific three-digit G-code for each unit.
- Tool sequence data succeed the block of the three-digit G-code of the unit data and are positioned between code G424, which denotes the beginning of the sequence data, and code G425, which denotes the end of the sequence data.
- The shape data, if present, succeed the block of the three-digit G-code of sequence data and are positioned between code G420, which denotes the beginning of the shape data, and code G421, which denotes the end of the shape data.
- The TPC data, if present, succeed the block of the three-digit G-code of the unit data and are positioned between code G422, which denotes the beginning of TPC data, and code G423, which denotes the end of TPC data.
- The order of output of the machining program data is predetermined. That is, unit data, TPC data, 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.

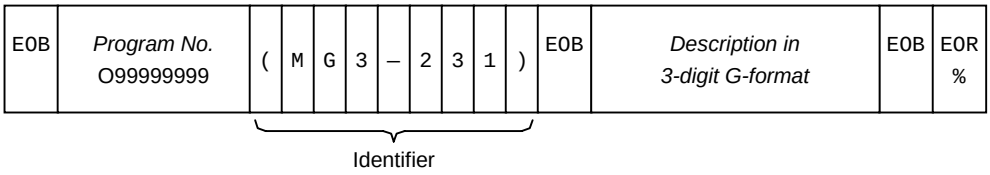
The codes for data identification are listed up and described in detail on the following pages.

10-3 Three-digit G-format of MAZATROL Program

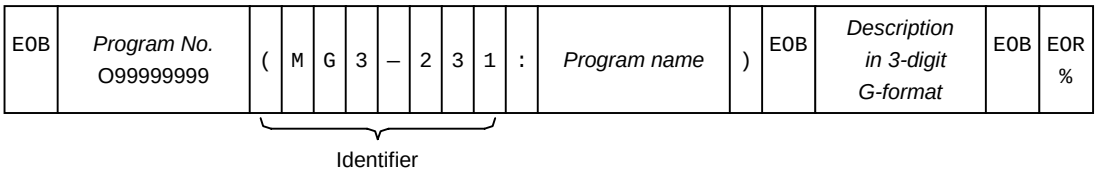
1. Program number and program name

In the three-digit G-format input/output of MAZATROL programs, the number and name of a program is described in the following format:

(Without program name)



(With program name)



- 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 “(MG3-231)” succeeding the program number identifies a MAZATROL program described in the three-digit G-format of the M640F.
- 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 G300

UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
U	( )	Z	A	B	C	X	Y

MULTI MODE
1: OFF
2: 5 * 2
3: OFFSET TYPE

- Basic coordinates system unit G379

UNo.	WPC No.	ADD. WPC	X	Y	th	Z	4	5
U	A	B	X	Y	E	Z	F	H

ADD. WPC B:	
1 - 6	G54 - G59
7 - 16	A - K
17 - 64	G54.1P1 - G54.1P48

- 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	ANGLE	Rotation direction	5
U	B	C	A	X	Y	Z	F	D	E	H

Rotation direction
0: CW
1: CCW
2: SHORTEST

- Sub-program unit G303

UNo.	WORK No.	$	REPEAT	Measurement flag
U	W	V	L	F

Measurement flag
0: Sub program contains no measurement
1: Sub program contains measurement

$ V:	
0 - 9	Unit skip (0 - 9)
–1 - –4	Multi-workpiece machining (A - D)

- Manual program mode unit G305

UNo.	TOOL	NOM-ϕ	ID code	Priority No.
U	T	D	S	P

* For nominal diameter of taps, refer to “Tapping unit.”

TOOL T:	Tool name
1	CTR-DR
2	DRILL
3	REAM
4	TAP (M)
5	TAP (UN)
6	TAP (PT)
7	TAP (PF)
8	TAP (PS)
9	TAP (OTHER)
10	BK FACE
11	BOR BAR
12	B-B BAR
13	CHF-C
14	F-MILL
15	E-MILL
16	OTHER
17	CHP VAC
18	T. SENS.
19	B-E-MIL

ID code S:	
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**

Priority No. P:	
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
U	P	MA	MB	MC	MD	ME	MF	MG	MH

Refer to “Manual program mode unit.”

- MMS 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.”

- Basic coordinate shift unit G306

UNo.	X	Y	Z	4	5	th
U	X	Y	Z	F	H	E

- Pallet changing unit G381

UNo.	PALLET No.	No. of next pallet
U	P	&P

- Process end unit G385

UNo.
U

- Indexing unit G382

UNo.	TURN POS X	TURN POS Y	TURN POS Z	ANGLE (Angle 1)	TURN DIR.	Angle 2
U	X	Y	Z	D	E	H
					0: CW	
					1: CCW	
					2: SHORTEST	

- 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
U	*	E	P	H	C	K

Example:

M10. A1D10.
UNn 1-2 A2D1V2
UN 1H-2 A3D1V2B1
PT 2Q A4D2B2

Tap screw type A:	
1	M
2	UNn
3	UN
4	PT
5	PF
6	PS
7	OTHER

Tap fraction B:	
1	1/2
2	1/4
3	1/8
4	1/16

Nominal diameter D:
Nominal diameter 2 V:

- 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

- 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

- 3-D plane definition G397

UNo.	LINE	PLANE	DISTANCE	ROT.-X	ROT.-Y	ROT.-Z	SHIFT-X	SHIFT-Y	SHIFT-Z
U	A	B	D	X	Y	Z	&X	&Y	&Z
	1: FL	1: X-Y							
	2: GL	2: Y-Z							
		3: X-Z							

- 3-D coordinates transfer G398

UNo.	ROT.-X	ROT.-Y	ROT.-Z	SHIFT-X	SHIFT-Y	SHIFT-Z
U	X	Y	Z	&X	&Y	&Z

- 3-D processing-area appointment G399

UNo.	X-MIN	X-MAX	Y-MIN	Y-MAX	Z-MIN	Z-MAX	IN/OUT
U	X	&X	Y	&Y	Z	&Z	A
							1: IN
							2: OUT

3. Machining sequence

`G420U_` Beginning of sequence data (U: unit No.)

