FANUC MANUAL GUIDE *i*

OPERATOR'S MANUAL

B-63874EN/05

SAFETY PRECAUTIONS

When using a machine equipped with the FANUC MANUAL GUIDE *i*, be sure to observe the following safety precautions.

1.1 DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to the degree of the risk or the severity of damage.
Also, supplementary information is described as Note.
Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

⚠ WARNING
Applied when there is a danger of the user being injured or when there is a damage of both the user being injured and the equipment being damaged if the approved procedure is not observed.

⚠ CAUTION
Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE
The Note is used to indicate supplementary information other than Warning and Caution.

- Read this manual carefully, and store it in a safe place.

1.2 GENERAL WARNINGS AND CAUTIONS

To ensure safety while using a machine featuring the MANUAL GUIDE *i* function, observe the following precautions:

⚠ WARNING

1 Confirm, on the screen, that the data has been entered correctly before proceeding to the next operation. Attempting operation with incorrect data may cause the tool to strike the workpiece or machine, possibly breaking the tool or machine or injuring the operator.
2 Before starting the machine using the tool compensation function, carefully determine the direction of compensation and the compensation value, and ensure that the tool will not strike the workpiece or machine. Otherwise the tool or machine may be damaged or the operator may be injured.
3 When using constant surface speed control, set the maximum rotating speed of the spindle to a value that is allowed for the workpiece and workpiece hold unit. Otherwise, the workpiece or hold unit may be removed by centrifugal force to damage the tool machine or injure the operator.
4 Set all necessary parameters and data items before starting MANUAL GUIDE *i* operations. Note that if the cutting conditions are not suitable for the workpiece, the tool may be damaged or the operator may be injured.
5 After creating a machining program using MANUAL GUIDE *i* functions, do no run the machine on that program immediately. Instead, confirm every step of the resultant program, and make sure that the tool path and machining operation are correct and that the tool will not strike the workpiece or machine. Before starting production machining, run the machine with no workpiece attached to the machine to make sure that the tool will not strike a workpiece or the machine. If the tool strikes the machine and/or work-piece, the tool and/or machine may be damaged, and even injuries the operator.

⚠ CAUTION

After pressing the power-on button, do not touch any keys on the keyboard until the initial screen appears. Some keys are used for maintenance or special operations such that pressing such a key may cause an unexpected operation.

TABLE OF CONTENTS

II. OPERATION

I. GENERAL

1 OVERVIEW OF THIS MANUAL

This manual describes the functions of "MANUAL GUIDE *i*" for the Series 16*i*/18*i*/21*i*-MODEL B or Series 30*i*-MODEL A and the MANUAL GUIDE *i* simulator for the personal computer.

For other functions, refer to the operator's manual for the Series 16*i*/18*i*/21*i*-MODEL B or Series 30*i*-MODEL A.
The specifications and usage of MANUAL GUIDE *i* may vary according to the specifications of the operator's panel of a machine tool. Be sure to read the manual provided by the machine tool builder.

The functions of the CNC machine tool system are determined not only by the CNC, but by the combination of the machine tool, the power magnetic circuit in the machine tool, the servo system, the CNC, and the operator's panel.

It is impossible to cover all possible combinations of all functions, programming methods, and operations in a single manual.

This manual explains only the MANUAL GUIDE *i* operations provided for the CNC. For individual CNC machine tools, refer to applicable manuals from the machine tool builders.

This manual explains as many detailed functions as possible. However, it is not possible to describe all of the items which cannot be done or which the operator must not do. Therefore, please assume that functions other than those described in this manual cannot be performed.

Detailed information and special conditions are explained in notes. The readers may encounter new technical terms in the notes not previously defined or described. In this case, read this manual through first, then review the details.

NOTE
MANUAL GUIDE *i* has options for supporting milling and turning cycles, animated simulation, and set-up guidance. This manual covers these functions assuming that your machine has all these functions. However, your machine may not actually be provided with some of these functions.
Refer to the manual published by the machine tool builder of your machine for details.

2 READ AT FIRST

In this chapter, you will find the explanation of the place where you should refer to when you operate MANUAL GUIDE *i*.
When trying to use a machine equipped with the FANUC MANUAL GUIDE *i*, be sure to observe the safety precautions written in this manual with sufficient caution.

What is MANUAL GUIDE *i* ?

MANUAL GUIDE *i* is the operation guidance to support an operator on many situations such as creating a part program, checking by machining simulation, set-up and actual machining. These operations can be done on the only one screen.
ISO code form is adopted as the part program format, it is widely used on many CNC machine tools, furthermore, machining cycles, which can realize complicated machining motions by simple programming, are provided.
As to the outline of MANUAL GUIDE *i*, refer to the following part.

- Features
  II 1.1 MAIN FEATURES OF MANUAL GUIDE *i*
- Displaying screens
  I 3. ALL-IN-ONE SCREEN

What kind of machines can be supported?

MANUAL GUIDE *i* can be used for lathe, machining center and compound machine including the following machine configurations.

- Lathe
  From 1-path to 3-path lathe, which includes Y-axis and tool inclining axis.
- Machining center
  Machining center including workpiece rotating table and tool inclining axis.
- Compound machine
  Compound machine with capability of both lathe machining and milling.

MANUAL GUIDE *i* has operation ability common to the above machines. So, you can operate on same screen commonly on the above machine excepting machining cycles and tool offset data setting which are used for each machine exclusively.

How to install MANUAL GUIDE *i* to CNC

In ordinary case, MANAUL GUIDE *i* is installed in an CNC and prepared by MTB such as parameter setting. In that case, you can use MANUAL GUIDE *i* as it is.
However, by some reasons, there is a case such like you must install MANUAL GUIDE *i* into CNC and set necessary parameter by yourself.
In order to support you in that case, parameter setting and set-up guidance explanations are prepared.

- Explanations for the operations for installing MANUAL GUIDE *i* into CNC newly.
- Appendix A. PARAMETER
- Appendix C. MANUAL GUIDE *i* SETUP METHOD

How to begin to operate MANUAL GUIDE *i*

In order to display the screen of MANUAL GUIDE *i*, press some function button on the MDI panel of CNC. As to which function button should be used, refer to the manual made by MTB because it depends on the parameter No.14794 and 14795 setting.

NOTE
MANUAL GUIDE *i* operations cannot be done during background editing mode of CNC program screen.

How to create a part program

In order to create a part program on MANUAL GUIDE *i*, refer to the following part,

- General operations flow of programming
  VI. EXAMPLE OF PROGRAMMING OPERATION:
  For lathe and machining center respectively, series of operations from setting up such as offset data to creating a part program and its checking are explained.
- Basic operations of part program editing
  II 3. EDITING MACHINING PROGRAMS:
  Various editing operations, which are used for the part program in form of ISO code, are explained.
- Details of machining cycles
  II 4. EDITING CYCLE MACHINING OPERATIONS
  II 5. DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES:
  Operations of various machining cycles, which can be used for realizing complicated machining, are explained.

In the ISO code form part program, similar programs are used frequently such as toll changing and spindle rotation. MANUAL GUIDE *i* supports those frequently used operations by the fixed form

sentence function. Fixed form sentences, programming template, are made in advance and can be used by selecting from the menu during programming operations. In to details, refer to the following part.

- Making and using of fixed form sentence
  II 3.14. FIXED FORM SENTENCE INSERTION
  II 10.3. REGISTERING FIXED FORM SENTENCES

How to check the created part program

As to the operations for checking the created part program on MANUAL GUIDE *i*, refer to the following part.

- Checking the part program
  II 9. OPERATIONS IN MACHINING SIMULATION:
  Operations of checking the part program by using machining simulation, animation or tool path drawing, are explained. And also, checking the tool path drawing of actual motions during machining is explained.

How to set the data needed for machining

In order to execute actual machining, tool offset data and workpiece coordinate data setting must be done in advance. These data are prepared as CNC standard function, but these data can be entered by using MANUAL GUIDE *i* too.

- Setting of tool offset data and workpiece coordinate data
  II 10.1 SETTING THE WORKPIECE COORDINATE DATA
  II 10.2 SETTING TOOL OFFSET DATA:
  Convenient operations for workpiece coordinate data and tool offset data setting are explained.

In order to execute animation or machining cycle on MANUAL GUIDE *i*, tool data necessary for these functions such as tool type must be defined in advance. As for necessary operations, refer to the following part.

- Setting of data necessary for animation and machining cycles.
  II 13 TOOL DATA BASE FUNCTION:
  Tool type, attaching direction, cutting edge and nose angle for lathe and so on are set as tool data for each offset number. Operation for tool data base is explained.

How to edit other part program during machining

By using a background editing function, editing of other part program during machining can be done.
As for background editing operation, refer to the following part.

- Operation of background editing
  II 11 BACKGROUND EDITING

How to modify part of cycle machining motions

While cycle machining which can be used in MANUAL GUIDE *i* can create the actual machining motions automatically by using entered cycle data, modifying part of the created machining motions cannot be done.
However, there is a case that part of created machining motions should be modified for more effective machining.
In this case, converting the cycle machining block to standard ISO code NC part program, motion blocks which are made up with G00/G01/G02/G03 and so on, is available.
The converted motion blocks are corresponded to each machining motion, so by modifying part of converted blocks, the actual machining motion can be changed.

NOTE
Hole machining and threading cycles are converted to canned cycles, so the detailed motions executed in a canned cycle cannot be changed.

As for NC program conversion operation, refer to the following part.

- Operations of NC program conversion
  II 12 NC PROGRAM CONVERSION FUNCTION

How to conserve part program and tool data

In order to use part program and tool data again later, these data can be conserved by outputting to memory card.
As for operations of outputting and inputting these data to and from memory card, refer to the following part.

- Outputting and inputting of part program and tool data
  II 17 MEMORY CARD INPUT/OUTPUT FUNCTION:
  In addition to the operations of outputting and inputting of part program and tool data, screen hard copy operations for MANUAL GUIDE *i* is explained.

How to operate more quickly

Almost all operations of MANUAL GUIDE *i* use soft-key excepting numeric data entering. However, if you are inured to MANUAL GUIDE *i* operations, you may feel you can operate more quickly if you use other key instead of soft-key one by one. This substitute key operation is called short cut key.

NOTE
On the small key board, short cut key cannot be used.

As for short cut key operation, refer to the following part.

- Details of short cut key operation
  II 15 SHORTCUT KEY OPERATIONS
- Displaying explanation of short cut key operation
  II 16 HELP SCREEN:
  Pressing HELP key on the MDI panel displays the window of HELP screen for short cut key operations.

How to calculate the entering data

In order to calculate the data before entering, pocket calculator type calculation function can be used.
As for the operations, refer to the following part.

- Details of operations for calculating data
  II 19 CALCULATOR FUNCTION

How are other operations of MANUAL GUIDE *i* ?

In addition to creating part program, many convenient functions, which can be used for operating a machine tool, are prepared on MANUAL GUIDE *i*.
As for those operations, refer to the following part.

- Other operations than creating part program
  II 6. OPERATIONS IN THE MEM MODE
  II 7. OPERATIONS IN THE MDI MODE
  II 8. OPERATIONS IN THE MANUAL MODE (HANDLE AND JOG):
  Operations for each mode are explained.

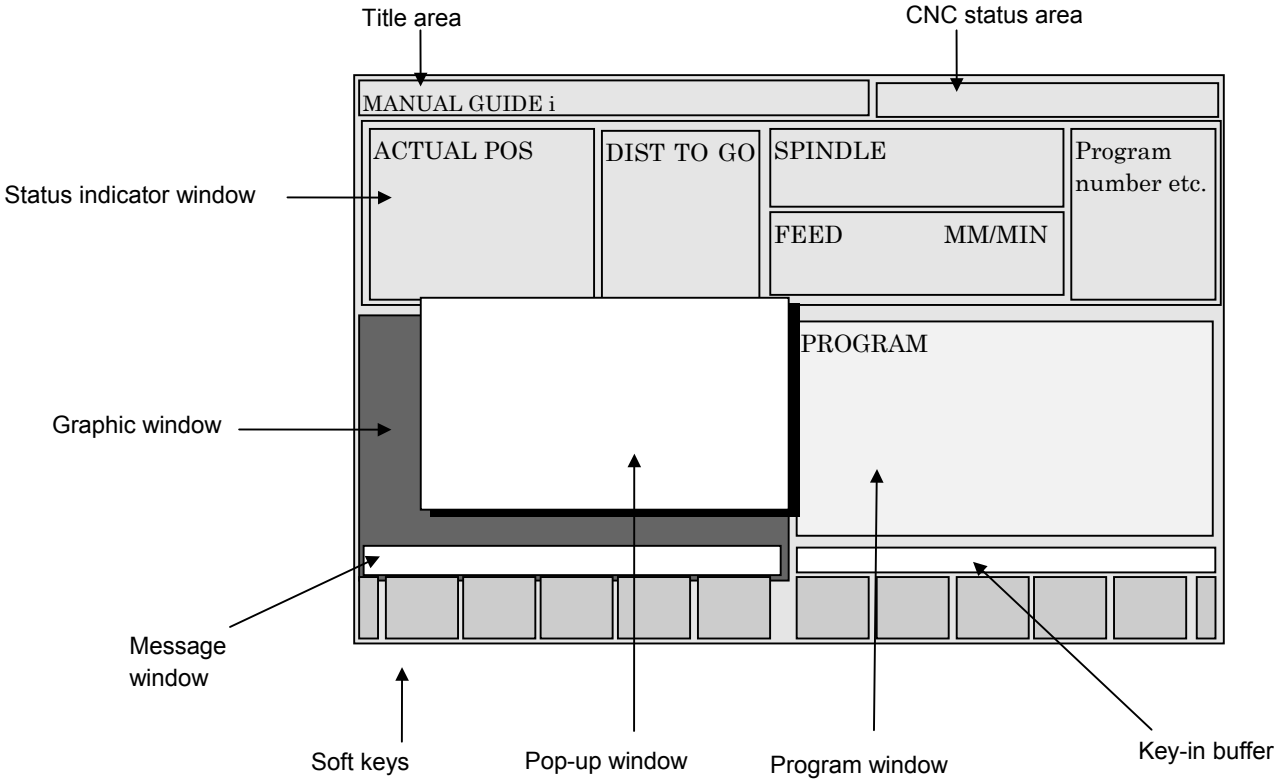
What kind of machining cycles can be used ?

On MANUAL GUIDE *i*, turning, lathe machining, and milling machining cycles are prepared as optional function.
As for details of machining cycle types and entering data, refer to the following part.

- Details of machining cycles
  III 1 MILLING
  III 2 TURNING

3 ALL-IN-ONE SCREEN

In MANUAL GUIDE *i*, basically, only one screen called the All-in-one Screen is used for all the operations from trial machining to actual machining.



Title area:

The title of MANUAL GUIDE *i* is always displayed.

CNC status area:

The following CNC statuses are displayed.

- Mode
- Alarm status
- Reset or emergency stop status
- Actual time

Status indicator window:

The following information about CNC is displayed.

- Actual machine position
- Remaining moving distance of the actual block

- Actual speed and load meter (for the axis with the maximum load)
  Remark) It is possible to display Actual speed in Feed per revolution. (Refer to the parameter No.14703#0.)
- Spindle rotating speed and spindle load meter
- Program number and process number
- Command values during automatic operation (M,S,T,F)

Graphic window:

The following graphical drawing is displayed as needed.

- Animated drawing with a solid model (machining simulation)
- Tool path drawing

Program window:

The machining program is displayed.

Pop-up window:

The following supplemental screens are displayed as needed.

- M-code menu
- Fixed format statement menu
- Set data (workpiece coordinate system, tool offset, fixed format statement registration)
- Program list
- Data input window for machining cycles

Key-in buffer:

Comments on data and input numerical data are displayed as needed.

Message window:

The following messages are displayed as needed.

- Meaning of the word which a cursor is allocated (Guidance Message)
  Remark) It is possible to display two Guidance Message of M/T mode in the CNC for compound lathe. (Refer to the parameter No.14703#2.)
- Meaning of Machine Cycle which is executing
- Content of Warning and Alarm

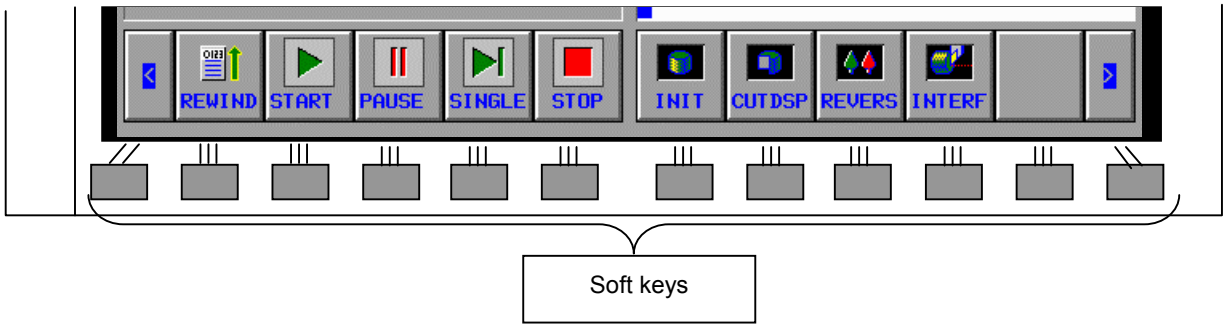
Soft keys :

The comments on the following soft-keys are displayed as needed.

- Editing operation menu
- Machining cycle menu
- Pop-up window menu

Remark)
The soft keys described in this manual are specified to 12 keys placed under the screen, LCD, as shown in the following example.
The meaning of each soft key is various by the displayed content on the screen, and will be displayed on the relevant part for each key.

Example of soft keys)



4 SYMBOLS USED

In this manual, the following conventions are used for keys.

(1) Function buttons are indicated in bold type:
Example) **PROGRM**, **OFSET**

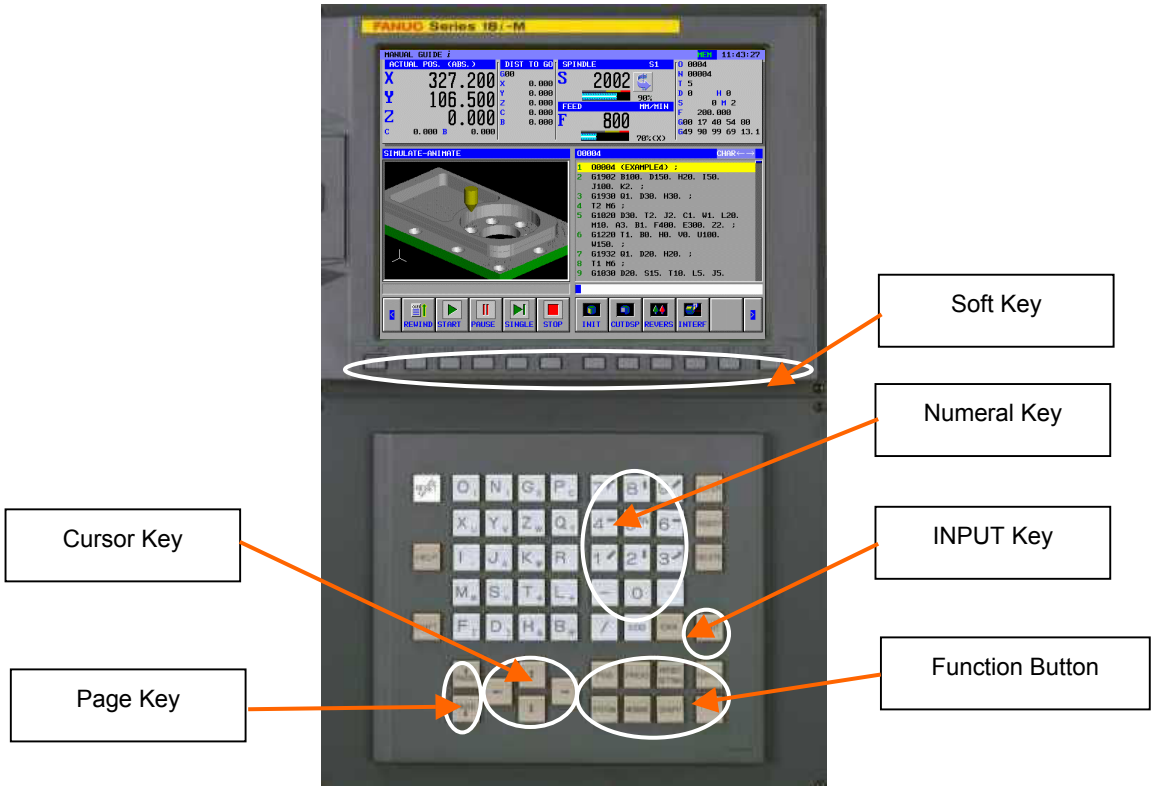
(2) The numbers to be input by numerical keys are underlined.
Example) 12.345

(3) The input key is indicated in bold type in the same way as the function buttons.
Example) 12.345 **INPUT**

(4) Soft-keys are enclosed in brackets [ ]:
Example) [LIST], [LINE]

(5) The cursor keys are indicated by the following symbols :
Example) ↑, ↓, ←, →

(6) The page keys are indicated by the following symbols :
Example) ⇩, ⇧



5 NOTES ON CREATING PROGRAMS

The notes that should be observed when creating a program are described below. Read the notes before creating a program.

1. General notes on machining programs
   <1> Use ISO code form (G code commands) basically for a machining program. Enter a command for simple motion directly with a G code, and use cycle machining for complicated machining.
   <2> Enter directly a tool change command involving the calling of the next tool by using a code such as a T code and M code or by using a subprogram call prepared by the machine tool builder, referring to the relevant operator's manual of the machine tool builder.
   <3> Enter directly a tool length compensation command used with a machining center by using a G code such as G43.
   <4> For changing tools or tool length compensation, a similar command is repeatedly entered. So, such a command can be entered easily by registering a fixed form sentence beforehand.

2. Programmed commands required before cycle machining input
   <1> When using a tool database with a CNC of the M series, be sure to enter a D command (offset number).
   <2> Be sure to enter a spindle rotation command. At the same time, specify whether to enable or disable constant surface speed control.
   <3> No feedrate command needs to be specified, because a feedrate command is separately entered as a cycle machining command. However, specify a command for feed per revolution or feed per minute as required. For example, operator know-how on executing milling in pocketing by feed per revolution can be utilized.
   <4> Enter an M code command for switching between the spindle and C-axis, and a command for C-axis reference position return as required.
   <5> Enter M codes such as for coolant ON/OFF as required.
   <6> By setting bit 2 of parameter No. 27000 to 1, a polar coordinate interpolation command (G12.1) required for machining (polar coordinate interpolation) on the XC plane can be automatically specified during cycle machining. Upon completion of the cycle machining, polar coordinate interpolation is automatically cancelled.

<7> By setting bit 3 of parameter No. 27000 to 1, a cylindrical interpolation command (G07.1) required for machining (cylindrical interpolation) on the ZC plane can be automatically specified during cycle machining. Upon completion of the cycle machining, cylindrical interpolation is automatically cancelled.

<8> Upon completion of cycle machining, the tool always returns to the position before the start of the cycle machining. By setting bit 7 of parameter No. 27002 to 1, the tool can be placed at a position different from the position before the start of the cycle machining.

<9> When using G code system B or C with a CNC of the T series (T mode for a CNC for complex machining) and specifying cycle machining (turning or milling), be sure to switch to the absolute coordinate system by specifying G90 beforehand.

<10>For the cycle machining mentioned above, a similar command is repeatedly entered. So, such a command can be entered easily by registering a fixed form sentence beforehand.

3. Cycle machining

<1> With a CNC of the T series as well, enter a coordinate value for milling (such as an I point coordinate in milling-based hole machining) as a radius value even if the value is an X coordinate.

<2> Specify an I point coordinate not as a distance from point R but as a coordinate value.

<3> Figure data should be entered as a subprogram for utilization in roughing, finishing, and chamfering. This eliminates the need to enter figure data each time.

<4> In machining of a projected portion (island figure), efficient machining can be achieved by using pocketing with an island instead of using contouring. In this case, specify a blank as the outer wall of a pocket.

<5> In contouring (side facing), enter a start point figure and end point figure so that both figures contact each other. This means that by starting not from a corner of a figure but from an intermediate point on a straight line, undercutting due to approaching the figure and retraction can be eliminated.

<6> If the C-axis operates as the hole machining axis in an attempt to execute hole machining by using the C-axis as the positioning axis, the C-axis may be set as an axis parallel to the X-axis with parameter No. 1022. In such a case, set the parameters below to execute hole machining in the axis direction normal to the plane including the C-axis.

When the parameter No. 5101#0 is set to 1 :

The hole machining axis in a hole machining canned cycle is the third axis for plane specification.

When the parameter No. 5103#2 is set to 1 :

If an axis normal to or parallel with a specified plane is specified during a hole machining canned cycle, the specification is regarded as a positioning command.

<7> With a CNC of the T series as well, enter a coordinate value for milling (such as an I point coordinate in milling-based hole machining) as a radius value even if the value is an X coordinate.

4. Machining program entered on a multipath lathe

<1> When creating a machining program for a multipath lathe, create necessary processes on the process list screen before creating machining operations. This allows the user to view the entire machining program more easily.

5. Animated simulation

<1> hen performing machining simulation of a program including a three-dimensional coordinate conversion command, increase the dry run feedrate (for example, by selecting the rapid traverse rate) to speed up the simulation.

<2> If a tool post is placed on the lower side when animated simulation is performed on a multipath lathe, the machining performed on the side opposite to the blank can be viewed during animated simulation by pressing the [REVERS] soft key.

6 MANUAL GUIDE *i* SIMULATOR FOR THE PERSONAL COMPUTER

This chapter describes the MANUAL GUIDE *i* simulator for the personal computer.

NOTE
The specifications of the MANUAL GUIDE *i* simulator for the personal computer are subject to change as a result of product improvement.

6.1 OPERATING ENVIRONMENT

6.1.1 Product Components

- CD-ROM disk
  MANUAL GUIDE *i* simulator software for the personal computer

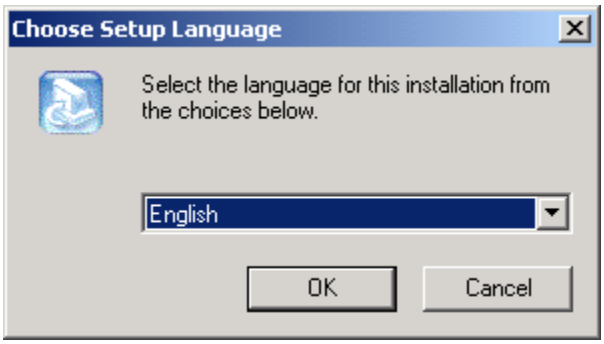
- Hardware protection key

6.1.2 Operating Environment

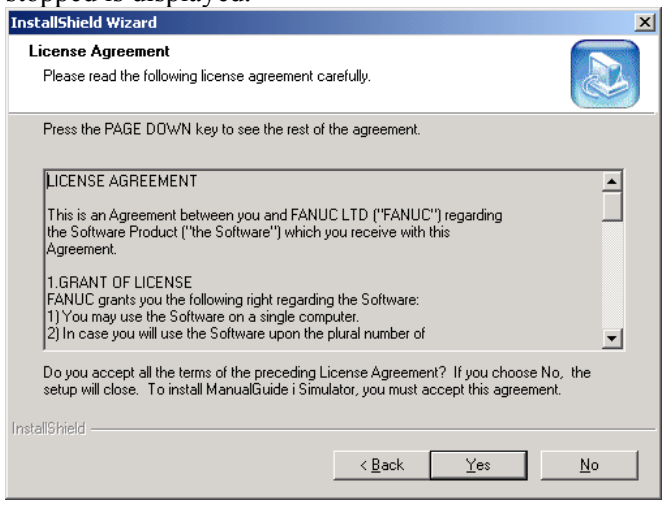
- Main computer unit
  - PC/AT-compatible machine on which Windows 2000/XP Professional can run
  - Japanese or English must be usable on the OS used.
  - Pentium(R) III, 1 GHz or higher
  - Memory: 256 MB or more
  - Hard disk: A free space of 150 MB is required.
  - A USB1.1 port must be provided.
  - An Ethernet port must be provided (not required in the case of package licensing).
  - For simulation using a 10.4-inch screen with no MDI buttons, XGA (1024×786) or a higher resolution is required.
  - For simulation using a 10.4-inch screen with MDI buttons, SXGA (1280×1024) or a higher resolution is required.
  - For simulation using a 15-inch screen with no MDI buttons, UXGA (1600×1200) or a higher resolution is required.
  - For simulation using a 15-inch screen with MDI buttons, QXGA (2048×1536) or a higher resolution is required.

6.2 METHOD OF INSTALLATION

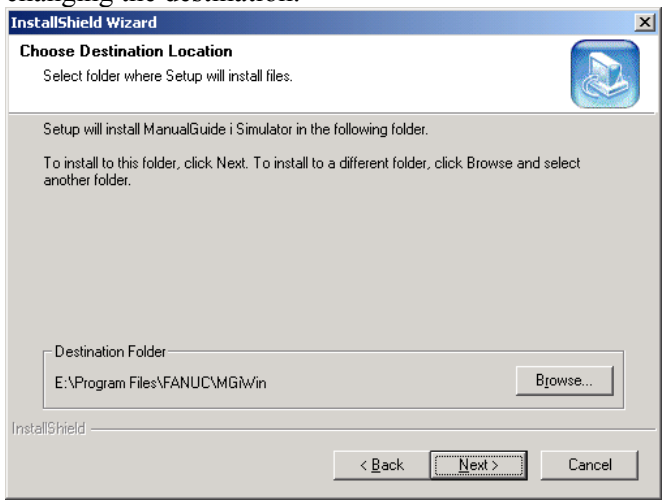
For installation, the administrator authority for the computer is required.
Insert the CD-ROM of the MANUAL GUIDE *i* simulator for the personal computer into the CD-ROM drive. Execute "SetUp.exe" on the CD-ROM drive to start the installer.
When the installer is started, a dialog box for selecting a language to be used for setup is displayed. Select a language to be used for setup from the list then click [OK].



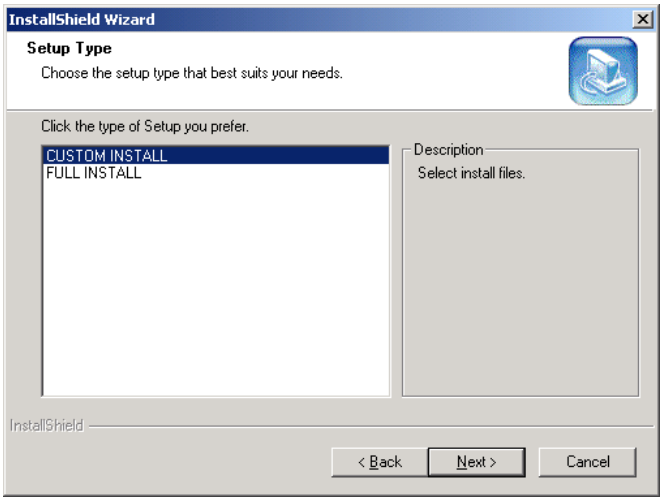
During installation, the system prompts you to agree upon the license agreement for using this software. If you agree, click [Yes]. If you select [No], a dialog box for checking if the installation may be stopped is displayed.



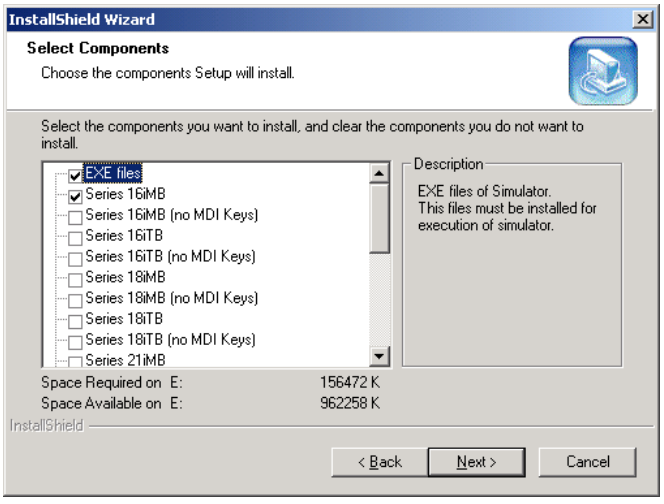
In the dialog box for installation destination selection, the installation destination can be changed to a desired destination from the standard one. Change the installation destination or click [Next] without changing the destination.



In the dialog box for setup type selection, you can select full installation or custom installation. When full installation is selected, a free space of about 700 MB is required. For custom installation, the required free space varies according to the components to be installed.



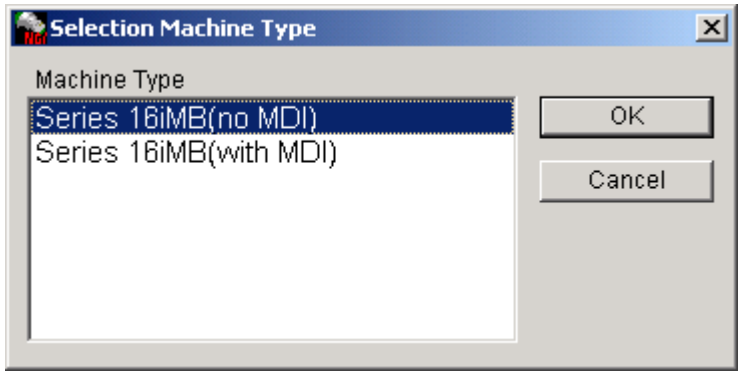
When custom installation is selected, a dialog box for selecting components to be installed is displayed. Check each component to be installed then click [Next]. This dialog box is displayed also when a component is added or deleted after installation. In this case, check a component to be added, and uncheck a component to be uninstalled.



Upon completion of installation, attach the hardware key to the USB port and execute the MANUAL GUIDE *i* simulator.
Do not attach the hardware key to the USB port before completion of software installation.

6.3 SIMULATION CNC SELECTION

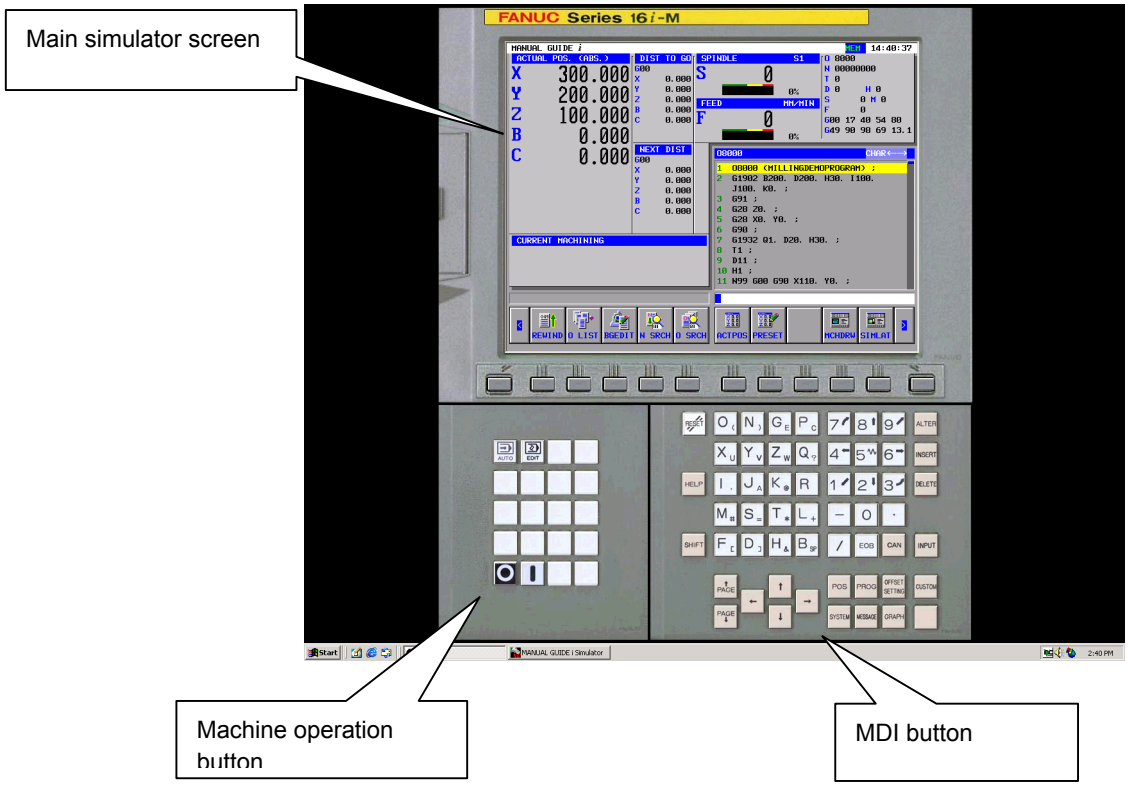
Start the MANUAL GUIDE *i* simulator by choosing [Start] menu → [Programs] → [FANUC] → [ManualGuide i Simulator]. When the MANUAL GUIDE *i* simulator is started, a dialog box for selecting a simulator definition file used to execute the MANUAL GUIDE *i* simulator is displayed. In this dialog box, select a definition to be simulated then click the [OK] button to start the MANUAL GUIDE *i* simulator.



The simulator definition file is a text file where information such as CNC model and display unit size information is written in a specified format. By editing this file, a simulator environment with a user-specific configuration can be defined.

6.4 FULL-SCREEN DISPLAY

When the MANUAL GUIDE *i* simulator is started, the simulator is displayed on the full screen of the personal computer.



With the MANUAL GUIDE *i* simulator, the main simulator screen, the MDI button screen operable using the mouse, and the machine operation button screen are displayed. A soft key operation can be performed by clicking a soft key on the main simulator screen or by pressing a key from F1 to F12 on the keyboard of the personal computer. Numeric values and uppercase letters can be entered through the MDI buttons or the keyboard of the personal computer. For CNC-specific key input, click MDI buttons or machine operation buttons with the mouse. To quit the simulator, click the machine operation button [O].

6.5 PARAMETER

1 Parameters created with the FS16*i*/18*i*/21*i* cannot be input.
2 When using a parameter of the FS16*i*/18*i*/21*i*, convert the parameter to the FS30*i* format.
3 Parameters in the FS30*i* format can be used without modification.
4 The parameters related to the MANUAL GUIDE i function are basically compatible.
5 Parameter data cannot be output.
6 For the method of parameter input, refer to "Standard_param¥Readme_j.txt" stored on the CR-ROM.
7 The standard parameters are stored on the CD-ROM. By editing the file with the text editor and reading the edited file, the simulator can be customized.

6.6 RESTRICTIONS ON USE

1 Simulation using full-screen display
The MANUAL GUIDE *i* simulator is executed using the full screen of the personal computer.

2 Memory card input/output function
A) For input to/output from the memory card, use the memcard folder under the installation folder. The following can be input/output using the memory card input/output function:
- NC programs
- Parameters
- Tool database

B) The file name of the file in the memcard folder should include the extension and be 12 characters or less.

6.7 NOTES

1 The operation and functions listed below described in the operator's manual cannot be used with the MANUAL GUIDE *i* simulator for the personal computer.

- Operation in the MDI mode
- Operation in a manual mode (handle, jog)
- Multi-path lathe function
- Tool management function
- Setup support function

2 The MANUAL GUIDE *i* simulator for the personal computer has a functional compatibility with the FS30*i*, but has some specification restrictions with the FS16*i*/18*i*/21*i*.

3 Those functions that input external signals cannot be used.

4 This does not correspond to the hyper threading technology function.

6.8 SIMULATOR DEFINITION FILE FORMAT

The simulator definition file is a text file where information such as CNC model and display unit size information is written in a specified format. The ini file format of Windows is used.

6.8.1 Comment

A comment can be written only in a continuous area at the top of a file.
A comment must start with a single quotation mark (').
A single quotation mark (') written at the start of a halfway data line does not comment out the character string that follows.

6.8.2 Section

The following two sections are used:
Simulator_MachineSetting_MaxNumber
Simulator_MachineSettingn
(The character n represents a number from 0 to maxnumber of Simulator_MachineSetting_MaxNumber.)

The character n in the "Simulator_MachineSettingn" section need not be successive, but must not be duplicate. If n is duplicate, the setting closer to the top of the file is selected. If a suffix greater than the maxnumber key is assigned to a setting, the setting is not processed.

6.8.3 Key

The following keys are used:
maxnumber
name
cnctype
displaytype
pathtype
cnctype_title
from_dat_filepath
fileall_dat_filepath
srammgi_dat_filepath
mgidef_folderpath
mgidef_machine_type
userdef_filepath1
userdef_filepath2
keyini_filepath
virtual_memcard_folderpath

6.8.3.1 [Simulator_MachineSetting_MaxNumber] section

Key name : maxnumber
Outline : Code a maximum subscript value for the simulator definitions to be found in the file.
Character string to be set : Maximum subscript number to be found
Explanation : Code a maximum subscript value for the simulator definitions in the simulator definition file. A search for a simulator definition is made until n of the Simulator_MachineSettingn section reaches maxnumber. A found simulator definition is displayed in a selection dialog box.

6.8.3.2 [Simulator_MachineSetting*n*] section

Key name : name
Outline : Code the name assigned to a simulator definition.
Character string to be set : Arbitrary
Explanation : Code the name assigned to a simulator definition. The character string coded here is displayed in the simulator definition selection dialog box displayed when the MGi manager is started.

Key name : cnctype
Outline : Code a CNC model.
Character string to be set : "FS16i", "FS18i", "FS21i", "FS30i"
Explanation : Code the model of the CNC. If an attempt to read a file described below such as a From.dat file fails, the model coded in the MGi manager ini file is used.

Key name : displaytype
Outline : Code a display unit type.
Character string to be set : "10.4", "15"
Explanation : Code the type of the display unit. At present, specify "10.4" or "15".

Key name : pathtype
Outline : Code a path type.
Character string to be set : "T", "TT", "TTT", "M", "TM", etc.
Explanation : Code a path type. Specify "T" for T series single-path control. Specify "TT" for T series two-path control. Specify "M" for the M series. Specify "TM" for compound control.

Key name : cnctype_title
Outline : Set a CNC model name for display.
Character string to be set : Arbitrary
Explanation : Set a CNC model name for display.

Key name : from_dat_filepath
Outline : Code the relative path of From.dat corresponding to a selected simulator definition.
Character string to be set : Relative path of From.dat
Explanation : Specify the relative path of From.dat corresponding to a selected simulator definition. The file coded here is copied under the name "From.dat" to the memcard directory under the simulator execution directory.

Key name : fileall_dat_filepath
Outline : Code the Relative path of file_all.dat corresponding to a selected simulator definition.
Character string to be set : Relative path of file_all.dat
Explanation : Specify the relative path of file_all.dat corresponding to a selected simulator definition. The file coded here is copied under the name "file_all.dat" to the simulator execution directory.

Key name : srammgi_dat_filepath
Outline : Code the relative path of sram_mgi.dat corresponding to a selected simulator definition.
Character string to be set : Relative path of sram_mgi.dat
Explanation : Specify the relative path of sram_mgi.dat corresponding to a selected simulator definition. The file coded here is copied under the name "sram_mgi.dat" to the simulator execution directory.

Key name : mgidef_folderpath
Outline : Code the relative path of the folder storing the MANUAL GUIDE definition file of a selected simulator definition.
Character string to be set :
Relative path of the folder storing a MANUAL GUIDE definition file (ending with "¥").
Explanation : Code the relative path of the folder storing the MANUAL GUIDE definition file of a selected simulator definition. From the folder coded here, a MANUAL GUIDE definition file matching the type of the definition file to be specified next is copied together with the language file to the simulator execution directory.

Key name : mgidef_machine_type
Outline : Code the type of a MANUAL GUIDE definition file.
Character string to be set :
"TURN_VERTICAL", "TURN_HORIZONTAL", "MILLING_VERTICAL", "MILLING_HORIZONTAL"
Explanation : Code the type (T/M series, vertical/horizontal) of a MANUAL GUIDE definition file. The file coded here is copied to the simulator execution directory.

Key name : userdef_folderpath1
Outline : Code the relative path of the user definition file of a selected simulator definition.
Character string to be set : Relative path of a user definition file
Explanation : Code the relative path of the user definition file of a selected simulator definition. The file coded here is copied under the name "cex2datu" to the simulator execution directory. Up to two user definition files can be specified. If no user definition file is necessary, specify a null character string.