`N1`
`N2`
⋮
⋮ Sequence data

`G421` End of sequence data

- Multi-workpiece machining sequence

SNo.	X	Y	th	Z
N	X	Y	E	Z

- Manual program mode sequence

SNo.	G1	G2	DATA 1	DATA 2	DATA 3	DATA 4	DATA 5	DATA 6	S	M/B
N	G	&G	?	?	?	?	?	?	S	M/B

→ M code M:, B code B:

→ Set address and data as specified, e.g. “X123.456”
(if 4th, 5th or 6th axis has been specified, set &4, &5 or &6 respectively).

- MMS sequence

SNo.	PTN	X	Y	Z	4	5	R	D/L	K
N	A	X	Y	Z	F	H	R	D	K

PTN A:		PTN A:		PTN A:	
1	X-FACE	6	X-STP	11	CAL
2	Y-FACE	7	Y-STP	12	Y-IN.
3	Z-FACE	8	XY-HOL	13	Z-IN.
4	X-GRV	9	XY-BOS		
5	Y-GRV	10	XYthCNR		

- Sub-program sequence

SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
N	?	?	?	?	?	?

→ Set address and data as specified, e.g. “X123.456”
(if macro variable has been specified, set “X#100”)

- 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

- Shape sequence, SQUARE (fixed form)

SNo.	PTN	P1X	P1Y	P3X	P3Y	CN1	CN2	CN3	CN4	Shape sequence type
N	A	X	Y	&X	&Y	CA/RA	CB/RB	CC/RC	CD/RD	B
	1: SQR									0: Fixed form

→ Set C_ or R_ for corner chamfering or rounding respectively.

- Shape sequence, CIRCLE (fixed form)

SNo.	PTN	CX	CY	R	Shape sequence type
N	A	X	Y	R	B
	2: CIR				0: Fixed form

- Shape sequence (arbitrary)

SNo.	PTN	X	Y	R/th	I	J	P	CNR	Start point	Shape sequence type
N	A	X	Y	V	I	J	P	C/R	S	B
	1: LINE								0: Not start point	1: Arbitrary
	2: CW								1: Start point	
	3: CCW									
	4: FIG-SH									
	5: CW-SH									
	6: CCW-SH									
	7: REP-EN									

4. Tool sequence

```
G424U_      Beginning of tool sequence (U: unit No.)
N1
N2          Tool sequence data
 :
 :
G425        End of tool sequence
```

- Point machining tool sequence

SNo.	TOOL	NOM-ϕ	ID code	Priority No.	HOLE-ϕ	HOLE-DEP	PRE-DIA	PRE-DEP	RGH	DEPTH	C-SP	FR	M	M
N	T	D	S	P	E	H	&E	&H	F	Z	I	J	MA	MB

E-MILL &H
0: CCW
1: CW
2: **CCW**
3: **CW**

CTR-DR F:
0: 90°
1: 118°
2: 60°

DRILL F:
1: DRIL
2: PCK2
3: PCK1
4: AUTO
18: **PCK1**
19: **PCK2**

REAM Z:
–1: G00
0: G01
1 - 9999
CTR-DR Z:
0: CTR-DR
1: CHAMF

BOR BAR &E
0: CYCLE 1
1: CYCLE 2
2: CYCLE 3
TAP &E
0: TAP
1: PECK
2: PLANET

→ For nominal diameter of taps, refer to “Tapping unit.”

→ Refer to “Manual program mode unit.”

- Line machining tool sequence

SNo.	TOOL	NOM-ϕ	ID code	Priority No.	APRCH-X	APRCH-Y	TYPE	ZFD	DEP-Z	C-SP	FR	M	M	R/F
N	T	D	S	P	X	Y	Q	F	Z	I	J	MA	MB	B
							7: CW	–1: G00						0: R
							8: CCW	0: G01						1: F
								0.1 - 9.9						

Refer to "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."

- 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."

- 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: R	
							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.)

⋮ ⎫
⋮ ⎬ TPC data
⋮ ⎭

G423 End of TPC data

A. TPC data

	A	B	C	D	E	F	H	I	J	K	L	M	P	Q	R	S	Y	Z
MMS	L1	L2																
DRILLING	D1	D3	D16	D17									D41	D42	D91		D45	D46
RGH CBOR	D1	D3	D16	D17		D19	D23						D41	D42	D91	D92	D45	D46
RGH RCB	D1	D3	D16	D17	D5	D19		D40					D41	D42	D91	D92	D45	D46
REAMING	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D29	D41	D42	D91	D92	D45	D46
TAPPING	D1	D3	D16	D17	D22			D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46
BK-CBORE	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28	D33	D41	D42	D91	D92	D45	D46
CIRC. MIL	D1		D16	D17		D19	D23						D41	D42	D91	D92		
CBOR-TAP	D1	D3	D16	D17	D22	D19	D23	D48	D31	D32	D49	D29	D41	D42	D91	D92	D45	D46
BORE T1	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46
BORE S1	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46
BORE T2	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46
BORE S2	D1	D3	D16	D17	D18	D19	D23	D24	D25	D26	D28		D41	D42	D91	D92	D45	D46
LINE CTR		E2		E7	E9		E17							E95				
LINE RGT		E2		E7	E9		E17			E22	E23	E24	E25	E95				
LINE LFT		E2		E7	E9		E17			E22	E23	E24	E25	E95				
LINE OUT	E1	E2	E5	E7	E9		E17		E21	E22	E23	E24	E25	E95				
LINE IN	E1	E2	E5	E7	E9		E17		E21	E22	E23	E24	E25	E95				
CHMF RGH		E2		E8	E9	E11	E17							E95				
CHMF LFT		E2		E8	E9	E11	E17							E95				
CHMF OUT	E1	E2		E8	E9	E11	E17		E21					E95				
CHMF IN	E1	E2		E8	E9	E11	E17		E21					E95				
FACE MIL					E9	E12	E15											
TOP EMIL				E7	E9	E13	E17							E97				
STEP	E1	E2	E5	E7	E9	E16	E17		E21	E22	E23	E24	E25	E91	E98			
POCKET	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E92				
PCKT MT	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E93				
PCKT VLY	E1	E2	E5	E7	E9		E17	E18	E21	E22	E23	E24	E25	E94	E98			
SLOT				E7	E9		E17		E21					E96				
ANG. FACE					E9	E12												

D1, E2 etc. denote parameter type.

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									

10-4 Various Data Description Using G10

"G10" is normally used to express the other various data than program data, and address "L" that follows denotes the type of the data.

```
G10L_
     ↑
     └──────── Data type
```

1. Tool data

A. MAZATROL data

```
G10L41T_P_C_D_S_I_E_H_K_R_J_Q_(_)_N_A_
```

1	PKNo.	TOOL	NOM-ϕ	ID code	Interference data	ACT-ϕ	LENGTH	COMP.	AUXIL.	LIFE	TIME	MAT.	STATUS	Drill border value
T	P	C	D	S	I	E	H	K		J	Q	( )	N	A

C	S	I
1: CTR-DR	0:	0:
2: DRILL	1: A	1:↓
3: REAM	2: B	2:↑
4: TAP (M)	3: C	3: L
5: TAP (UN)	4: D	4: S
6: TAP (PT)	5: E	
7: TAP (PF)	6: F	
8: TAP (PS)	7: G	
9: TAP(OTHER)	8: H	
10: BK FACE	9: J	
11: BOR BAR	–1: A	
12: B-B BAR	–2: B	
13: CHF-C	–3: C	
14: F-MILL	–4: D	
15: E-MILL	–5: E	
16: OTHER	–6: F	
17: CHP VAC	–7: G	
18: T. SENS.	–8: H	
19: B-E-MIL	–9: J	

For tap

End-mill corner R
R

Tap function
Y
0: FLOAT
1: FIX

Number of drill cutting edges
B

Tap screw type A:	
1	M
2	UNn
3	UN
4	PT
5	PF
6	PS
7	OTHER

Tap fraction B:	
1	1/2
2	1/4
3	1/8
4	1/16

Nominal diameter D:
Nominal diameter 2 V:

Example

M10.	A1D10.
UNn 1-2	A2D1V2
UN 1H-2	A3D1V2B1
PT 2Q	A4D2B2

B. EIA/ISO and extended data

```
G10L42T_J_I_D_H_E_N_P_K_S_
```

TNo.	GROUP No.	IDNo.	NOM-ϕ CORR.	ACT-ϕ CORR.	NOM-ϕ COR. No.	ACT-ϕ COR. No.	THR.	HP	REV.
T	J	I	D	H	E	N	P	K	S

Tap returning speed rate
R

2. Tool offset

A. Type A

G10L10P_R_

B. Type B

Geometric compensation for tool length ..... G10L10P_R_

Defacement for tool length ..... G10L11P_R_

Geometric compensation for tool radius ..... G10L12P_R_

Defacement for tool radius ..... G10L13P_R_

Offset No.	OFFSET
P	R

3. Tool file

A. End mill, face mill and ball-end mill

G10L49P_C_D_S_(_) R_H_

Tool file No.	TOOL	NOM-ϕ	ID code	MAT.	DEPTH	No.
P	C	D	S	( )	R	H

B. Chamfering cutter

G10L49P_C_D_S_E_(_) A_

→ Refer to “Tool data.”

Tool file No.	TOOL	NOM-ϕ	ID code	MIN-ϕ	MAT.	ANG
P	C	D	S	E	( )	A

4. Cutting condition

A. Cutting condition (WORK-MAT)

G10L52P_(_)

Material No.	WORK-MAT
P	( )

B. Cutting condition (TOOL-MAT, C-SP, FR)

G10L_P_S_(_)

G10L_P_S_F_

DRILL	CTR-DR	REAM	TAP	BOR BAR	B-B BAR	BK FACE	CHF-C	E-MILL	F-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	( )

5. 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
P	X	Y	Z	A	B	C

6. Parameter

A. User parameter

G10L50 User parameter setting start

D1Z1 1 is set to D1 (example).