Key name : userdef_folderpath2
Outline : Code the relative path of the user definition file of a selected simulator definition.
Character string to be set : Relative path of a user definition file
Explanation : Code the relative path of the user definition file of a selected simulator definition. The file coded here is copied under the name "cex3datu" to the simulator execution directory. Up to two user definition files can be specified. If no user definition file is necessary, specify a null character string.

Key name : keyini_filepath
Outline : Code the relative path of an MDI key setting file.
Character string to be set : Relative path of an MDI key setting file
Explanation : Code the relative path of a setting file specifying the layout of MDI keys, and so forth. By using information such as character strings for input key determination and button display positions stored in the file specified here, the MGi manager displays the MDI keys on the MGi manager.

6.9 DISPLAY DATA ini FILE FORMAT

The display data ini file is a text file where information about images and buttons to be arranged on the screen is specified. The ini file format of Windows is used.

6.9.1 Comment

A comment can be written only in a continuous area at the top of a file.
A comment must start with a single quotation mark (').
A single quotation mark (') written at the start of a halfway data line does not comment out the character string that follows.

6.9.2 Section

The following sections are used:
[settings]
[frame_mainscreen]
[cnctitle]
[softkey]
[frame_mdikey]
[mdikey]
[frame_functionkey]
[functionkey]

[settings]	: Specifies general screen layout information.
[frame_mainscreen]	: Specifies information such as screen component positions on the CNC display section. [cnctitle] and [softkey] below are included in this frame.
[cnctitle]	: Setting for CNC model title section display
[softkey]	: Setting for soft key display
[frame_mdikey]	: Frame display setting for MDI key display
[mdikey]	: Setting for MDI key display
[frame_functionkey]	: Setting for display of keys such as the mode switch key other than the MDI keys
[functionkey]	: Setting for display of keys such as the mode switch key

6.9.3 Key

The keys are described on a section-by-section basis.

6.9.3.1 [settings] section

Key name : bgcolor
Outline : Specify a background color.
Setting method : bgcolor = *r*, *g*, *b*
Specify a number from 0 to 255 in *r*, *g*, and *b*.
Example: Grey is specified as the background color.
bgcolor = 128,128,128
Details : Specify a background color by the lightness values of red (r), green (g), and blue (b). A lightness value ranges from 0 to 255. If a value greater than 255 is specified, the specification of 255 is assumed. The background color specified here is used as the background color for the overall screens and frames.

Key name : cncscrn_offsetx, cncscrn_offsety
Outline : Specify the display position of a CNC display section application.
Setting method : cncscrn_offsetx = *x*
: cncscrn_offsety = *y*
Each of *x* and *y* represents the number of pixels from the top-left corner of the screen.
Example: CNCScrn.exe is displayed at (320,60) from the top-left corner of the screen.
cncscrn_offsetx=320
cncscrn_offsety=60
Details : Specify the position of a CNC display section application in pixels. Specify (x,y) in pixels, with (x,y) = (0,0) for the top-left corner of the screen. When an NC frame image is displayed, for example, specify (x,y) to display the CNC display section at a proper position.

Key name : layout
Outline : Code the layout of MDI keys.
Setting method : layout = QWERTY | ONG-M | ONG-T | MINI-M | MINI-T
Example: The QWERTY layout is selected for the MDI keys.
layout=QWERTY
Details : Set the layout of MDI keys by using a specified character string. Here, only a layout name is specified. The actual layout of keys is not determined here.

6.9.3.2 [frame_mainscreen] section

Key name : image
Outline : Specify the path of an image to be displayed in the CNC display section area of the MGi manager.
Setting method : image = *drive:¥dir1・・・¥filename*
Example: image=.¥image¥frameFS30i.bmp
Details : Specify the path of a frame image to be displayed around the CNC display section. When no fame is to be displayed, delete the coding of the key itself. Either an absolute path or a relative path may be used. The usable types of image files are bmp, jpg, and gif. The size of an image must match the size of the frame. Those portions that exceed the frame are not displayed. Those portions that do not reach the frame size are filled using the color specified by bgcolor.

Key name : visible
Outline : Specify whether to display the main frame.
Setting method : visible = 0 | 1
Example: When the main frame is displayed
visible=1
Details : Specify whether to display the main frame. When 0 is specified, the main frame is not displayed. When 1 is specified, the main frame is displayed. The main frame includes the NC title and soft keys. So, when 0 is specified not to display the main frame, the CNC display section is displayed, but the title and soft keys are not displayed.

Key name : left, top, width, height
Outline : Specify the display position and size of the main frame.
Setting method : left = left
top = top
width = width
height = height
Example: When the position of the main frame is (230,20), and the size of the main frame is (800,600)
left=230
top=20
width=800
height=600
Details : Specify the position and size of the main frame in pixels. For display position specification, assume that the top-left corner of the screen is at (left,top) = (0,0). When an image is to be pasted, the size must match the size of the image.

6.9.3.3 [cnctitle] section

Key name : image
Outline : Specify the path of a title image to be displayed within the main frame.
Setting method : image = *drive:¥dir1・・・¥filename*
Example: image=.¥image¥titleFS30i.bmp
Details : Specify the path of the file storing a title image to be displayed. Either an absolute path or a relative path may be used. When the image key itself is deleted, no image is displayed, but the character string of the title specified in the simulator definition is displayed as text. The usable types of image files are bmp, jpg, and gif. The size of an image must match the size of the frame. Those portions that exceed the frame are not displayed. Those portions that do not reach the frame size are filled using the color specified by bgcolor.

Key name : visible
Outline : Specify whether to display the title section.
Setting method : visible = 0 | 1
Example: When the title section is displayed
visible=1
Details : Specify whether to display the title. When 0 is specified, the title is not displayed. When 1 is specified, the title is displayed. Specify 0, for example, when an image pasted to the main frame already includes a title, and no title needs to be drawn additionally.

Key name : left, top, width, height
Outline : Specify the display position and size of the title section.
Setting method : left = *left*
top = *top*
width = *width*
height = *height*
Example: When the display position of the title section is (540,180), and the size of the title section is (7000,400)
left=540
top=180
width=7000
height=400
Details : Specify the display position and size of the title section in twip. Specify a desired display position as relative coordinates on the main frame, with (left,top) = (0,0) for the top-left corner of the main frame. When an image is to be pasted, the size must match the size of the image.

6.9.3.4 [softkey] section

Key name : keynum
Outline : Specify the number of buttons that are displayed in the main frame and operate as soft keys.
Setting method : keynum = n
Example: When 12 soft keys are used
keynum=12
Details : Specify the number of buttons that are displayed in the main frame and used as function keys. The number of buttons specified here is used to specify the number of key pressing information items to be read next.

Key name : key*n* (*n*: 0 to keynum-1)
Outline : Code a setting to be assigned to a button used as a key.
Setting method : key*n* = *key1*, *key2*, *x*, *y*, *imagefile*
Example: When the first soft key is assigned to the operation of the F1 key, the display position is (900,8220), and an image to be pasted is specified
key0={F1},{F1},900,8220,.¥image¥skey_l.bmp
Details : Code information for setting of a button to be assigned as a soft key. The following information is to be set:

- key1: Specify key information (to be described later) to be passed to the CNC display section application when a button is pressed.
- key2: Specify key information to be passed to the CNC display section application when a button is pressed while the SHIFT button is held down.
- x, y: Specify the display position of a button in twip. For display position setting, specify relative coordinates on the main frame, with (x,y) = (0,0) for the top-left corner of the main frame.
- Imagefile: Specify the path of the image file to be pasted to a button. The size of an image for a soft key is 28×28 pixels when the CNC model specified in the simulator definition is FS30i, and is 33×18 pixels for other cases.

6.9.3.5 [frame_mdikey] section

Key name : image
Outline : Specify the path of an image to be displayed as a frame for MDI key display.
Setting method : image = *drive:¥dir1・・¥filename*
Example: image=.¥image¥frameQWERTY.bmp
Details : Specify the path of a frame image to be displayed around the MDI keys. When no frame is to be displayed, delete the coding of the key itself. Either an absolute path or a relative path may be used. The usable types of image files are bmp, jpg, and gif. The size of an image must match the size of the frame. Those portions that exceed the frame are not displayed. Those portions that do not reach the frame size are filled using the color specified by bgcolor.

Key name : visible
Outline : Specify whether to display a frame for MDI key display.
Setting method : visible = 0 | 1
Example: When a frame for MDI key display is displayed
visible=1
Details : Specify whether to display a frame for MDI key display. When 0 is specified, no frame for MDI display is displayed. When 1 is specified, a frame for MDI key display is displayed. When 0 is specified, the MDI keys are not displayed.

Key name : left, top, width, height
Outline : Specify the display position and size of a frame for MDI key display.
Setting method : left = *left*
top = *top*
width = *width*
height = *height*
Example: When the display position of a frame for MDI key display is (230,20), and the size is (800,600)
left=230
top=20
width=800
height=600
Details : Specify the display position and size of a frame for MDI key display in pixels. For display position specification, assume that the top-left corner of the screen is at (left,top) = (0,0). When an image is to be pasted, the size must match the size of the image.

6.9.3.6 [mdikey] section

Key name : keynum
Outline : Specify the number of buttons that are displayed in a frame for MDI key display and operate as MDI keys.
Setting method : keynum = *n*
Example: When 66 MDI keys are used
keynum=66
Details : Specify the number of buttons that are displayed in a frame for MDI key display and operate as MDI keys. The number of buttons specified here is used to specify the number of key pressing information items to be read next.

Key name : key*n* (*n*: 0 to keynum-1)
Outline : Code a setting to be assigned to a button used as a key.
Setting method : key*n* = *key1*, *key2*, *x* , *y*, *imagefile*
Example: When the first MDI key is assigned to the operation of "O", the display position is (6020,1350), and an image to be pasted is specified
key0=O,{(),6020,1350,.¥image¥o.bmp
Details : Code information for setting of a button to be assigned as an MDI key. The following information is to be set:

key1 Specify key information (to be described later) to be passed to the CNC display section application when a button is pressed.

key2 Specify key information to be passed to the CNC display section application when a button is pressed while the SHIFT button is held down.

x, y Specify the display position of a button in twip. For display position setting, specify relative coordinates on the frame for MDI key display, with (x,y) = (0,0) for the top-left corner of the frame for MDI key display.

Imagefile Specify the path of the image file to be pasted to a button. The size of an image for a soft key is 36×36 pixels.

6.9.3.7 [frame_functionkey] section

Key name : image
Outline : Specify the path of an image to be displayed as a frame for function key display.
Setting method : image = *drive:¥dir1・・¥filename*
Example: image=.¥image¥frameFunc.bmp
Details : Specify the path of a frame image to be displayed around the function keys. When no frame is to be displayed, delete the coding of the key itself. Either an absolute path or a relative path may be used. The usable types of image files are bmp, jpg, and gif. The size of an image must match the size of the frame. Those portions that exceed the frame are not displayed. Those portions that do not reach the frame size are filled using the color specified by bgcolor.

Key name : visible
Outline : Specify whether to display a frame for function key display.
Setting method : visible = 0 | 1
Example: When a frame for function key display is displayed
visible=1
Details : Specify whether to display a frame for function key display. When 0 is specified, no frame for function display is displayed. When 1 is specified, a frame for function key display is displayed. When 0 is specified, the function keys are not displayed.

Key name : left, top, width, height
Outline : Specify the display position and size of a frame for function key display.
Setting method : left = *left*
top = *top*
width = *width*
height = *height*
Example: When the display position of a frame for function key display is (230,20), and the size is (800,600)
left=230
top=20
width=800
height=600
Details : Specify the display position and size of a frame for function key display in pixels. For display position specification, assume that the top-left corner of the screen is at (left,top) = (0,0). When an image is to be pasted, the size must match the size of the image.

6.9.3.8 [functionkey] section

Key name : keynum
Outline : Specify the number of buttons that are displayed in a frame for function key display and operate as function keys.
Setting method : keynum = *n*
Example: When two function keys are used
keynum=2
Details : Specify the number of buttons that are displayed in a frame for function key display and operate as function keys. The number of buttons specified here is used to specify the number of key pressing information items to be read next.

Key name : key*n* (*n*: 0 to keynum-1)
Outline : Code a setting to be assigned to a button used as a key.
Setting method : key*n* = *key1*, *key2*, *x* , *y*, *imagefile*
Example: When the first function key is assigned to the operation of the "auto" key, the display position is (450,660), and an image to be pasted is specified
key0=^r,^r,450,60,.¥image¥auto.bmp
Details : Code information for setting of a button to be assigned as a function key. The following information is to be set:

key1 Specify key information (to be described later) to be passed to the CNC display section application when a button is pressed.

key2 Specify key information to be passed to the CNC display section application when a button is pressed while the SHIFT button is held down.

x, y Specify the display position of a button in twip. For display position setting, specify relative coordinates on the frame for function key display, with (x,y) = (0,0) for the top-left corner of the frame for function key display.

Imagefile Specify the path of the image file to be pasted to a button. The size of an image for a soft key is 36×36 pixels.

6.9.3.9 Information to be passed when a button is pressed

When specifying buttons such as soft keys, MDI keys, and function keys in a display data ini file, specify key information to be passed to the CNC display section application when the buttons are pressed . The format for key specification is described below.

The format for button setting is as follows:
Setting method : key*n* = *key1*, *key2*, *x* , *y*, *imagefile*

To assign the pressing of "a" when the key*n* button is pressed, code "a" in key1. To input "b" when the key is pressed while the SHIFT key is held down, code "b" in key2.
Example: key*n* = a, b, *x* , *y*, *imagefile*

* A character coded here is input to the CNC display section application. So, if a character not supported by the CNC display section application is coded, an unpredictable operation results.
The table below indicates the CNC keys and corresponding character strings. In an ini file, code the character strings corresponding to CNC keys to be operated.

CNC key	Corresponding character string	CNC key	Corresponding character string
A	a	1	1
B	b	2	2
C	c	3	3
D	d	4	4
E	e	5	5
F	f	6	6
G	g	7	7
H	h	8	8
I	i	9	9
J	j	0	0
K	k	-	―
L	l	.	.
M	m	/	/
N	n	(	{(}
O	o	)	{)}
P	p	?	?
Q	q	,	COMMA
R	r	@	(No corresponding character string)
S	s	#	#
T	t	=	=
U	u	*	*
V	v	+	{+}
W	w	[	[
X	x	]	]
Y	y	&	(No corresponding character string)
Z	z	SP	SPACE

CNC key	Corresponding character string	CNC key	Corresponding character string
RESET	@	SOFTKEY 1	Q
HELP	{'}	SOFTKEY 2	A
SHIFT	SHIFT	SOFTKEY 3	Z
ALTER	`	SOFTKEY 4	X
INSERT	^n	SOFTKEY 5	C
DELETE	DEL	SOFTKEY 6	V
INPUT	{ENTER}	SOFTKEY 7	B
PAGEUP	PDUP	SOFTKEY 8	N
PAGEDOWN	PGDN	SOFTKEY 9	M
UP	E	SOFTKEY 10	L
DOWN	D	SOFTKEY 11	R
LEFT	S	SOFTKEY 12	T
RIGHT	F	SOFTKEY 13	Y
MEM MODE	^r	SOFTKEY 14	U
EDIT MODE	^d	SOFTKEY 15	I
EXIT	%{F4}	SOFTKEY 16	O
SOFTKEY L	W	SOFTKEY 17	G
SOFTKEY R	P	SOFTKEY 18	H
		SOFTKEY 19	¥

II. OPERATION

1 OVERVIEW OF THE PROCEDURE

1.1 MAIN FEATURES OF MANUAL GUIDE *i*

By using MANUAL GUIDE *i*, the operator can carry out routine machining easily.

1) Integrated operation screen that enables almost all routine machining operations
A single integrated operation screen enables routine machining operations including machining program input/editing, animated simulation-based machining program checks, production machining, MDI operations, and manual operations with JOG and HANDLE.

2) Machining programs in ISO code format
Using ISO code machining programs, which are in wide use, enables the operator to specify simple operations with simple commands, such as those for straight lines and arcs, and complicated machining operations with machining cycles easily.

3) High affinity with CAD/CAM
ISO code machining programs created using CAD/CAM can be used without modification. Adding advanced machining cycles to these machining programs makes them perfect machining programs. They can be checked easily, using animated simulation.

4) Advanced machining program editing
Using advanced editing functions, such as substring search and cut/paste via the clipboard, enables easy editing of machining programs.

5) Advanced machining using machining cycles (option)
Advanced machining cycles are available which cover various types of machining including milling and turning. With these machining cycles, it is possible to perform complex machining by creating and running programs easily.

6) Fixed format program menu-driven simple program entry
Register a series of frequently used machining operations previously as a menu, and select necessary machining operations from the menu when creating a machining program. This method can eliminate the trouble of entering similar machining operations repeatedly.

7) M code menu
It is possible to input M codes easily by referencing explanations displayed in an M code menu. Machine tool builders can create the explanations easily.

8) Realistic animated simulation (option)

Machining programs can be checked easily, using an animated simulation method that can realistically show what the surface machined with a specific type of tool tip is like. In addition, you can check a simulated workpiece as if you were looking at a real workpiece because the animated simulation method uses solid models for all operations, from milling to turning, for the workpiece.

9) Advanced set-up guidance (option)

It is possible to set up machining operations and check the precision of machined workpieces easily, using an advanced set-up guidance function that can handle all measurements, from tool offset measurement to the measurement of workpieces in machine tools.

10) Supporting a wide variety of machine types including lathes and machining centers

MANUAL GUIDE *i* supports lathes 1 to 3, vertical and horizontal machining centers, and tilting head-equipped machining centers with 3 + 2 axes. It also supports compound machine tools that are capable of turning and milling.

2 MACHINING PROGRAM FORMAT

Machining programs used with MANUAL GUIDE *i* are created using the ISO code format, which is widely used in CNC machine tools. They use 4-digit G code machining and measurement cycles to implement further advanced machining operations.
These 4-digit G code machining and measurement cycles can be input and edited easily, using menu windows in which machining data can be entered interactively.
When a machining program is being edited, an explanation about an address pointed to with the cursor is displayed at the bottom of the screen.

Remark)
"Word" is the minimum data unit of NC part program, and specified by the combination of address (A to Z) and numerical data (0 to 9, +, -, decimal point).
"Address" means the content of numerical data such as moving axis name.

Remark)
"Cursor" of the part program means the part whose background area is displayed by yellow. The part displayed by yellow is called "selected by the cursor", and program editing operation will be dealt on this part.
There are 2 case cases for the selected part by the cursor.

(1) 1 block
When the cursor moved the top of the next block by pressing the cursor key →, the whole word data included in the block are selected.
When the cursor selects the top word data of some block, the whole word data included in the block are selected by pressing the cursor key ←.
In case that the whole word data include in the block are selected by the cursor, the next or previous 1 block will be selected by pressing the cursor key ↓ or ↑.

(2) 1 word
In case that the whole word data in 1 block are selected by the cursor as described above, if you push the cursor key ← or →, the cursor will select 1 word only. By pressing the cursor key ← or → again, the cursor will select the next 1 word.
When the 1 word is selected, the cursor will select the 1 word of same order in the nest or previous block by pressing the cursor key ↑ or ↓.

Remark)

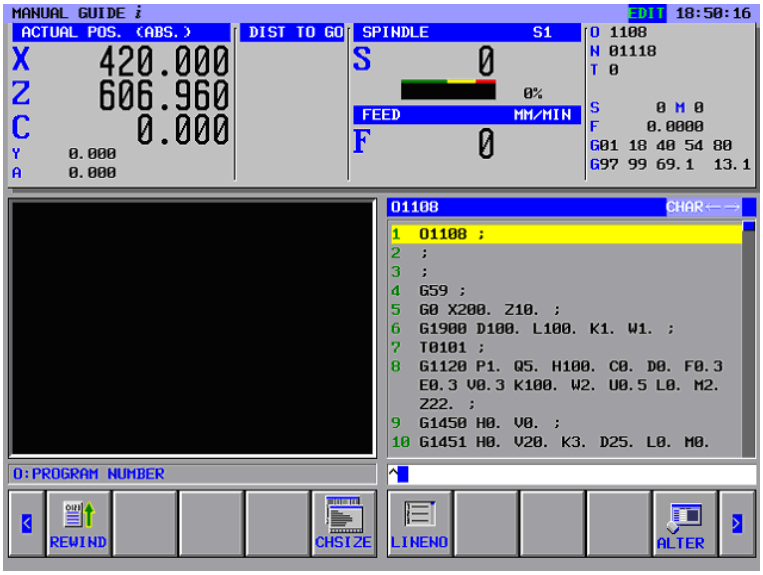
In the screen, in which numerical data are directly entered such as offset data or cycle machining data, the cursor is specified by displaying the data frame by blue.

The part specified by blue frame is called “data item selected by the cursor”, and you can enter the proper data by pressing **INPUT** key after entering numeric data by numeric keys.

By pressing the cursor key ← ↑ → ↓, you can select the next or previous data item.

3 EDITING MACHINING PROGRAMS

3.1 MACHINING PROGRAM WINDOW AND EIDITNG



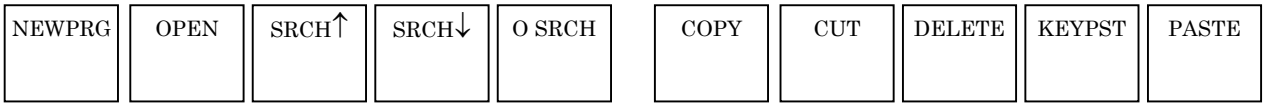
MANUAL GUIDE *i* uses a program window to input and edit machining programs (in ISO code format).
The program window is operated using the following soft keys, which are displayed by pressing the leftmost soft key [<] or rightmost soft key [>] several times.

REWIND				CHSIZE	LINENO				SETING

[REWIND] : This soft key takes you to the beginning of the program.
[CHSIZE] : This soft key zooms the program window to full screen.
[LINENO] : This soft key selects whether to display the line number at the beginning of each block. The line number is not memorized in the machining program.
[SETING] : This soft key opens the setting screen.

The scroll bar displayed on the right edge of the program window indicates the approximate position of the cursor throughout the machining program.
The cursor is moved around in the program window, using the cursor keys ←, ↑, ↓, and →. Pressing → moves the cursor right in a block. To move the cursor to the next block, you need to select the block entirely once and then select an individual address. Pressing. ← has the same effect as for → except for the direction in which the cursor moves.

The program is edited using the following soft keys, which are displayed by pressing the leftmost soft key [<] or rightmost soft key [>] several times.



Remark)

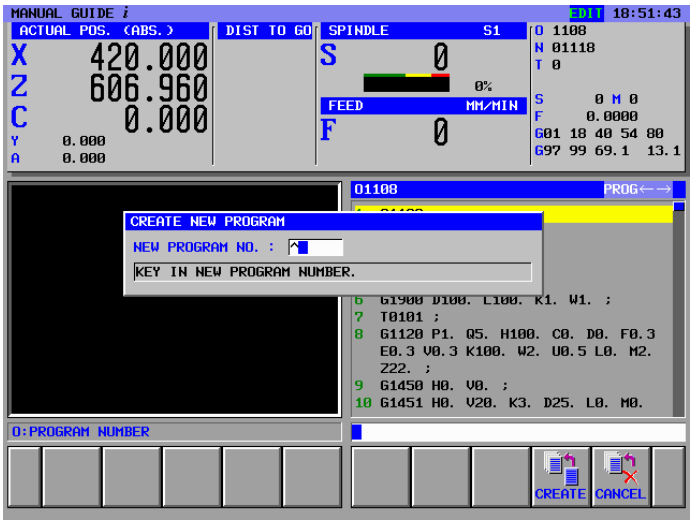
Basically, soft keys are placed in form of 1 line. By pressing the right end key [>], the soft keys shifted to right direction by 10 will appear. By pressing the left end key [<], the left side soft keys shifted by 10 will appear.

When the right end soft keys are displayed, the left end soft keys will appear by pressing [>]. When the left end soft keys are displayed, the right end soft keys will appear by pressing [<].

3.2 CREATING MACHINING PROGRAMS

NEWPRG	O LIST	SRCH↑	SRCH↓	O SRCH		COPY	CUT	DELETE	KEYPST	PASTE

Pressing [NEWPRG] displays the program creation window.



For the program creation window, the following soft keys are displayed.

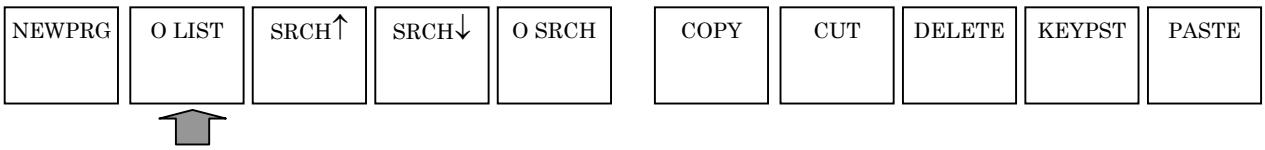
								CREATE	CANCEL

In this window, enter a program number, using numeric keys, and then press [CREATE].
Usually, the program number is 4 digits, but you can use 8 digits program number when the corresponding optional function is added.
Pressing [CREATE], new program with entered program number only is created, and it is displayed in the program window. In that case, the following soft keys for editing operation are displayed.

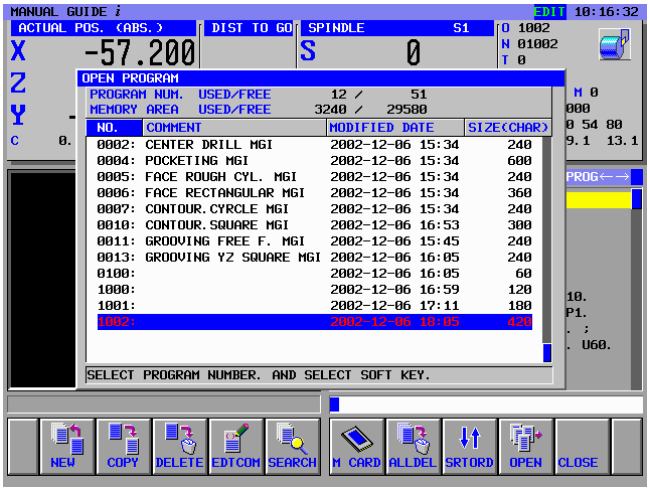
NEWPRG	O LIST	SRCH↑	SRCH↓	O SRCH		COPY	CUT	DELETE	KEYPST	PASTE

In addition to the above operation, you can create new part program by entering new program number after the address “O”, and pressing **INSERT** key.

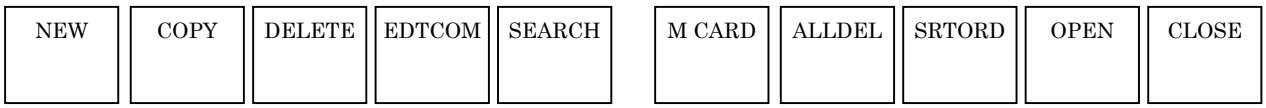
3.3 EDITING IN A PROGRAM LIST



Pressing [O LIST] displays a window that lists registered machining programs.
By pressing the ← or → cursor key, a sort type (sort by number, sort by date and time of updating, or sort by size) can be selected.



The following soft keys are displayed in the window. They can be used for editing such as copying or deleting machining programs.



[NEW] : This soft key displays the program creation window, in which you can create machining programs.

[COPY] : This soft key displays the program copy window. After a program number is entered using numeric keys, pressing [COPY] copies the specified program.

[DELETE] : This soft key displays a message for asking you whether you really want to delete a specified program. Pressing [YES] deletes the specified program. Pressing [NO] cancels the request to delete the specified program.

[EDTCOM] : This soft key displays the comment (program name) edit window. If you move the cursor, using ← or →, and enter a character, using MDI keys, the character appears at the left of the cursor. Using the **CAN** key can delete the character at the left. Pressing [ALTER] causes the comment to be changed.

[SEARCH] : This soft key displays the program search window. After entering a desired program number in the window, using numeric keys, press [SEARCH].

[M CARD] : This soft key enables input/output to and from the memory card.

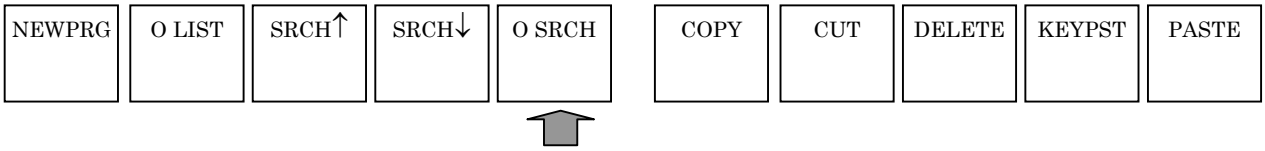
[ALLDEL] : This soft key displays a message for checking if all programs may be deleted. Pressing [YES] deletes all programs. Pressing [NO] cancels the deletion of all programs.

[SRTORD] : This soft key switches the sort order for displaying programs between ascending order and descending order.

[OPEN] : After placing the cursor on a program you want to edit, using the ↑ or ↓ cursor key, pressing this soft key selects the program and closes the list window. Then, the soft keys described at the top of this subchapter are displayed again.

[CLOSE] : This soft key closes the program list window.

3.4 SEARCHING FOR A MACHINING PROGRAM TO BE EDITED



After entering a desired program number, using numeric keys, pressing [O SRCH] can select the program.
Pressing [O SRCH] without entering a program number causes the next program to be selected.
Pressing the O key on the MDI panel, the soft keys changes to the above one, so enter the program number to be searched and press [O SRCH].
And as another way, you can select the program in the program list by pressing [OPEN] after placing the cursor on the program number to be selected.
When the part program is selected, the content of the part program is displayed in the program window, and the above soft keys will be displayed.

3.5 BASIC EDITING OPERATIONS OF PART PROGRAM

Since MANUAL GUIDE *i* uses ISO-code form part program, editing of 1 word, minimum unit of the program and made from address and numerical data, are available by using **INSERT**, **ALTER** and **DELETE** keys, which are placed on the MDI panel.

3.5.1 Entering a Word (INSERT key)

Operation

(1) Select the word, before which you want to insert the new word, by placing the cursor. Otherwise, select the block, before which you want to insert the new word, by placing the cursor.
(2) Enter new word by MDI keys. Plural words can be entered at once.
(3) Press **INSERT**.

NOTE
1 In the case a parameter No.14852#2(G4E) is set to 0, when a cursor is placed on the word in the cycle machining block and an operator is going to insert a new word, the warning to confirm an operator whether it can be done or not will be displayed.

3.5.2 Modifying a Word (ALTER key)

Operation

(1) Select the word to be modified by placing the cursor on the word.
(2) Enter new word by MDI keys. Plural words can be entered at once.
(3) Press **ALTER**.

3.5.3 Modifying the Numeric Value of a Word (ALTER key)

Operation

(1) Select the word to be modified by placing the cursor on the word.
(2) Enter new numeric data only by MDI keys. Plural words can not be entered.
(3) Press **ALTER**.

3.5.4 Deleting a Word (DELETE key)

Operation

(1) Select the word to be deleted by placing the cursor on the word.
(2) Press **DELETE**.

NOTE
1 As the deleting operation in the CNC program screen, no prompting message for deleting a word is displayed.
2 Program number, Oxxxx, and end of program, %, cannot be deleted.

3.5.5 Modifying a Block (ALTER key)

Operation

(1) Select the block to be modified by placing the cursor on the block.
(2) Enter new word by MDI keys. Plural words can be entered at once.
(3) Press **ALTER**.

3.5.6 Deleting a Block (DELETE key)

Operation

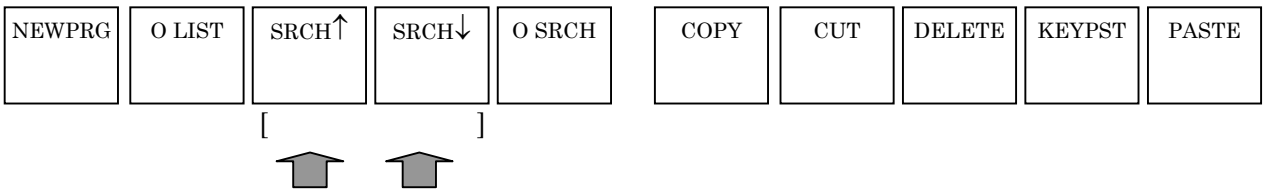
(1) Select the block to be deleted by placing the cursor on the block.
(2) Press **DELETE**.

3.5.7 Changing the Program Number (ALTER key)

Operation

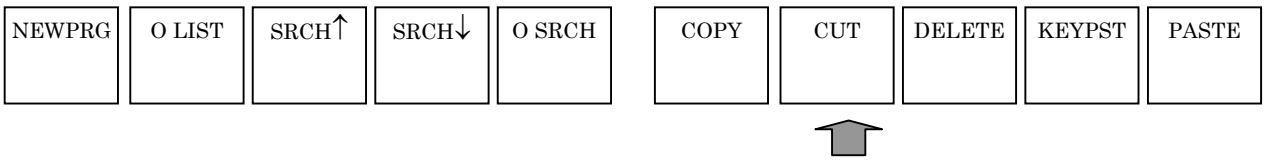
(1) Select the program number word, Oxxxx, by placing the cursor on the word.
(2) Enter the address “O” and new program number by MDI keys.
(3) Press **ALTER**.

3.6 SEARCH (FORWARD AND BACKWARD)



After a character string is entered using MDI keys, pressing [SRCH↑] (backward search) or [SRCH↓] (forward search) searches for the specified character string. A substring can also be searched for.
If the character string is found, key-in buffer is cleared, but the string is displayed in the [ ] under [SRCH↑] and [SRCH↓].
This displayed string can be used for the next time searching operation. Only pressing [SRCH↑] or [SRCH↓] without entering same string again, you can search the same string over and over.

3.7 CUT



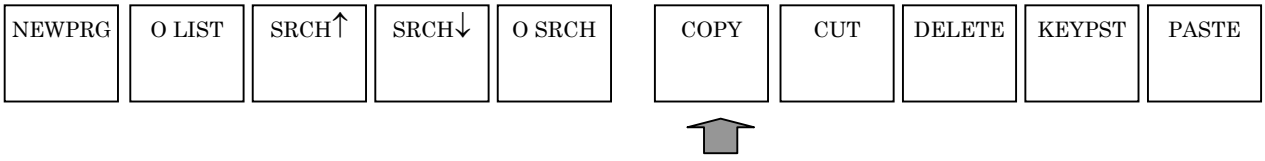
Pressing [CUT] displays a message that prompts you to select a range of data to be cut. First select the cut range (by displaying it in yellow), using cursor keys, and then press [CUT]. The data cut off from the specified range is stored on the clipboard. The contents of the clipboard can be pasted to other places in the program and to other programs.

Clip board is the memory, in which the content of part program can be stored temporary. By pressing [CUT] or [COPY], the content of the cursor selected is stored in it, then the old content is cleared. The content of the clip board is cleared at the power off of CNC.
A clip board size can be selected using bits 4 and 5 of parameter No. 14701.

Bit 4 = 0, bit 5 = 0: Sets a clip board size of 1024 bytes.
Bit 4 = 1, bit 5 = 0: Sets a clip board size of 2048 bytes.
Bit 4 = 0, bit 5 = 1: Sets a clip board size of 4096 bytes.
Bit 4 = 1, bit 5 = 1: Sets a clip board size of 8192 bytes.

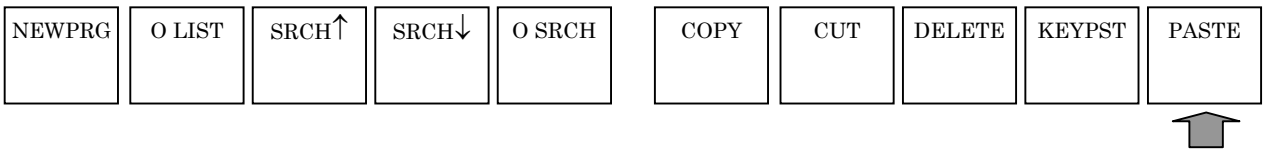
NOTE
The contents of the clipboard are preserved until the power is shut off or other data is stored on the clipboard. So, the clipboard contents can be used any number of times.

3.8 COPY



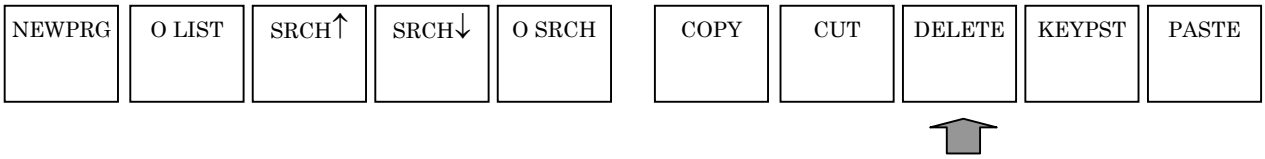
Pressing [COPY] displays a message that prompts you to select a range of data to be copied. First select the copy range (by displaying it in yellow), using cursor keys, and then press [COPY]. The data copied from the specified range is stored on the clipboard. The contents of the clipboard can be pasted to other places in the program and to other programs.

3.9 PASTE



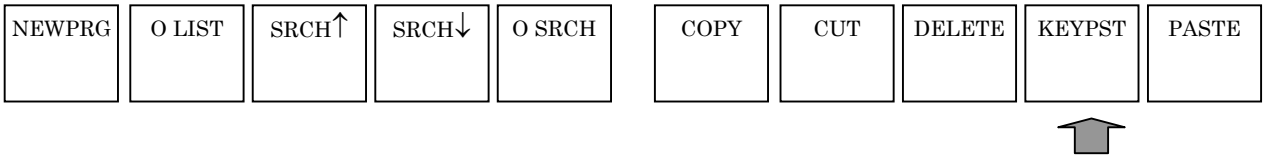
Pressing [PASTE] pastes the contents of the clipboard to the place that immediately follows the current cursor position.
The clipboard contents are preserved.

3.10 DELETE



Pressing [DELETE] displays a message that prompts you to select a range of data to be deleted. First select the delete range (by displaying it in yellow), using cursor keys, and then press [SELECT]. A message appears which asks whether you really want to delete the selected data. Pressing [YES] deletes the data.
The deleted data is not stored on the clipboard. The clipboard preserves the previous contents.

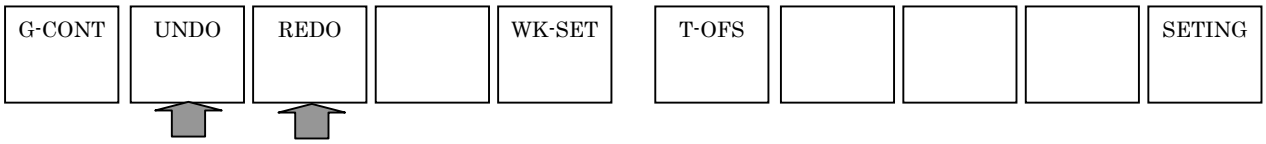
3.11 KEY-IN PASTE



Pressing [KEYPST] copies the contents of a range selected (displayed in yellow) using the cursor to the key-in buffer.
Using the ← and → cursor keys can move the cursor around in the key-in buffer, so a character entered using an MDI key can be placed immediately before the cursor. Pressing the **CAN** key deletes the character at the left of the cursor.
The character strings in the key-in buffer can be used in the same manner as for ordinary editing. Pressing the **ALTER** key, for example, replaces the cursor-selected current contents in the program with the current contents of the key-in buffer. Pressing the **INSERT** key inserts the key-in buffer contents to the place that immediately follows the cursor-selected contents in the program.
Pressing **CAN** after **SHIFT** clears the contents of the key-in buffer.

NOTE
Using key-in paste enables a very long comment program and part of a custom macro program block to be changed easily.

3.12 UNDO, REDO



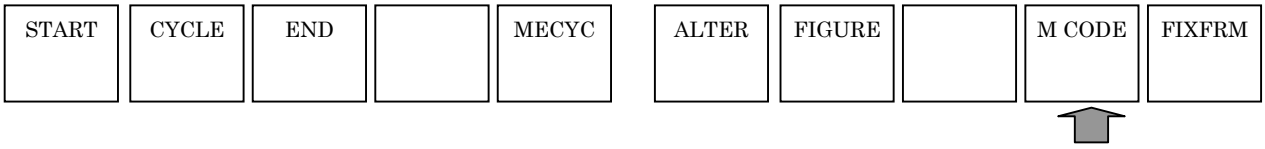
Pressing [REDO] during editing in the MDI mode, EDIT mode, or MEM mode can cancel (undo) a program editing operation using the MANUAL GUIDE *i*. Pressing [UNDO] can cancel (redo) the cancellation of an operation. This function can be used, for example, when a block has been deleted by mistake.

NOTE

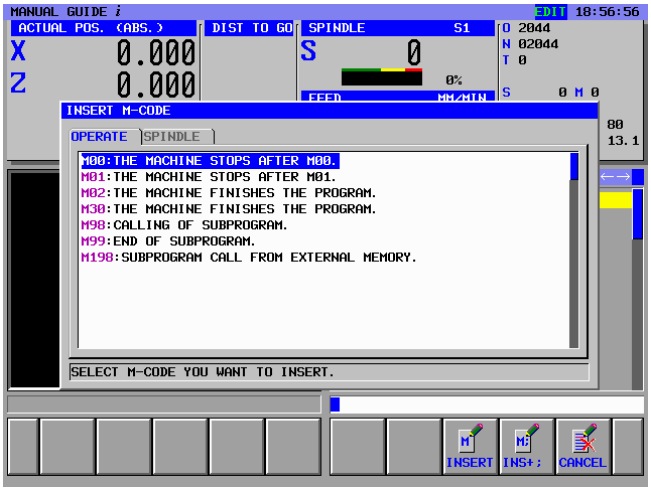
1 Before the undo function or redo function can be used with MANUAL GUIDE *i*, the size of a buffer for storing operations must be set in parameter No. 14861. When 0 is set in the parameter, a 5-KB buffer is allocated.
2 The undo and redo functions can be used for editing operations in a selected program. This means that the creation of a new program and the deletion of a program cannot be undone, and the editing of the key-in buffer cannot be undone.
3 When the screen display is switched or an operation for opening a program is performed, the buffer is cleared.

3.13 M-CODE MENU

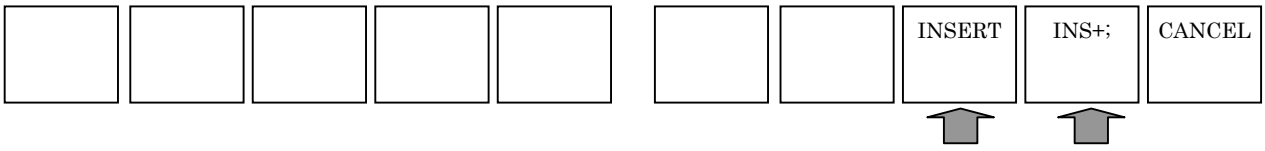
Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays [M CODE] as follows:



Pressing [M CODE] displays the M code menu.



The following soft keys are displayed for M-code menu.



Select an M code group, using the ← and → cursor keys, and then select an M code from the M code group, using the ↑ and ↓ cursor keys.
Pressing [INSERT] inserts the selected M code to the place that follows immediately the cursor position in the program. Pressing [INS+;] inserts an EOB to the place that follows immediately the M code at the same time.
Entering M-code continuously and pressing [INS+;] at last, plural M-code can be entered in one block.

By the following operations, M-code in a part program can be altered directly by using M-code menu.

1) Place a cursor on the M-code should be altered.
2) Press [ALTER] or INPUT-key, then M-code menu will be displayed.
3) Select a M-code by placing a cursor in the M-code menu window.

4) Pressing [ALTER], then M-code will be replaced to the newly selected one.

NOTE
1 In many cases, M codes in the M code menu are set up to a machine tool by the machine tool builder. So, the M code menu varies from one machine tool to another.
2 If the bit 4 of parameter No. 14850 is set to 1, M code menu is disabled.