D2Z2
⋮

G11 User parameter setting end

POINT CUTTING D:	LINE/FACE/3-D CUTTING E:	USER No. 1 F:	USER No. 2 G:	USER No. 3 H:	USER No. 4 I:
D1 - D108	E1 - E108	F1 - F108	Q1 - Q132	H1 - H108	I1 - I96

B. Machine parameter

G10L51 Machine parameter setting start

J1Z1 1 is set to J1 (example).

J2Z1
⋮

G11 Machine parameter setting end

MACHINE PARA. No. 1 J:	MACHINE PARA. No. 2 K:	MACHINE PARA. No. 3 L:	MACHINE PARA. No. 4 M:
J1 - J108	K1 - K108	L1 - L108	M1 - M108

MACHINE PARA. No. 5 N:	MACHINE PARA. No. 6 S:	CUTTING PARA. No. 1	CUTTING PARA. No. 2
N1 - N96	S1 - S96	A1 - A108	B1 - B108

CUTTING PARA. No. 3	OPTION PARA.	SPINDLE PARA.	SERVO PARA.	R REGISTER PARA.
C1 - C108	Y1 - Y108	P1 - P384	V1 - V480	R2100 - R2527

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					
5: 5					
6: 6					

Parameter data should not be changed until you have fully understood the meaning of the respective parameter.

7. Additional WPC

```
G10L43P_X_Y_E_Z_F_H_
```

WPC No.	X	Y	th	Z	4	5
P	X	Y	E	Z	F	H
1: A						
2: B						
3: C						
4: D						
5: E						
6: F						
7: G						
8: H						
9: J						
10: K						