3.14 FIXED FORM SENTENCE INSERTION

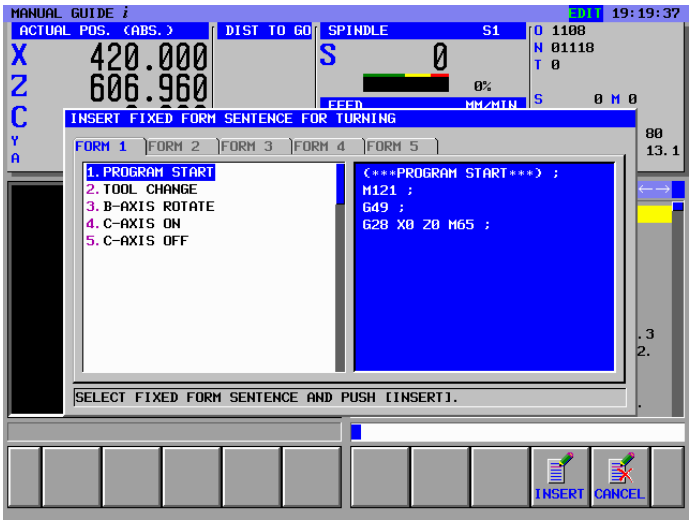
Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays [FIXFRM] for milling or turning.
However, there is a case that either of them is displayed depending on the machine construction, so into details, refer to the document made by machine tool builder.

(Example of softy key menu for milling)

(Example of softy key menu for turning)

Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays [FIXFRM].
If [FIXFRM] is displayed together with a milling soft key menu, pressing it displays a milling fixed form sentence menu. If [FIXFRM] is displayed together with a turning soft key menu, pressing it displays a turning fixed form sentence menu.

START	CYCLE	END		MECYC	ALTER	FIGURE		M CODE	FIXFRM



The following soft keys are displayed for the fixed form sentence menu.

								INSERT	CANCEL

Select a fixed form sentence group, using the ← and → cursor keys, and then select a fixed form sentence from the fixed form sentence group, using the ↑ and ↓ cursor keys.
Pressing [INSERT] inserts the selected fixed form sentence to the place that follows immediately the cursor position in the program.
Using this function makes it easy to enter machining program patterns (fixed form sentences) that may be used frequently.

As the following example, you can register the fixed form sentence, in which undefined data is replaced to the special character such as "?".

```
1. TOOL CHANGE
    G28 G91 X0. Y0. ;
    G28 Z0. ;
    T? ;
    M03 S? ;
```

When the fixed form sentence, in which this special character is included, is inserted to the part program, warning to urge an operator to enter the defined data.
Entering the data that is used actually and pressing **ALTER** can make the correct part program.

This special character is displayed in red in the program window.
This special character and displaying color can be modified by setting corresponding parameters.
Set ASCII code in decimal value of the special character to the parameter No.14860. If 0 is set to this parameter, "?" is used as the special character.
Set the color code of the displaying color to the parameter No.14480 in form of "AABBCC". AA is red color value, BB is green color value and CC is blue color value. If 0 is set to this parameter, the character is displayed in red.

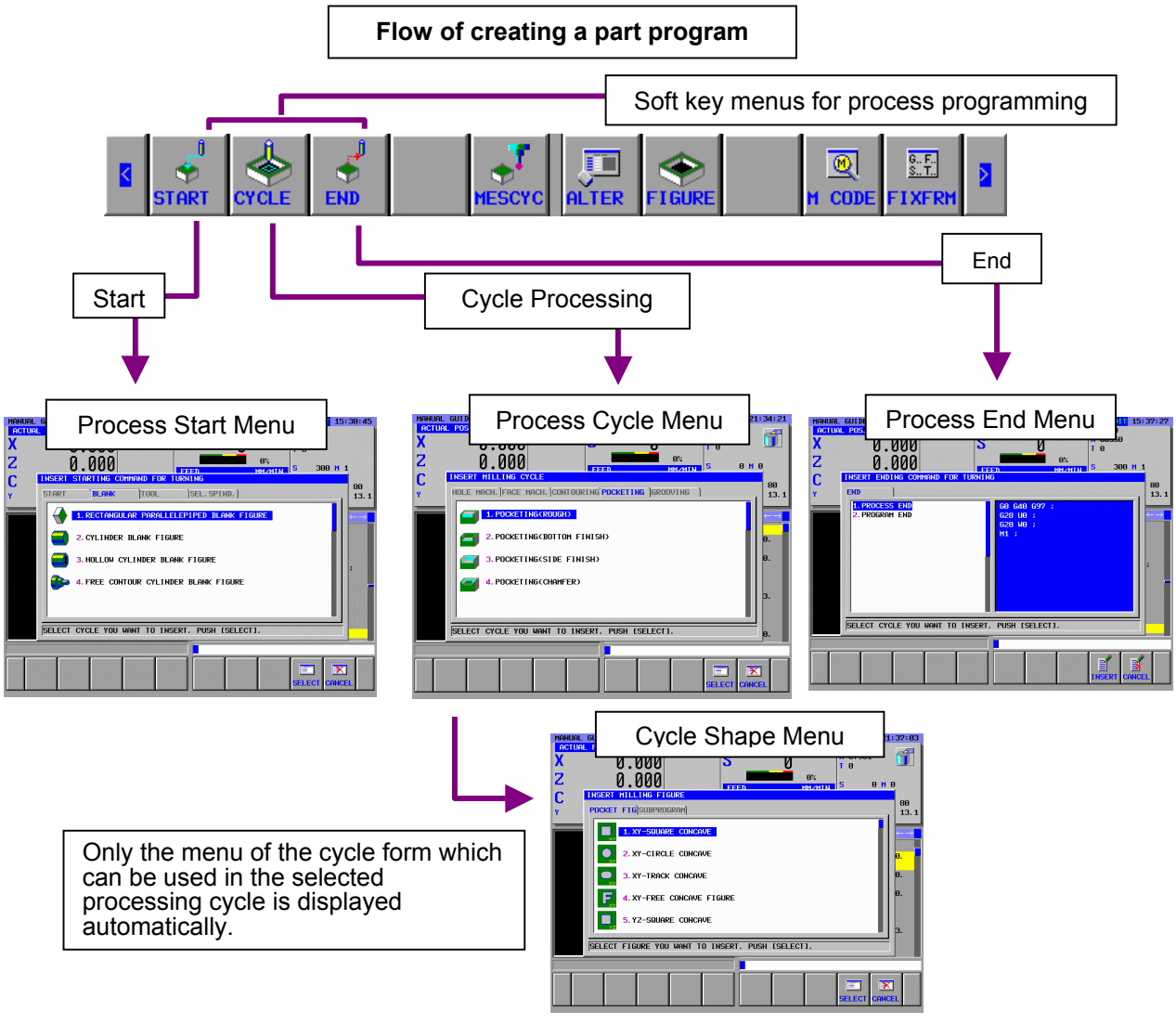
NOTE
1 In many cases, fixed form sentences in the fixed form sentence menu are set up to a machine tool by the machine tool builder. So, the fixed form sentence menu varies from one machine tool to another.
Operators can make changes and additions to the menu. For details, see the descriptions about [SETING].
2 Start and end command fixed form sentences can be called from the fixed form sentence group separately, using, respectively, [START] and [END].

4 EDITING CYCLE MACHINING OPERATIONS

Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the following cycle machining soft key menu.
Two cycle machining types, milling and turning, are optionally supported. A soft key menu is available for each cycle machining type.

START	CYCLE	END		MESCYC	ALTER	FIGURE		M CODE	FIXFRM

See Sections 3.13 and 3.14 for explanations about [M CODE] and [FIXFRM].



4.1 ENTERING THE START COMMAND

START	CYCLE	END		MESCYC	ALTER	FIGURE		M CODE	FIXFRM

Pressing [START] displays the start command fixed form sentence menu.

(Example of the fixed form sentence menu for milling start)

(Example of the fixed form sentence for turning start)

Select a fixed form sentence, using the ↑ and↓ cursor keys.
Pressing [INSERT] inserts the selected fixed form sentence to the place that follows immediately the current cursor position in the program.
This operation makes it easy to enter machining program patterns (fixed form sentences) that are used frequently at machining program start.

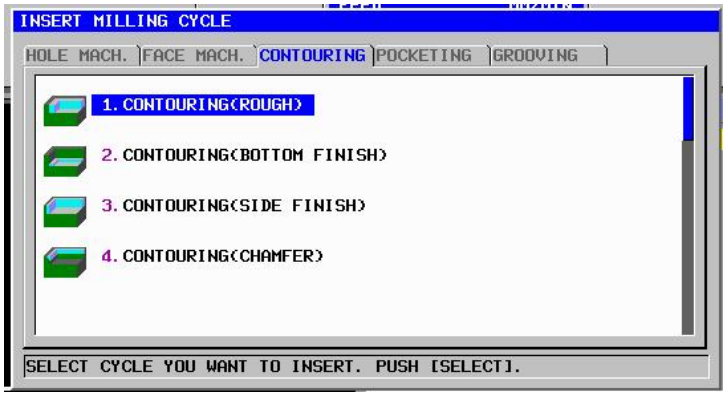
NOTE

In many cases, fixed form sentences in the fixed form sentence menu are set up to a machine tool by the machine tool builder. So, the fixed form sentence menu varies from one machine tool to another.
Operators can make changes and additions to the menu. For details, see the descriptions about [SETING].

4.2 SELECTING A CYCLE MACHINING TYPE

START	CYCLE	END		MESCYC	ALTER	FIGURE		M CODE	FIXFRM

Pressing [CYCLE] displays the cycle machining menu.



The following soft keys are displayed for the cycle machining menu.

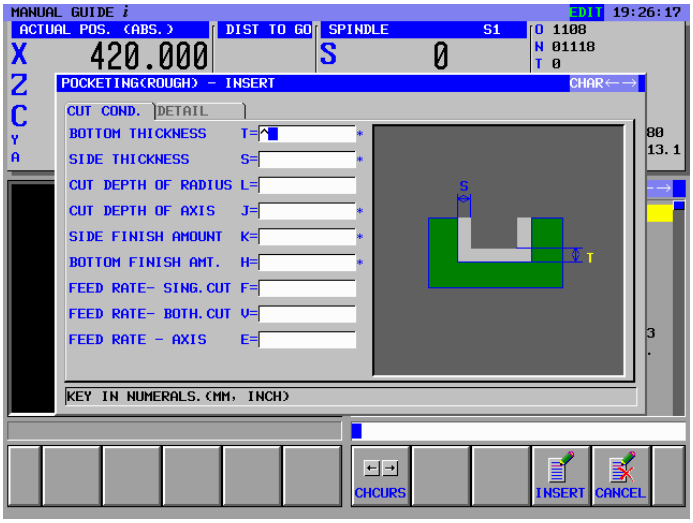
								SELECT	CANCEL

Select a cycle machining group, using the ← and → cursor keys, and then select a cycle machining type from the cycle machining group, using the ↑ and ↓ cursor keys.
Pressing [SELECT] displays the data entry window for the selected cycle machining type.
After a menu number is inputted, it is possible that cycle machining is selected by pressing **INPUT** key.

NOTE

The scroll bar displayed on the right edge of the cycle machining menu window indicates the approximate position of the cursor throughout the cycle machining menu.
If the scroll bar marker is on the middle of the scroll bar, therefore, it is likely that part of the cycle machining menu is hidden behind the window. If this is the case, pressing the ↓ cursor key makes the hidden part visible.

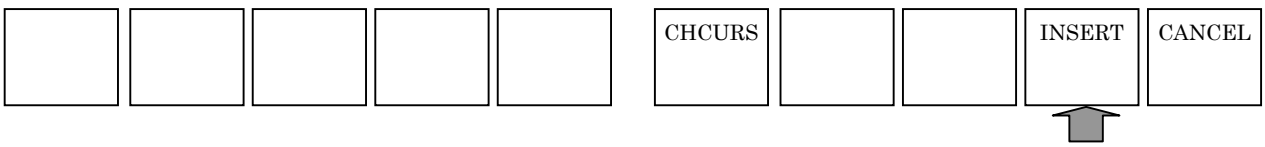
4.3 ENTERING CYCLE MACHINING DATA



The cycle machining data entry window is divided into two sections, one section for cutting conditions and the other for detailed data.

Pressing the ← or → cursor key switches between the two sections and changes the displayed tab. The character in the selected tab is displayed in blue.

Pressing the ↑ or ↓ cursor key can place the cursor on a desired data input item.
There are two types of data input items. One type includes those entered as numbers, and the other type, those selected from a menu displayed using a soft key. For the first type, the message "KEY IN NUMERALS." is displayed in the lower section of the window. For the second type, the message "SELECT SOFT KEY" is displayed.
Necessary data can be entered by referencing a comment for the data input item and a guide chart displayed in the window.
The following soft keys are displayed for the machining cycle data window.



When the necessary data has been entered, pressing [INSERT] inserts a block for the related cycle machining into a machining program.

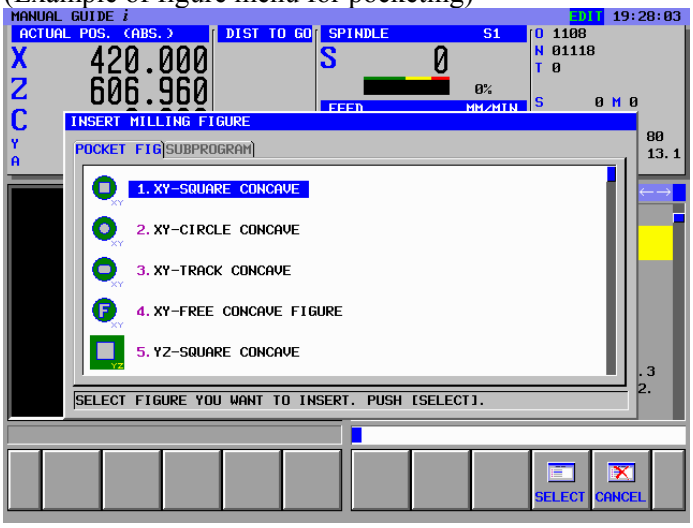
NOTE

1 Among the data item displayed in the cutting condition window, there are data should be danger if they are set automatically such as cutting amount or feedrate. These data should be entered by an operator always. Other data are set automatically.
2 Data items displayed in the detail window are usually set automatically by duplicating the value entered last time. So, check these automatically set value and modify them if necessary.
3 Executing cycle machining requires a cycle machining block and a figure block, which will be explained later. Once a cycle machining block has been entered, therefore, be sure to enter a figure block in succession.
4 A data item with an asterisk displayed at its right end has a default value. You need to enter no data if you accept the default value.
5 The [CHCURS] soft key is displayed in data entry windows for cycle machining, figure, and contour programs. Pressing this soft key can select whether the ← and → cursor keys are to be used for tab switching or cursor movement within the entered data. If tab switching is selected, "Tab ←→" is displayed in the upper right section of the window. If cursor movement is selected, "Character ←→" is displayed.

4.4 SELECTING FIGURES

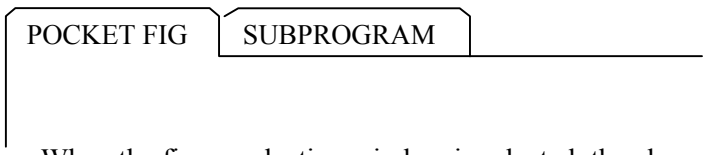
In usual case, entering a cycle motion block displays continuously the following figure menu exclusively used for the already entered cycle machining.

(Example of figure menu for pocketing)

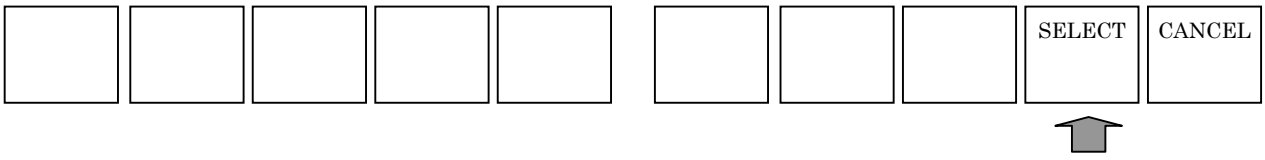


The cycle figure menu window is divided into two sections, one section for selecting figure and the other for subprogram.

(Example of tab for pocketing)



When the figure selection window is selected, the characters of the tab are displayed in blue. Select a desired figure by moving the cursor with the cursor keys ↑ and ↓.

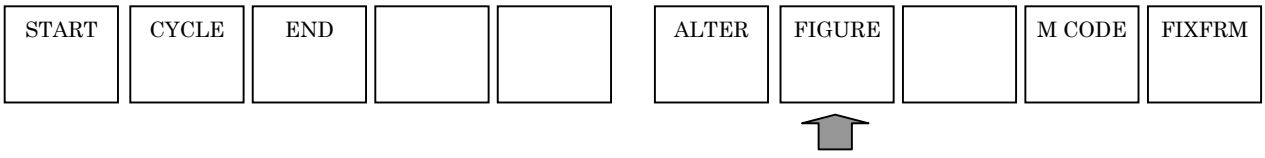


Pressing [SELECT] displays a data entry window for the selected figure.
There are 2 types of figures used for cycle machining, fixed form figure and free form figure.
The former one is predefined figure such as square and circle, and can be specified by entering minimum data such as side length or circle radius.
The later one is specified by entering figures by line or arc along the figures written on a blue print one by one.

Pressing the → cursor key displays the menu window of subprogram and the character in the selected tab is displayed in blue.
If some figure blocks were created as subprogram in advance, the subprogram number and name are displayed in the subprogram menu, and suitable subprogram can be selected by placing the cursor on it.
Pressing [SELECT] create the block for calling the selected subprogram as "M98 Pxxxx ;" in the actual part program.

It is specified by program number to be displayed in the subprogram menu or not.
For turning cycle machining figure, the minimum and maximum program number of the programs should be used as the subprogram are set in the parameter No.14720 (TFIGSNO) and No.14721 (TFIGENO).
For milling cycle machining figure, the minimum and maximum program number of the programs should be used as the subprogram are set in the parameter No.14722 (MFIGSNO) and No.14723 (MFIGENO).
Subprogram whose number is within these range is displayed in the subprogram menu for turning or milling cycle figure.

Over 2 figure blocks can be specified continuously for one cycle machining block. If adding another figure block to the part program in which cycle machining block and figure block were already inserted, press [FIGURE] in the soft-keys of program screen and make the figure menu window display.
But, in this case, all figure menus for all kind of cycle machining are displayed in one screen. This is different from the figure menu displayed at the time some cycle machining block is inserted.

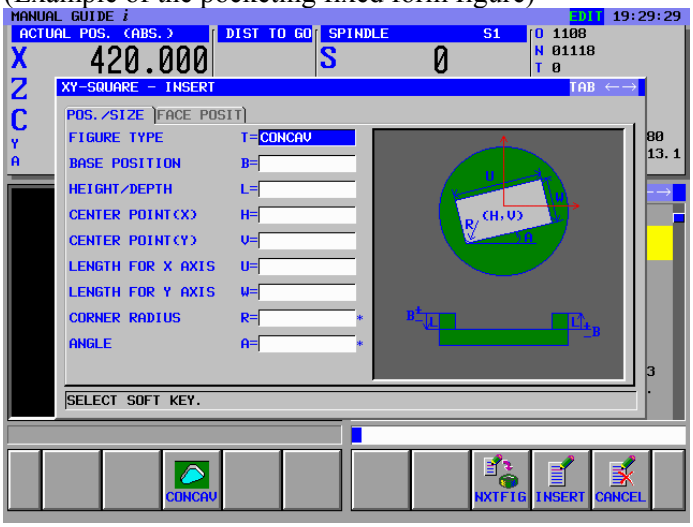


NOTE
The scroll bar displayed on the right edge of the figure menu window indicates the approximate position of the cursor throughout the figure menu. If the scroll bar marker is on the middle of the scroll bar, therefore, it is likely that part of the figure menu is hidden behind the window. If this is the case, pressing the ↓ cursor key makes the hidden part visible.

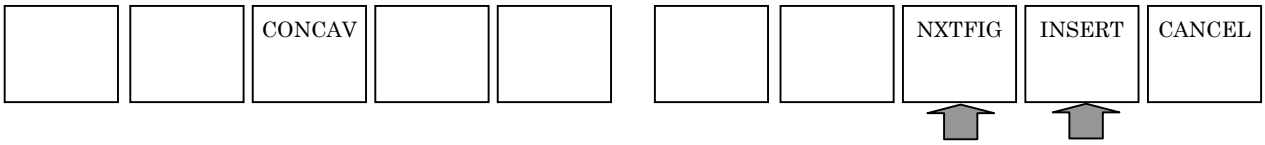
4.5 ENTERING FIXED FORM FIGURE DATA FOR CYCLE MACHINING

Selecting the fixed form figure displays the data entry window for cycle machining fixed form data entry window.

(Example of the pocketing fixed form figure)



A data entry window for hole positions, fixed form figures (milling), turning groove figures, or screw figures (turning) may be extended over two pages as necessary.
Pressing the ← or → cursor key switches between the two pages. It also changes the displayed tab.
Pressing the ↑ or ↓ cursor key can place the cursor on a desired data input item.
There are two types of data input items. One type includes those entered as numbers, and the other type, those selected from a menu displayed using a soft key. For the first type, the message "KEY IN NUMERALS." is displayed in the lower section of the window. For the second type, the message "SELECT SOFT KEY" is displayed.
Necessary data can be entered by referencing a comment for the data input item and a guide chart displayed in the window.
The following soft keys are displayed for the fixed form figure data entry windows.



When the necessary data has been entered, press [INSERT] or [NXTFIG], then a block for the related figure block is inserted into a machining program.
Pressing [INSERT] returns to the program window. Pressing [NXTFIG] displays figure selecting menu again.

NOTE

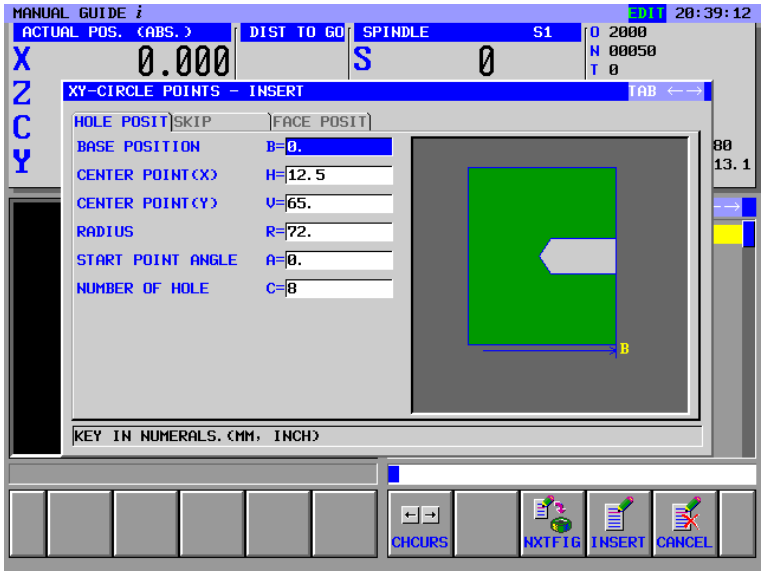
1 More than one figure can be entered in succession for a single cycle machining type.
Cycle machining is executed for each of the specified figures sequentially.

2 An ordinary ISO code block can be entered between cycle machining and figure blocks. If this is done, no machining operation takes place in the cycle machining block. Instead, the ISO code block entered before the figure block is executed. Cycle machining is executed only in the figure block after the ISO code block has been executed.

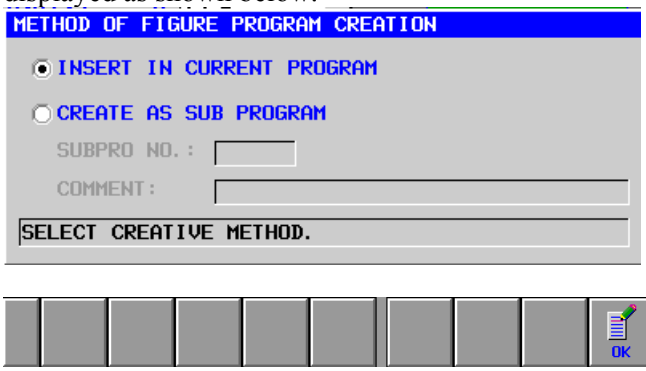
Output as a subprogram

When bit 1 of parameter No. 14851 is set to 1, fixed form figure data can be output as a subprogram. The operation procedure is described below.

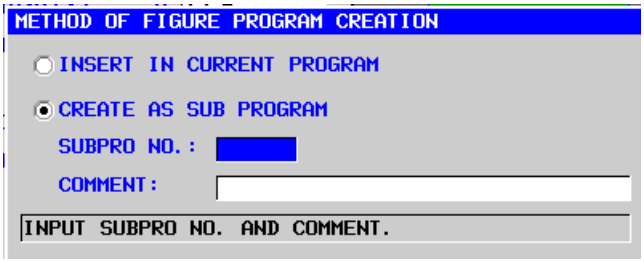
<1> Press the [INSERT] soft key in the fixed form figure data input window.



<2> A screen for selecting fixed form figure data as a subprogram is displayed as shown below.



<3> When creating fixed form figure data as a subprogram, select "CREATE AS SUB PROGRAM".



<4> If a comment is entered, the entered comment is added to the M98P**** block of the main program and the subprogram at the time of program output.

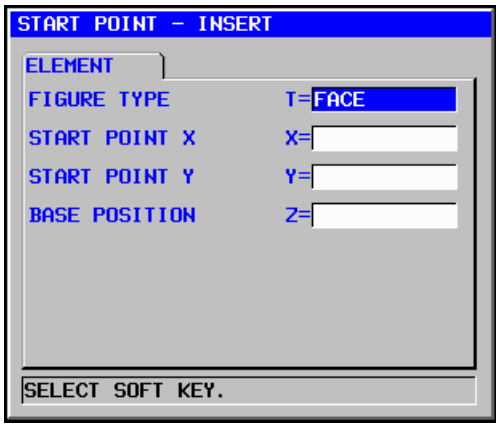
<5> Pressing the [OK] soft key outputs a created fixed form figure program as a subprogram to the machining program.

4.6 ENTERING ARBITRARY FIGURE DATA FOR CYCLE MACHINING

For cycle machining, an arbitrary figure consisting of circles and straight lines can be entered by performing automatic calculation on entered data to obtain the end point of each figure even if no end point is specified on the drawing.

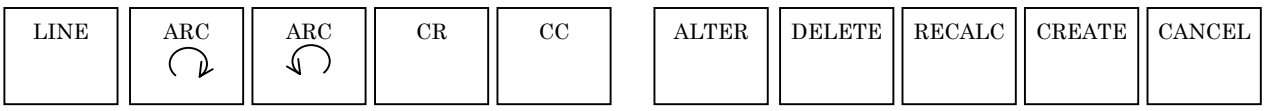
(1) Entering start points

Be sure to enter a start point for the beginning of an arbitrary figure.
After entering necessary data, press [OK].



(2) Selecting a figure

A menu for figure selection is displayed on soft keys as shown below. Select whatever is necessary.



(3) Entering figure data

When a figure is selected, a figure data entry window appears. In this window, enter the figure data specified on the drawing.
There are two types of data input items. One type includes those entered as numbers, and the other type, those selected from a menu displayed using a soft key. For the first type, the message "KEY IN NUMERALS." is displayed in the lower section of the window. For the second type, the message "SELECT SOFT KEY" is displayed.
After entering the necessary data, press [OK].

NOTE
As figures are entered, they are drawn in the figure entry window.
In the upper section of the window, symbols for entered figures are displayed sequentially, starting at the left. The ← or → cursor key can be used to select an entered figure; the lines that correspond to the selected figure will be displayed in yellow.

(4) Changing figure data

Place the cursor on the figure that contains the figure data you want to change, and press [ALTER]. The figure data entry window appears. In this window, place the cursor on the data input item you want to change, and enter new data.
If you want to delete data, press the **CAN** and **INPUT** keys in the stated sequence.
Pressing [OK] closes the figure data entry window. Press [RECALC] to perform calculation for all entered figures, and check drawn figures to see if the intended changes have been made.

(5) Inserting new figures

After placing the cursor on the figure that precedes immediately the place where your want to add a figure, select the figure to be added from the soft key menu.
In the figure data entry window that appears for the new figure, enter figure data necessary for the figure, and press [OK].
Press [RECALC] to perform calculation for all entered figures, and check drawn figures to see if the intended additions have been made.

(6) Deleting unnecessary figures

After placing the cursor on the figure you want to delete, press [DELETE].
Press [RECALC] to perform calculation for all entered figures, and check drawn figures to see if the intended deletions have been made.

(7) Changing figures

If you want to change the type of an entered figure, delete it and enter a new figure.

(8) Writing entered arbitrary figures to machining programs

After all necessary figures have been entered, they can be written to machining programs.
There are two methods for writing figures to machining programs. The first method writes directly to the currently selected machining program. The second method writes to a newly created subprogram.
After necessary data has been entered, pressing [CREATE] displays a window for selecting one of the creation methods.

Select whichever creation method you want, using the ↑ and ↓ cursor keys.
To write to the machining program that has been selected, simply press [OK].
To create a subprogram, enter a new subprogram number to the subprogram number item, and then press the **INPUT** key. You can enter character strings as a comment if necessary. The comment is displayed as a program name in the list window.
Pressing [OK] now creates a subprogram and writes figures to the subprogram.

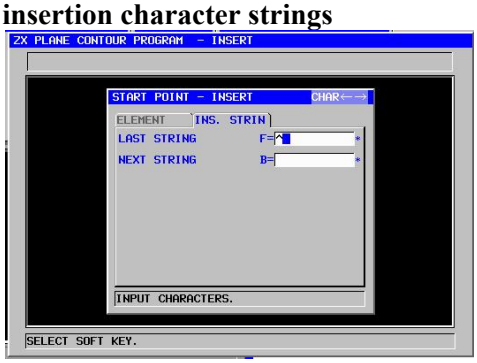
NOTE
See Chapter 5, "Detailed Descriptions about Entering Arbitrary Figures," for descriptions about data for arbitrary figures.

4.7 ENTERING CONTOUR PROGRAMS

It is possible to enter arbitrary figures consisting of circles and straight lines (contour programs), which are different from cycle machining.
Pressing [G-CONT] displays the same window as for the arbitrary figure data described earlier. After performing the same operations to enter figures, write them to the last machining program.
In this case, a G01/G02/G03 program in ISO code format is created.

NOTE
See Chapter 5, "Detailed Descriptions about Entering Arbitrary Figures," for descriptions about data entered for contour programs.

(1) Entering insertion character strings



Any character string can be entered to all figure blocks entered for a contour program.
The "INS. STRIN" tab is added to the entry window for each figure block of the contour program.

Insertion character string		
Data item		**Meaning**
F*	LAST STRING	Character string (consisting of up to 16 characters) entered at the beginning (immediately after an EOB) of each figure block
B*	NEXT STRING	Character string (consisting of up to 16 characters entered) at the end (immediately before an EOB) of each figure block

Using "INSERT STRING" can enter G codes and feedrates into figure blocks.

NOTE
"INSERT STRING" can enter neither a comment based on a ( ) pair nor an EOB.

(2) Editing figure blocks in contour programs

A figure block entered to a contour program can be edited by performing figure calculation in the same manner as when it was entered.

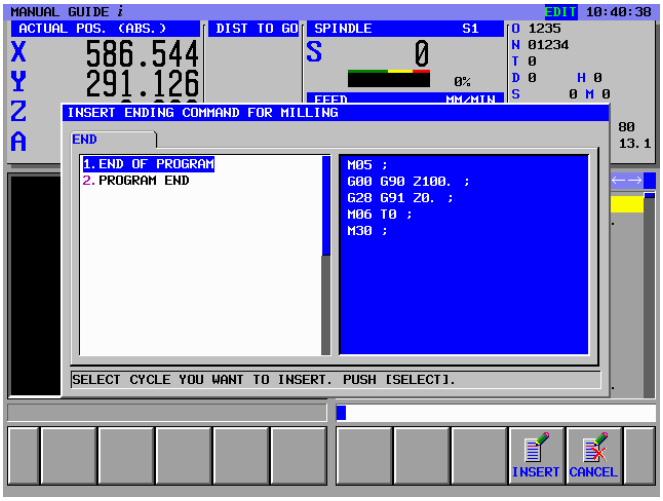
For editing figure blocks, entered figure data is written as a comment to each figure block.

There is a start point G code (G1200, G1300, G1450, G1500, or G1600) in the first figure block in contour programming. Place the cursor on the block, and press the [ALTER] soft key.
This operation displays a screen for editing contour programs. On this screen, perform the same editing operation as for other arbitrary figures.

4.8 ENTERING THE END COMMAND

START	CYCLE	END		MESCYC	ALTER	FIGURE		M CODE	FIXFRM

Pressing [END] displays the end command fixed form sentence menu.



Select a fixed form sentence, using the ↑ and ↓ cursor keys.
Pressing [INSERT] inserts the selected fixed form sentence to the place that follows immediately the current cursor position in the program.
This operation makes it easy to enter machining program patterns (fixed form sentences) that are used frequently at machining program start.

4.9 CHANGING CYCLE MACHINING AND FIGURE DATA

To change the contents of a machining cycle and fixed form figure, place the cursor on the related block, and press [ALTER].
To change the contents of an arbitrary figure and contour program, place the cursor at the first of the related blocks, and press [ALTER].
In any case, a data entry window for these items appears. After making the necessary data changes, press [CREATE] in the same manner as for new entry. New figures are written to the machining program.

5 DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES

This chapter explains data for arbitrary figures entered with MANUAL GUIDE *i*.

NOTE

1 When entering arbitrary figures, enter all the data for each figure specified on the drawing.
If superfluous data is entered, it will not be used in figure calculation; only the data necessary for figure calculation is used.
2 If all the data necessary for figure calculation is specified on the drawing, entering it enables figure calculation. If data is insufficient, however, it is necessary to calculate and enter the necessary figure data separately.
3 If arbitrary-figure calculation according to MANUAL GUIDE *i* produces more than one candidate figure, the figure that passes the shortest path will be selected automatically.
In calculating a contact between an arc and straight line or between two arcs, a figure that links two figures smoothly is selected automatically.
4 On the program editing screen, after moving the cursor on the sub program call command (M98 P****) which is composed by the arbitrary figure blocks, pressing the [INPUT] key displays the window for editing the arbitrary figures to edit directly.
5 When positioning the cursor on the block of subprogram calling in the program-editing screen, arbitrary figures of the subprogram is drawn on the graphic window.

5.1 INCREMENTAL PROGRAMMING

In entering arbitrary figures of element “LINE” or “ARC”, the end point can be set as an incremental programming.

When positioning the cursor on “END POINT”, the soft key [ST.P+I] and [ST.P-I] are displayed.

(1) When the operator pushes the soft key [ST.P+I], “END POINT” is set to

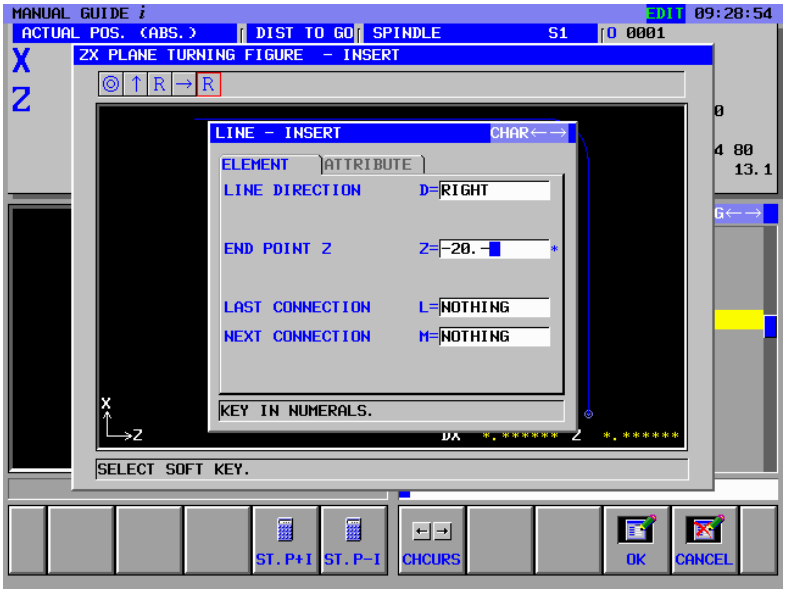
“Start Point of the figure (=End Point of last figure) +”.

And input the increment or decrement in succession. As result of pushing [INPUT] key, the plus incremental value is set.

(2) When the operator pushes the soft key [ST.P-I], “END POINT” is set to

“Start Point of the figure (=End Point of last figure) -”.

And input the increment or decrement in succession. As result of pushing [INPUT] key, the minus incremental value is set.



NOTE

1 If the last figure element is “CORNER R” or “CHAMFER”, The point to set as the start point in this function is as follows.

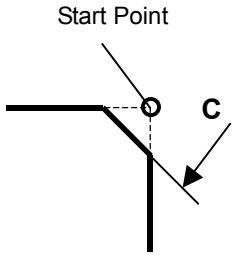


Fig1 Last figure is a chamfer.

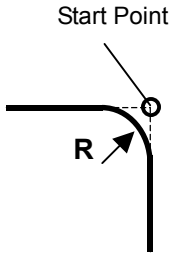


Fig2 Last figure is a corner R.

2 If “END POINT” of the last figure is pending, “END POINT” is not set to the start point by pressing the software key [ST.P+I] and [ST.P-I]. The warning message “START POINT IS PENDING.” is displayed.

5.1.1 Arbitrary Figures for the XY Plane

Arbitrary figures in the XY plane can be used in the following types of milling.

1. Facing
2. Contouring (Side cutting)
3. Pocketing
4. Grooving
5. Emboss machining

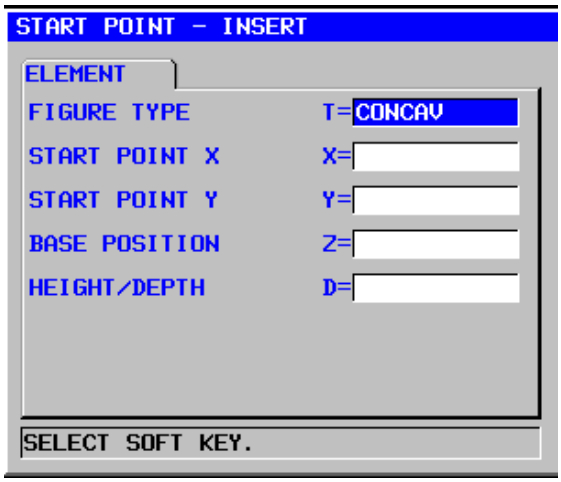
NOTE

1 See Chapter 1, "Milling," in Part III for detailed descriptions about data to be entered for each type of cycle machining.

2 When entering an arbitrary figure for pocketing, make its end point coincide with its start point. To put another way, combine parts figures with blank figures in such a way that a closed curve is created.
Pocketing is carried out in this closed curve.

3 It is possible to specify "PART" and "BLANK" as "ELEMENT TYPE" for an individual arbitrary figure. If a pocket is open in part, this "BLANK" can be used to enter an arbitrary figure resembling a blank figure for the open portion, thus carrying out optimum pocketing.

Start point: G1200 (XY plane)



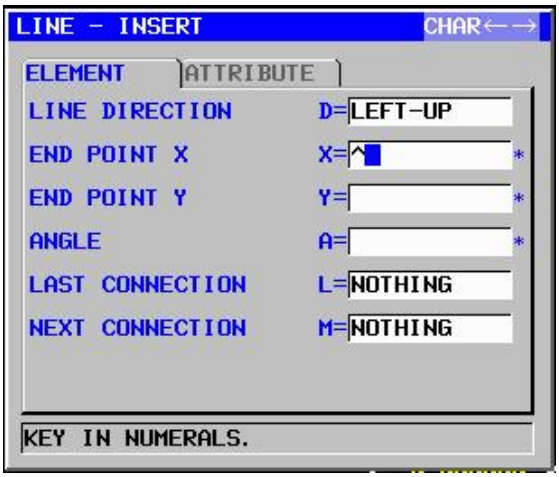
ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE **(Note 2)**	[FACE] : Used as a figure in facing [CONVEX] : Used as an outer-perimeter figure in contouring [CONCAV] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [GROOVE] : Used as a figure in grooving
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
Z	BASE POSITION	Position of the machining surface of an arbitrary figure
D	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
W	GROOVE WIDTH	Groove width (Positive value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[RIGHT] : The right side of an entered figure as cutting [LEFT] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.
2 Select, with a figure type for a start point, which milling operation is to use an arbitrary figure. Data to be entered for the start point varies depending on what figure type is selected. For details, see descriptions about the respective milling types.

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		Meaning
T	FIGURE ATTRIBUTE	[1] : Used as a figure in facing [2] : Used as an outer-perimeter figure in contouring [3] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [4] : Used as a figure in grooving (input value)
H	START POINT X	X coordinate of the start point (input value)
V	START POINT Y	Y coordinate of the start point (input value)
B	BASE POSITION	Position of the machining surface (input value)
L	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface (input value) Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
D	GROOVE WIDTH	Groove width (Positive value) (input value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[1] : The right side of an entered figure as cutting [2] : The left side of an entered figure as cutting (input value) Remarks) This item is displayed in Open figure of Contouring.

NOTE
3 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Straight line: G1201 (XY plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key.
X*	END POINT X	X coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
Y*	END POINT Y	Y coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
A*	ANGLE	Straight-line angle Remarks) This data may not be displayed depending on the value entered as the direction of the straight line.
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)

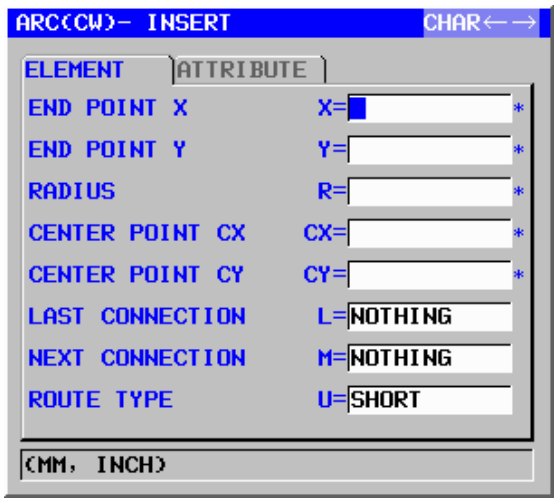
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT Y	Y coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT X	X coordinate of the end point of a straight line (input value)
D*	END POINT Y	Y coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [0] : Not in contact with the immediately following figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arc (CW): G1202 (XY plane)
Arc (CCW): G1203 (XY plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
X*	END POINT X	X coordinate of an arc end point Remarks) Incremental programming is possible.
Y*	END POINT Y	Y coordinate of an arc end point Remarks) Incremental programming is possible.
R*	RADIUS	Arc radius
CX*	CENTER POINT CX	X coordinate of an arc center
CY*	CENTER POINT CY	Y coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
U	ROUTE TYPE	[LONG] : An arc with a short route is made [SHORT] : An arc with a long route is made (initial value)

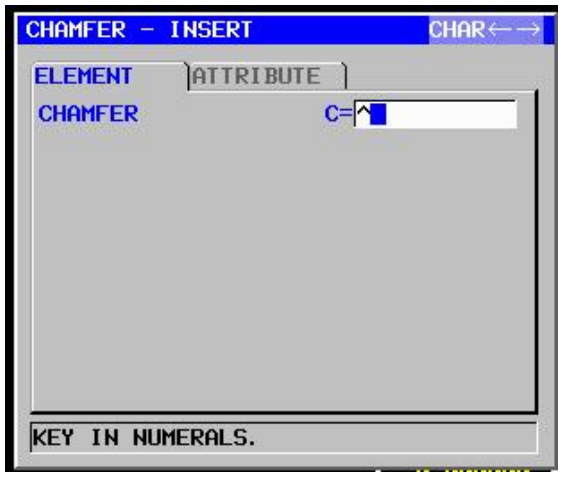
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END PINT Y	Y coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Y	Y coordinate of an arc center (calculation result)
C*	END POINT X	X coordinate of an arc end point (input value)
D*	END POINT Y	Y coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CX	X coordinate of an arc center (input value)
Q*	CENTER POINT CY	Y coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
U	ROUTE TYPE	[1] : An arc with a long route is made [0] : An arc with a short route is made (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Chamfering: G1204 (XY plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
C	CHAMFER	Amount of chamfering (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		Meaning
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

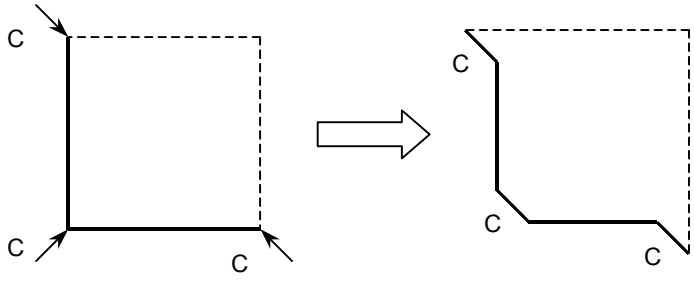
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT Y	Y coordinate of the end point of a straight line (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

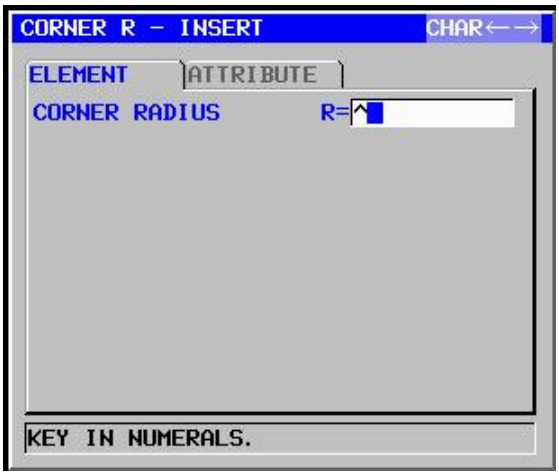
NOTE

2 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.