8. 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.
```

9. 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

10. 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						

11. Maintenance check

A. Regular check item

G10L70P_T_C_Y_M_D_( )

Check No.	Target time	Current time	Year	Month	Day	Check item
P	T	C	Y	M	D	( )

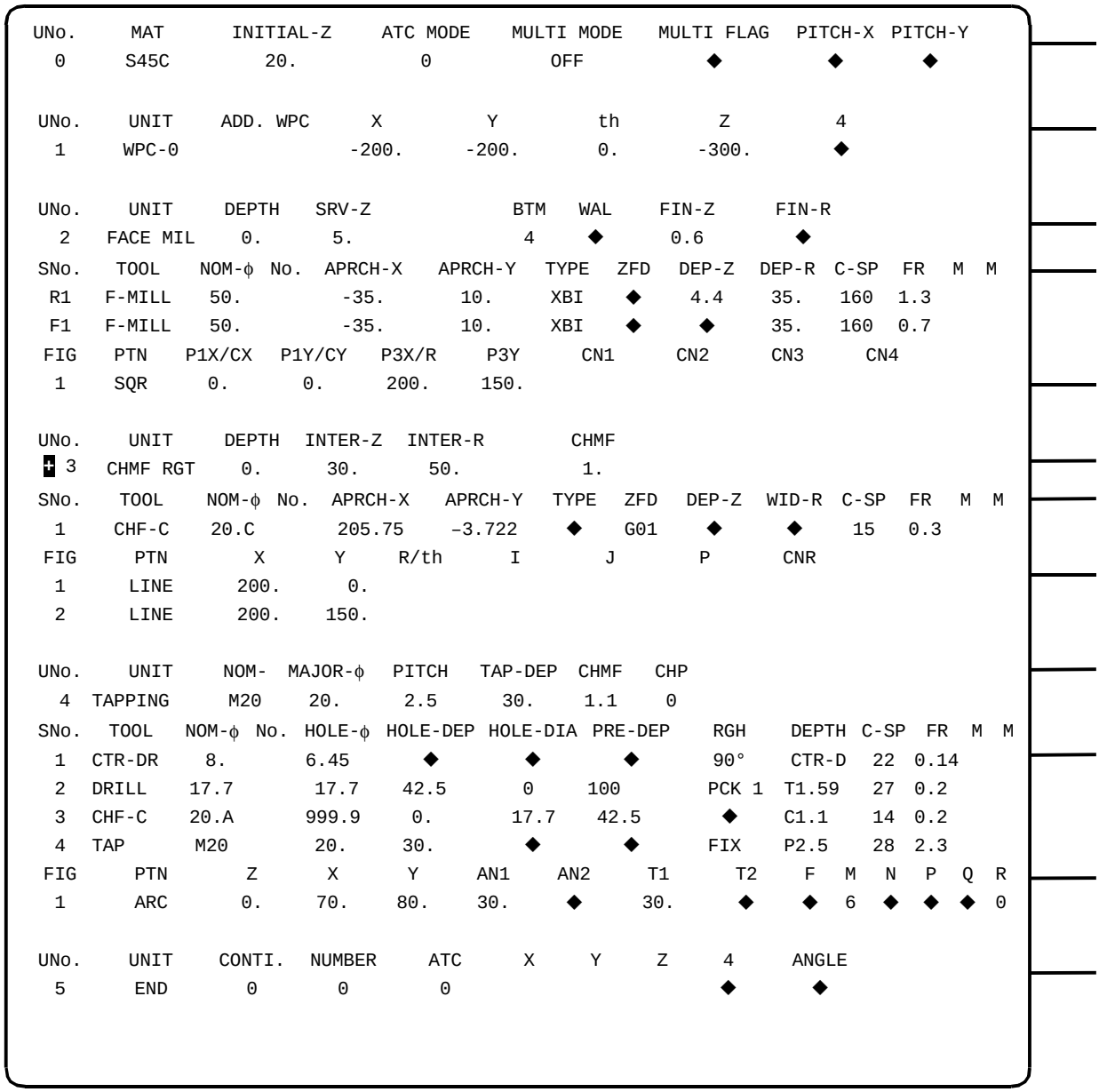
B. Long-term check item

G10L70P_( )

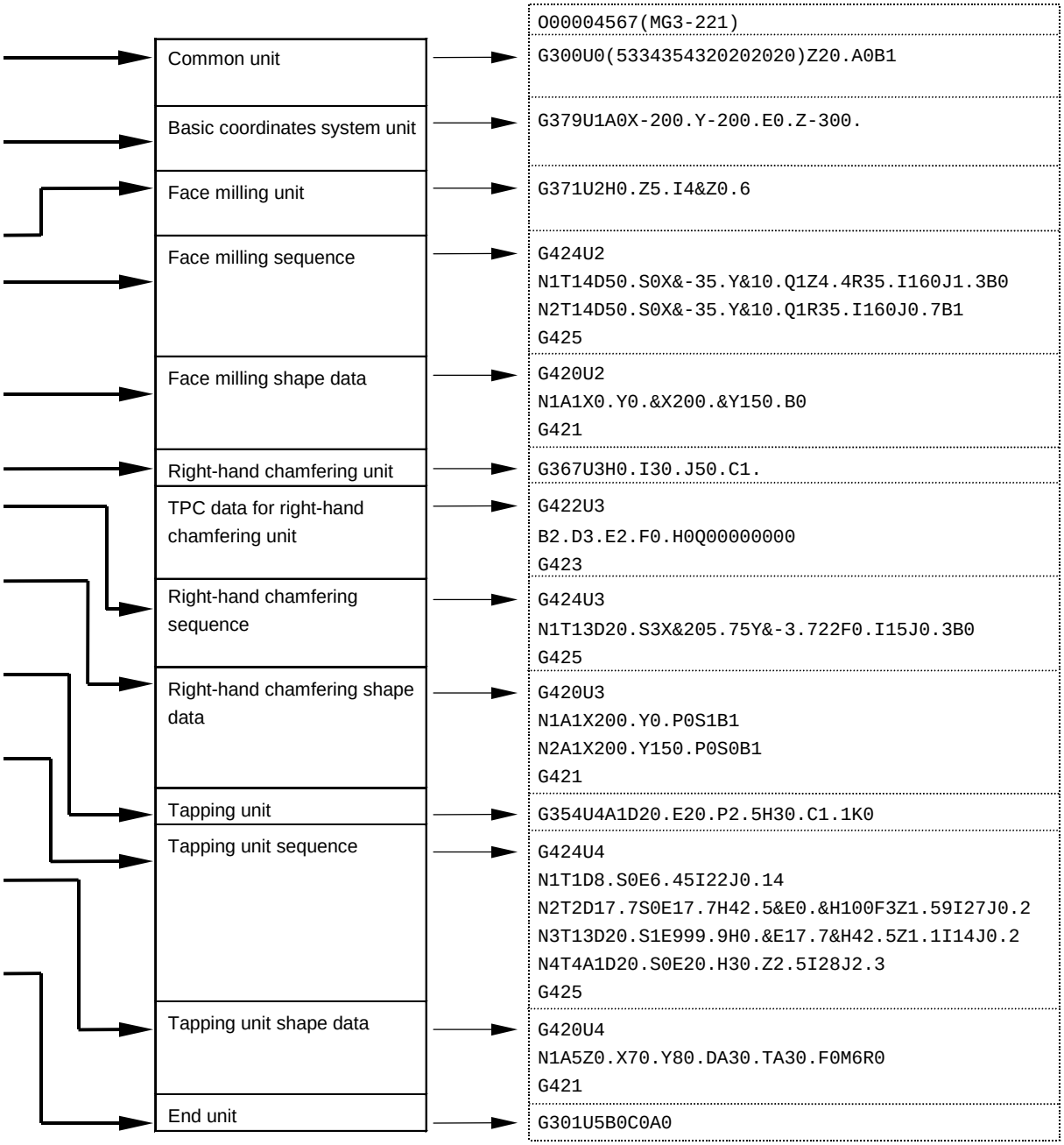
Check No.	Check item
P	( )

- NOTE -

Example of output in use of 3-digit G-format



```
UNo.    MAT      INITIAL-Z    ATC MODE    MULTI MODE    MULTI FLAG   PITCH-X  PITCH-Y
  0     S45C        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   FACE MIL    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   F-MILL    50.         -35.       10.     XBI     ◆     4.4     35.    160   1.3
 F1   F-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    CHF-C    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   CHF-C    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                                  ◆       ◆
```


O00004567(MG3-221)
Common unit
G300U0(5334354320202020)Z20.A0B1
Basic coordinates system unit
G379U1A0X-200.Y-200.E0.Z-300.
Face milling unit
G371U2H0.Z5.I4&Z0.6
Face milling sequence
G424U2
N1T14D50.S0X&-35.Y&10.Q1Z4.4R35.I160J1.3B0
N2T14D50.S0X&-35.Y&10.Q1R35.I160J0.7B1
G425
Face milling shape data
G420U2
N1A1X0.Y0.&X200.&Y150.B0
G421
Right-hand chamfering unit
G367U3H0.I30.J50.C1.
TPC data for right-hand chamfering unit
G422U3
B2.D3.E2.F0.H0Q00000000
G423
Right-hand chamfering sequence
G424U3
N1T13D20.S3X&205.75Y&-3.722F0.I15J0.3B0
G425
Right-hand chamfering shape data
G420U3
N1A1X200.Y0.P0S1B1
N2A1X200.Y150.P0S0B1
G421
Tapping unit
G354U4A1D20.E20.P2.5H30.C1.1K0
Tapping unit sequence
G424U4
N1T1D8.S0E6.45I22J0.14
N2T2D17.7S0E17.7H42.5&E0.&H100F3Z1.59I27J0.2
N3T13D20.S1E999.9H0.&E17.7&H42.5Z1.1I14J0.2
N4T4A1D20.S0E20.H30.Z2.5I28J2.3
G425
Tapping unit shape data
G420U4
N1A5Z0.X70.Y80.DA30.TA30.F0M6R0
G421
End unit
G301U5B0C0A0

- NOTE -

APPENDIX

CONTENTS

Page

APPENDIX-1 LIST OF M CODES 1

APPENDIX-2 PROGRAM EXAMPLES.....4

APPENDIX-3 WHAT TO DO IN SUCH A CASE? 8

- NOTE -

APPENDIX

APPENDIX-1 LIST OF M CODES

The following list shows the general M codes used in the machining centers. It must be noted however, that certain codes can not be used in certain machines and that other codes can be ordered. For details, refer to the Operating Manual for the machine.

M code	Function
0	Programmed stop
1	Optional stop
2	End of program (EIA/ISO)
3	Rotation of spindle (to the right)
4	Rotation of spindle (to the left)
5	Stop of the spindle
6	Changing of tool (EIA/ISO)
7	Mist spray during operation
8	Liquid spray during operation
9	Stop of all spray and compressed air