Dotted line : blank element
Normal line : part element

Corner rounding: G1205 (XY plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
R	CORNER RADIUS	Corner rounding (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		Meaning
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END PINT Y	Y coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Y	Y coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : clockwise [3] : counterclockwise
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.
3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.

End of arbitrary figures: G1206 (XY plane)

This block is output at the end of a series of arbitrary-figure blocks.

NOTE
A series of arbitrary figures must be enclosed between G1200 (start point) and G1206 (end of arbitrary figures).

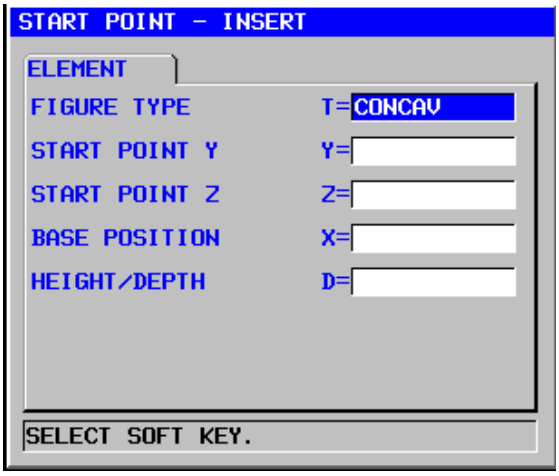
5.2 ARBITRARY FIGURES FOR THE YZ PLANE

Arbitrary figures in the YZ plane can be used in the following types of milling.

1. Facing
2. Contouring (Side cutting)
3. Pocketing
4. Grooving
5. Emboss machining

NOTE
See Chapter 1, "Milling," in Part III for detailed descriptions about data to be entered for each type of cycle machining.

Start point: G1300 (YZ plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE **(Note 2)**	[FACE] : Used as a figure in facing [CONVEX] : Used as an outer-perimeter figure in contouring [CONCAVE] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [GROOVE] : Used as a figure in grooving
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
Z	START POINT Z	Z coordinate of the start point of an arbitrary figure
X	BASE POSITION	Position of the machining surface of an arbitrary figure
D	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
W	GROOVE WIDTH	Groove width (Positive value) Remarks) This item is displayed in Grooving.

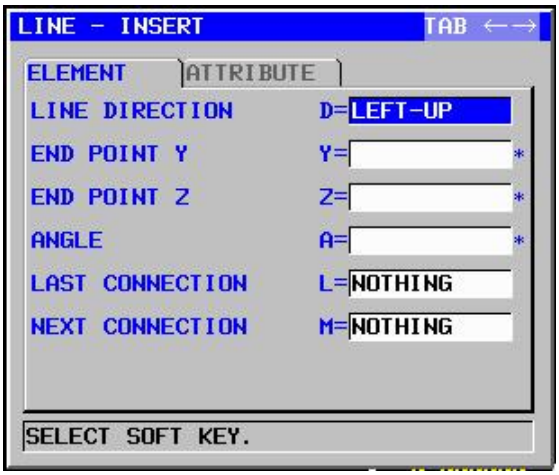
ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
P	FIGURE ATTRIBUTE	[RIGHT] : The right side of an entered figure as cutting [LEFT] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.
2 Select, with a figure type for a start point, which milling operation is to use an arbitrary figure. Data to be entered for the start point varies depending on what figure type is selected. For details, see descriptions about the respective milling types.

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE	[1] : Used as a figure in facing [2] : Used as an outer-perimeter figure in contouring [3] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [4] : Used as a figure in grooving (input value)
H	START POINT Y	Y coordinate of the start point (input value)
V	START POINT Z	Z coordinate of the start point (input value)
B	BASE POSITION	Position of the machining surface (input value)
L	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface (input value) Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
D	GROOVE WIDTH	Groove width (Positive value) (input value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[1] : The right side of an entered figure as cutting [2] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring. (input value)

NOTE
3 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Straight line: G1301 (YZ plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key.
Y*	END POINT Y	Y coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
Z*	END POINT Z	Z coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
A*	ANGLE	Straight-line angle Remarks) This data may not be displayed depending on the value entered as the direction of the straight line.
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)

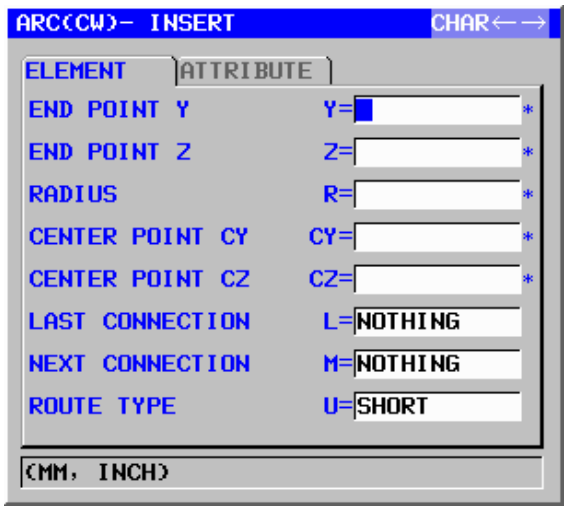
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Y	Y coordinate of the end point of a straight line (calculation result)
V	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT Y	Y coordinate of the end point of a straight line (input value)
D*	END POINT Z	Z coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [0] : Not in contact with the immediately following figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arc (CW): G1302 (YZ plane)
Arc (CCW): G1303 (YZ plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
Y*	END POINT Y	Y coordinate of an arc end point Remarks) Incremental programming is possible.
Z*	END POINT Z	Z coordinate of an arc end point Remarks) Incremental programming is possible.
R*	RADIUS	Arc radius
CY*	CENTER POINT CY	Y coordinate of an arc center
CZ*	CENTER POINT CZ	Z coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
U	ROUTE TYPE	[LONG] : An arc with a long route is made [SHORT] : An arc with a short route is made (initial value)

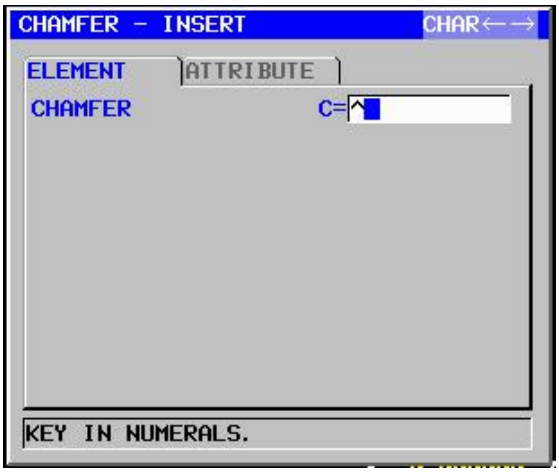
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Y	Y coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT Y	Y coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
C*	END POINT Y	Y coordinate of an arc end point (input value)
D*	END POINT Z	Z coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CY	Y coordinate of an arc center (input value)
Q*	CENTER POINT CZ	Z coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
U	ROUTE TYPE	[1] : An arc with a long route is made [0] : An arc with a short route is made (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Chamfering: G1304 (YZ plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
C	CHAMFER	Amount of chamfering (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		Meaning
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

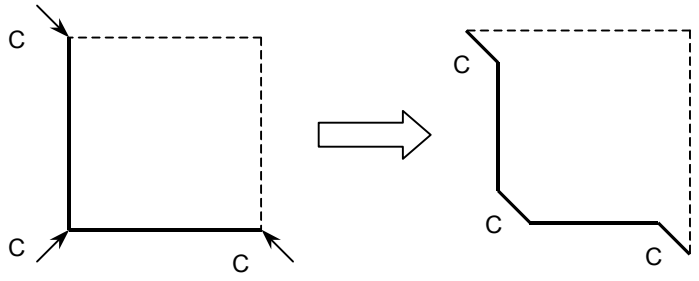
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT Y	Y coordinate of the end point of a straight line (calculation result)
V	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

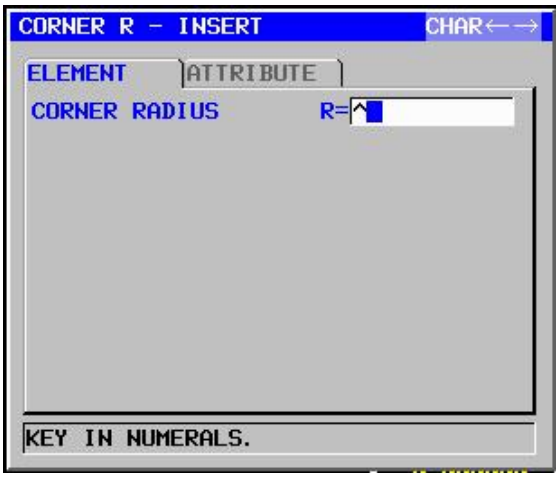
NOTE

3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.



Dotted line : blank element
Normal line : part element

Corner rounding: G1305 (YZ plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
R	CORNER RADIUS	Corner rounding (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Y	Y coordinate of an arc end point (calculation result)
V	END PINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT Y	Y coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : clockwise [3] : counterclockwise
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.
3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.

End of arbitrary figures: G1306 (YZ plane)

This block is output at the end of a series of arbitrary-figure blocks.

NOTE
A series of arbitrary figures must be enclosed between G1300 (start point) and G1306 (end of arbitrary figures).

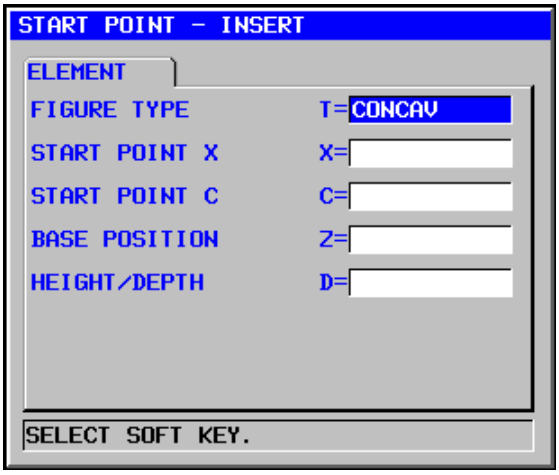
5.3 ARBITRARY FIGURES FOR THE POLAR COORDINATE INTERPOLATION PLANE (XC PLANE)

The following types of milling can be specified also for the polar coordinate interpolation plane (XC plane), and arbitrary figures in the XC plane can be used in these milling types.

1. Facing
2. Contouring (Side cutting)
3. Pocketing
4. Grooving
5. Emboss machining

NOTE
1 See Chapter 1, "Milling," in Part III for detailed descriptions about data to be entered for each type of cycle machining.
2 X coordinate values specified for the XC plane must also be diameters.

Start point: G1500 (XC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE **(Note 2)**	[FACE] : Used as a figure in facing [CONVEX] : Used as an outer-perimeter figure in contouring [CONCAV] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [GROOVE] : Used as a figure in grooving
X	START POINT X	X coordinate of the start point of an arbitrary figure
C	START POINT C	C coordinate of the start point of an arbitrary figure
Z	BASE POSITION	Position of the machining surface of an arbitrary figure
D	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.

ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
W	GROOVE WIDTH	Groove width (Positive value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[RIGHT] : The right side of an entered figure as cutting [LEFT] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring.
F	FACE POSITION	[+FACE] : References the figure (+end face) [-FACE] : References the figure (-end face) Remarks) This item is displayed when the parameter No.27000#4=1.
Y	ROTATION AXIS NAME	[C] : The rotation axis is the C axis. [A] : The rotation axis is the A axis (No.27000#1=1) [B] : The rotation axis is the B axis (No.27000#2=1) [E] : The rotation axis is the E axis (No.27000#3=1) Remarks) This item is displayed when the parameter No.27000#0=1.

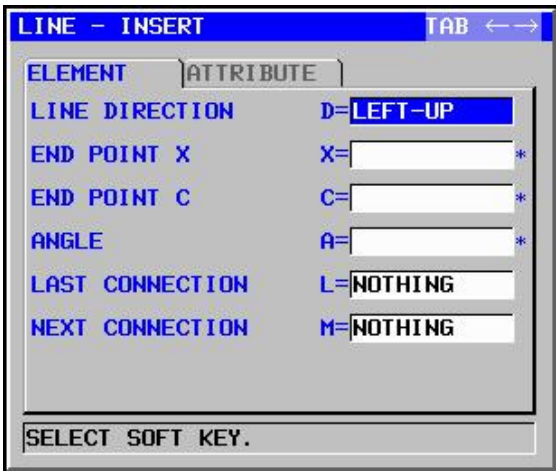
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.
2 Select, with a figure type for a start point, which milling operation is to use an arbitrary figure. Data to be entered for the start point varies depending on what figure type is selected. For details, see descriptions about the respective milling types.

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE	[1] : Used as a figure in facing [2] : Used as an outer-perimeter figure in contouring [3] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [4] : Used as a figure in grooving (input value)
H	START POINT X	X coordinate of the start point (input value)
V	START POINT C	C coordinate of the start point (input value)
B	BASE POSITION	Position of the machining surface (input value)
L	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface (input value) Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
D	GROOVE WIDTH	Groove width (Positive value) (input value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[1] : The right side of an entered figure as cutting [2] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring. (input value)

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		Meaning
Y	ROTATION AXIS NAME	[1] : The rotation axis is the C axis. [2] : The rotation axis is the A axis (No.27000#1=1) [3] : The rotation axis is the B axis (No.27000#2=1) [4] : The rotation axis is the E axis (No.27000#3=1) (input value)
Z	FACE POSITION	[1] : References the figure (+end face) [2] : References the figure (-end face) (input value)

NOTE
3 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Straight line: G1501 (XC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key.
X*	END POINT X	X coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
C*	END POINT C	C coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
A*	ANGLE	Straight-line angle Remarks) This data may not be displayed depending on the value entered as the direction of the straight line.
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)

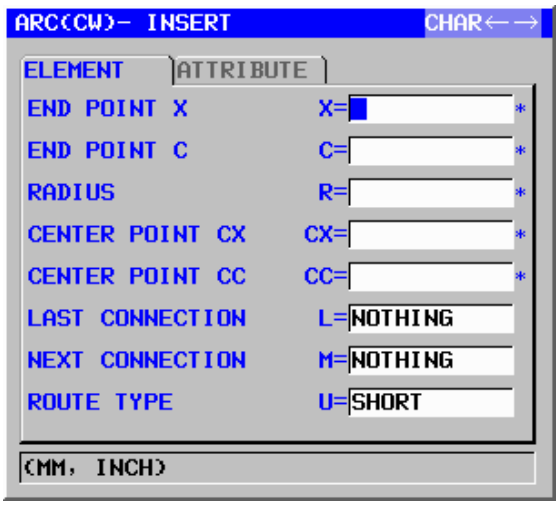
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT C	C coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT X	X coordinate of the end point of a straight line (input value)
D*	END POINT C	C coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [0] : Not in contact with the immediately following figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arc (CW): G1502 (XC plane)
Arc (CCW): G1503 (XC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
X*	END POINT X	X coordinate of an arc end point Remarks) Incremental programming is possible.
C*	END POINT C	C coordinate of an arc end point Remarks) Incremental programming is possible.
R*	RADIUS	Arc radius
CX*	CENTER POINT CX	X coordinate of an arc center
CC*	CENTER POINT CC	C coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
U	ROUTE TYPE	[LONG] : An arc with a long route is made [SHORT] : An arc with a short route is made (initial value)

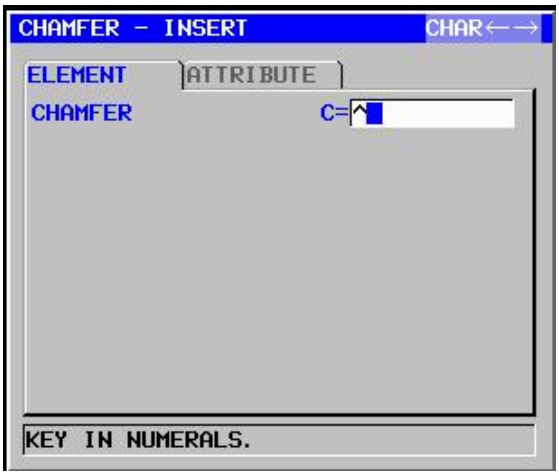
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT C	C coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT C	C coordinate of an arc center (calculation result)
C*	END POINT X	X coordinate of an arc end point (input value)
D*	END POINT C	C coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CX	X coordinate of an arc center (input value)
Q*	CENTER POINT CC	C coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
U	ROUTE TYPE	[1] : An arc with a long route is made [0] : An arc with a short route is made (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Chamfering: G1504 (XC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
C	CHAMFER	Amount of chamfering (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		Meaning
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

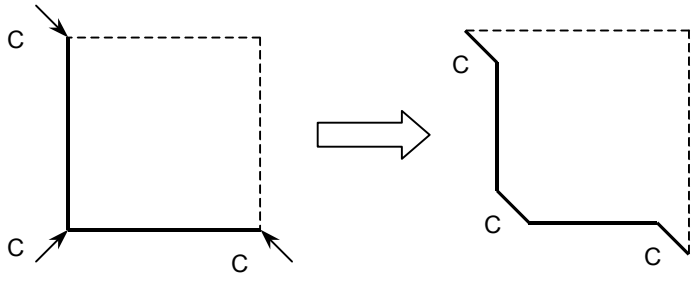
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT C	C coordinate of the end point of a straight line (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

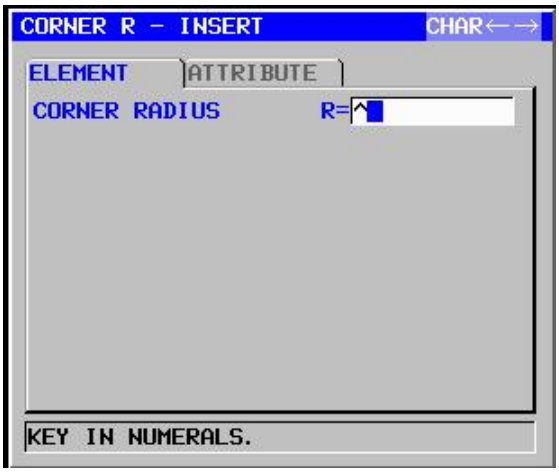
NOTE

3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.



Dotted line : blank element
Normal line : part element

Corner rounding: G1505 (XC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
R	CORNER RADIUS	Corner rounding (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT C	C coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT C	C coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : clockwise [3] : counterclockwise
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.
3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.

End of arbitrary figures: G1506 (XC plane)

This block is output at the end of a series of arbitrary-figure blocks.

NOTE
A series of arbitrary figures must be enclosed between G1500 (start point) and G1506 (end of arbitrary figures).

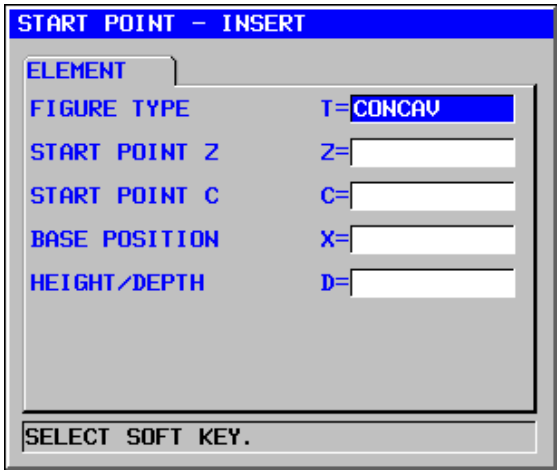
5.4 ARBITRARY FIGURES FOR THE CYLINDRICAL SURFACE (ZC PLANE)

The following types of milling can be specified also for the cylindrical surface (ZC plane), and arbitrary figures in the ZC plane can be used in these milling types.

1. Facing
2. Contouring (Side cutting)
3. Pocketing
4. Grooving
5. Emboss machining

NOTE
See Chapter 1, "Milling," in Part III for detailed descriptions about data to be entered for each type of cycle machining.

Start point: G1600 (ZC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE **(Note 2)**	[FACE] : Used as a figure in facing [CONVEX] : Used as an outer-perimeter figure in contouring [CONCAV] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [GROOVE] : Used as a figure in grooving Remarks) Be sure to select [FACE] if facing is selected.
Z	START POINT Z	Z coordinate of the start point of an arbitrary figure
C	START POINT C	C coordinate of the start point of an arbitrary figure
X	BASE POSITION	Position of the machining surface of an arbitrary figure

ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining.
W	GROOVE WIDTH	Groove width (Positive value) Remarks) This item is displayed in Grooving.
P	FIGURE ATTRIBUTE	[RIGHT] : The right side of an entered figure as cutting [LEFT] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring.
Y	ROTATION AXIS NAME	[C] : The rotation axis is the C axis. [A] : The rotation axis is the A axis (No.27000#1=1) [B] : The rotation axis is the B axis (No.27000#2=1) [E] : The rotation axis is the E axis (No.27000#3=1) Remarks) This item is displayed when the parameter No.27000#0=1.

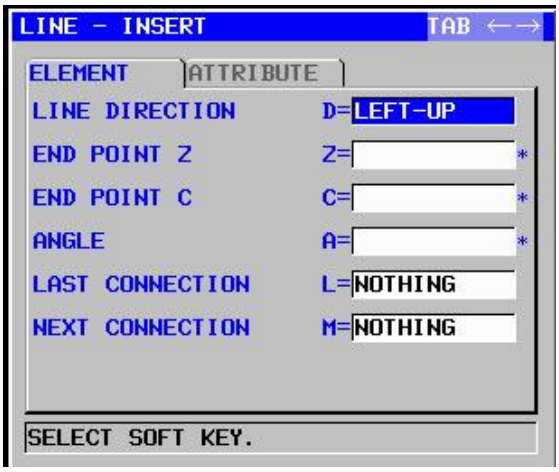
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.
2 Select, with a figure type for a start point, which milling operation is to use an arbitrary figure. Data to be entered for the start point varies depending on what figure type is selected. For details, see descriptions about the respective milling types.

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		**Meaning**
T	FIGURE ATTRIBUTE	[1] : Used as a figure in facing [2] : Used as an outer-perimeter figure in contouring [3] : Used as an inner-perimeter figure in contouring and emboss machining or a figure in pocketing [4] : Used as a figure in grooving (input value)
H	START POINT Z	Z coordinate of the start point (input value)
V	START POINT C	C coordinate of the start point (input value)
I	START POINT (Z AXIS)	Z coordinate of an arc end point (calculation result)
J	START POINT (C AXIS)	C coordinate of an arc end point (calculation result)
B	BASE POSITION	Position of the machining surface (input value)
L	HEIGHT/DEPTH	Height or Depth from Base position to cutting surface Remarks) This item is displayed in Contouring, Pocketing, Grooving and Emboss machining. (input value)
D	GROOVE WIDTH	Groove width (Positive value) Remarks) This item is displayed in Grooving. (input value)

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		Meaning
P	FIGURE ATTRIBUTE	[1] : The right side of an entered figure as cutting [2] : The left side of an entered figure as cutting Remarks) This item is displayed in Open figure of Contouring. (input value)
Y	FACE POSITION	[1] : The rotation axis is the C axis. [2] : The rotation axis is the A axis (No.27001#1=1) [3] : The rotation axis is the B axis (No.27001#2=1) [4] : The rotation axis is the E axis (No.27001#3=1) (input value)

NOTE
3 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Straight line: G1601 (ZC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key.
Z*	END POINT Z	Z coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
C*	END POINT C	C coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
A*	ANGLE	Straight-line angle Remarks) This data may not be displayed depending on the value entered as the direction of the straight line.
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)

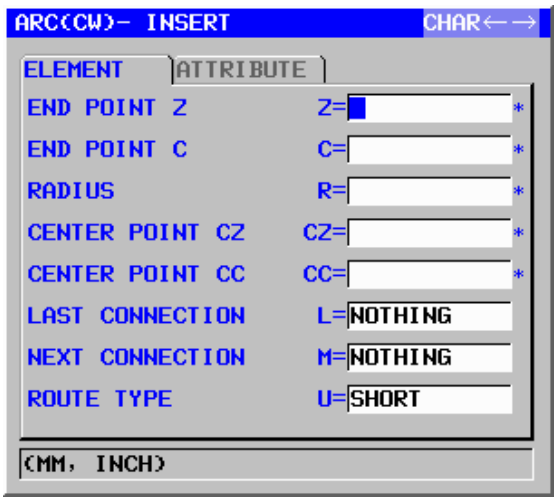
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
V	END POINT C	C coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT Z	Z coordinate of the end point of a straight line (input value)
D*	END POINT C	C coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [0] : Not in contact with the immediately following figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arc (CW): G1602 (ZC plane)
Arc (CCW): G1603 (ZC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
Z*	END POINT Z	Z coordinate of an arc end point Remarks) Incremental programming is possible.
C*	END POINT C	C coordinate of an arc end point Remarks) Incremental programming is possible.
R*	RADIUS	Arc radius
CZ*	CENTER POINT CZ	Z coordinate of an arc center
CC*	CENTER POINT CC	C coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
U	ROUTE TYPE	[LONG] : An arc with a long route is made [SHORT] : An arc with a short route is made (initial value)

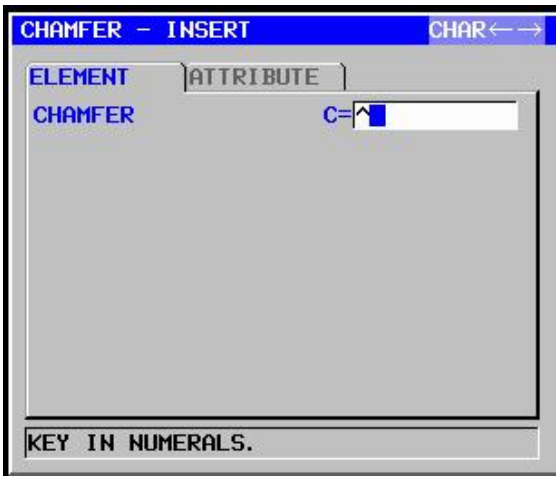
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Z	Z coordinate of an arc end point (calculation result)
V	END POINT C	C coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT Z	Z coordinate of an arc center (calculation result)
J	CENTER POINT C	C coordinate of an arc center (calculation result)
C*	END POINT Z	Z coordinate of an arc end point (input value)
D*	END POINT C	C coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CZ	Z coordinate of an arc center (input value)
Q*	CENTER POINT CC	C coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
U	ROUTE TYPE	[1] : An arc with a long route is made [0] : An arc with a short route is made (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Chamfering: G1604 (ZC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
C	CHAMFER	Amount of chamfering (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		Meaning
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

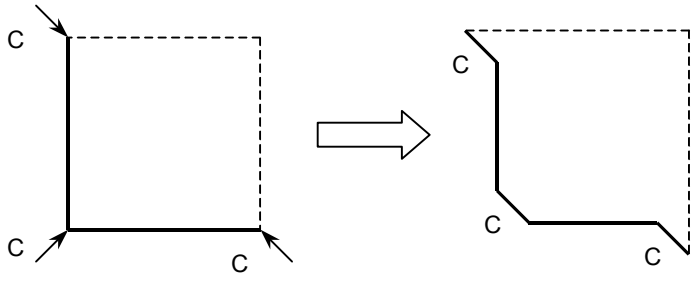
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
V	END POINT C	C coordinate of the end point of a straight line (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

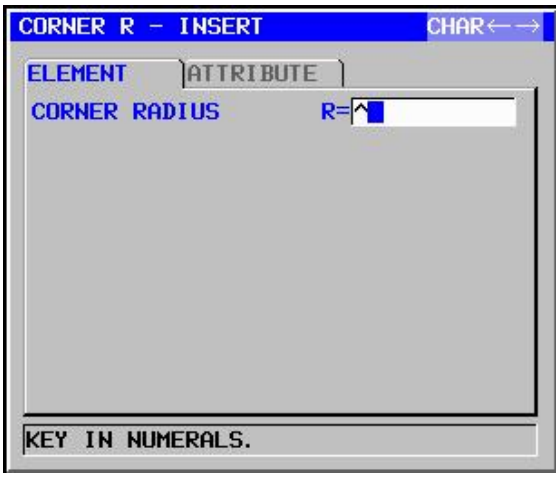
NOTE

3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.



Dotted line : blank element
Normal line : part element

Corner rounding: G1605 (ZC plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
R	CORNER RADIUS	Corner rounding (radius value, positive value)

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT Z	Z coordinate of an arc end point (calculation result)
V	END POINT C	C coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT Z	Z coordinate of an arc center (calculation result)
J	CENTER POINT C	C coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : Clockwise [3] : Counterclockwise
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.
3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.

End of arbitrary figures: G1606 (ZC plane)

This block is output at the end of a series of arbitrary-figure blocks.

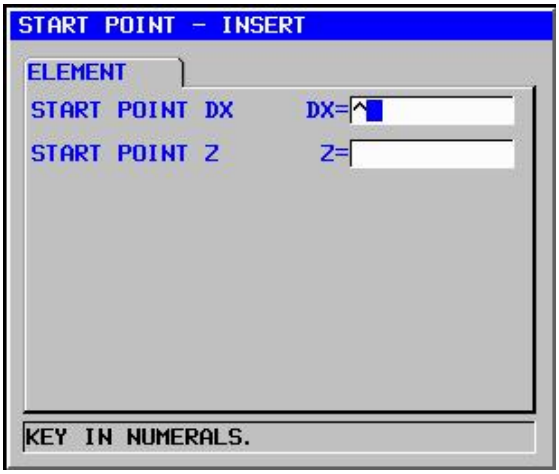
NOTE
A series of arbitrary figures must be enclosed between G1600 (start point) and G1606 (end of arbitrary figures).

5.5 ARBITRARY FIGURES FOR TURNING (ZX PLANE)

Arbitrary figures in the ZX plane can be used in turning.
1. Outer surface rough/semifinish/finish turning
2. Inner surface rough/semifinish/finish turning
3. End surface rough/semifinish/finish turning

NOTE
1 See Section 2.1, "Turning," in Part III for detailed descriptions about data to be entered for each type of cycle machining.
2 When entering an arbitrary figure for turning, make its end point coincide with its start point. To put another way, combine parts figures with blank figures in such a way that a closed curve is created.
Turning is carried out in this closed curve.
3 It is possible to specify "PART" and "BLANK" as "ELEMENT TYPE" for an individual arbitrary figure. This "BLANK" can be used to enter an arbitrary figure resembling a blank figure to be actually machined, so any portion other than the blank portion will not be cut. This way, optimum rough turning can be realized.

Start point: G1450 (ZX plane)



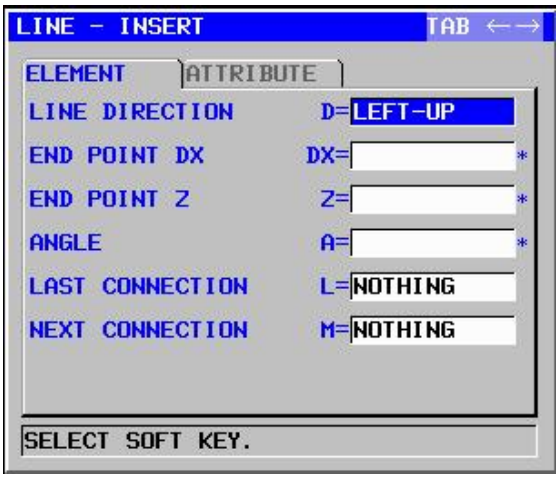
ELEMENT (INPUT DATA) (Note 1)		
Data item		Meaning
DX	START POINT DX	X coordinate of the start point of an arbitrary figure
Z	START POINT Z	Z coordinate of the start point of an arbitrary figure

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	START POINT DX	X coordinate of the start point (input value)
V	START POINT Z	Z coordinate of the start point (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Straight line: G1451 (ZX plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key.
DX*	END POINT DX	X coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
Z*	END POINT Z	Z coordinate of the end point of a straight line Remarks1) This data may not be displayed depending on the value entered as the direction of the straight line. Remarks2) Incremental programming is possible.
A*	ANGLE	Straight-line angle Remarks) This data may not be displayed depending on the value entered as the direction of the straight line.
L	LAST CONNECTION	[TANGNT]: In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT]: In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

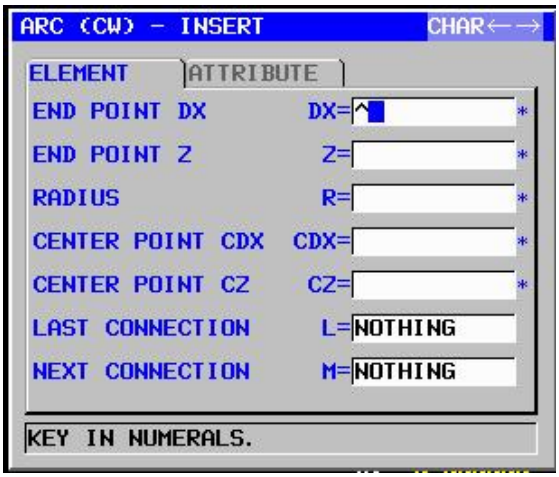
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
F	FINISH FEEDRATE	Cutting feedrate for finishing (positive value) Remarks) It is possible to set the feedrate to each block on Finishing.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT DX	X coordinate of the end point of a straight line (input value)
D*	END POINT Z	Z coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [0] : Not in contact with the immediately following figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)
F	FINISH FEEDRATE	Cutting feedrate for finishing (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arc (CW): G1452 (ZX plane)
Arc (CCW): G1453 (ZX plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
DX*	END POINT DX	X coordinate of an arc end point Remarks) Incremental programming is possible.
Z*	END POINT Z	Z coordinate of an arc end point Remarks) Incremental programming is possible.
R*	RADIUS	Arc radius
CDX*	CENTER POINT CDX	X coordinate of an arc center
CZ*	CENTER POINT CZ	Z coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : In contact with the immediately preceding figure [NO SET] : Not in contact with the immediately preceding figure (initial value)
M	NEXT CONNECTION	[TANGNT] : In contact with the immediately following figure [NO SET] : Not in contact with the immediately following figure (initial value)
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

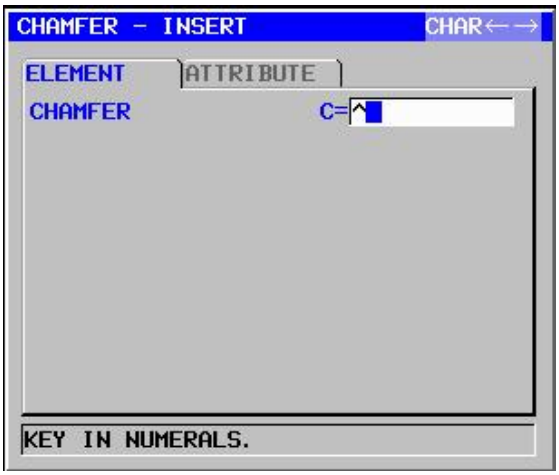
ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
F	FINISH FEEDRATE	Cutting feedrate for finishing (positive value) Remarks) It is possible to set the feedrate to each block on Finishing.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
C*	END POINT X	X coordinate of an arc end point (input value)
D*	END POINT Z	Z coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CDX	X coordinate of an arc center (input value)
Q*	CENTER POINT CZ	Z coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [0] : Not in contact with the immediately preceding figure (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)
F	FINISH FEEDRATE	Cutting feedrate for finishing (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Chamfering: G1454 (ZX plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
C	CHAMFER	Amount of chamfering (radius value, positive value)
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
F	FINISH FEEDRATE	Cutting feedrate for finishing (positive value) Remarks) It is possible to set the feedrate to each block on Finishing.

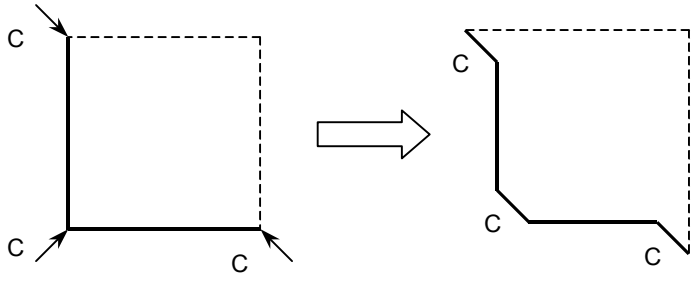
NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
F	FINISH FEEDRATE	Cutting feedrate for finishing (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

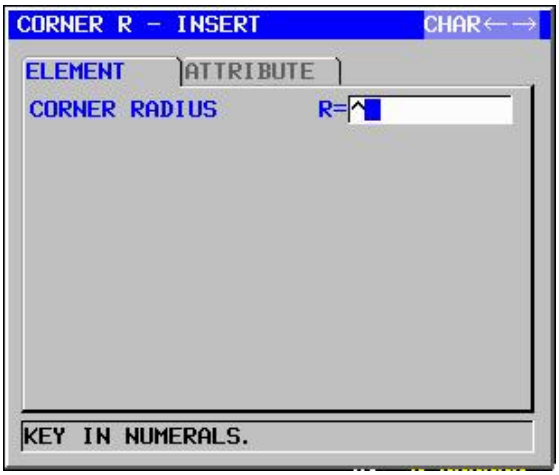
NOTE

3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.



Dotted line : blank element
Normal line : part element

Corner rounding: G1455 (ZX plane)



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
R	CORNER RADIUS	Corner rounding (radius value, positive value)
T	ELEMENT TYPE	[PART] : Cut as parts [BLANK] : Cut as a blank portion Remarks) The system sets the last data as an initial value.

ATTRIBUTE (INPUT DATA) (Note 1)		
Data item		**Meaning**
F	FINISH FEEDRATE	Cutting feedrate for finishing (positive value) Remarks) It is possible to set the feedrate to each block on Finishing.

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT & ATTRIBUTE (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : Clockwise [3] : Counterclockwise
T	ELEMENT TYPE	[1] : Cut as parts [2] : Cut as a blank portion (input value)
F	FINISH FEEDRATE	Cutting feedrate for finishing (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.
3 By setting the parameter No.14851#0=1, Corner element between a blank element and a part element can be created in the opposite direction.

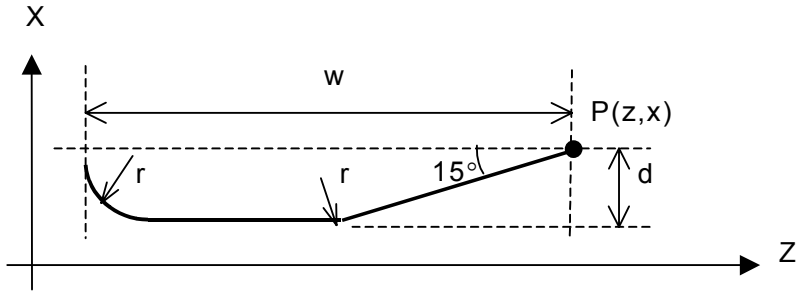
End of arbitrary figures: G1456 (ZX plane)

This block is output at the end of a series of arbitrary-figure blocks.

NOTE
A series of arbitrary figures must be enclosed between G1450 (start point) and G1456 (end of arbitrary figures).

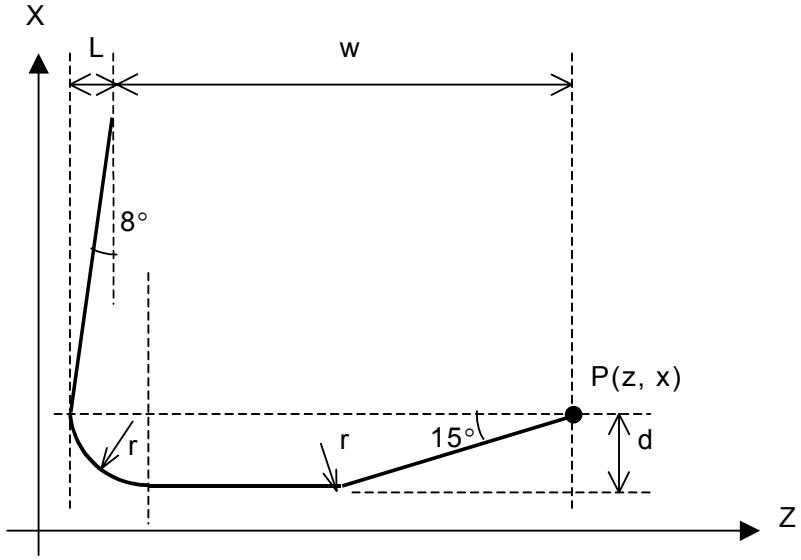
DIN509 :

Pressing the [DIN509] soft key displays the sub-window. By entering necessary data, a neck figure for DIN509 can be created.



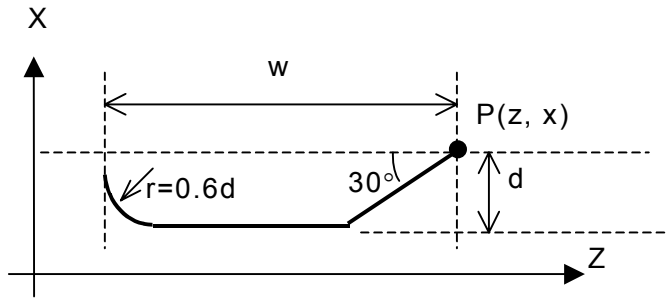
DIN509F :

Pressing the [D509-F] soft key displays the sub-window. By entering necessary data, a neck figure for DIN509-F can be created.



DIN76 :

Pressing the [DIN76] soft key displays the sub-window. By entering necessary data, a neck figure for DIN76 can be created.



5.6 ARBITRARY FIGURE COPY FUNCTIONS

A specified area of figure can be copied (parallel copy, mirror copy, or rotational copy) for addition as a new figure on the arbitrary figure creation screen or contour program input screen.

1) Parallel copy
A specified area of arbitrary figure is copied specified times after the selected area.

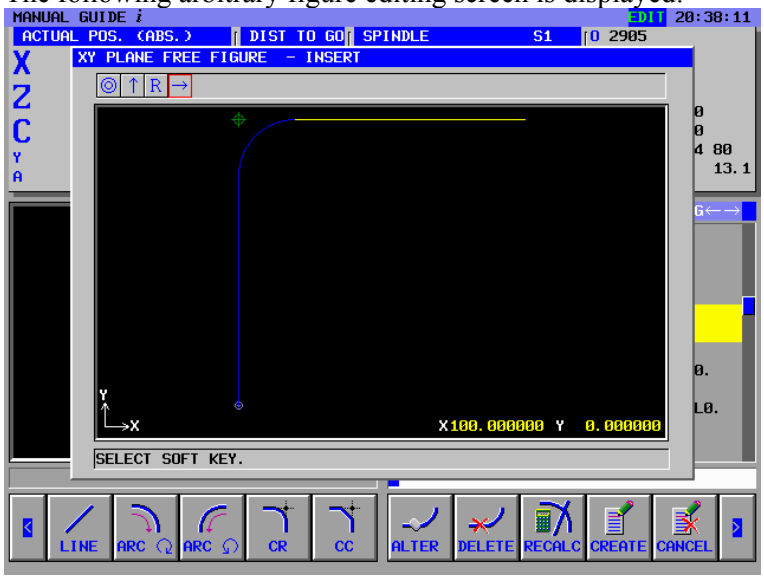
2) Rotational copy
A specified area of arbitrary figure is copied specified times around specified coordinates after the selected area.

3) Mirror copy
A specified area of arbitrary figure is mirror-copied relative to a specified symmetry axis.

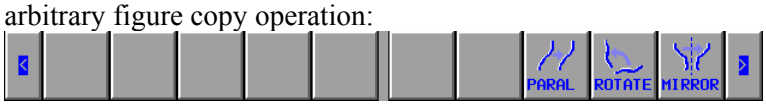
5.6.1 Selecting an Arbitrary Figure Copy Function

An arbitrary figure copy function can be selected by pressing the [PARAL], [ROTATE], or [MIRROR] soft key displayed when the [>] soft key is pressed on the arbitrary figure editing screen.

<1> The following arbitrary figure editing screen is displayed:

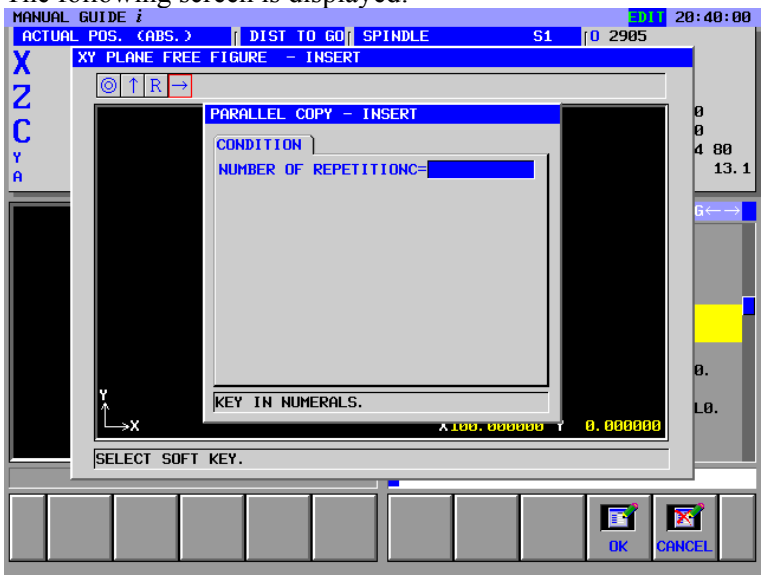


<2> Pressing the [>] soft key displays the following soft keys for arbitrary figure copy operation:



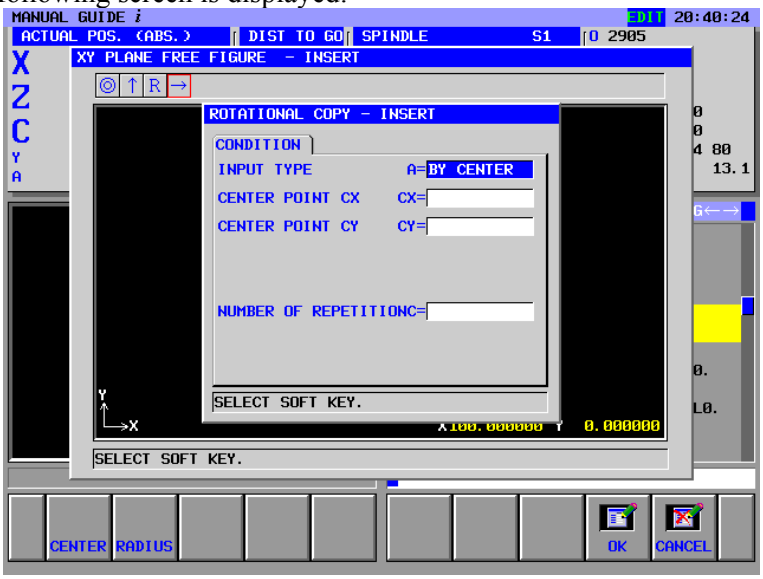
5.6.2 Copy Condition Input Screen

1) Parallel copy
Parallel copy can be selected by pressing the [PARAL] soft key.
The following screen is displayed:



NUMBER OF REPETITIONC = :
Enter the number of times a selected figure is to be copied.

2) Rotational copy
Rotational copy can be selected by pressing the [ROTATE] soft key.
The following screen is displayed:



INPUT TYPE :
Select [CENTER] or [RADIUS] to specify the method of condition input.

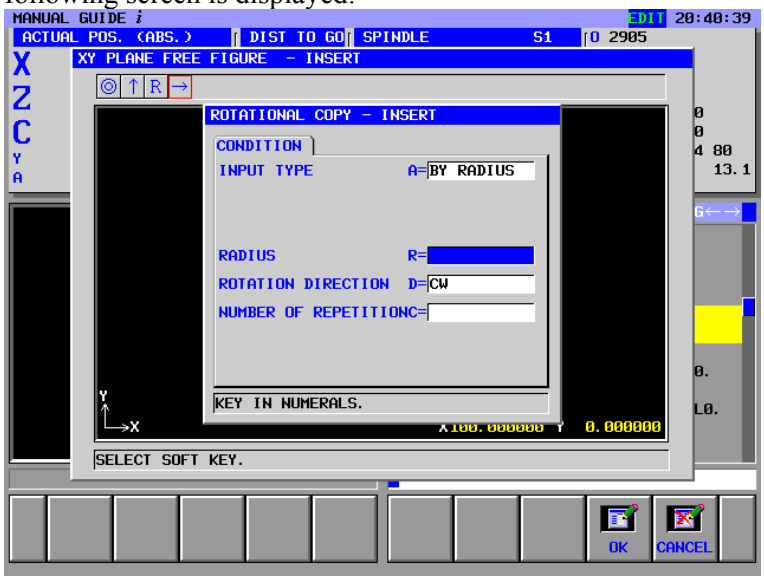
CENTER POINT CX, CENTER POINT CY :

Enter the X coordinate and Y coordinate of a rotation center around which a rotation is made.

NUMBER OF REPETITIONC = :

Enter the number of times a selected figure is to be copied.

When "BY RADIUS" is selected in "INPUT TYPE", the following screen is displayed:



RADIUS :

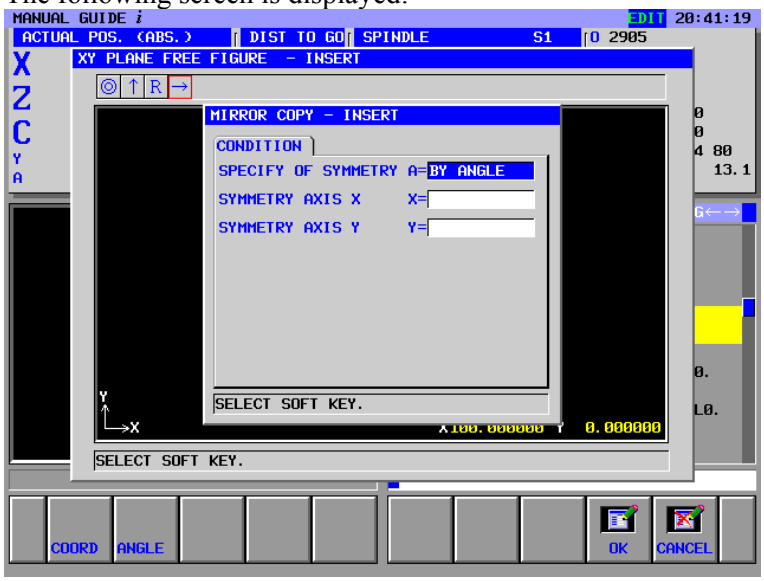
Enter the distance from the start or end point of a selected figure to the rotation center point.

ROTATION DIRECTION :

Select [CW] or [CCW] to specify the direction of rotation copy operation.

3) Mirror copy

Mirror copy can be selected by pressing the [MIRROR] soft key. The following screen is displayed:



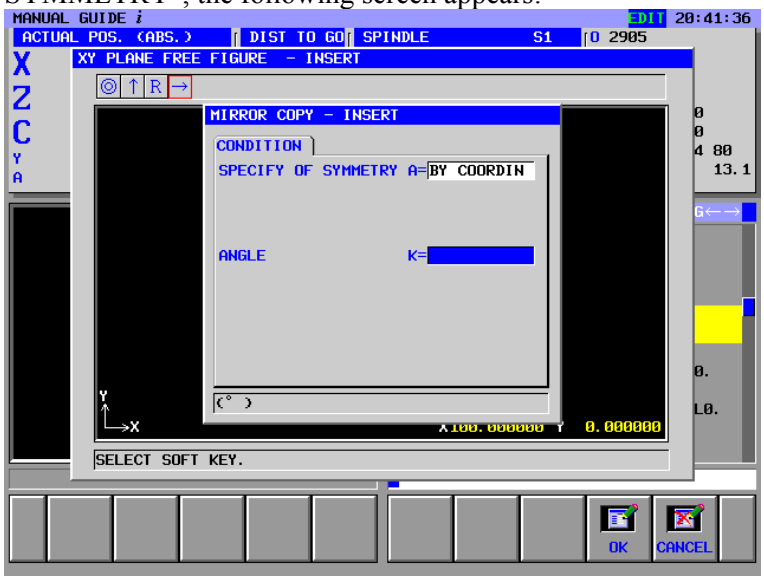
SPECIFY OF SYMMETRY :

Use the [COORD] or [ANGLE] soft key to select the method for specifying a symmetry axis.

SYMMETRY AXIS X, SYMMETRY AXIS Y :

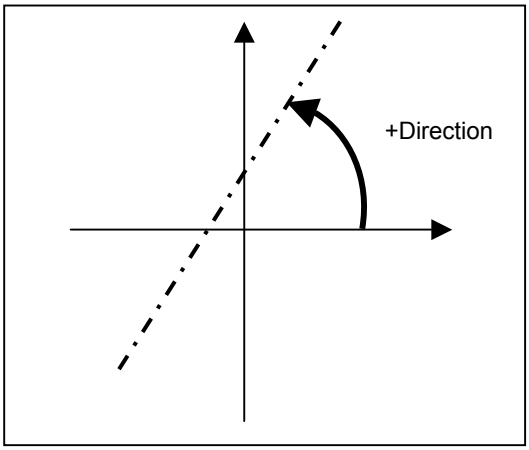
Enter X and Y coordinates through which the symmetry axis for mirror copying passes. The symmetry axis should pass through the end point of a specified figure and these X and Y coordinates.

When "BY ANGLE" is selected in "SPECIFY OF SYMMETRY", the following screen appears:



ANGLE :

Enter the angle between a symmetry axis used for mirror copy operation and the horizontal axis. The plus direction of the horizontal axis represents 0°. Enter a positive value for an angle made toward the plus vertical axis direction. Enter a negative value for an angle made toward the minus vertical axis direction.



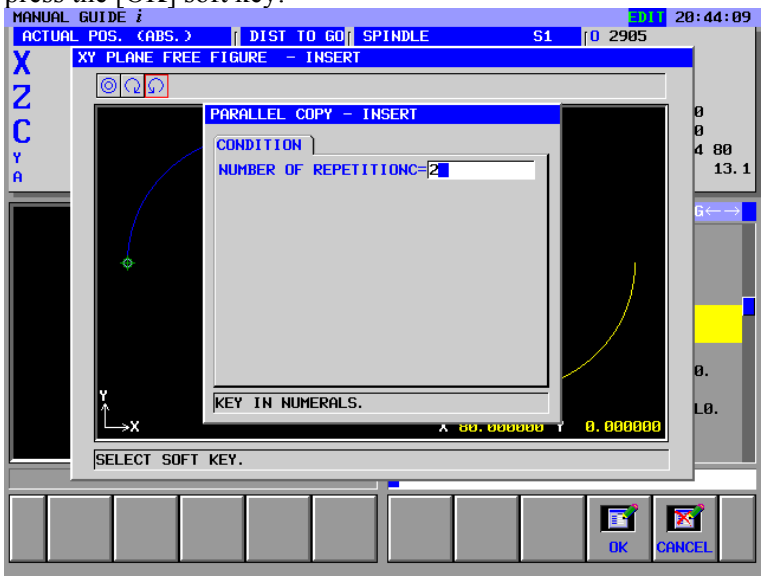
NOTE

Data entered on the parallel copy input screen, rotation copy input screen, or mirror copy input screen is not output to the program.
Each figure entered by a type of copy operation is automatically expanded as an arbitrary figure element, so that each figure can be modified. However, the state before copy operation cannot be automatically resumed.

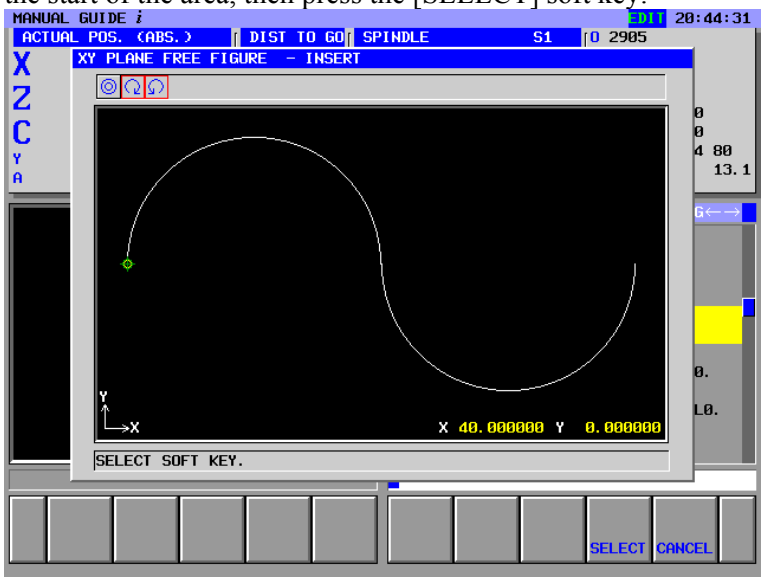
5.6.3 Executing Arbitrary Figure Copy Operation

Use the operation procedure described below.
(Example) Parallel copy

<1> Enter the number of repetition on the copy input screen then press the [OK] soft key.



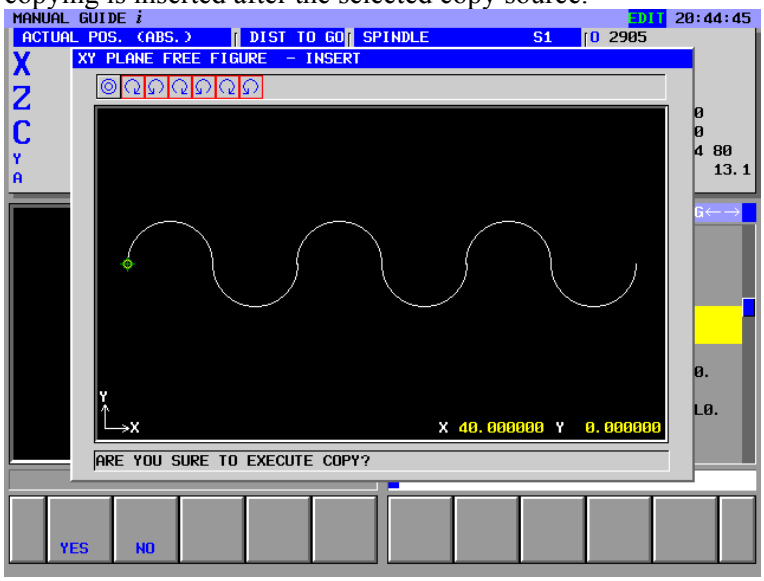
<2> The screen for specifying an area of figure used as a copy source appears. Move the cursor to the icon of the figure element at the start of the area, then press the [SELECT] soft key.



Pressing the [CANCEL] soft key returns the screen display to the arbitrary figure editing screen.

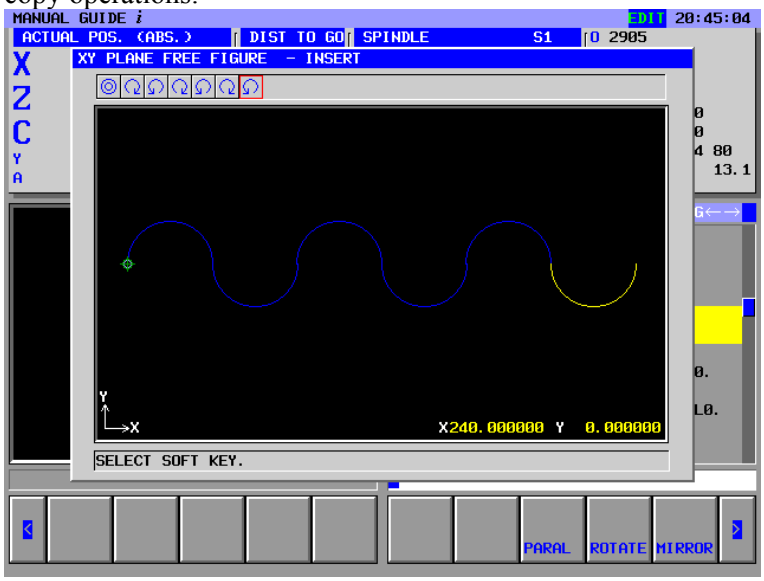
NOTE
A selected copy source ranges from the figure where the cursor is placed when the [SELECT] soft key is pressed to the last figure.

<3> At this time, the graphic window displays a figure produced by executing copy operations repeatedly. A figure produced by copying is inserted after the selected copy source.



Pressing the [NO] soft key returns the screen display to the previous step.

<4> Pressing [YES] at the end fixes a figure produced by repeated copy operations.



5.6.4 Figure after Arbitrary Figure Copy Execution

When the screen used for entering a copied figure is opened by selecting [ALTER], the set input items differ from those set for the figure before being copied, as described below.

For a straight line:

Only the input items "LINE DIRECTION" and "END POINT" are set.

For an arc:

Only the input items "RADIUS" and "END POINT " are set.

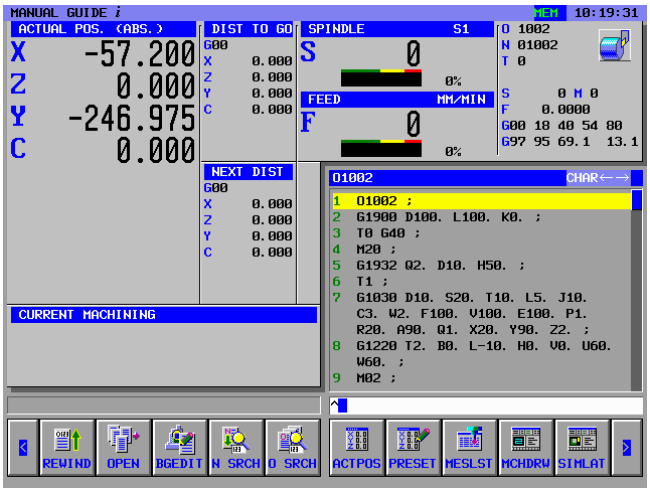
If a copy source figure includes a corner R and chamfer, the corner R and chamfer are an arc and straight line, respectively, in a figure produced by copying.

This is because parallel, rotation, or mirror copy operation is performed according to the end point information of the copy source figure. So, information such as "NEXT CONNECTION" set for a copy source figure is not reflected in a figure produced by copying.

5.6.5 Notes on Arbitrary Figure Copy Operation

- A warning is output if the result of expansion based on an attempt for copy operation exceeds the maximum allowable number of blocks.
- The end point of the last selected copy source figure needs to be fixed.
- The element immediately before a selected copy source figure must not be a corner R or chamfer. Moreover, the starting element and ending element of a selected copy source figure must not be a corner R and chamfer.

6 OPERATIONS IN THE MEM MODE



When the MEM mode is selected with the machine operator's panel, the soft keys shown below appear on the screen of MANUAL GUIDE *i*. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second page.

REWIND	O LIST	BGEDIT	N SRCH	O SRCH	ACTPOS	PRESET	MESLST	MCHDRW	SIMLAT
NC CNV	P TYPE (Note)	Q TYPE (Note)		WRK CO	T-OFS				SETING

For [SETING], [WRK CO], and [T-OFS], see Chapter 10, "SETTING DATA," in Part II.

NOTE
When the optional function of PROGRAM RE-START, the soft-key of [P TYPE] and [Q TYPE] are displayed on the screen.

6.1 REWINDING A MACHINING PROGRAM

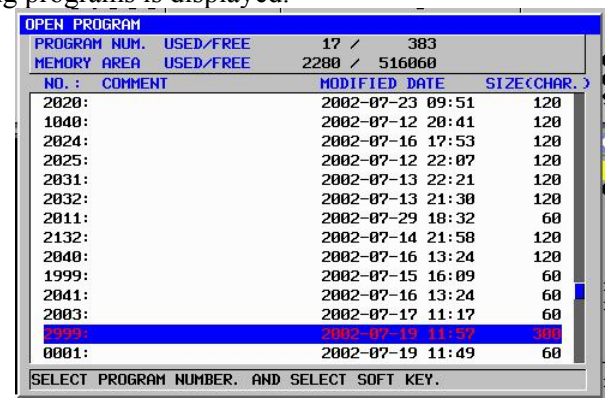
REWIND	O LIST	BGEDIT	N SRCH	O SRCH		ACTPOS	PRESET	MESLST	MCHDRW	SIMLAT

By pressing [REWIND], you can return to the beginning of a selected program.

6.2 EDITING WITH THE MACHINING PROGRAM LIST

REWIND	O LIST	BGEDIT	N SRCH	O SRCH		ACTPOS	PRESET	MESLST	MCHDRW	SIMLAT

By pressing [O-LIST], a window showing a list of registered machining programs is displayed.



The following soft keys appear, allowing you to select machining programs.

				SEARCH			SRTORD	OPEN	CANCEL

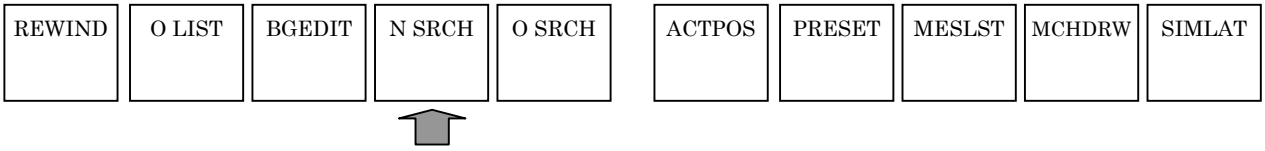
[SEARCH] : A window for searching programs appears. Enter the number of the program you want to search for with numeric keys, then press [SEARCH]. The search window can be closed using [CANCEL].

[OPEN] : After performing the above search operation or moving the cursor to a desired program number by using the cursor keys ↑ and ↓, press [OPEN]. Then the program is selected.

[CANCEL] : Close the list window.

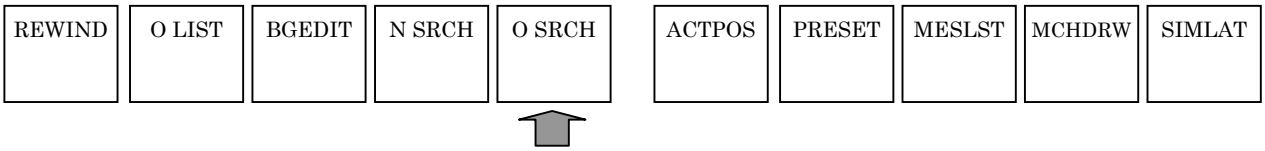
[SRTORD] : This soft key switches the sort order for displaying programs between ascending order and descending order.

6.3 SEARHING FOR A SEQUENCE NUMBER IN A PROGRAM



When you enter the sequence number you want to search for by using numeric keys then press [N SRCH], you can search for the block having the sequence number.
When you press [N SRCH] without entering any data, the sequence number (N) is searched for sequentially.
If the sequence number is not found, a warning is displayed. It can be released by pressing a key.

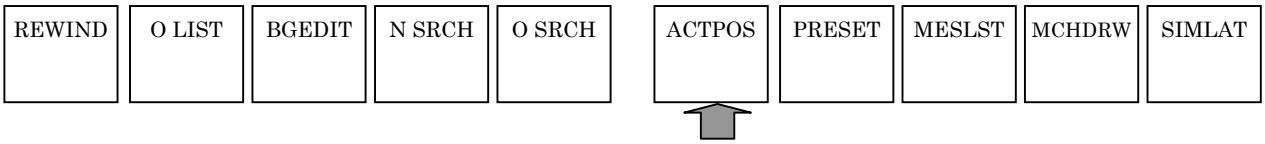
6.4 SELECTING A PROGRAM FOR MACHINING OPERATION



When you enter the program number you want to select by using numeric keys then press [O SRCH], the program with the program number is selected and displayed in the program window.
By pressing [O SRCH] without entering any key, the next program is selected.

NOTE
If the parameter No.3202#3 is set to “1”, pressing [O SRCH] without entering any data, the next program is not selected.

6.5 CHANGING THE CURRENT POSITION DISPLAY

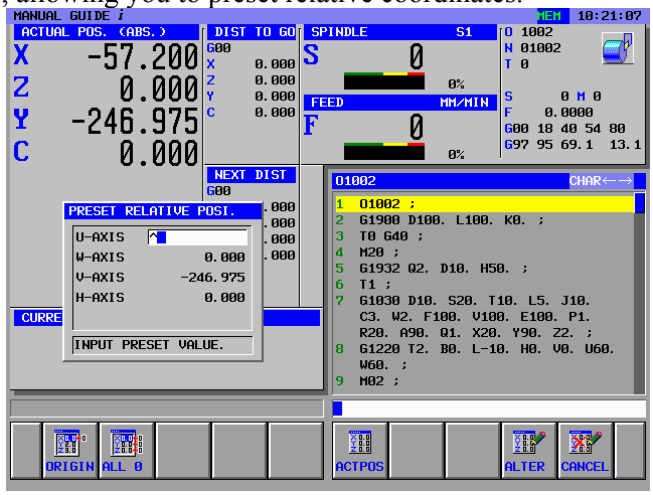


By pressing [ACTPOS], the current position display in the status display window in the upper part of the screen changes in the sequence "ACTUAL POS. (ABS)" → "ACTUAL POS. (RELATIVE)" → "ACTUAL POS. (MACHINE)"
If the manual handle interrupt option function is attached, the position display changes in the sequence "ACTUAL POS. (ABS)" → "ACTUAL POS. (RELATIVE)" → "ACTUAL POS. (MACHINE)" → “HANDLE INTER.(INPUT)” → “HANDLE INTER.(OUTPUT)”

6.6 PRESETTING RELATIVE COORDINATES

REWIND	O LIST	BGEDIT	N SRCH	O SRCH	ACTPOS	PRESET	MESLST	MCHDRW	SIMLAT

By pressing [PRESET], a relative coordinates presetting window appears, allowing you to preset relative coordinates.



When the relative coordinates presetting window is displayed, the following soft keys are displayed:

ORIGIN	ALL 0				ACTPOS			ALTER	CANCEL

[ORIGIN] : Set the axis selected with the cursor keys to "0." At this time, no relative coordinates are preset.

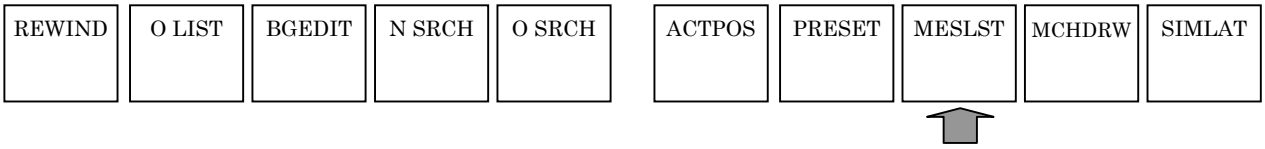
[ALL 0] : Set all axes to "0." At this time, no relative coordinates are preset.

Remark) You can set a desired coordinate value for an axis selected by the cursor by entering the value you want to preset by using numeric keys then pressing the **INPUT** key. At this time, as described above, the relative coordinates are not yet preset.

[ACTPOS] : Change the current position display in the status display window in the upper part of the screen in the sequence "ACTUAL POS. (ABS)" → "ACTUAL POS. (RELATIVE)" → "ACTUAL POS. (MACHINE)."
If the manual handle interrupt option function is attached, the position display changes in the sequence "ACTUAL POS. (ABS)" → "ACTUAL POS. (RELATIVE)" → "ACTUAL POS. (MACHINE)" → "HANDLE INTER.(INPUT)" → "HANDLE INTER.(OUTPUT)"

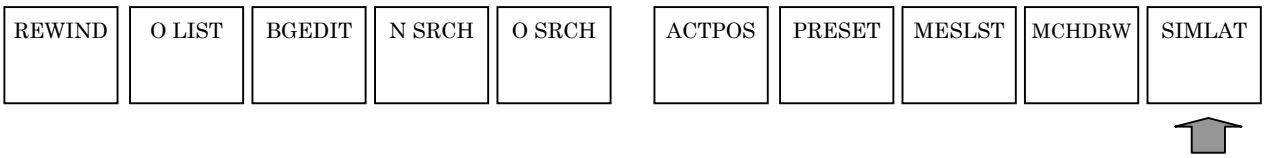
[ALTER] : Preset the relative coordinates to coordinate values set by the above operation. This soft key also closes the relative coordinates presetting window.
[CANCEL] : Cancel presetting of coordinates and just close the window.

6.7 DISPLAYING MEASUREMENT RESULTS



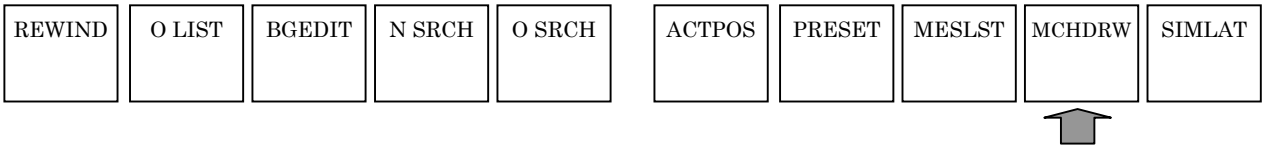
By pressing [MESLST], a window showing a list of measurement results can be displayed. For details of this window, see "MANUAL GUIDE *i* Operator's Manual (Set-up Guidance Function)" or the manual made by machine tool builder.

6.8 DISPLAYING THE MACHINING SIMULATION WINDOW



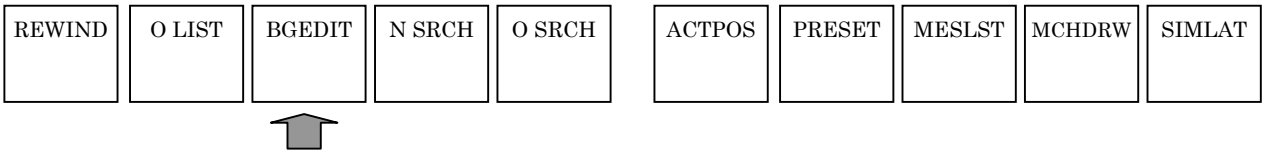
By pressing [SIMLAT], a machining simulation window can be displayed. For machining simulation, see Chapter 9, "OPERATIONS IN MACHINING SIMULATION," in Part II.
Press [GRPOFF] to close the machining simulation window.

6.9 DISPLAYING THE DRAWING-DURING-MACHINING WINDOW



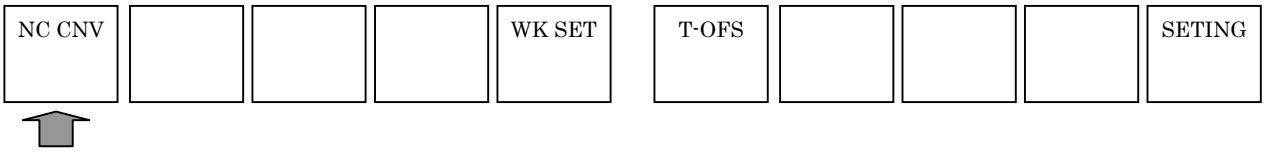
By pressing [MCHDRW], the drawing-during-machining window can be displayed. For details of drawing during machining, see II.9, "OPERATIONS IN MACHINING SIMULATION".
Press [GRPOFF] to close the drawing-during-machining window.

6.10 BG EDITING



By pressing [BGEDIT], the background editing function can be used. For details of the background editing function, see II.11, "OPERATIONS IN BACKGROUND EDITING".

6.11 NC PROGRAM CONVERSION



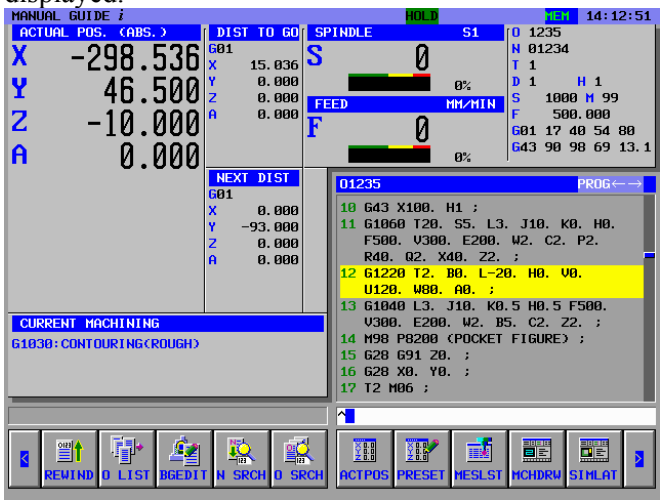
By pressing [NC CNV], the NC program conversion function can be used. For details of the NC program conversion function, see II.12, "OPERATIONS WITH THE NC PROGRAM CONVERSION FUNCTION".

6.12 NEXT-BLOCK DISPLAY FUNCTION

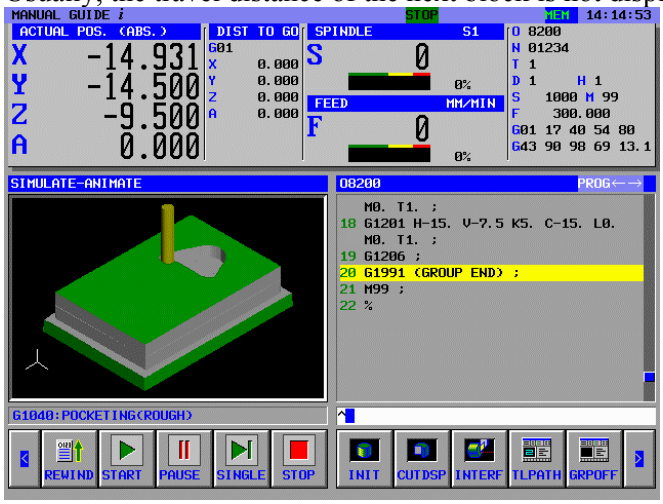
During simulation execution or operation in the MEM mode or MDI mode, the travel distance data of the block to be executed next is displayed.

NOTE
1 During actual machining, the travel distance of the actually executed block is displayed in the “DIST TO GO” columns, but during machining simulation, 0 value is displayed always in those columns.
2 In order to display the travel distance data of the block to be executed next during single block operation, set the parameter No.3106#2 to “1”.

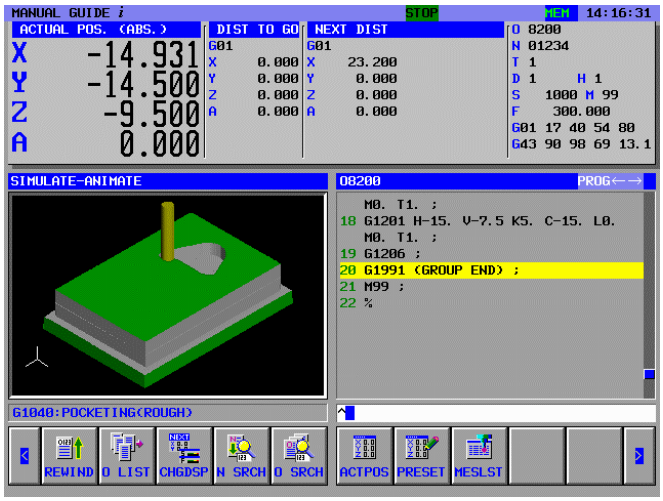
- In case of machining simulation nor path drawing during actual machining is not executed
  The travel distance data of the next block is automatically displayed.



- In case of machining simulation or path drawing during actual machining is executed
  Usually, the travel distance of the next block is not displayed.



Pressing [CHGDSP] erases the display of spindle and actual feedrate information and displays the travel distance data of the next block. Pressing [CHGDSP] again returns the screen to the display of spindle and actual feedrate information.

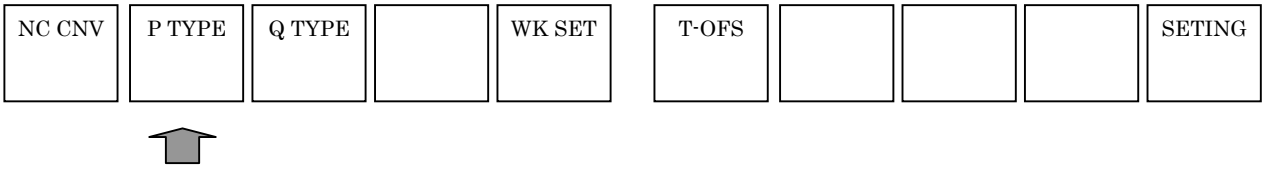


6.13 PROGRAM RESTART FUNCTION

When a tool is broken, or machining is to be restarted after holidays, for example, the block number or sequence number of a block from which machining is to be restarted can be specified using this function to enable machining to be restarted from the block. This function can also be used as a high-speed program check function.
Two methods of restarting are available: P type and Q type.

NOTE
To use this function, the order of movement to a program restart position needs to be set in parameter No. 7310.

6.13.1 [P TYPE] Soft-key



(1) When the program restart signal G006#0 turns to 0 :
Pressing the [P TYPE] soft key has no effect. (Nothing occurs.)

(2) When the program restart signal G006#0 turns to 1 :
<1> Enter one of the following in the key-in buffer:
1. Sequence number
2. N sequence number
3. B block number
4. Nxxxyyyyy (xxx: Number of repetition (3-digit) yyyyy:Sequence number (5-digit))
5. xxxyyyyy (xxx: Number of repetition (3-digit) yyyyy:Sequence number (5-digit))

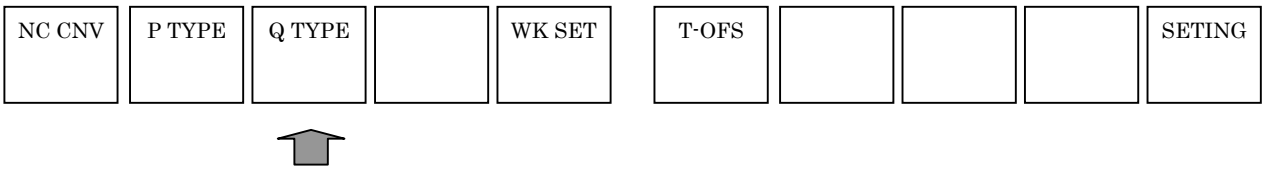
(*) For items 4 or 5, enter an 8-digit number. For sequence number 3 for two times, for example, enter N00200003.

<2> Press the [P TYPE] soft key.
If a value entered into the key-in buffer satisfies none of the items 1 through 5, an error is indicated.

<3> The program restart screen is displayed.

NOTE
If [P TYPE] is pressed without performing an automatic operation even once after the power is turned on, an emergency stop is reset, or a P/S alarm (No. 094 to No. 097) is reset, P/S alarm 097 is issued.

6.13.2 [Q TYPE] Soft-key



(1) When the program restart signal G006#0 turns to 0 :
Pressing the [Q TYPE] soft key has no effect. (Nothing occurs.)

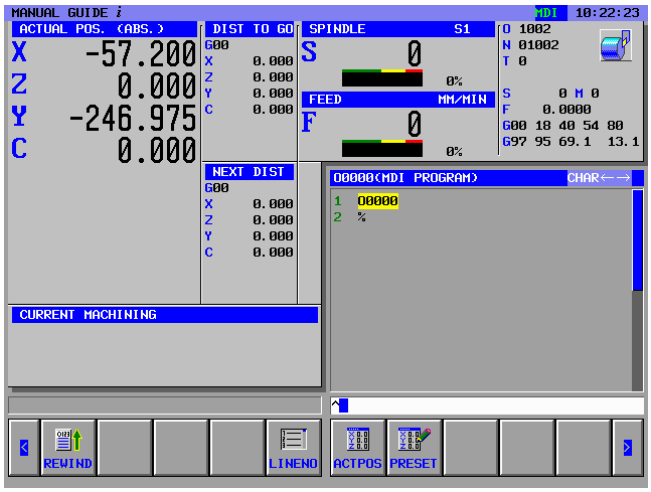
(2) When the program restart signal G006#0 turns to 1 :
<1> Enter one of the following in the key-in buffer:
1. Sequence number
2. N sequence number
3. B block number
4. Nxxxyyyyy (xxx: Number of repetition (3-digit) yyyyy:Sequence number (5-digit))
5. xxxyyyyy (xxx: Number of repetition (3-digit) yyyyy:Sequence number (5-digit))

(*) For items 4 or 5, enter an 8-digit number. For sequence number 3 for two times, for example, enter N00200003.

<2> Press the [Q TYPE] soft key.
If a value entered into the key-in buffer satisfies none of the items 1 through 5, an error is indicated.

<3> The program restart screen is displayed.

7 OPERATIONS IN THE MDI MODE



When the MDI mode is selected with the machine operator's panel, the soft keys shown below appear on the screen of MANUAL GUIDE *i*. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second and third pages sequentially.

REWIND				LINENO	ACTPOS	PRESET			

REWIND		SRCH↑	SRCH↓	CHSIZE	COPY	CUT	DELETE	KEYPST	PASTE

				WRK CO	OFFSET				SETING

For soft keys [ACTPOS] and [PRESET] on the first page, see the following sections:

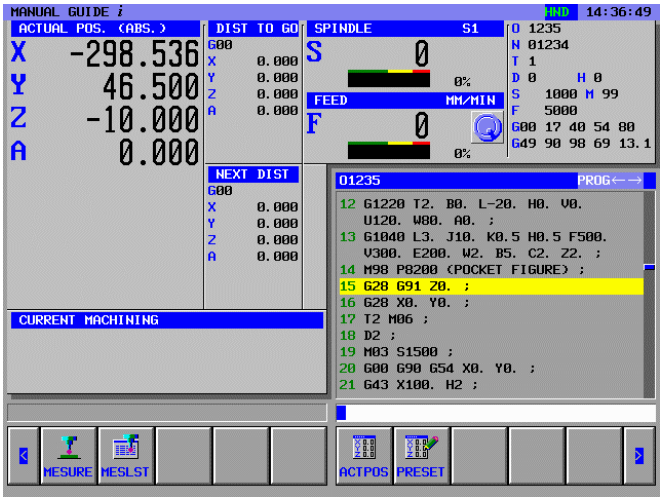
6.5 CHANGING THE CURRENT POSITION DISPLAY
6.6 PRESETTING RELATIVE COORDINATES

The soft keys on the second and third pages are used for editing machining programs entered by MDI. For details on these soft keys, see the following sections:

3.1 MACHINING PROGRAM WINDOW AND EDITING
3.6 SEARCH (FORWARD AND BACKWARD)
3.7 CUT
3.8 COPY
3.9 PASTE
3.10 DELETION
3.11 KEY-IN PASTE

For [SETING], see Chapter 10, "SETTING DATA," in Part II.

8 OPERATIONS IN THE MANUAL MODE (HANDLE AND JOG)



When the handle or jog mode is selected with the machine operator's panel, the following soft keys appear on the screen of MANUAL GUIDE *i*:

MESURE	MESLST				ACTPOS	PRESET			SETING

NOTE
1 [MESURE] and [MESLST] are displayed only when the Set-up guidance optional function is attached.