10	Clamping of tool in the spindle
11	Unclamping of tool in the spindle
15	Magazine cover closed
16	Magazine cover open
19	Orientation of spindle
23	Error detection valid
24	Error detection invalid
30	End of program and rewinding of strip (EIA/ISO)
33	Tool length measurement unit feed
34	Tool length measurement unit withdrawal
35	Detection of tool rupture
36	Selection of spindle speed range (Low)
37	Selection of spindle speed range (Low/Medium low)
38	Selection of spindle speed range (Low/Medium/Medium high)
39	Selection of spindle speed range (High)
40	Selection of spindle speed range (Neutral)
42	Inverted rotation of indexing table
43	External command of M code 3
44	External command of M code 4
45	External command of M code 5
48	Spindle speed correction — Valid
49	Spindle speed correction — Invalid

M code	Function
50	Air blast in operation
51	Spraying crossing the spindle in operation
52	Spraying for tapping in operation
58	Check of tool life
64	Closing of pallet door
65	Opening of pallet door
68	Clamping of pallet
69	Unclamping of pallet
70	Pallet unload
71	Selection of pallet No. 1
72	Selection of pallet No. 2
73	Selection of pallet No. 3
74	Selection of pallet No. 4
75	Selection of pallet No. 5
76	Selection of pallet No. 6
90	Cancellation of symmetrical image (MAZATROL)
91	Symmetrical image WPC-X — Valid (MAZATROL)
92	Symmetrical image WPC-Y — Valid (MAZATROL)
93	Symmetrical image WPC-4 — Valid (MAZATROL)
98	Call-up of subprogram (EIA/ISO)
99	End of subprogram (EIA/ISO)
100	External command of M code 1
101	External command of M code 2
122	Play take-up device — Valid
123	Play take-up device — Invalid
130	Niagara spraying in operation
132	Compressed air crossing the spindle in operation

EIA/ISO: valid in EIA/ISO program

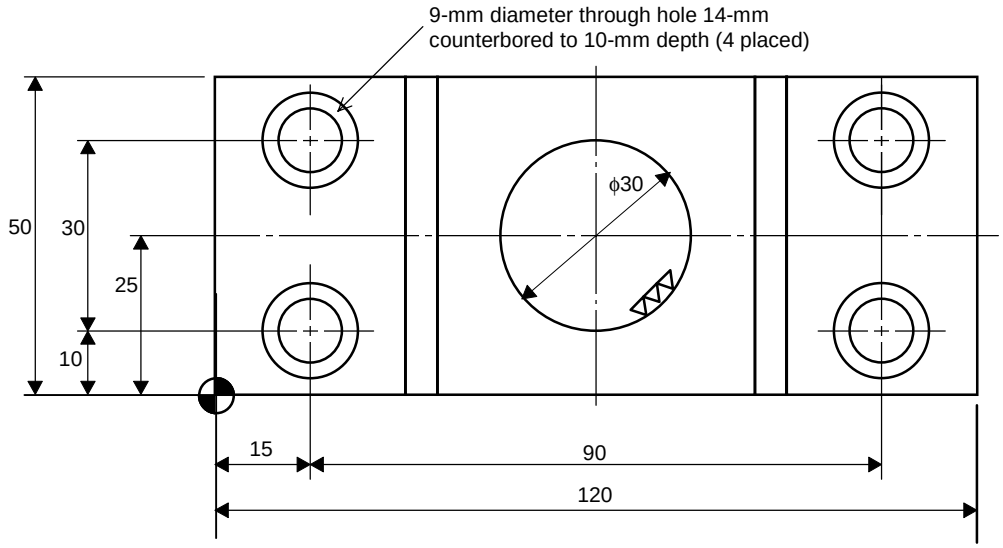
MAZATROL: valid in MAZATROL program

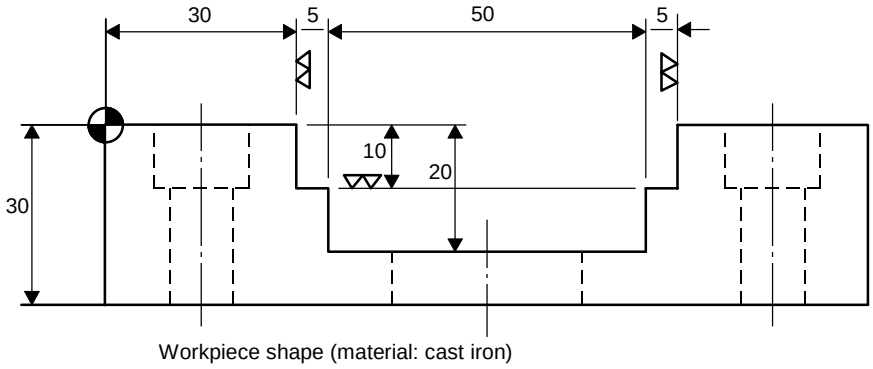
- NOTE -

APPENDIX-2 PROGRAM EXAMPLES

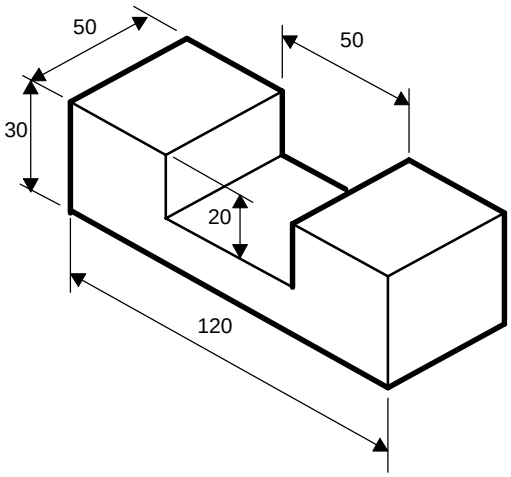
Program examples are given below.

Example 1:





Workpiece shape (material: cast iron)

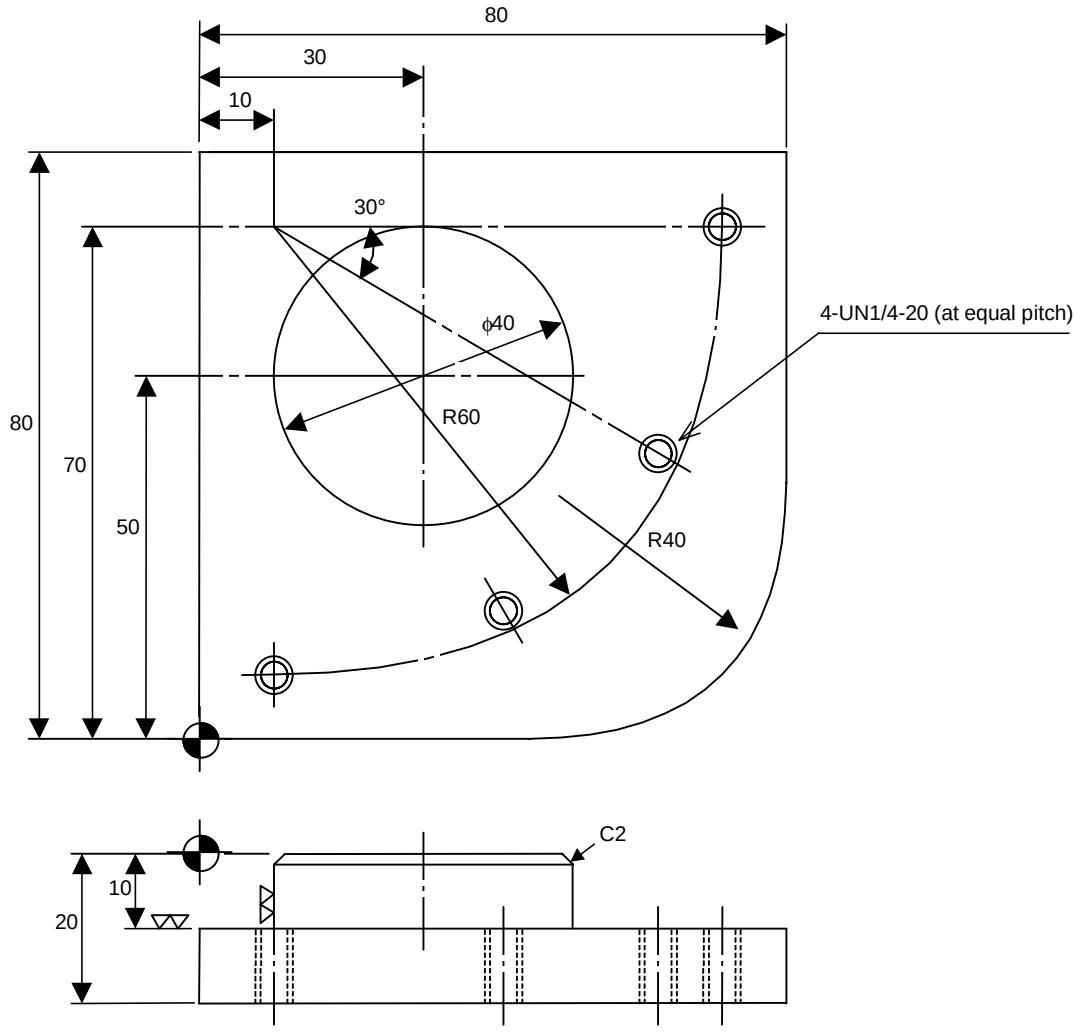


TOOL FILE display

No.	TOOL	NOM-ϕ	MAT.	DEPTH	No.
1	E-MILL	10. A	CBD	30.	2

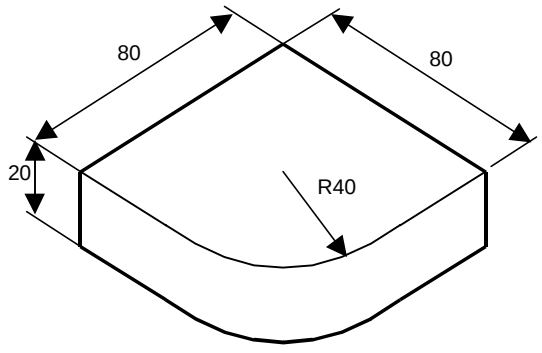
M3P381

```
      MAT     INITIAL-Z    ACT MODE     MULTI MODE      MULTI FLAG    PITCH-X  PITCH-Y
  0  CST IRN      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   E-MIL    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     E-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      E-MIL       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
```



Workpiece shape (material: alminium)

TOOL FILE display



No.	TOOL	ACT-ϕ	MAT.	DEPTH	No.
1	E-MILL	20. A	CBD	30.	2

No.	TOOL	NOM-ϕ	MIN-ϕ	MAT	No.	ANG
1	CHF-C	20. A	5.	HSS	2	45.

M3P382

UNo.	MAT	INITIAL-Z	ACT MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	ALUMINUM	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	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	E-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	CHF-C	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						

APPENDIX-3 WHAT TO DO IN SUCH A CASE?

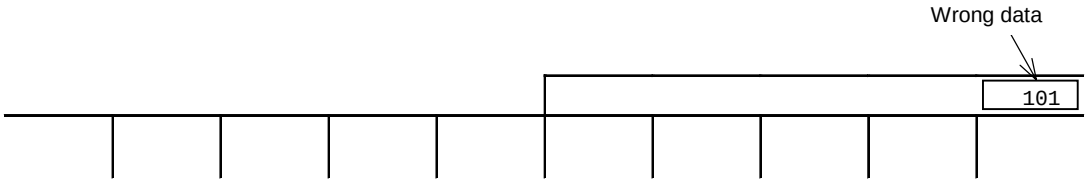
If an erroneous key operation is carried out during creation of a program, you may be placed at a loss what to do. Consult this Appendix in such cases; it offers a description of how to act in case of trouble.

Here, the following states of trouble are covered:

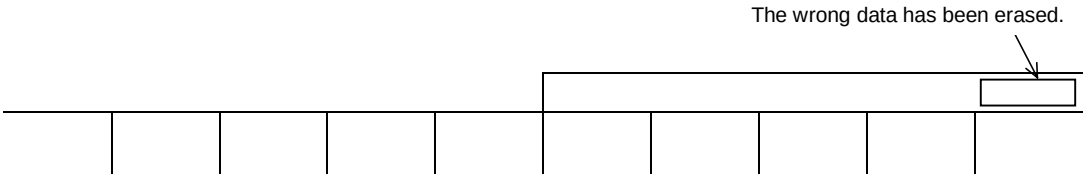
1. A wrong numeric key has been pressed.
2. Wrong data has been input (data change).
3. Wrong data has been input (data erasure).
4. The display selector key or the menu selector key has been pressed inadvertently.
5. How to add a unit(s).
6. How to erase unit(s).
7. How to add a tool sequence(s).
8. How to erase a tool sequence(s).

1. Wrong numeric key has been pressed.

⇨ Operate as follows:



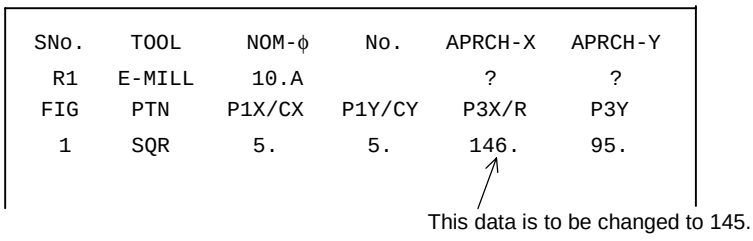
(1) Press the clear key to erase the data in the data selection area .



(2) Press the correct numeric key(s).

2. Wrong data has been input (data change).

⇨ Operate as follows:



(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 1 4 5 → in this order.

```
SNo.    TOOL      NOM-ϕ     No.    APRCH-X   APRCH-Y
 R1    E-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 CANCEL.

```
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.”

- NOTE -