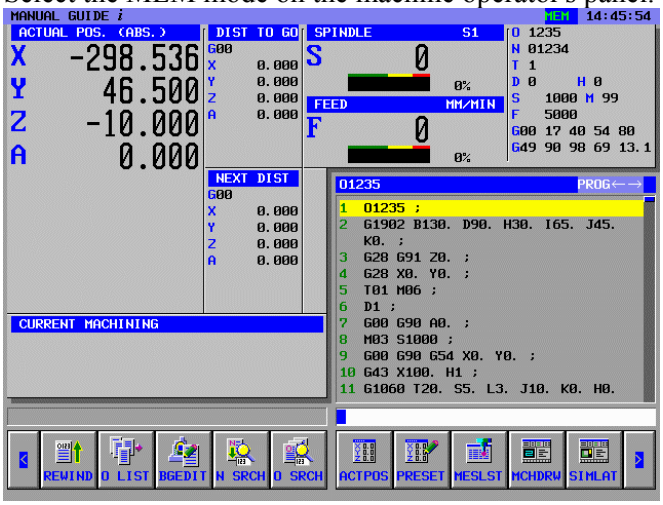
By pressing [MESURE], a window for set-up guidance functions that are performed in the manual mode is displayed. By pressing [MESLST], a window showing a list of measurement results is displayed. For details of these windows, see "Set-up Guidance" in Part IV.
For soft keys [ACTPOS] and [PRESET], see the following sections:

6.5 CHANGING THE CURRENT POSITION DISPLAY
6.6 PRESETTING RELATIVE COORDINATES

For [SETING], see Chapter 10, "SETTING DATA," in Part II.

9 MACHINING SIMULATION AND DRAWING DURING MACHINING

Select the MEM mode on the machine operator's panel.



REWIND	O LIST	BGEDIT	N SRCH	O SRCH		ACTPOS	PRESET	MESLST	MCHDRW	SIMLAT

When you press [MCHDRW], the DRAWING-TOOL PATH screen is displayed.

When you press [SIMLAT], the SIMULATE-ANIMATE screen appears with the following soft keys:

REWIND	START	PROCES	SINGLE	STOP		INIT	CUTDSP	INTERF	TLPATH	GRPOFF

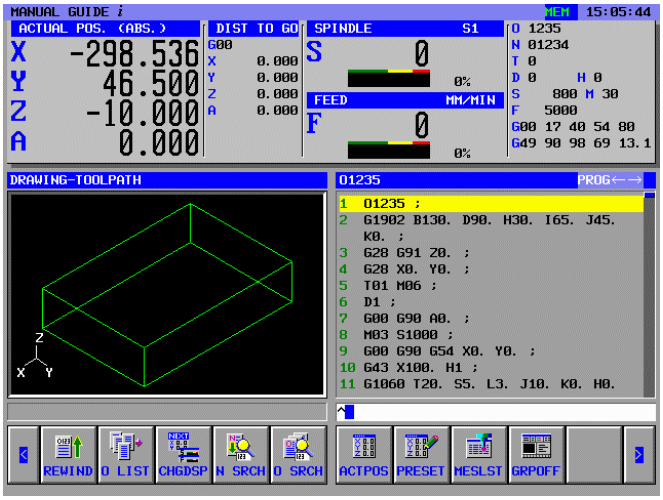
When you press [TLPATH], the SIMULATE-TOOL PATH screen appears with the following soft keys:

REWIND	START	PROCES	SINGLE	STOP		DISP	NODISP	CLEAR	ANIME	GRPOFF

Pressing [ANIME] selects the mode for machining simulation (animated) again.

Pressing [AGRPOFF] in the machining simulation, animation and tool path drawing, or drawing during machining mode, the screen gets back to the memory mode screen.

9.1 DRAWING DURING MACHINING (TOOL PATH)



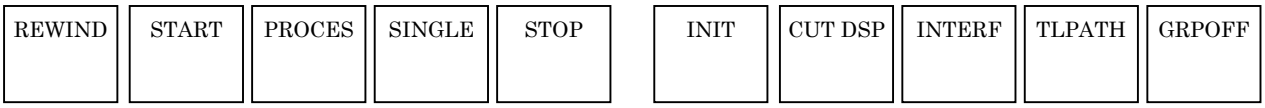
While a machining operation is being performed on the machine, the tool path can be drawn. This function is available also during machine lock and dry run operation.

NOTE

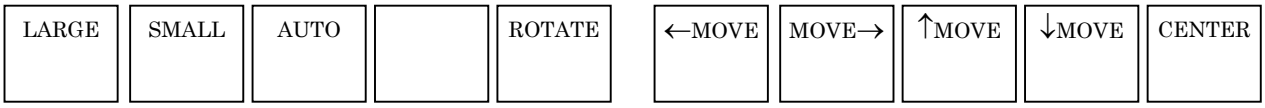
1 To perform drawing during machining, open the window for drawing during machining by pressing [GRP ON] before starting machining operation.
2 If a coordinate system change is specified in the program during drawing, drawing is performed without changing the coordinate system.
3 In the drawing during machining, when the workpiece specifying block is executed, form of a workpiece is drawn in form of wire frame. Drawing color of the workpiece can be specified by the parameter No.14773.
4 When a simulation window is opened, a blank figure drawing at the last drawing is displayed. However, if bit 4 of parameter No. 27310 is set to 1, no blank figure is displayed when a simulation window is opened.
5 Tool path drawing with a multi-path system is performed only at the selected path for displaying. And a simulation window is initialized if the selected path for displaying is changed. So if the selected path for displaying is changed in displaying a free blank figure, the drawing is initialized for a column figure which encloses it.
6 Path drawing is performed using values in the workpiece coordinate system. Coordinates that allow for tool compensation (cutter compensation, tool length compensation, geometry compensation, and wear compensation), tool tip control, or the like are not used for drawing.
7 This function cannot be used with the MANUAL GUIDE *i* simulator for the personal computer.

When drawing during machining (tool path) is selected, the soft keys shown below appear. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second or third page.

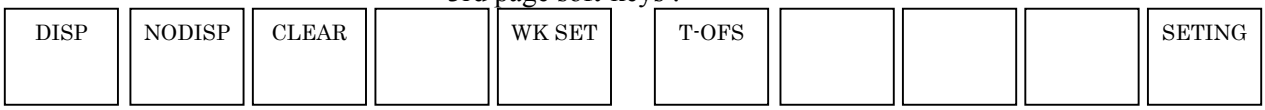
1st page soft-keys :



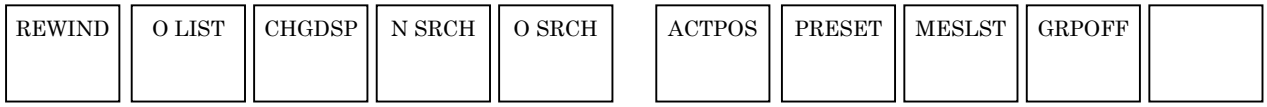
2nd page soft-keys :



3rd page soft-keys :



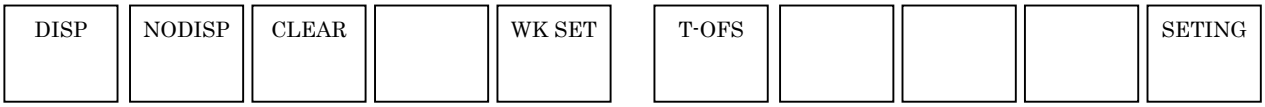
9.1.1 Program Selection Operation and Other Operations in Drawing during Machining



1st page Soft keys are used for operations such as selection of a program. These key operations are the same as the operations in the MEM mode, so see the sections listed below.
To close the window for drawing during machining and stop the mode for drawing during machining, press [GRPOFF].

6.1 REWINDING A MACHINING PROGRAM
6.2 EDITING WITH THE MACHINING PROGRAM LIST
6.3 SEARCHING FOR A SEQUENCE NUMBER IN A PROGRAM
6.4 SELECTING A PROGRAM FOR MACHINING OPERATION
6.5 CHANGING THE CURRENT POSITION DISPLAY
6.6 PRESETTING RELATIVE COORDINATES
6.7 DISPLAYING MEASUREMENT RESULTS

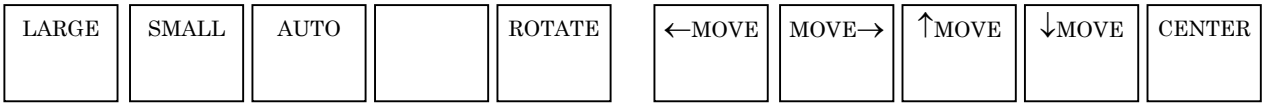
9.1.2 Selecting Whether to Display the Tool Path or Not in Drawing during Machining



3rd page soft keys mainly allow you to select whether to display the tool path or not.
For [SETING], see Chapter 10, "SETTING DATA," in Part II.

[NODISP] : As soon as this soft key is pressed, drawing of the tool path is stopped. The tool path is not drawn until [DISP] is pressed next.
[DISP] : As soon as this soft key is pressed, drawing of the tool path is started.
Remark) Only necessary tool path portions can be drawn by using [DISP] and [NODISP].
[CLEAR] : Erase the tool path drawn so far. Immediately after this soft key is pressed, tool path drawing is displayed.

9.1.3 Scaling, Movement, and Other Operations in Drawing during Machining



2nd soft keys allow you to perform scaling and movement of a tool path drawing as well as selection of a drawing coordinate system.

NOTE
The operations explained below must be performed before tool path drawing is started.

1) Scaling and movement

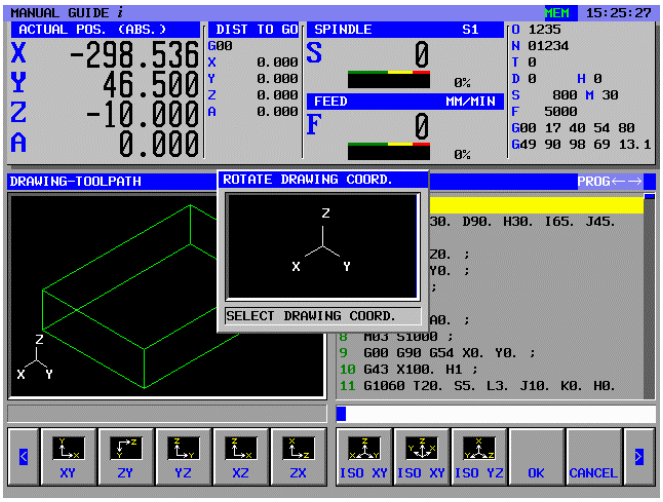
[LARGR] : Increase the drawing magnification.
[SMALL] : Decrease the drawing magnification
[AUTO] : When the blank form block for animation has been input to the machining program, perform automatic scaling so that the blank figure is fit in the window.

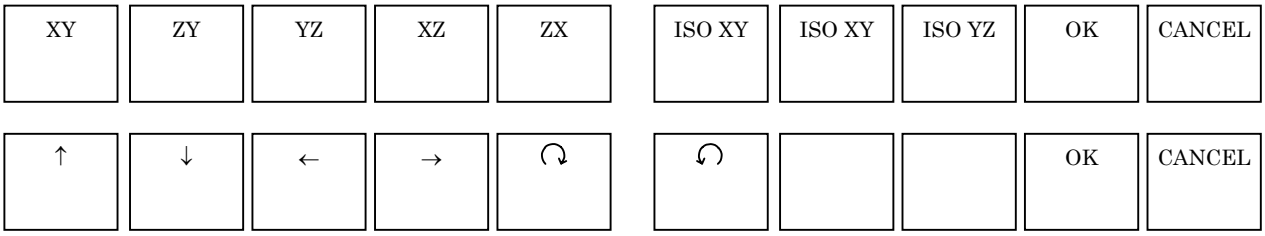
NOTE
If a blank form for animation has been entered, automatic scaling is performed when the corresponding block is executed for the first time.

[←MOVE] : Move the viewpoint leftward. As a result, the tool path drawn moves rightward.
[MOVE→] : Move the viewpoint leftward. As a result, the tool path drawn moves leftward.
[↑MOVE] : Move the viewpoint upward. As a result, the tool path drawn moves downward.
[↓MOVE] : Move the viewpoint downward. As a result, the tool path drawn moves upward.
[CENTER] : Set the center of the tool path to the center of the window.

2) Selecting a drawing coordinate system

[ROTATE] : Display the soft keys shown below for selecting the drawing coordinate system. A window for describing drawing coordinate systems is also displayed.

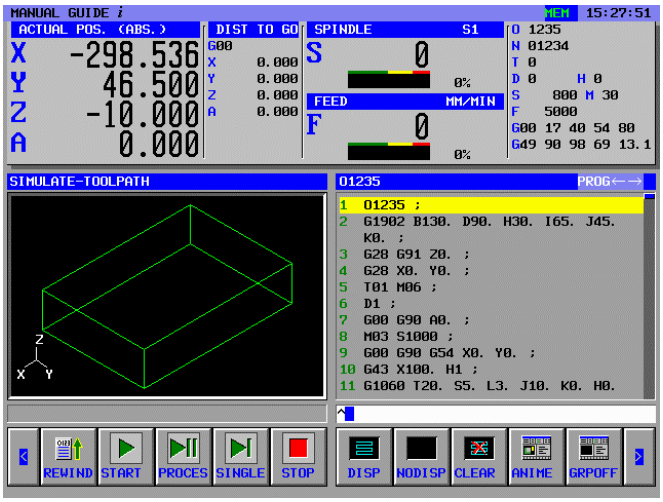




[XY] : Select the XY plane.
[ZY] : Select the ZY plane.
[YZ] : Select the YZ plane.
[XZ] : Select the XZ plane.
[ZX] : Select the ZX plane.
[ISO XY] : Select an equiangular coordinate system with the positive direction of the Z-axis facing up.
[ISO XY] : Select an equiangular coordinate system with the positive direction of the Z-axis facing up. But, the view point is opposed to the above one.
[ISO YZ] : Select an equiangular coordinate system with the positive direction of the X-axis facing up.
[OK] : Perform drawing in the coordinate system selected by one of the above soft keys.
[CANCEL] : Cancel the coordinate system selected by one of the above soft keys and use the original coordinate system for drawing.
[↑] : Make a counterclockwise rotation by selecting the right direction viewed when you face the screen as the center axis.
[↓] : Make a clockwise rotation by selecting the right direction viewed when you face the screen as the center axis.
[←] : Make a counterclockwise rotation by selecting the upward direction viewed when you face the screen as the center axis.
[→] : Make a clockwise rotation by selecting the upward direction viewed when you face the screen as the center axis.
[↷] : Make a clockwise rotation by selecting the direction toward you viewed when you face the screen as the center axis.
[↶] : Make a counterclockwise rotation by selecting the direction toward you viewed when you face the screen as the center axis.

NOTE
A rotation is made according to the unit specified in parameter No. 14716.

9.2 MACHINING SIMULATION (TOOL PATH) (FOR Series 16*i*/18*i*/21*i*)



The path of the tool in a machining program can be drawn without performing actual machining operation on the machine (machining simulation). This section is an explanation for Series 16*i*/18*i*/21*i* model B. In the case of Series 30*i*, see Section 9.4.

NOTE

1 To perform machining simulation, select the MEM mode with the mode switch on the machine operator's panel.
2 If a coordinate system change is specified in the program during simulation, simulation is performed without changing the coordinate system.
3 In the drawing during machining, when the workpiece specifying block is executed, form of a workpiece is drawn in form of wire frame. Drawing color of the workpiece can be specified by the parameter No.14773.
4 When a simulation window is opened, a blank figure drawing at the last drawing is displayed. However, if bit 4 of parameter No. 27310 is set to 1, no blank figure is displayed when a simulation window is opened.
5 Tool path drawing with a multi-path system is performed only at the selected path for displaying. And a simulation window is initialized if the selected path for displaying is changed. So if the selected path for displaying is changed in displaying a free blank figure, the drawing is initialized for a column figure which encloses it.

NOTE

6 Path drawing is performed using values in the workpiece coordinate system. Coordinates that allow for tool compensation (cutter compensation, tool length compensation, geometry compensation, and wear compensation), tool tip control, or the like are not used for drawing.

7 Simulation cannot be performed in any of control modes of the high-speed and high-precision functions (commands for advanced preview control, AI advanced preview control, AI contour control, AI nano contour control, high-precision contour control, AI high-precision contour control, and AI nano high-precision contour control).

8 In simulation of a program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

To close the machining simulation (tool path) window and stop the drawing operation of machining simulation, press [GRPOFF].

When machining simulation (tool path) is selected, the soft keys shown below appear. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second, third, or fourth page.

1st page soft-keys :

REWIND	START	PROCES	SINGLE	STOP		DISP	NODISP	CLEAR	ANIME	GRPOFF

2nd page soft-keys :

LARGE	SMALL	AUTO		ROTATE		←MOVE	MOVE→	↑MOVE	↓MOVE	CENTER

3rd page soft-keys :

				WK SET		T-OFS				SETING

4th page soft-keys :

REWIND	O-LIST	CHGDSP	N SRCH	O SRCH		ACTPOS	PRESET	MESLST		

9.2.1 Program Selection Operation and Other Operations in Machining Simulation (Tool Path)

				WK SET	T-OFS				SETING
REWIND	O-LIST	CHGDSP	N SRCH	O SRCH	ACTPOS	PRESET	MESLST		

The soft keys on the 3rd and 4th pages are used for operations such as program selection and offset data setting. These key operations are the same as the operations in the MEM mode, so see the sections listed below.

6.1 REWINDING A MACHINING PROGRAM
6.2 EDITING WITH THE MACHINING PROGRAM LIST
6.3 SEARCHING FOR A SEQUENCE NUMBER IN A PROGRAM
6.4 SELECTING A PROGRAM FOR MACHINING OPERATION
6.5 CHANGING THE CURRENT POSITION DISPLAY
6.6 PRESETTING RELATIVE COORDINATES
6.7 DISPLAYING MEASUREMENT RESULTS

9.2.2 Execution Operations in Machining Simulation (Tool Path)

REWIND	START	PAUSE	SINGLE	STOP	DISP	NODISP	CLEAR	ANIME	GRPOFF

On the 1st page soft key, you can perform operations related to execution in machining simulation (tool path).
For [SETING], see Chapter 10, "SETTING DATA," in Part II.

[REWIND] : Return to the beginning of the machining program selected for machining simulation.
[START] : Start machining simulation for the currently selected machining program.
[PAUSE] : Stop machining simulation temporarily.
[SINGLE] : Cause a single-block stop when machining simulation is performed in the continuous mode. When machining simulation is in the stopped state, this soft key starts machining simulation in the single-block mode.
[STOP] : End machining simulation.
[NODISP] : From the block immediately after this soft key is pressed, suppress tool path drawing. Tool path drawing is not performed until [DISP] is pressed next.

[DISP] : From the block immediately after this soft key is pressed, start drawing of the tool path.
Remark) Only necessary tool path portions can be drawn by using [DISP] and [NODISP].
[CLEAR] : Erase the tool path drawn so far. Tool path drawing immediately after this soft key is pressed is displayed.

NOTE
By setting bit 0 (PRC) of parameter No. 27310 to 1, [PAUSE] can be changed to [PROCES] (for temporarily stopping machining simulation at a block specifying M01;).

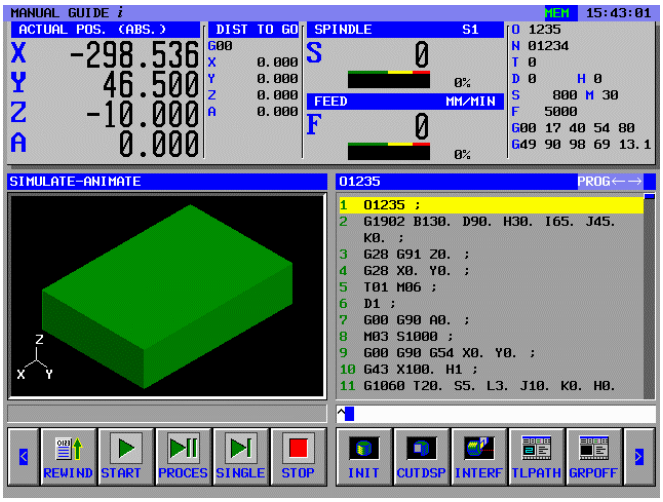
9.2.3 Scaling, Movement, and Other Operations in Machining Simulation (Tool Path)

LARGE	SMALL	AUTO		ROTATE	←MOVE	MOVE→	↑MOVE	↓MOVE	CENTER

2nd page soft keys allow you to perform scaling and movement for machining simulation (tool path) as well as selection of a drawing coordinate system. These operations are the same as those in drawing during machining. For details, see Subsection 9.1.3.

NOTE
1 Operations such as scaling, movement, and selection of a drawing coordinate system must be performed before machining simulation (tool path) is started.
2 If a coordinate system change is specified in the program during simulation, simulation is performed without changing the coordinate system.

9.3 MACHINING SIMULATION (ANIMATED) (FOR Series 16*i*/18*i*/21*i*)



Animated simulation of a machining operation by a machining program can be performed without performing actual machining operation on the machine. This section is an explanation for Series 16*i*/18*i*/21*i*-MODEL B. In the case of Series 30*i*, see Section 9.5.

NOTE

1 To perform machining simulation, select the MEM mode with the mode switch on the machine operator's panel.
2 If a coordinate system change is specified in a program under simulation, whether to perform simulation with the same coordinate system or a changed coordinate system can be chosen using bit 1 (ACD) of parameter No. 27311. For details, see Section 9.8.
3 When a program that is composed by repeated minute line segments like a program outputted by CAD/CAM is executed, sometimes the speed of simulation much slows down. In this case, please set tool motion unit to parameter No. 27323 by dot. When tool moves over the value of this parameter on screen, the screen is updated and the speed will be accelerated. But, if the value of the parameter is too large, the resolution will decrease.

NOTE

4 A tool tip position in animated simulation has coordinates of values in the workpiece coordinate system. Coordinates that allow for tool compensation (cutter compensation, tool length compensation, geometry compensation, and wear compensation), tool tip control, or the like are not used for drawing.

5 Simulation cannot be performed in any of control modes of the high-speed and high-precision functions (commands for advanced preview control, AI advanced preview control, AI contour control, AI nano contour control, high-precision contour control, AI high-precision contour control, and AI nano high-precision contour control).

6 In simulation of a program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

To close the machining simulation (animated) window and stop the drawing operation of machining simulation, press [GRPOFF].
When machining simulation (animated) is selected, the soft keys shown below appear. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second, third, or fourth page.

1st page soft-keys :

REWIND	START	PROCES	SINGLE	STOP		INIT	CUT DSP	INTERF	TLPATH	GRPOFF

2nd page soft-keys :

LARGE	SMALL	AUTO	REVERS	ROTATE		←MOVE	MOVE→	↑MOVE	↓MOVE	CENTER

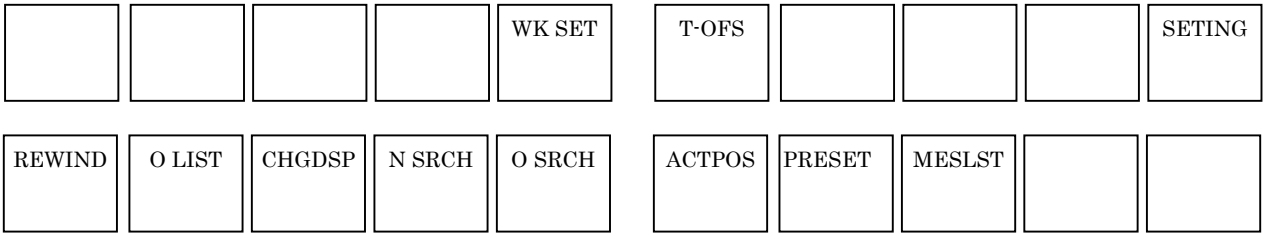
3rd page soft-keys :

				WK SET		T-OFS				SETING

4th page soft-keys :

REWIND	O LIST	CHGDSP	N SRCH	O SRCH		ACTPOS	PRESET	MESLST		

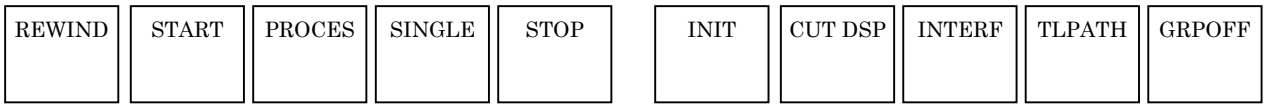
9.3.1 Program Selection Operation and Other Operations in Machining Simulation (Animated)



The soft keys on the 3rd and 4th pages are used for operations such as program selection and offset data setting. These key operations are the same as the operations in the MEM mode, so see the sections listed below.

6.1 REWINDING A MACHINING PROGRAM
6.2 EDITING WITH THE MACHINING PROGRAM LIST
6.3 SEARCHING FOR A SEQUENCE NUMBER IN A PROGRAM
6.4 SELECTING A PROGRAM FOR MACHINING OPERATION
6.5 CHANGING THE CURRENT POSITION DISPLAY
6.6 PRESETTING RELATIVE COORDINATES
6.7 DISPLAYING MEASUREMENT RESULTS

9.3.2 Execution Operations in Machining Simulation (Animated)



On the 1st page soft keys, you can perform operations related to execution in machining simulation (animated). The operations of [REWIND], [START], [PAUSE], [SINGLE], and [STOP] are the same as the operations in machining simulation (tool path). For details, see Subsection 9.2.3.

[INIT] : Initialize the machined blank used for animation.
[CUT DSP] : Lets you switch between 1/4 workpiece and entire periphery for rod blanks and drilled rod blanks. Before starting animated simulation, be sure to switch if necessary.
[INTERF] : Lets you select whether to make an interference check during animated simulation. If an interference check is enabled, a warning is displayed when the tool tip collides with a workpiece during rapid traverse, and the portion collided with the tool tip is displayed in the same color as that of the tool.

NOTE
With bit 0 (ITF) of parameter No. 27311, you can select continued operation (ITF = 0) or temporary stop (ITF = 1) if tool interferes with the workpiece during animation.

9.3.3 Scaling, Movement, and Other Operations in Machining Simulation (Animated)

LARGE	SMALL	AUTO	REVERS	ROTATE	←MOVE	MOVE→	↑MOVE	↓MOVE	CENTER

The soft keys on the 2nd page allow you to perform scaling and movement for machining simulation (animated) as well as selection of a drawing coordinate system. These operations are the same as those in drawing during machining. For details, see Subsection 9.1.3.

[REVERS] : Change the viewpoint of the blank in animation to the exactly opposite position. You can use this soft key when, during machining with a sub-spindle or machining with the C-axis rotated, for example, you want to see animation from the opposite side.

NOTE
1 Although [REVERS] may be used even during machining simulation, switching will sometimes be delayed depending on the CNC status.
2 Operations such as scaling, movement, and selection of a drawing coordinate system must be performed before machining simulation (tool path) is started.

9.3.4 Notes on Machining Simulation

During the execution of drawing, the system is automatically placed in the machine locked state. During the execution of drawing, no machine control signals are output by auxiliary and other functions, but control signals such as "OP," "STL," "SPL," "RST," and "AL" may be output. During the execution of drawing, the drawing-in-progress signal "CKGRP," shown below, is output. If the control signals that may be output during the execution of drawing affect machine control, the PMC Ladder program must be modified so that these signals are ignored, by using the drawing-in-progress signal.

	#7	#6	#5	#4	#3	#2	#1	#0
F62			CKGRP					

CKGRP Indicates that a dynamic graphics display is being drawn for check purposes.
In order to distinguish the executing state of the program, machining

custom macro program can refer to the following system variable #3010.

System variable	Value	Executing State
#3010	0	Normal condition(Other than the following status)
	1	Executing automatic operation(Including Drawing during Machining)
	4	Executing machining simulation(Animated, Tool Path)

9.4 MACHINING SIMULATION (TOOL PATH) (FOR Series 30*i*)

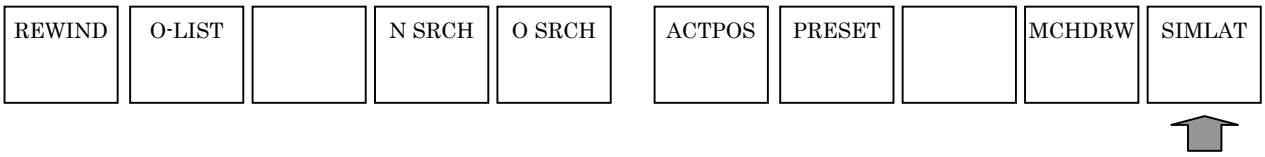
During machining, the tool path of another program can be drawn.
With Series 30*i* MANUAL GUIDE *i*, the terms related to operation and drawing are defined as follows:

Automatic operation	Operation performed for actual machining
Drawing during machining	Tool path drawing during automatic operation

Background operation	Virtual operation performed for simulation. This operation can be performed simultaneously with automatic operation.
Machining simulation	Machining simulation in background operation (generic term for tool path drawing and animated simulation)
Tool path drawing	Machining simulation for drawing the tool path in background operation
Animated drawing	Machining simulation for animated simulation of background operation

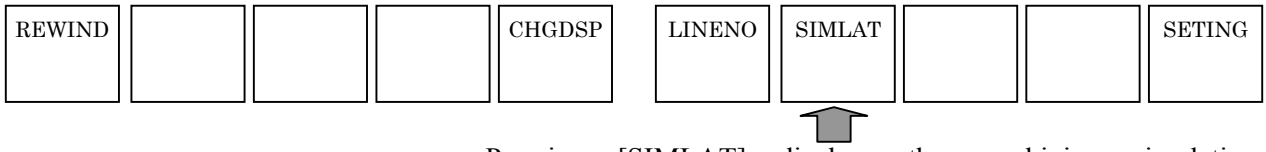
With FANUC Series 30*i* MANUAL GUIDE *i*, machining simulation can be performed in the MEM mode and EDIT mode. When each mode is selected, the machining simulation screen can be displayed using the operation described below.

- MEM mode



Pressing [SIMLAT] displays the machining simulation (animated) screen and the soft keys for machining simulation.

- EDIT mode



Pressing [SIMLAT] displays the machining simulation (animated) screen and the soft keys for machining simulation.

NOTE

1 A program subject to simulation is placed in the background editing selection state. So, if background editing is in progress when the [SIMLAT] soft key is pressed, the simulation screen cannot be displayed. (The warning message "CAN NOT OPERATE ON BG EDIT" is displayed.) Terminate background editing then press the [SIMLAT] soft key.
2 A program selected when the [SIMLAT] soft key is pressed is the program being selected in the foreground.
3 When machining simulation is terminated by pressing the [GRPOFF] soft key, the program selected for drawing is closed, and the program selected in the foreground is placed in the foreground selection state.
4 If the screen display is switched to another screen during drawing, machining simulation is terminated.
5 If the machining simulation screen is displayed again after the machining simulation screen is terminated, the results of the previous simulation are erased.
6 Animated simulation assumes that the tool tip is placed at the current position. So, if the tool is inclined with the tool inclination axis, drawing is performed at a position different from the actual tool position until the next move command is specified for the tool tip. Coordinates that allow for tool compensation (cutter compensation, tool length compensation, geometry compensation, and wear compensation), tool tip control, or the like are not used for drawing.
7 Simulation cannot be performed in any of control modes of the high-speed and high-precision functions (commands for advanced preview control, AI advanced preview control, AI contour control, AI nano contour control, high-precision contour control, AI high-precision contour control, and AI nano high-precision contour control).
8 In simulation of a program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

To close the machining simulation (tool path) window and stop the drawing operation of machining simulation, press [GRPOFF].

When machining simulation (tool path) is selected, the soft keys shown below appear. Pressing the leftmost soft key [<] or the rightmost soft key [>] changes the page of the soft key display to the second, third, or fourth page.

1st page soft-keys :

REWIND	START	PAUSE	SINGLE	STOP	DISP	NODISP	CLEAR	ANIME	GRPOFF

2nd page soft-keys :

LARGE	SMALL	AUTO		ROTATE	↑MOVE	MOVE→	↑MOVE	↓MOVE	CENTER

3rd page soft-keys :

				WK SET	T-OFS				SETING

4th page soft-keys

REWIND	O LIST	↑ SRCH	↓ SRCH	O SRCH	ACTPOS	PRESET	MESLST		

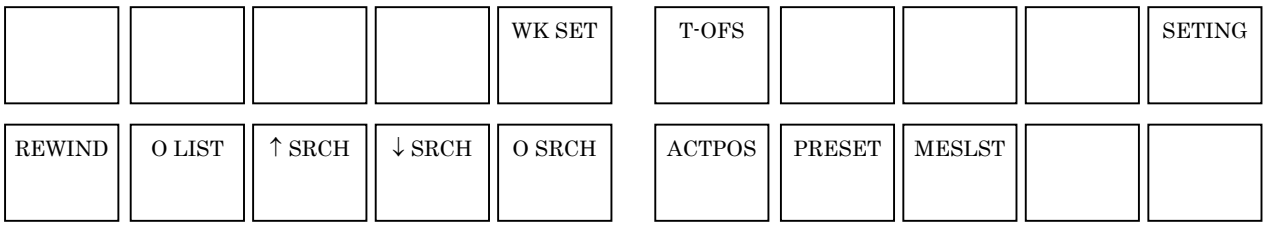
NOTE

1 The data displayed on the machining simulation screen such as the current position and remaining travel distance is not automatic operation state data but is machining simulation data.
2 The machining simulation screen does not display spindle information and feed information but displays the next travel distance only.
3 On the machining simulation screen, the remaining travel distance is 0 at all times.
4 To stop machining simulation, press the [STOP] soft key. If **RESET** key is pressed, foreground operation is reset.
5 If a blank registration command is executed in tool path drawing during machining simulation, the blank figure is displayed using a line (wire frame). Use parameter No. 14773 to specify the display color of the blank figure.
6 When the simulation window is displayed, the blank figure data previously displayed is used for drawing. However, when bit 4 of parameter No. 27310 is set to 1, no blank figure is displayed if the simulation window is open.
7 In machining simulation performed with a multipath system, the tool path of the displayed path only is drawn. If the displayed path is switched during path drawing, the drawing screen is initialized. So, if the displayed path is switched when an arbitrary figure blank is displayed, the blank is initialized to a round rod figure enclosing the arbitrary figure.
8 If the animated simulation option is not selected, a blank figure command is ignored, and no blank figure (wire frame) is displayed.
9 When simulation is started from a midway point, the modal state of the NC is not always executed up to the midway point. So, if simulation is started from a midway point, correct drawing operation may not be performed.

In order to distinguish the executing state of the program, machining custom macro program can refer to the following system variable #3010.

System variable	Value	Executing State
#3010	0	Normal condition(Other than the following status)
	-1	Executing machining simulation(Animated, Tool Path)

9.4.1 Program Selection Operation and Other Operations in Machining Simulation (Tool Path)



The soft keys on the 3rd and 4th pages are used for operations such as program selection and offset data setting. These operations are the same as operations in the MEM mode. However, no data can be set during machining simulation.

NOTE
1 A program subject to simulation is placed in the background editing selection state. So, the selected program is placed in the background editing selection state.
2 Data such as offset data can be edited at all times except during machining simulation. A modification to data can affect program execution for automatic operation. So, before editing data, ensure that the data is not being used.
3 Offset data and coordinate system data is copied for machining simulation, and such simulation-dedicated data is used during machining simulation. Even if such data is rewritten using G10, for example, the actual data is not modified.

9.4.2 Execution Operations in Machining Simulation (Tool Path)

REWIND	START	PAUSE	SINGLE	STOP		DISP	NODISP	CLEAR	ANIME	GRPOFF

On the 2nd page soft key, you can perform operations related to execution in machining simulation (tool path).

[REWIND] : Return to the beginning of the machining program selected for machining simulation.
[START] : Start machining simulation for the currently selected machining program.
[PAUSE] : Stop machining simulation temporarily.
[SINGLE] : Cause a single-block stop when machining simulation is performed in the continuous mode. When machining simulation is in the stopped state, this soft key starts machining simulation in the single-block mode.
[STOP] : End machining simulation.
[NODISP] : From the block immediately after this soft key is pressed, suppress tool path drawing. Tool path drawing is not performed until [DISP] is pressed next.
[DISP] : From the block immediately after this soft key is pressed, start drawing of the tool path.
Remark) Only necessary tool path portions can be drawn by using [DISP] and [NODISP].
[CLEAR] : Erase the tool path drawn so far. Tool path drawing immediately after this soft key is pressed is displayed.

NOTE
1 By setting bit 0 (PRC) of parameter No. 27310 to 1, [PAUSE] can be changed to [PROCES] (for temporarily stopping machining simulation at a block specifying M01;).

9.4.3 Scaling, Movement, and Other Operations in Machining Simulation (Tool Path)

The operations are the same as those operations performed with Series 16*i*/18*i*/21*i*-MODEL B. For details, see Subsection 9.2.3.

9.5 MACHINING SIMULATION (ANIMATED) (FOR Series 30*i*)

During machining, animated simulation can be performed for another program.
The terms related to operation and drawing for tool path drawing described in Section 9.4 are applicable. See Section 9.4.
With Series 30*i* MANUAL GUIDE *i*, machining simulation (animated) can be performed in the MEM mode and EDIT mode. The operation for displaying the machining simulation screen in each mode is the same as for tool path drawing. See Section 9.4.

In order to distinguish the executing state of the program, machining custom macro program can refer to the following system variable #3010.

System variable	Value	Executing State
#3010	0	Normal condition(Other than the following status)
	-1	Executing machining simulation(Animated, Tool Path)

9.5.1 Program Selection Operation and Other Operations in Machining Simulation (Animated)

The operations are the same as those operations performed for tool path drawing described in Section 9.4. See Section 9.4.

9.5.2 Execution Operations in Machining Simulation (Animated)

REWIND	START	PAUSE	SINGLE	STOP		INIT	CUTDSP	INTERF	TLPATH	GRPOFF

On the 1st page soft keys, you can perform operations related to execution in machining simulation (animated). The operations of [REWIND], [START], [PAUSE], [SINGLE], and [STOP] are the same as the operations in machining simulation (tool path). For details, see Subsection 9.2.3.

[INIT] : Initialize the machined blank used for animation.
[CUTDSP] : Lets you switch between 1/4 workpiece and entire periphery for rod blanks and drilled rod blanks. Before starting animated simulation, be sure to switch if necessary.
[INTERF] : Lets you select whether to make an interference check during animated simulation. If an interference check is enabled, a warning is displayed when the tool tip collides with a workpiece during rapid traverse, and the

portion collided with the tool tip is displayed in the same color as that of the tool.

NOTE
With bit 0 (ITF) of parameter No. 27311, you can select continued operation (ITF = 0) or temporary stop (ITF = 1) if tool interferes with the workpiece during animation.

9.5.3 Scaling, Movement, and Other Operations in Machining Simulation (Animated)

The operations are the same as those operations performed with Series 16*i*/18*i*/21*i*-MODEL B. For details, see Subsection 9.3.3.

9.6 DATA HANDLED DURING MACHINING SIMULATION (FOR Series 30*i*)

During machining simulation (background operation), data is handled as indicated below.

<1>	Parameter	The same parameters are used for machining simulation and automatic operation.
<2>	Tool compensation value Workpiece origin offset value Extended workpiece origin offset value Macro variable Fixture offset data	Data used for machining simulation differs from data used for automatic operation.
<3>	Tool life management data Tool number offset data 3-dimensional error compensation data	These data items are not used for machining simulation.

<1> Parameter
The same parameters as used for automatic operation are used for machining simulation. However, no parameter can be rewritten in background operation. (If an attempt is made to rewrite a parameter by programmable parameter input (G10L50), a warning (NC statement error (B.G.) is output.)

<2> Tool compensation value, workpiece origin offset value, extended workpiece origin offset value, macro variable, etc.
Data used for background operation differs from data used for automatic operation. When background operation is started, the data for automatic operation is copied to produce data for background operation. Afterwards, the data for background operation and the data for automatic operation are handled separately from each other. So, even when data is rewritten using G10, for example, background operation and automatic operation do not affect each other. Note, however, that data rewritten in background operation is erased without being reflected in the actual data (foreground data).

<3> Tool life management data, tool number offset data, 3-dimensional error compensation data
These data items are not used for background operation. So, if an attempt is made to rewrite data by using G10, a warning (NC statement error (B.G.)) is output.

9.7 FUNCTIONS OPERATING DIFFERENTLY BETWEEN MACHINING SIMULATION AND AUTOMATIC OPERATION (FOR Series 30*i*)

The functions listed below are major functions that operate in background operation and automatic operation differently from each other. There are additional functions that operate differently.

<1> Custom macro

1) Interface signal
   #1000 to #1035 are always assumed to be 0 at all times.
2) Message output
   A message output with #3006 is not displayed but is ignored.
3) Clock
   #3001 and #3002 are ignored. So, note that if a command as indicated below is specified, for example, drawing does not proceed:
   ```
   #3001=1;
   WHILE[#3001 LE100]DO1;
   END1;
   ```
4) Mirror image
   #3007 is assumed to be 0 at all times.
5) State during program restart operation
   #3008 is assumed to be 0 at all times.
6) External output command
   BPRNT, DPRNT, POPEN, and PCLOS are ignored.

<2> Functions ignored

1) G04 (Dwell)
2) G20, G21 (Inch/metric conversion)
3) Auxiliary function (M, S, T, B)
4) G22,G23 (Stored stroke limit on/off)
5) G10.1 (PMC data setting)
6) G10.6 (Tool retraction data setting)
7) G10.9 (Programmable diameter/radius switching)
8) G81.1 (Chopping)
9) G25/G26 (Spindle speed fluctuation detection on/off)

<3> Functions that operate differently in part

1) G28 (automatic reference position return) is drawn up to a midway point.
2) G29 (automatic return from the reference position) is drawn from a midway point.
3) G27 (reference position return check) does not make a reference position return check.
4) Stored stroke limit checks are not made.
5) G31 (skip function), and G31.1, G31.2, and G31.3 (multi-step skip) are drawn up to a specified position, regardless of the skip signal.
6) G60 (single direction positioning) is drawn directly up to a specified position at all times even when the positioning direction is reverse.

9.7.1 Functions That Cannot Be Used for Machining Simulation

<1> Functions that operate differently in background drawing
When the functions below are specified, the operations described below are performed.
G02.2/G03.2 : Involute interpolation
Circular interpolation is performed.
G06.1 : Spline interpolation
Linear interpolation is performed.
When the functions below are specified, the operations described below are performed.
G02.1/G03.1 : Circular threading B
Circular interpolation is performed. No rotation axis can be drawn.
G02.3/G03.3 : Exponential interpolation
Linear interpolation is performed for a straight line only.
G07 : Hypothetical axis interpolation
Circular interpolation is performed.

<2> Functions that disable background drawing
When the functions below are specified, a warning (NC statement error (B.G.)) is output, and background drawing stops.
G10 : Data setting Part of the function can be used.
M198 : External sub program call

9.7.2 Functions That Can Be Used for Machining Simulation (Milling System)

G00 : Positioning
G01 : Linear interpoaltion
G02/G03 : Circular interpolation (Helical interpolation cannot be drawn.)
G17/G18/G19 : Plane selection
G33 : Threading (Drawn as linear interpolation)
G40/G41/G42 : Cutter compensation / Cancel
G52 : Local coordinate system
G53 : Machine coordinate system select
G54 to G59 : Workpiece coordinate system select
G54.1 : Extended workpiece coordinate system select
G65 : Macro call
G68/G69 : Coordinate system rotation, 3-dimensional coordinate conversion / Cancel
G90/G91 : Absolute/incremental programming
G92 : Workpiece coordinate system change
G92.1 : Workpiece coordinate system preset
G94 : Feed per minute
G95 : Feed per revolution
G96/G97 : Constant surface speed control / Cancel
M98 : Sub program call
G07.1 : Cylindrical interpolation
G12.1 : Polar coordinate interpolation

9.8 CHANGING WORKPIECE COORDINATE DURING MACHINING SIMULATION (ANIMATION, TOOL PATH DRAWING)

If a coordinate system is changed in the part program during machining simulation, animation or tool path drawing, the drawing is performed on the changed coordinate system.
But in the path drawing during machining, the drawing is performed without changing the coordinate system.
Drawing of Machining Simulation with this function is performed on the standard coordinate system for drawing which is initialized by G1900, G1901, G1902 (drawing definition of blank form block) or G1998(Spindle definition block for drawing). Workpiece rotated axis (usually C-axis) is the rotated axis around Z-axis of the standard coordinate system. The standard coordinate system is by drawing definition of blank form block.

NOTE
1 This function is not available in drawing during machining.
2 Even if the coordinate system is changed in machining simulation, the axis drawing of coordinate system is not changed.
3 Drawing of machining simulation is performed on the premise that the top point of tool (or the center point of tool diameter) is on a commanded point at the workpiece coordinate system. So, there are some case where machining simulation is different from the movement of real machining before a position command after slanting a tool.
4 Rotary axis about a blank, C-axis, is one about Z axis of the basic three axis. So, in the case with rotary axis about a blank, a “WORK ORIGIN” point of rectangular solid blank must be set on the rotary axis by a blank form block. Rotary axis about a column blank is one about a center point of the diameter.
5 If 3-dimentional coordinate conversion command or angular axis’s machining command are performed in slanting a tool, the slanted angle of angular axis is made ineffective and a tool is drawn on the slant of the workpiece coordinate system changed by 3-dimentional coordinate conversion command or angular axis’s machining command.

NOTE

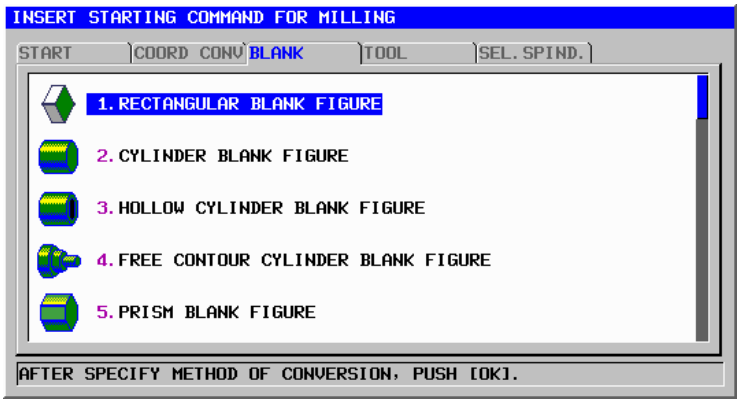
6 When the blank registration command for animated simulation or the spindle switching command G1998 is specified, the modal workpiece coordinate system is displayed to match the workpiece coordinate system set with the blank registration command (the C-axis position is also initialized).
7 If drawing of machining simulation is performed from an intermediate point of the program, it is performed as the continuous-state workpiece coordinate system is same as one of the last simulation.
8 Machining simulation is performed assuming that the tool tip position (radius of the tool nose or tool radius center) is at specified workpiece coordinates (absolute coordinates). So, if the tool tip is not placed at a specified position, drawing is not performed correctly. Moreover, drawing is not performed in a coordinate system where tool compensation (such as cutter compensation, tool length compensation, geometry compensation and wear compensation), tool nose control, and so forth are considered.
9 Rotary C-axis, rotary axis set by parameter No.14717, always rotates about a blank. And in the case of a rectangular solid blank, rotary C-axis rotates about Z-axis on a “WORK ORIGIN” point.
10 Tool drawing is not slant by the rotate angle in drawing of machining simulation of polar coordinate interpolation.

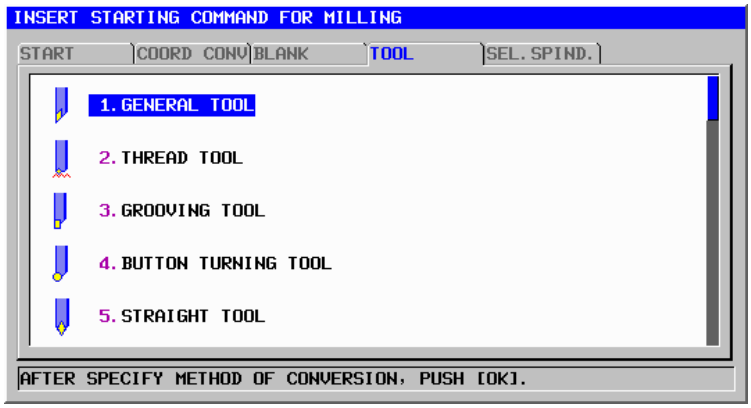
In the case of multi-paths T system, G1998 (spindle definition block for drawing) and G1992 (top of process definition) are as follows.

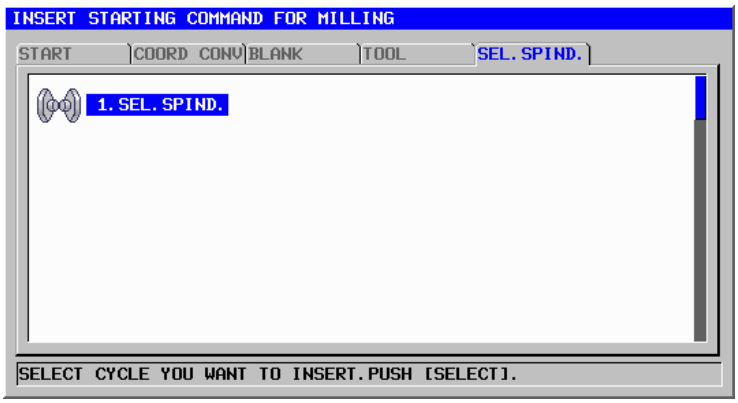
1) Parameter No.27311#1(ACD)=1
   - Spindle at which animated simulation is performed is defined only by G1998 command.
   - Spindle at which animated simulation is performed is the last commanded by G1998 command at one of paths.
   - If Spindle of G1998 or G1992 commanded at a path is same as the last commanded by G1998 command, animated simulation is performed at the path.
2) Parameter No.27311#1(ACD)=0
   - Spindle at which animated simulation is performed is defined only by G1998 or G1992 commands.
   - Spindle at which animated simulation is performed is the last commanded by G1998 or G1992 commands at one of paths.
   - If Spindle of G1998 or G1992 commanded at a path is same as the last commanded by G1998 or G1992 commands, animated simulation is performed at the path.

9.9 SETTING OF DATA FOR ANIMATION

When animation can be performed, a blank figure and tool figure must be set. Such animation data must be set in the DRAWING DEFINITION block, which is to be entered in a machining program.
To display the window for entering DRAWING DEFINITION data, press [START] in the soft key menu for editing machining programs.



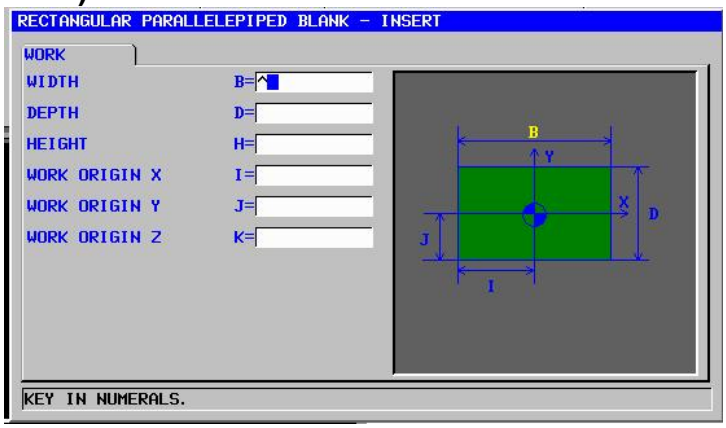




DRAWING DEFINITION			
	Blank form block	G1902	Rectangular solid
		G1900	Column
		G1906	Column (around X)
		G1901	Column with a hole
		G1907	Column with a hole (around X)
		G1903	Prism
		G1904	Prism with a hole
	Arbitrary blank figure block	G1970	Start point
		G1971	Line
		G1972	Arc (CW)
		G1973	Arc (CCW)
		G1974	Chamfering
		G1975	Corner rounding
		G1976	End
	Arbitrary blank figure block (around X)	G1970	Start point
		G1971	Line
		G1972	Arc (CW)
		G1973	Arc (CCW)
		G1974	Chamfering
		G1975	Corner rounding
		G1976	End
	Tool definition block	G1910	General-purpose tool (turning)
		G1911	Threading tool (turning)
		G1912	Grooving tool (turning)
		G1913	Round-nose tool (turning)
		G1914	Point nose straight tool (turning)
		G1921	Drill (turning, milling)
		G1931	Counter sink tool (milling)
		G1932	Flat end mill (milling)
		G1933	Ball end mill (milling)
		G1922	Tap (turning, milling)
		G1923	Reamer (turning, milling)
		G1924	Boring tool (turning, milling)
		G1930	Face mill (milling)
	Spindle selection block	G1998	Spindle selection

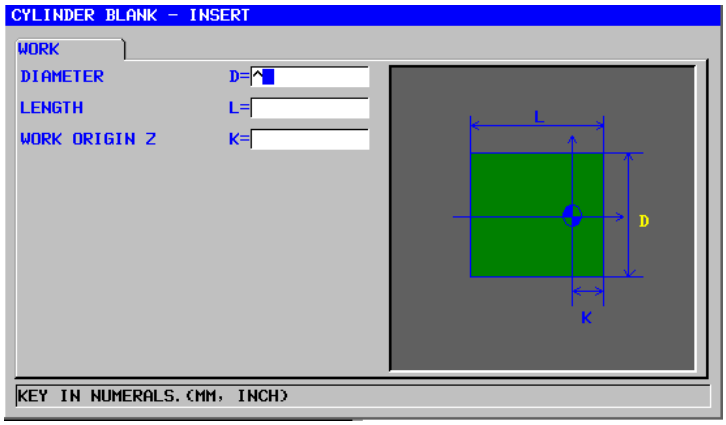
NOTE
1 A tool figure during animation is drawn by a tool definition block.
2 A blank is initialized by a blank figure block.
3 When you use ‘Tool Data Base Function’, you do not need to specify ‘Tool Definition Block’.

Blank form block (rectangular solid): G1902



WORK		
Data item		**Meaning**
B	WIDTH	Width of the rectangular solid blank. Length in the X-axis direction (positive value)
D	DEPTH	Depth of the rectangular solid blank. Length in the Y-axis direction (positive value)
H	HEIGHT	Height of the rectangular solid blank. Length in the Z-axis direction (positive value)
I	WORK ORIGIN X	Distance in the X-axis direction from the lower left corner of the rectangular solid to workpiece origin (positive or negative value)
J	WORK ORIGIN Y	Distance in the Y-axis direction from the lower left corner of the rectangular solid to workpiece origin (positive or negative value)
K	WORK ORIGIN Z	Distance in the Z-axis direction from the lower left corner of the rectangular solid to workpiece origin (positive or negative value)

Blank form block (column): G1900



WORK		
Data item		Meaning
D	DIAMETER	Diameter of the column blank (positive value)
L	LENGTH	Length of the column blank (positive value)
K	WORK ORIGIN Z	Cutting allowance of the end face of the blank (Z-axis distance between the end face and workpiece origin) (positive value)
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of the back end face of the blank (Z-axis distance between the back end face and workpiece origin) (positive value)

NOTE
The input item of “WORK ORIGIN Z(SPN 2) (W)“ is prepared for a machine with a subspindle, and is displayed when the parameter No.14702 #1 = 1.

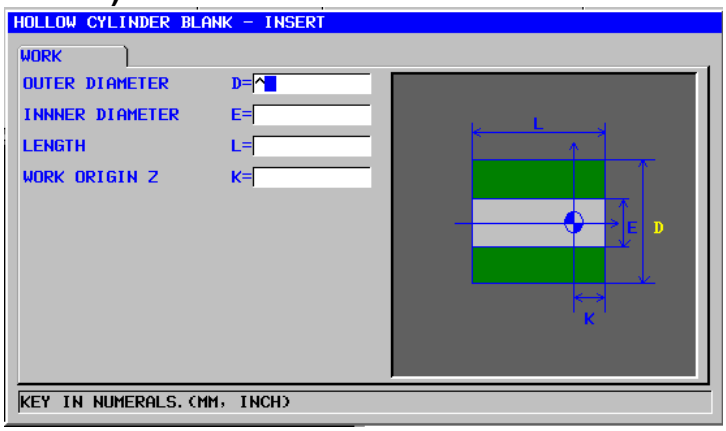
Blank form block (column(around X)): G1906

This blank is used for the machining simulation whose workpiece rotates around X-axis in machining centers.
Data setting is the same as the above column.

NOTE
1 There is the following limitations when the simulation whose workpiece rotates around X axis is executed.
- This function is available in only machining centers.
- Turning cycle can’t be simulated.
- Polar coordinate interpolation can’t be simulated.
- Under simulation as workpiece rotates workpiece around X-axis, coordinate is displayed in X, Y, -Z.

2 The following parameter setting is necessary.
- No.27003#2=1, 27003#1=0 and 27003#0=0
- No.14717 : The axis number of work rotation axis

Blank form block (column with a hole): G1901



WORK		
Data item		**Meaning**
D	DIAMETER	Diameter of the column blank (positive value)
E	INNER DIAMETER	Inner diameter of the column blank (positive value)
L	LENGTH	Length of the column blank (positive value)
K	WORK ORIGIN Z	Cutting allowance of the end face of the blank (Z-axis distance between the end face and workpiece origin) (positive value)
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of the back end face of the blank (Z-axis distance between the back end face and workpiece origin) (positive value)

NOTE
The input item of “WORK ORIGIN Z (SPN 2) (W)” is prepared for a machine with a subspindle, and is displayed when the parameter No.14702#1 = 1.

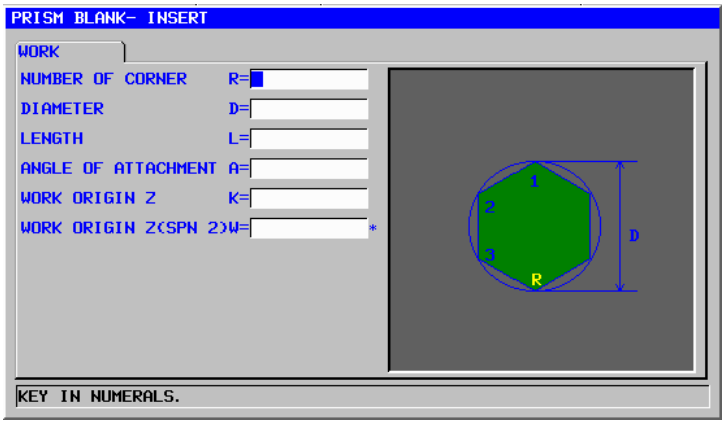
Blank form block (column with a hole (around X)): G1907

This blank is used for the machining simulation whose workpiece rotates around X-axis in machining centers.
Data setting is the same as the above column with a hole.

NOTE
1 There is the following limitations when the simulation whose workpiece rotates around X axis is executed.
- This function is available in only machining centers.
- Turning cycle can’t be simulated.
- Polar coordinate interpolation can’t be simulated.
- Under simulation as workpiece rotates workpiece around X-axis, coordinate is displayed in X, Y, -Z.

2 The following parameter setting is necessary.
- No.27003#2=1, 27003#1=0 and 27003#0=0
- No.14717 : The axis number of work rotation axis

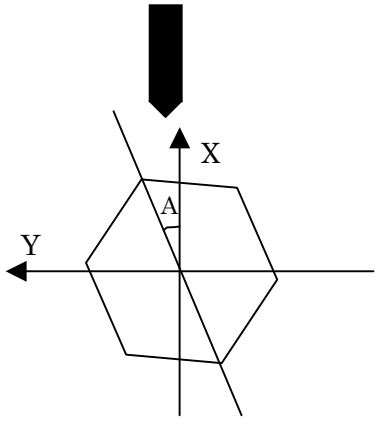
Blank form block (prism): G1903



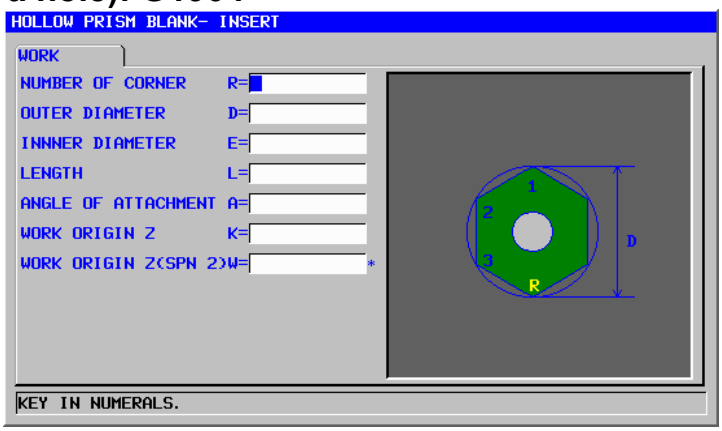
WORK		
Data item		Meaning
R	NUMBER OF CORNER	The number of corner This must be a integer, larger than 2 and smaller than 100.
D	DIAMETER	Diameter of the prism blank (positive value)
L	LENGTH	Length of the prism blank (positive value)
A	Angle of attachment	The angle between a corner and X axis
K	WORK ORIGIN Z	Cutting allowance of the end face of the blank (Z-axis distance between the end face and workpiece origin) (positive value)
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of the back end face of the blank (Z-axis distance between the back end face and workpiece origin) (positive value)

NOTE

1 The input item of “WORK ORIGIN Z(SPN 2) (W)“ is prepared for a machine with a subspindle, and is displayed when the parameter No.14702 #1 = 1.
2 Only equilateral prism can be displayed.
3 Angle of attachment is the angle that is formed by X-axis and one comer of blank figure.(Refer to the following figure.)



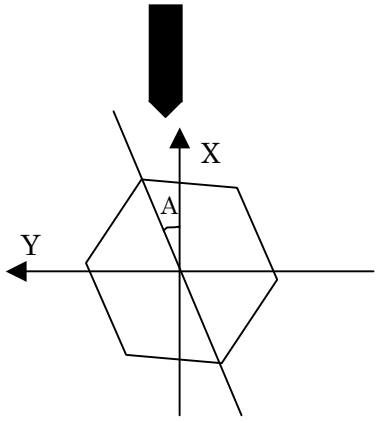
Blank form block (column with a hole): G1904



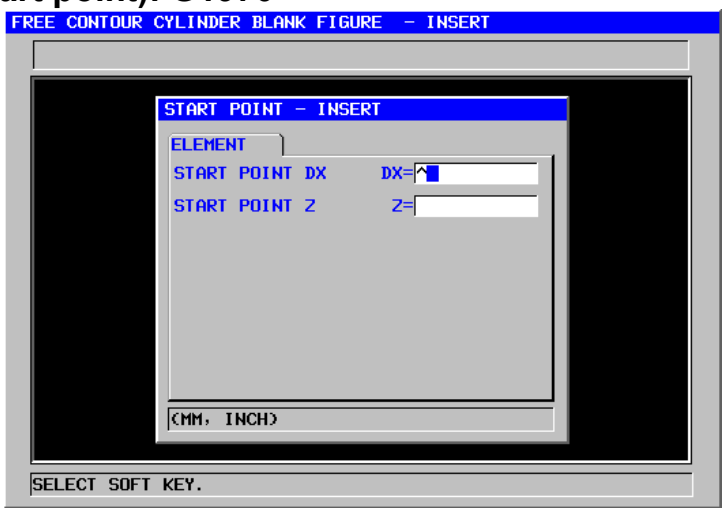
WORK		
Data item		**Meaning**
R	NUMBER OF CORNER	The number of corner This must be a integer, larger than 2 and smaller than 100.
D	DIAMETER	Diameter of the prism blank (positive value)
E	INNER DIAMETER	Inner diameter of the prism blank (positive value)
L	LENGTH	Length of the prism blank (positive value)
A	Angle of attachment	The angle between a corner and X axis
K	WORK ORIGIN Z	Cutting allowance of the end face of the blank (Z-axis distance between the end face and workpiece origin) (positive value)
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of the back end face of the blank (Z-axis distance between the back end face and workpiece origin) (positive value)

NOTE

1 The input item of “WORK ORIGIN Z(SPN 2) (W)“ is prepared for a machine with a subspindle, and is displayed when the parameter No.14702 #1 = 1.
2 Only equilateral prism can be displayed.
3 Angle of attachment is the angle that is formed by X-axis and one comer of blank figure.(Refer to the following figure.)



Arbitrary blank figure block (start point): G1970



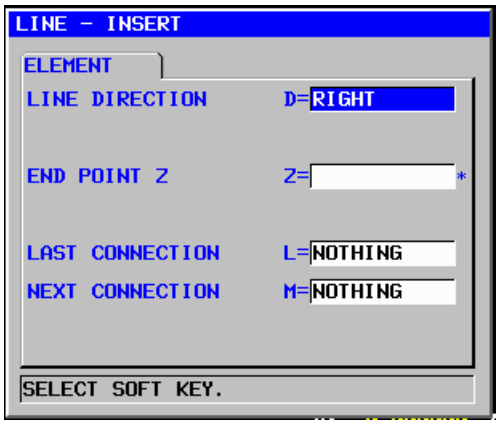
ELEMENT (INPUT DATA) (Note 1, 2)		
Data item		**Meaning**
DX	START POINT DX	X coordinate of the start point of an arbitrary figure (positive value)
Z	START POINT Z	Z coordinate of the start point of an arbitrary figure
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of a blank rear end face (distance between the rear end face and workpiece origin in the Z-axis direction) (positive value) **(Note 3)**

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.
2 Figure input is performed based on the +X-side cross section of the ZX plane of a round rod.
3 The input item of "WORK ORIGIN Z (SPN 2) (W)" is prepared for a machine with a subspindle, and is displayed when the parameter No.14702 #1 = 1.

ELEMENT (OUTPUT DATA) (Note 3)		
Data item		**Meaning**
H	START POINT DX	X coordinate of the start point (input value)
V	START POINT Z	Z coordinate of the start point (input value)
W	WORK ORIGIN Z (SPN 2)	Cutting allowance of a blank rear end face (distance between the rear end face and workpiece origin in the Z-axis direction) (positive value) (input value)

NOTE
3 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arbitrary blank figure block (line): G1971



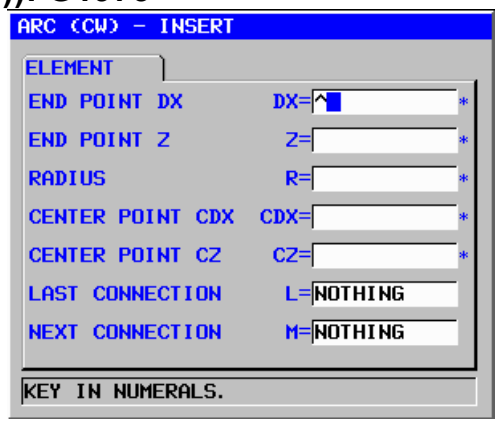
ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
D	LINE DIRECTION	Select a line direction from the displayed soft key menu.
DX	END POINT DX	X coordinate of a line end point Remark) This item may not be displayed, depending on the input line direction value.
Z	END POINT Z	Z coordinate of a line end point Remark) This item may not be displayed, depending on the input line direction value.
A	ANGLE	Line angle Remark) This item may not be displayed, depending on the input line direction value.
L	LAST CONNECTION	[TANGNT] : Contacts the immediately preceding figure. [NOTHING] : Does not contact the immediately preceding figure. (initial value)
M	NEXT CONNECTION	[TANGNT] : Contacts the next figure. [NOTHING] : Does not contact the next figure. (initial value)

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT (OUTPUT DATA) (Note 2)		
Data item		Meaning
H	END POINT X	X coordinate of the end point of a straight line (calculation result)
V	END POINT Z	Z coordinate of the end point of a straight line (calculation result)
K	LINE DIRECTION	The direction of a straight line is selected from a menu indicated on a soft key. (input value)
C*	END POINT DX	X coordinate of the end point of a straight line (input value)
D*	END POINT Z	Z coordinate of the end point of a straight line (input value)
A*	ANGLE	Straight-line angle (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [2] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately following figure [2] : Not in contact with the immediately following figure (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arbitrary blank figure block (arc(CW)): G1972
Arbitrary blank figure block (arc(CCW)): G1973



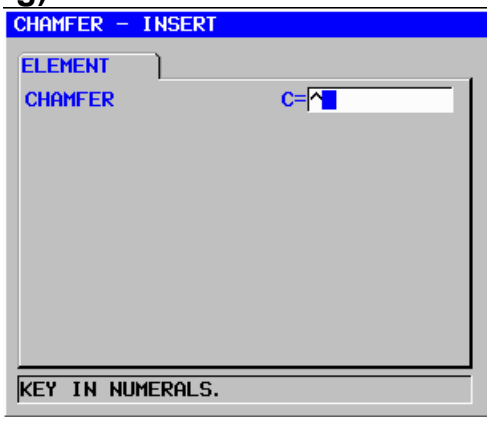
ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
DX	END POINT DX	X coordinate of an arc end point
Z	END POINT Z	Z coordinate of an arc end point
R	RADIUS	Radius of an arc
CDX	CENTER POINT CDX	X coordinate of an arc center
CZ	CENTER POINT CZ	Z coordinate of an arc center
L	LAST CONNECTION	[TANGNT] : Contacts the immediately preceding figure. [NOTHING] : Does not contact the immediately preceding figure. (initial value)
M	NEXT CONNECTION	[TANGNT] : Contacts the next figure. [NOTHING] : Does not contact the next figure. (initial value)

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
C*	END POINT X	X coordinate of an arc end point (input value)
D*	END POINT Z	Z coordinate of an arc end point (input value)
E*	RADIUS	Arc radius (input value)
P*	CENTER POINT CDX	X coordinate of an arc center (input value)
Q*	CENTER POINT CZ	Z coordinate of an arc center (input value)
L	LAST CONNECTION	[1] : In contact with the immediately preceding figure [2] : Not in contact with the immediately preceding figure (input value)
M	NEXT CONNECTION	[1] : In contact with the immediately preceding figure [2] : Not in contact with the immediately preceding figure (input value)
S	SELECT FIG. INFO.	In the case of plural intersection or contact, the operator sets a candidate. (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arbitrary blank figure block (chamfering): G1974



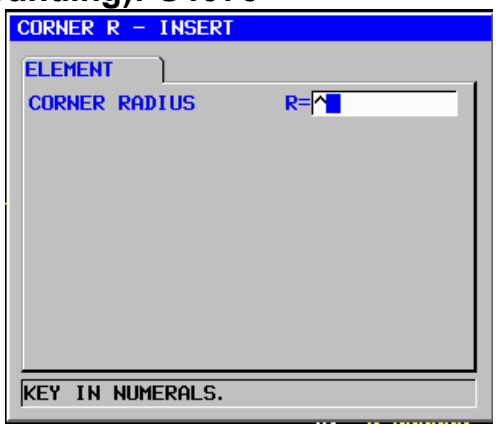
ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
C	CHAMFER	Chamfer (radius value, positive value)

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
C	CHAMFER	Amount of chamfering (radius value, positive value) (input value)

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arbitrary blank figure block (corner rounding): G1975



ELEMENT (INPUT DATA) (Note 1)		
Data item		**Meaning**
R	CORNER RADIUS	Corner R radius (radius value, positive value)

NOTE
1 'INPUT DATA' means the items, which are displayed on the input data window in editing or altering.

ELEMENT (OUTPUT DATA) (Note 2)		
Data item		**Meaning**
H	END POINT X	X coordinate of an arc end point (calculation result)
V	END POINT Z	Z coordinate of an arc end point (calculation result)
R	RADIUS	Arc radius (calculation result)
I	CENTER POINT X	X coordinate of an arc center (calculation result)
J	CENTER POINT Z	Z coordinate of an arc center (calculation result)
K	ROTATION DIRECTION	[2] : clockwise [3] : counterclockwise

NOTE
2 'OUTPUT DATA' means the items, which are displayed on the program window as creating program. It can be referenced only for program display purposes.

Arbitrary blank figure block (end) : G1976

This block is output at the end of a series of arbitrary figure blocks.

Arbitrary blank figure block (around X) (start point): G1970
Arbitrary blank figure block (around X) (line): G1971
Arbitrary blank figure block (around X) (arc(CW)): G1972
Arbitrary blank figure block (around X) (arc(CCW)): G1973
Arbitrary blank figure block (around X) (chamfering): G1974
Arbitrary blank figure block (around X) (corner rounding): G1975
Arbitrary blank figure block (around X) (end) : G1976

This blank is used for the machining simulation whose workpiece rotates around X-axis in machining centers.
Data setting is the same as the above arbitrary blank figure.

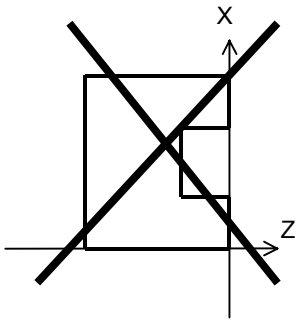
NOTE

1 There is the following limitations when the simulation whose workpiece rotates around X axis is executed.
- This function is available in only machining centers.
- Turning cycle can't be simulated.
- Polar coordinate interpolation can't be simulated.
- Under simulation as workpiece rotates workpiece around X-axis, coordinate is displayed in X, Y, -Z.

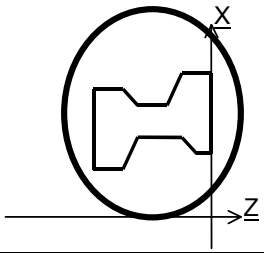
2 The following parameter setting is necessary.
- No.27003#2=1, 27003#1=0 and 27003#0=0
- No.14717 : The axis number of work rotation axis

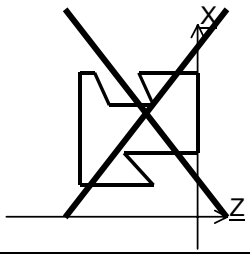
NOTE

1 The arc commands (G1972 and G1973) and corner rounding command (G1975) are changed to linear elements with several blocks, and then displayed. Depending on the figure, it may require a longer time before being completely displayed.
2 Input the end point of an arbitrary blank figure such that it matches with the start point.
3 A series of arbitrary figures must be enclosed with G1970 (start point) and G1976 (arbitrary figure end).
4 To edit an arbitrary figure, place the cursor on the block of G1970 (start point), then press [ALTER].
5 A figure for an end face portion cannot be specified. An end face portion must contain vertical lines only.

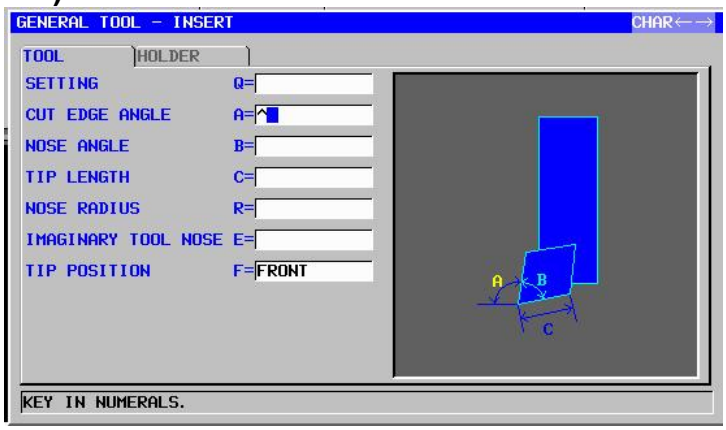


6 A figure in the Z direction from an end face portion must be a monotonously increasing or decreasing figure.





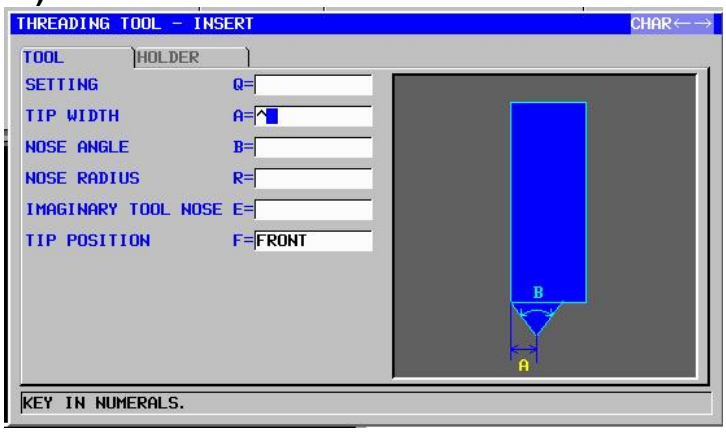
Tool definition block (general tool): G1910



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
A	CUT EDGE ANGLE	Angle of the cutting edge (positive value) Remark) Even when the same tool is used, the location of the cutting edge angle varies with the cutting direction (for example, outer surface machining and end facing).
B	NOSE ANGLE	Angle of the tool nose (positive value) Remark) Generally, the nose angle remains unchanged even when the cutting direction changes.
C	TIP LENGTH	Length of the tool nose portion that can actually cut (positive value)
R	NOSE RADIUS	Radius of the tool nose (positive value)
E	IMAGINARY TOOL NOSE	Imaginary tool nose position. Select a number from the menu indicated in the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
F	TIP POSITION	[FRONT] : Display the tip in front of the holder. (tool for forward spindle rotation) [REAR] : Display the tip in the rear of the holder. (tool for reverse spindle rotation)

HOLDER		
Data item		**Meaning**
L*	HOLDER LENGTH	Length of the holder (positive value)
W*	HOLDER WIDTH	Width of the holder (positive value)
I*	HOLDER LENGTH 2	When the tip is installed in the direction opposite to the holder installation direction, the distance between the longitudinal end of the holder and tip center (positive value)
J*	HOLDER WIDTH 2	When the tip is installed in the direction opposite to the holder installation direction, the distance between the lateral end of the holder and tip center (positive value)

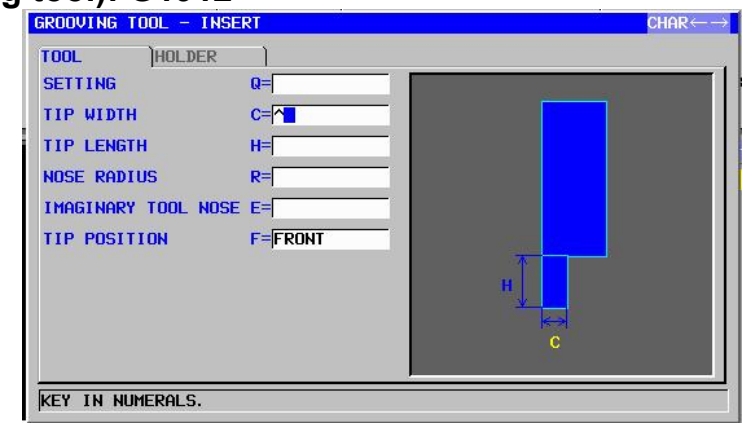
Tool definition block (thread tool): G1911



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
A	TIP WIDTH	Tool tip width
B	NOSE ANGLE	Angle of the tool nose (positive value)
R	NOSE RADIUS	Radius of the tool nose (positive value)
E	IMAGINARY TOOL NOSE	Imaginary tool nose position. Select a number from the menu indicated in the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
F	TIP POSITION	[FRONT] : Display the tip in front of the holder. (tool for forward spindle rotation) [REAR] : Display the tip in the rear of the holder. (tool for reverse spindle rotation)

HOLDER		
Data item		**Meaning**
L*	HOLDER LENGTH	Length of the holder (positive value)
W*	HOLDER WIDTH	Width of the holder (positive value)

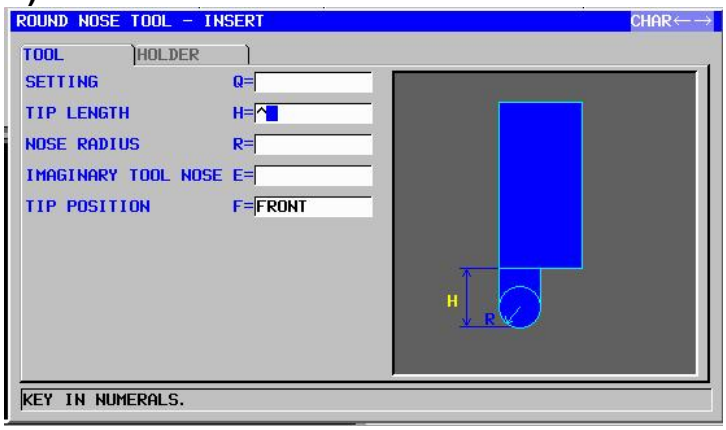
Tool definition block (grooving tool): G1912



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
C	TIP WIDTH	Tip width of the grooving tool (positive value)
H	TIP LENGTH	Length of the cutting portion of the grooving tool (positive value)
R	NOSE RADIUS	Radius of the tool nose (positive value)
E	IMAGINARY TOOL NOSE	Imaginary tool nose position. Select a number from the menu indicated in the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
F	TIP POSITION	[FRONT] : Display the tip in front of the holder. (tool for forward spindle rotation) [REAR] : Display the tip in the rear of the holder. (tool for reverse spindle rotation)

HOLDER		
Data item		**Meaning**
L*	HOLDER LENGTH	Length of the holder (positive value)
W*	HOLDER WIDTH	Width of the holder (positive value)

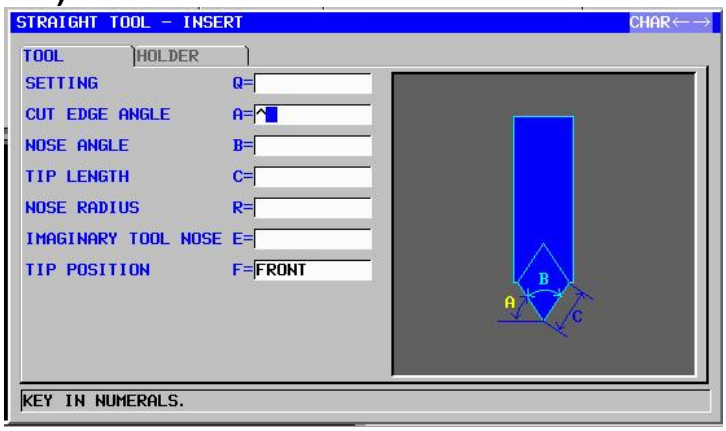
Tool definition block (round tool): G1913



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
H	TIP LENGTH	Length of the cutting portion of the round-nose tool (positive value)
R	NOSE RADIUS	Radius of the tool nose (positive value)
E	IMAGINARY TOOL NOSE	Imaginary tool nose position. Select a number from the menu indicated in the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
F	TIP POSITION	[FRONT] : Display the tip in front of the holder. (tool for forward spindle rotation) [REAR] : Display the tip in the rear of the holder. (tool for reverse spindle rotation)

HOLDER		
Data item		**Meaning**
L*	HOLDER LENGTH	Length of the holder (positive value)
W*	HOLDER WIDTH	Width of the holder (positive value)

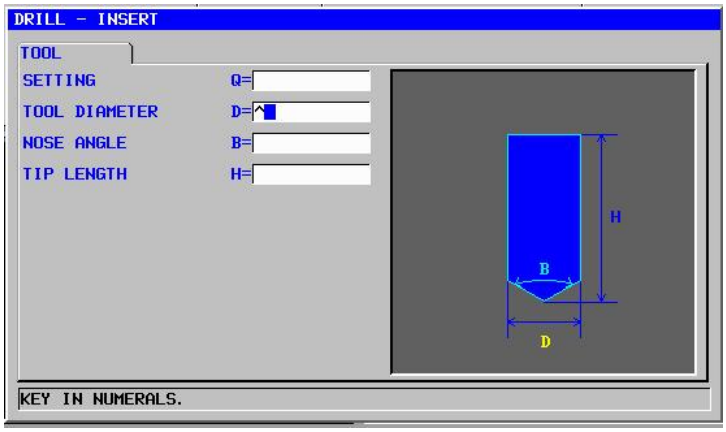
Tool definition block (straight tool): G1914



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
A	CUT EDGE ANGLE	Angle of the cutting edge (positive value)
B	NOSE ANGLE	Angle of the tool nose (positive value)
C	TIP LENGTH	Length of the tool nose portion that can actually cut (positive value)
R	NOSE RADIUS	Radius of the tool nose (positive value)
E	IMAGINARY TOOL NOSE	Imaginary tool nose position. Select a number from the menu indicated in the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
F	TIP POSITION	[FRONT] : Display the tip in front of the holder. (tool for forward spindle rotation) [REAR] : Display the tip in the rear of the holder. (tool for reverse spindle rotation)

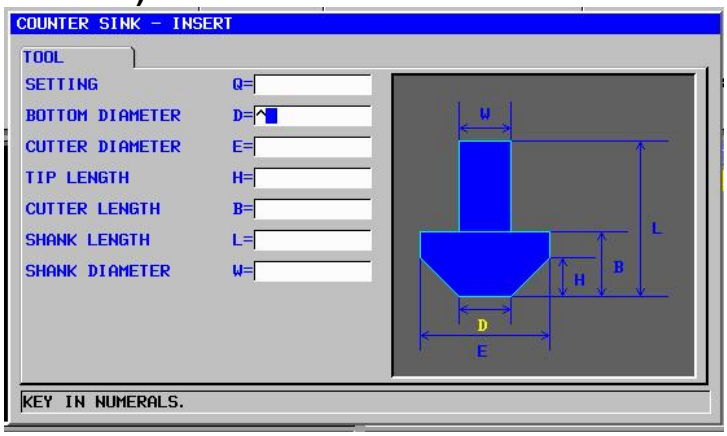
HOLDER		
Data item		**Meaning**
L*	HOLDER LENGTH	Length of the holder (positive value)
W*	HOLDER WIDTH	Width of the holder (positive value)
I*	HOLDER LENGTH 2	When the tip is installed in the direction opposite to the holder installation direction, the distance between the longitudinal end of the holder and tip center (positive value)
J*	HOLDER WIDTH 2	When the tip is installed in the direction opposite to the holder installation direction, the distance between the lateral end of the holder and tip center (positive value)

Tool definition block (drill): G1921



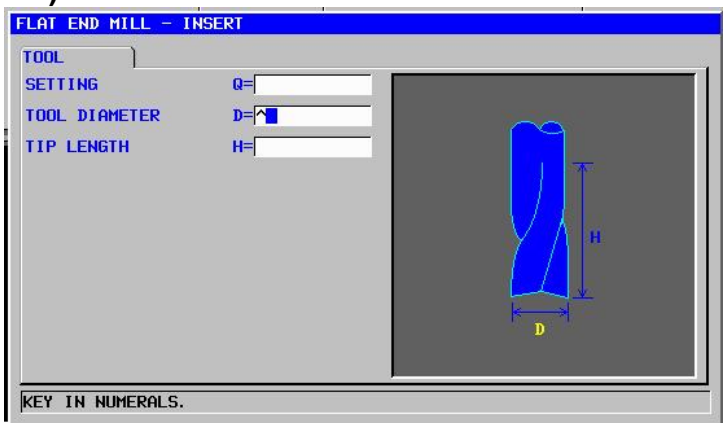
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	Drill diameter (positive value)
B	NOSE ANGLE	Drill nose angle (positive value)
H	TIP LENGTH	Drill length (positive value)

Tool definition block (counter sink tool): G1931



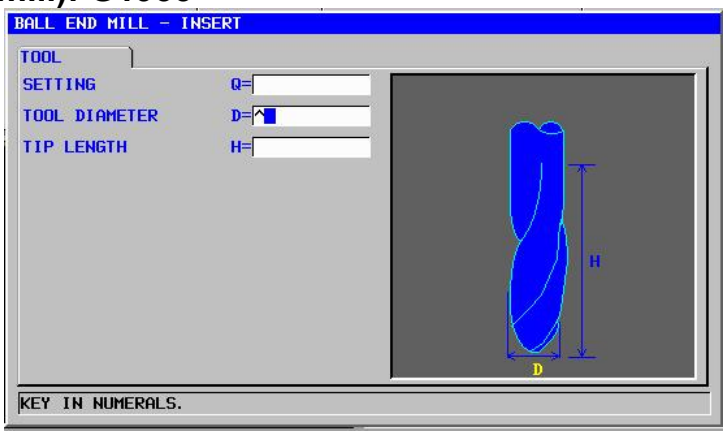
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	BOTTOM DIAMETER	Diameter of the end of the counter sink tool end (positive value)
E	CUTER DIAMETER	Diameter of the counter sink tool (positive value)
H	TIP LENGTH	Length of the cutting portion of the counter sink tool (tool axis direction, positive value)
B	CUTTER LENGTH	Cutter length of the counter sink tool (tool axis direction, positive value)
L	SHANK LENGTH	Entire length of the counter sink tool (positive value)
W	SHANK DIAMETER	Diameter of the shank of the counter sink tool (positive value)

Tool definition block (flat end mill): G1932



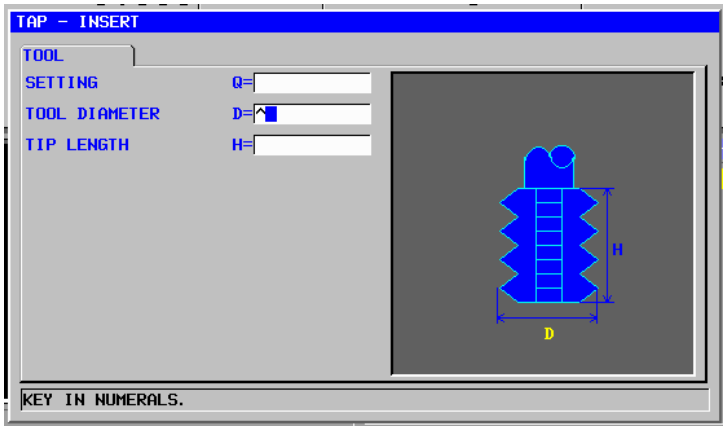
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	End mill diameter (positive value)
H	TIP LENGTH	End mill tool length (positive value)

Tool definition block (ball end mill): G1933



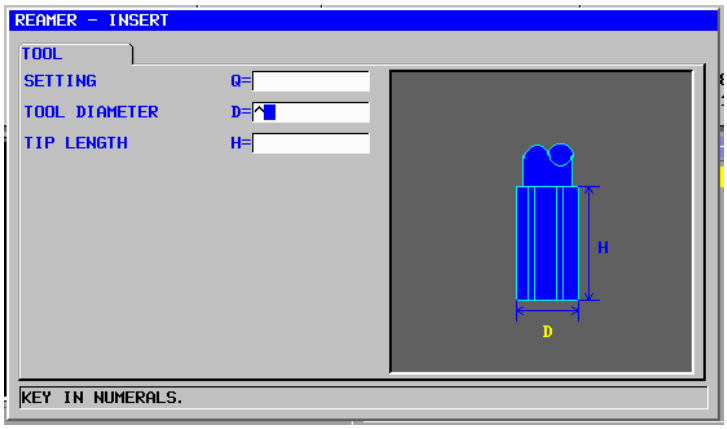
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	End mill diameter (positive value)
H	TIP LENGTH	End mill tool length (positive value)

Tool definition block (tap): G1922



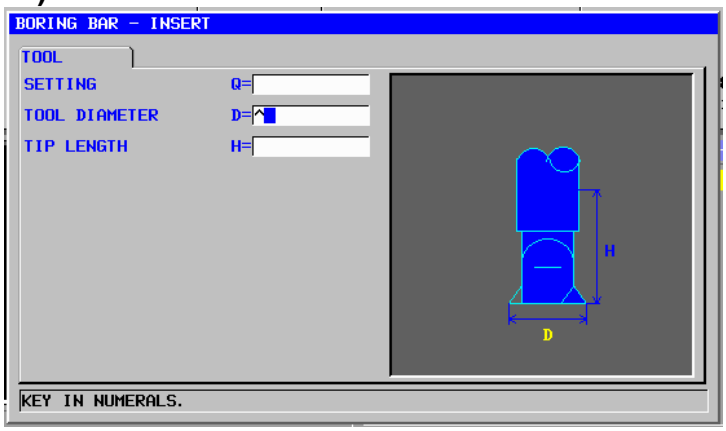
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	Tap diameter (positive value)
H	TIP LENGTH	Tap length (positive value)

Tool definition block (reamer): G1923



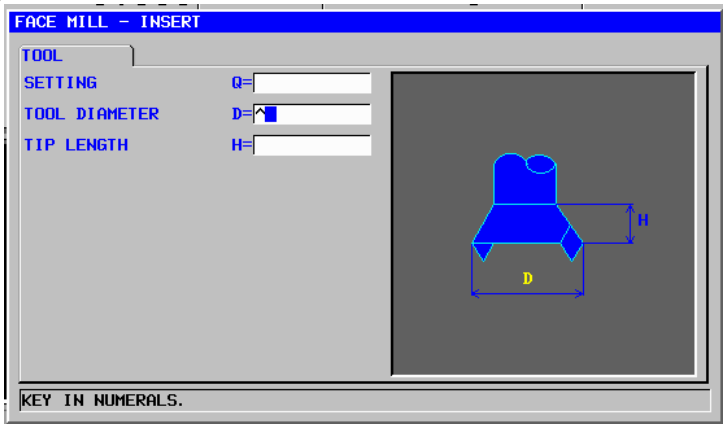
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	Reamer diameter (positive value)
H	TIP LENGTH	Reamer length (positive value)

Tool definition block (boring tool): G1924



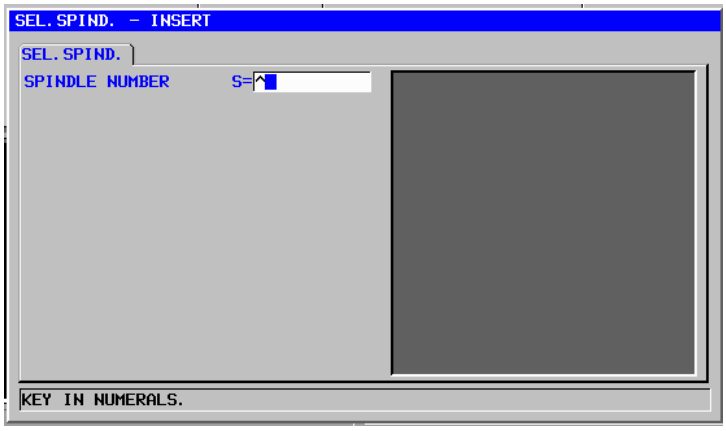
TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	Boring tool diameter (positive value)
H	TIP LENGTH	Boring tool length (positive value)

Tool definition block (face mill): G1930



TOOL		
Data item		**Meaning**
Q	SETTING	Tool installation direction. Select the number of an installation method from the illustration. Remark) To be selected visually for both vertical and horizontal lathes.
D	TOOL DIAMETER	Face mill diameter (positive value)
H	TIP LENGTH	Face mill length (positive value)

Spindle selection block: G1998



SEL. SPIND.		
Data item		**Meaning**
S	SPINDLE NUMBER	Spindle number of a subspindle (positive number) Remark) Enter 2 when the subspindle has the spindle number 2. Enter 3 when the subspindle has the spindle number 3.

NOTE
The menu above is prepared for a machine with a subspindle, and is displayed when bit 1 of parameter No. 14702 = 1.

9.10 SPINDLE MOVEMENT ANIMATION FOR AUTOMATIC LATHES

This is the additional animation function to simulate machining that utilizes movement of spindle for automatic lathes.
The option of "spindle movement animation for automatic lathe" is necessary to use this function.

NOTE
This function is available for only Series 16*i*/18*i*/21*i*.

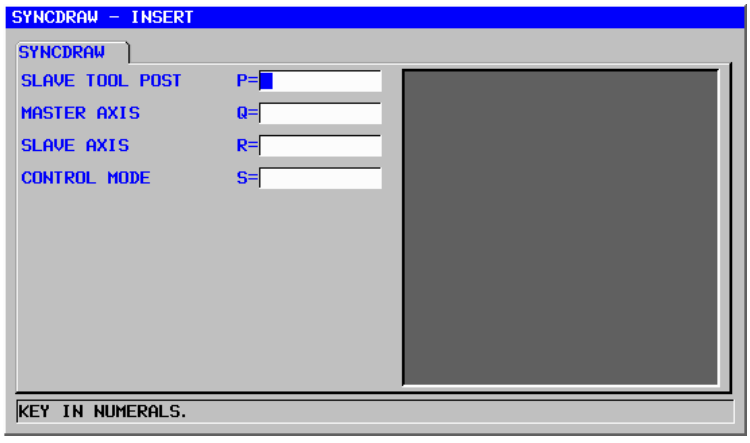
9.10.1 Function

If the command that reflects movement of master in movement of slave is entered in program and the command is executed, slave tool moves too. From this movement, machining that utilizes movement of spindle will be simulated.

9.10.2 Start Command of Reflection

The "SYNCDRAW" tab will be displayed by pressing [START]. The following window for inputting start command of reflection will be displayed, after the "SYNCDRAW" tab is selected, the cursor is placed on "START SYNCHRONIZ-ATION CONTROL" and **INPUT** key is pushed. The reflection in slave starts by the command.

Start command of reflection : G1994

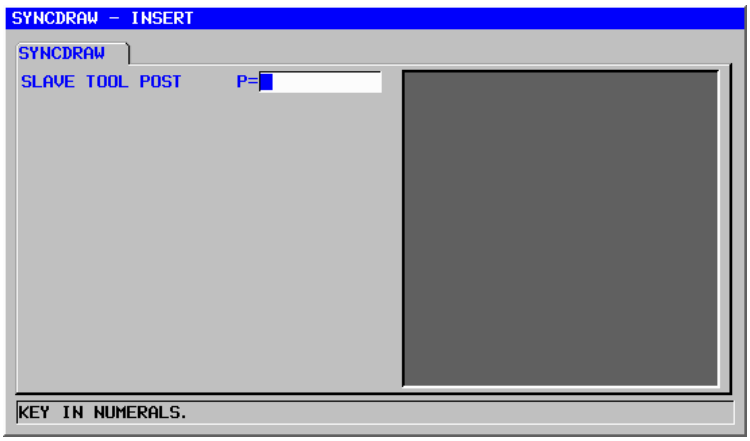


Condition		
Data item		Meaning
P	SLAVE TOOL POST	Number of slave tool post in which Manual Guide *i* reflects movement of master.
Q	MASTER AXIS	Number of axis that is reflected (X, Y, Z, or C axis)
R	SLAVE AXIS	Number of axis in which Guide *i* reflects movement of master (X, Y, Z, or C axis)
S	CONTROL MODE	[SYNC]:Under synchronization control mode [NOSYNC]:Not Under synchronization control mode

9.10.3 End Command of Reflection

The following window for inputting end command of reflection will be displayed, after the cursor is placed on "END SYNCHRONIZATION CONTROL" in the "SYNCDRAW" tab and **INPUT** key is pushed. The reflection in slave finishes by the command.

End command of reflection : G1995



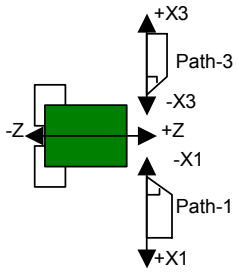
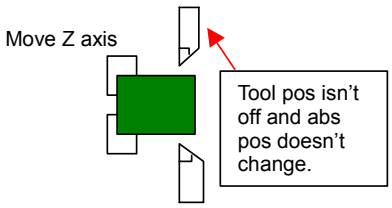
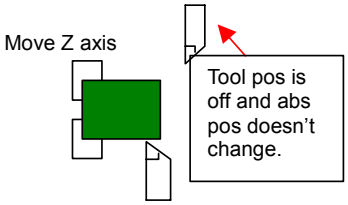
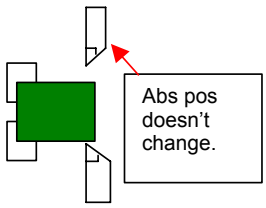
Condition		
Data item		Meaning
P	SLAVE TOOL POST	Number of slave tool post in which Manual Guide *i* reflects movement of master.

NOTE
1 Setting the parameter No27310#5 to "1" is necessary to display "SYNCDRAW" tab.

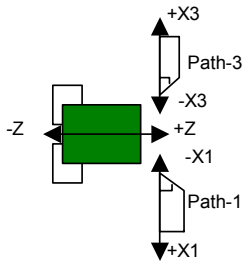
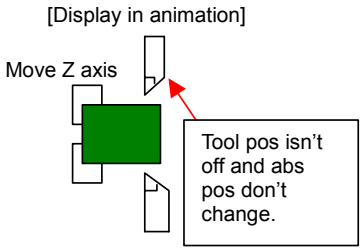
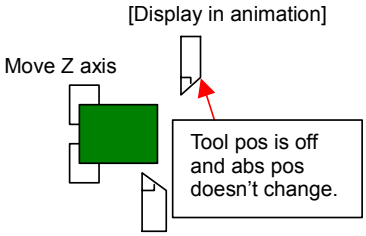
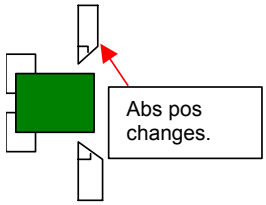
9.10.4 Simulation and Actual Working of the Machine

Between G1994 and G1995, the movement of axis number that is designated at Q is reflected in the movement of axis that is designated by R. And R is a number of tool post that is designated by P.

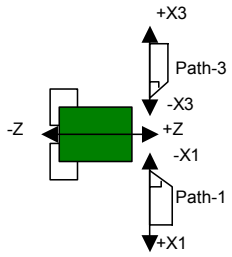
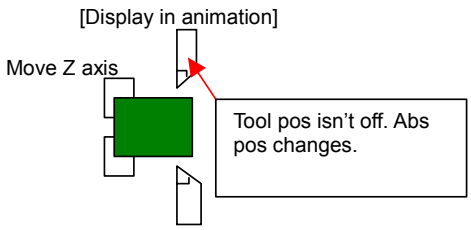
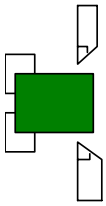
Ex1. Working not under synchronous, composite, or superimposed control

Path-1(There is reflect command)	Path-1(There isn't reflect command)	Working of actual machine
Not under synchronous, composite, or superimposed control	Not under synchronous, composite, or superimposed control	
G1994 P3.Q2. R2. S0.; Reflect movement of path-1 second axis (Z-axis) in movement of path-3 second axis (Z-axis).	There isn't command.	
[Display in animation] 	[Display in animation] 	
G1995 P3.; End reflection in path-3.	There isn't command.	

Ex.2 Working under synchronization control

<table>
<tr><td>Path-1(There isn't reflect command)</td><td>Path-1(There isn't reflect command)</td><td>Working of actual machine</td></tr>
<tr><td>Under synchronization control</td><td>Under synchronization control</td><td> 
</td></tr>
<tr><td>G1994 P3.Q2. R2. S1.;
Reflect movement of path-1 second axis (Z-axis) in movement of path-3 second axis (Z-axis).</td><td>There isn't command.</td><td></td></tr>
<tr><td> 
</td><td> 
</td><td> 
</td></tr>
<tr><td>G1995 P3.;
End reflection in path-3.</td><td>There isn't command.</td><td></td></tr>
</table>

Ex3. Working under composite control or superimposed control

Path-1(There isn't reflect command)	Working of actual machine
Under composite control or superimposed control	
G1995 P3.; End reflection in path-3.	
	
Not under composite control or superimposed control	

NOTE

1 The option of spindle movement animation for automatic lathe is necessary to use this function.
2 Set parameter No.27311#1 to "1" when this function is used.
3 Master axis must be only basic three axes and C axis. And slave axis must be the same axis as master.
4 In animation, slave absolute axis isn't renewed though under Synchronization control (S1. is commanded in G1994).
5 When machining simulation is finished by reset key as so on, the reflection will be canceled.
6 Master tool post can't become slave tool post though G1994 that designates the tool post as the slave is command from other tool post.
7 During reflecting movement, when second start command of reflection is executed and control mode that is designated by second command is different from former one, former start command will be canceled. When control mode is same as former one, slave axis will be newly added and former command won't be canceled.

10 SETTING DATA

<1> BASIC
1. WORK COORDINATE DATA
2. TOOL ODFFSET DATA
3. FIXED FORM SENTENCE FOR MILLING
4. FIXED FORM SENTENCE FOR TURNING
5. SETTING OF OFFSET NO. AND TOOL NO.
6. TOOL MANAGEMENT DATA
7. TOOL LIFE MANAGEMENT DATA
Remark) For items 5, 6, and 7, see the description of "V. Tool Management Function" .

<2> MEASURE COND
1. SETTING

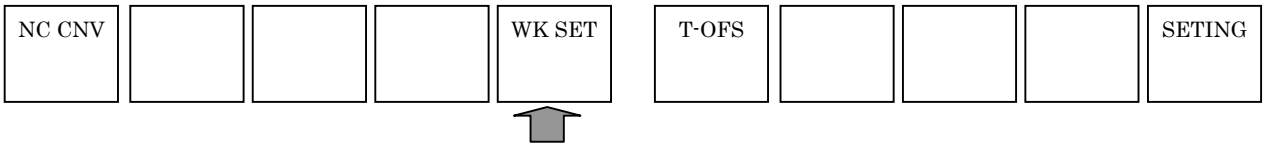
<3> CALIBRATION
1. SETTING
Remark) For <2> and <3>, refer to "MANUAL GUIDE *i* OPERATOR'S MANUAL Set-up Guidance Function."

NOTE
The menu mentioned above shows only items that are required depending on the provided options and the machine type; all the items above are not indicated.
For details, refer to the relevant manual issued by the machine tool builder.

10.1 SETTING THE WORKPIECE COORDINATE DATA

[WK SET] to open the workpiece coordinate data window can be displayed on all mode such as MEM, EDIT and manual mode.
Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the soft-keys including [WK SET]

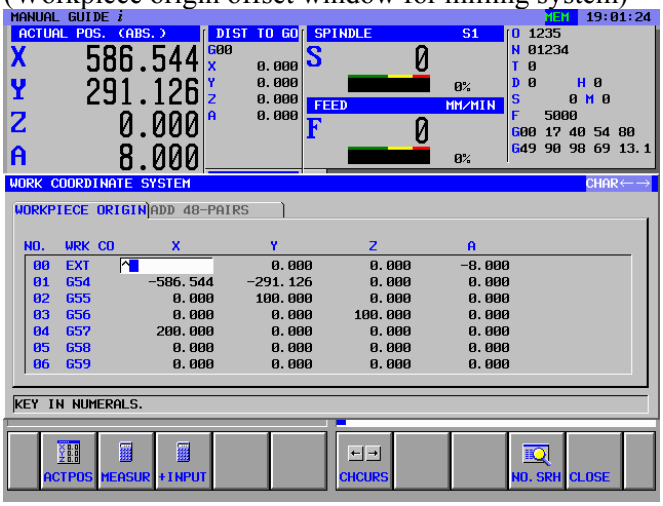
Example of MEM mode soft-keys)



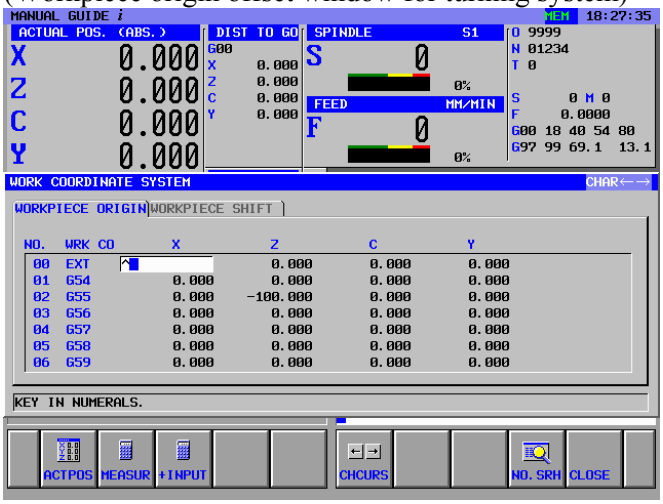
Pressing [WK SET] displays the workpiece coordinate data window.

For compound machine tools, the workpiece coordinate setting windows for the T mode and M mode can be selected with corresponding tabs.
For lathes, a window for setting the workpiece origin offset data and workpiece coordinate shift data is displayed.
For machining centers, a window for setting the workpiece origin offset is displayed.

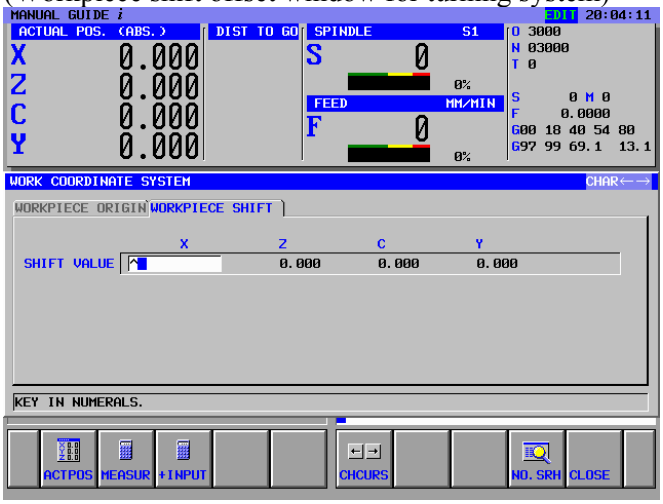
(Workpiece origin offset window for milling system)



(Workpiece origin offset window for turning system)



(Workpiece shift offset window for turning system)



The data items to be set and displayed are common to the corresponding data items of the CNC. So, for details, refer to the operator's manual of the CNC.

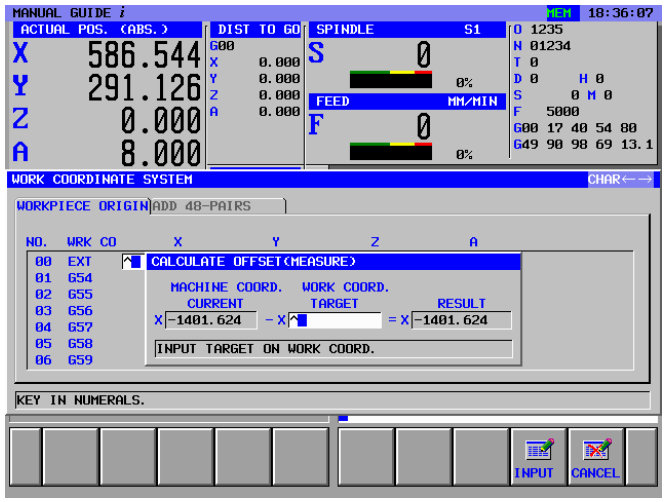
10.1.1 [MEASUR] Soft Key

ACTPOS	MEASUR	+INPUT			CHCURS			NO.SRH	CLOSE

By pressing [MEASUR], the calculations below can be made.

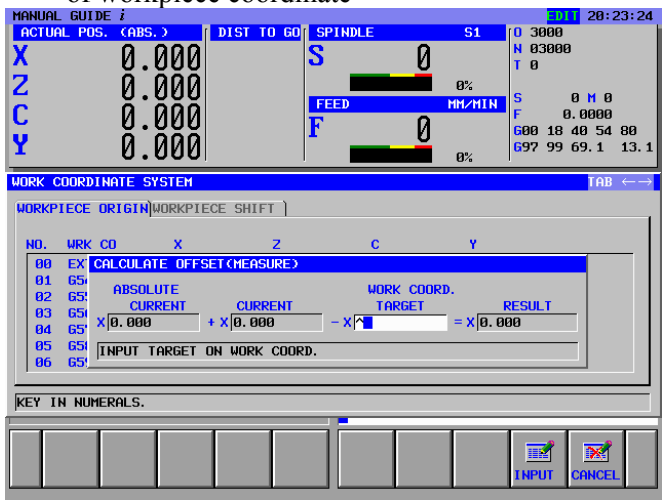
(Workpiece origin offset window for milling system)

Current machine coordinate value - Target value of workpiece coordinate

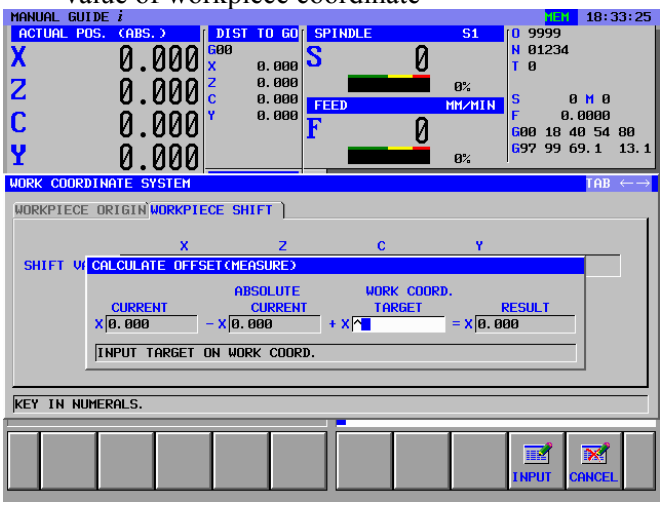


(Workpiece origin offset window for turning system)

Current absolute coordinate value + Current setting - Target value of workpiece coordinate



(Workpiece coordinate system shift amount with the turning system)
Current setting - Current value of absolute coordinate + Target value of workpiece coordinate

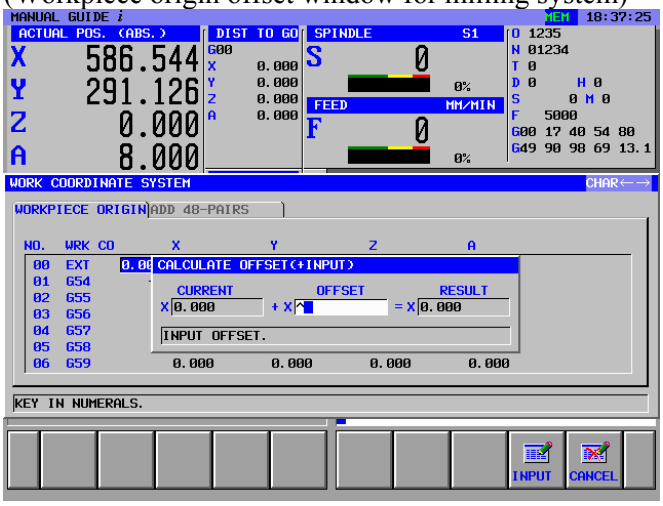


10.1.2 [+INPUT] Soft Key

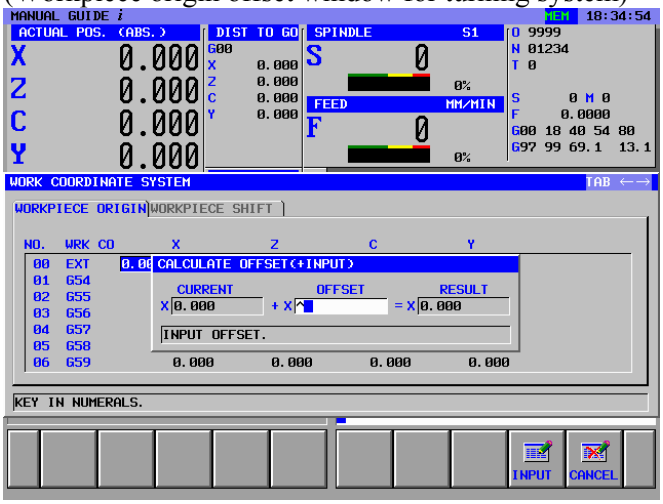
ACTPOS	MEASUR	+INPUT			CHCURS			NO.SRH	CLOSE

By pressing the [+INPUT], "current value + offset value" can be calculated.

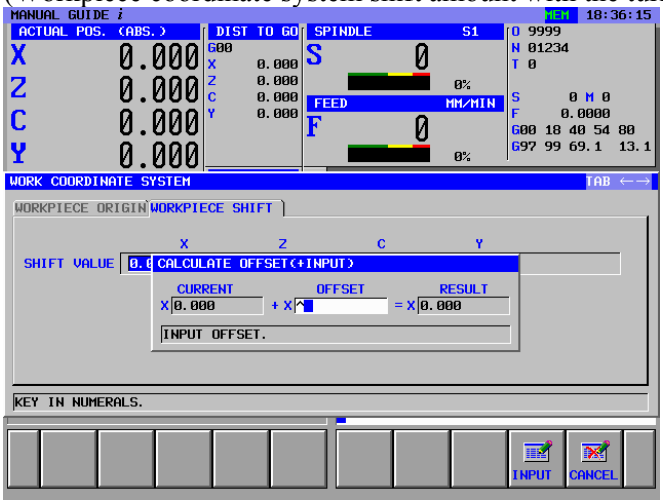
(Workpiece origin offset window for milling system)



(Workpiece origin offset window for turning system)



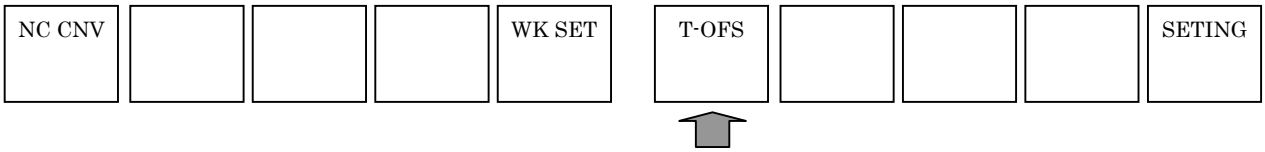
(Workpiece coordinate system shift amount with the turning system)



10.2 SETTING TOOL OFFSET DATA

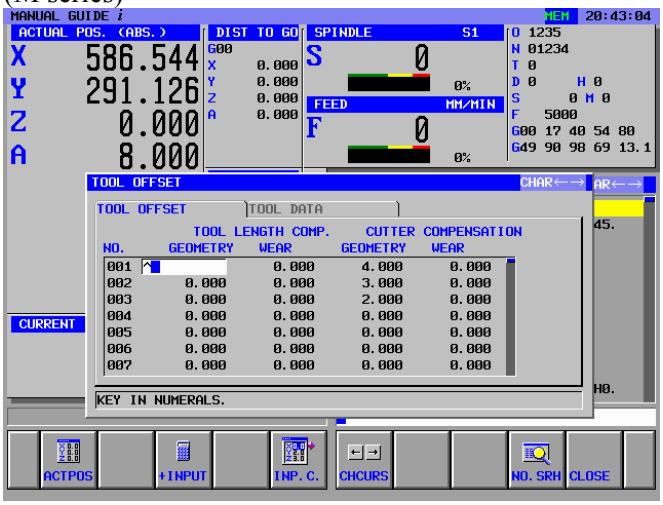
[T-OFS] to open the tool offset data window can be displayed on all mode such as MEM, EDIT and manual mode.
Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the soft-keys including [T-OFS]

Example of MEM mode soft-keys)

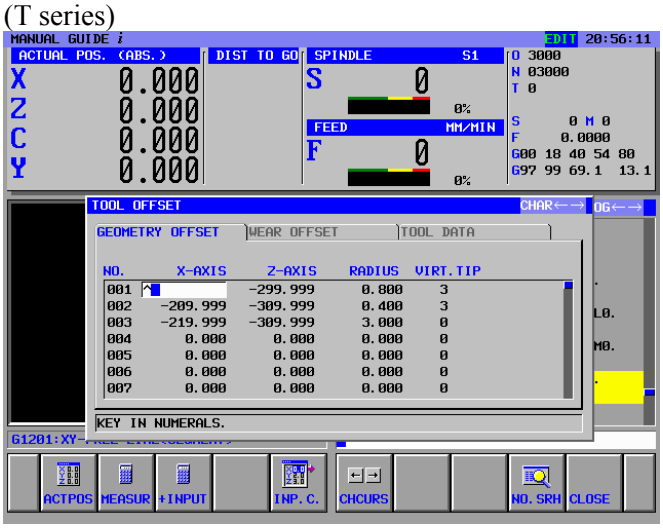


Pressing [T-OFS] displays the tool offset data window.

(M series)



(T series)



For compound machine tools, the following data items are displayed for the T mode:

<1> T: GEOMETRY OFFSET
<2> T: WEAR OFFSET
<3> T: GEOMETRY TOOL TYPE OFFSET
<4> T: GEOMETRY WEAR TYPE OFFSET

The following data items are displayed for the M mode:

<5> M: TOOL OFFSET (TOOL LENGTH COMP. / CUTTER COMPENSATION)
<6> M: TOOL TYPE OFFSET (TOOL LENGTH COMP. / CUTTER COMPENSATION)

Data for each mode can be selected with a corresponding tab.

For lathes, data items <1> to <4> are displayed. For machining centers, data items <5> and <6> are displayed.

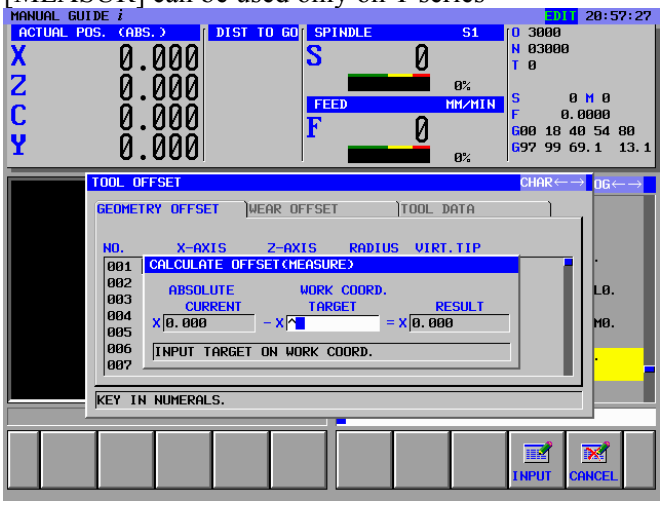
The data to be set and displayed in <1>, <2>, and <5> is common to the corresponding data in the CNC. For details, refer to the operator's manual of the CNC.
For data items <3>, <4>, and <6>, which are related to the tool management function, see the description of "V. Tool Management Function".

10.2.1 [MEASUR] Soft Key

ACTPOS	MEASUR	+INPUT		INP.C.	CHCURS			NO.SRH	CLOSE

By pressing [MEASUR], "Current machine coordinate value - Target value of workpiece coordinate" can be calculated.
With the [WEAR OFFSET] tab usable when tool geometry/wear compensation option is specified, "Current machine coordinate value - Current geometry offset value - Target workpiece coordinate value" can be calculated.

[MEASUR] can be used only on T series

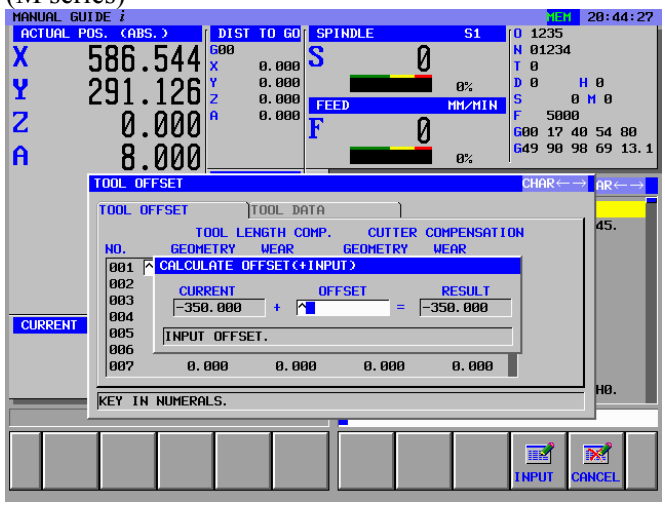


10.2.2 [+INPUT] Soft Key

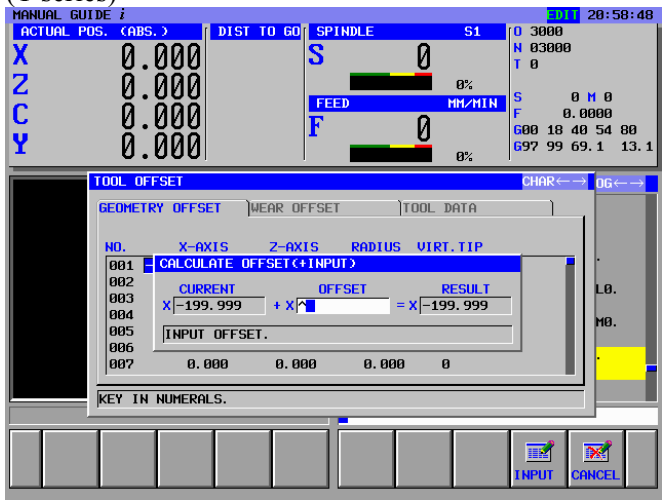
ACTPOS	MEASUR	+INPUT		INP.C.	CHCURS			NO.SRH	CLOSE

By pressing the [+INPUT] soft key, "Current value + Offset value" can be calculated.

(M series)



(T series)

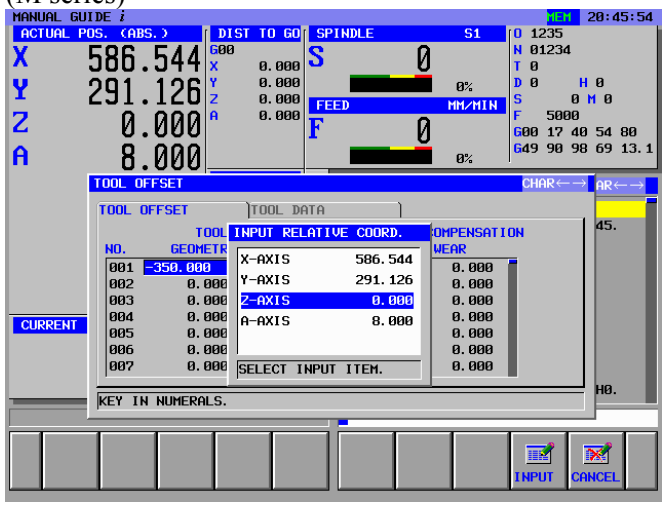


10.2.3 [INP.C.] Soft Key

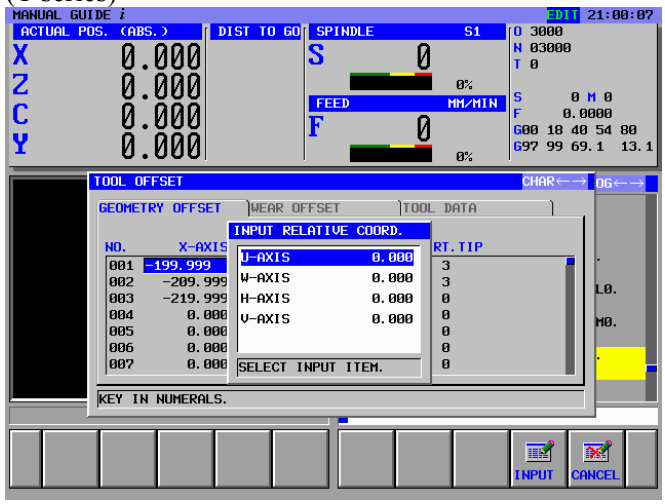
ACTPOS	MEASUR	+INPUT		INP.C.	CHCURS			NO.SRH	CLOSE

By pressing the [INP.C.] soft key, "Relative coordinate value" can be entered to the offset value directly.

(M series)



(T series)



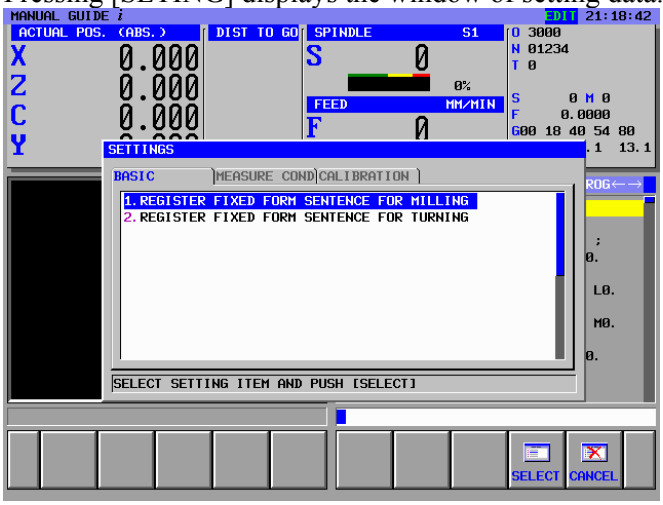
10.3 REGISTERING FIXED FORM SENTENCES

[SETING] to open the setting window can be displayed on all mode such as MEM, EDIT and manual mode.
Pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the soft-keys including [SETING]

Example of MEM mode soft-keys)

NC CNV				WK SET	T-OFS				SETING

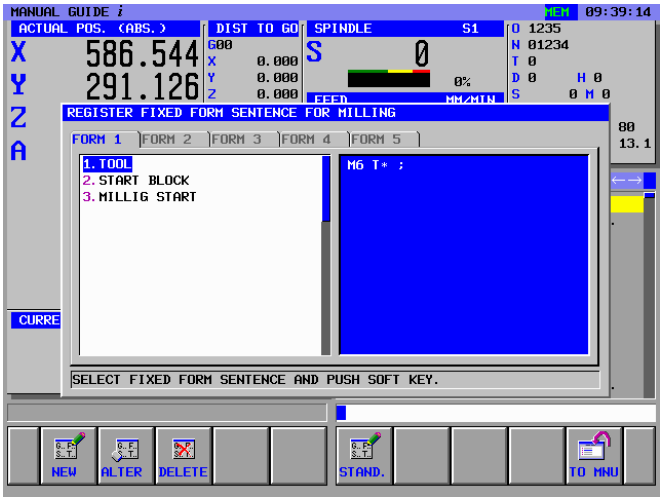
Pressing [SETING] displays the window of setting data.



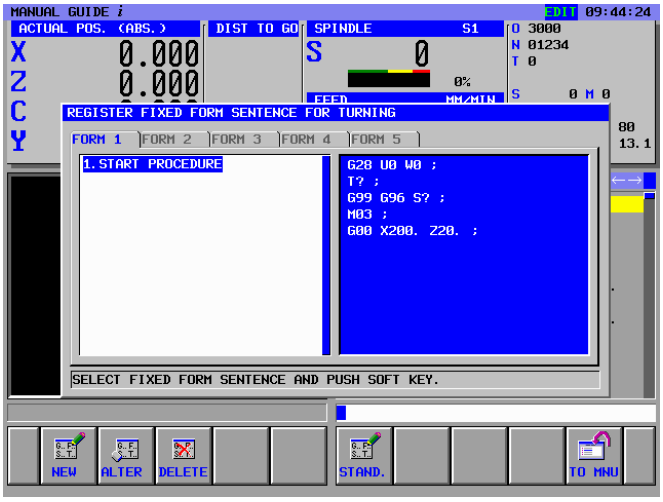
NOTE
1 On the T series CNC, the “REGISTER FIXED FORM SENTENCE” menu for milling is displayed only when the milling cycle optional function is attached. The menu for turning is always displayed.
2 On the M series CNC, the REGISTER FIXED FORM SENTENCE” menu for turning is displayed only when the turning cycle optional function is attached. The menu for milling is always displayed.

After selecting the menu of “REGISTER FIXED FORM SENTENCE FOR MILLING” by placing the cursor, pressing [SELECT] displays the window of data setting.
By similar operations, the window for turning can be displayed.

With "FIXED FORM SENTENCE FOR MILLING," which is called by [FIXFRM] displayed together with the milling menu, you can modify the contents of a selected fixed form sentence or add a new sentence.
Selecting “FIXED FORM SENTENCE FOR MILLING” displays the following screen.



Selecting “FIXED FORM SENTENCE FOR TUNING” displays the following screen.



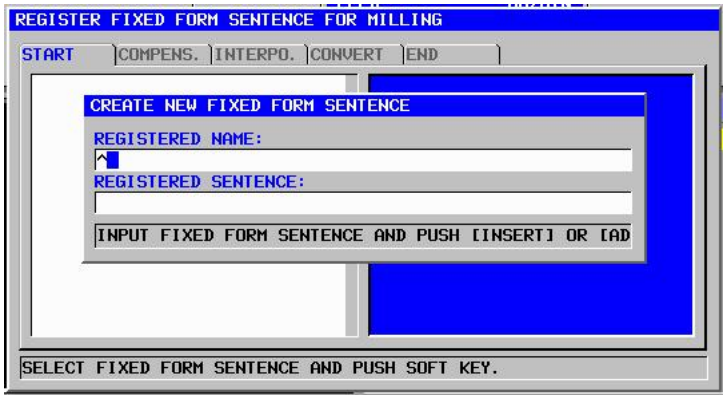
NOTE
1 The fixed form sentence menu displayed in the tab of “FORM1” has same contents with the one displayed in the “START” menu. Into detail, refer to the II 4.1 “ENTERING THE START COMMAND”.
2 The fixed form sentence menu displayed in the tab of “FORM5” has same contents with the one displayed in the “END” menu. Into detail, refer to the II 4.8 “ENTERING THE END COMMAND”.
3 When the parameter No. 14850#3 is ‘1’, prohibiting registering fixed form sentences by memory protection key will be enabled.

10.3.1 Registering a New Fixed Form Sentence

When the REGISTER FIXED FORM SENTENCE MILLING / TURNING window is displayed on a screen, the following soft-keys are displayed.

NEW	ALTER	DELETE			STAND.				TO MNU

By pressing [NEW], a window for registering a new fixed form sentence appears. At the same time, the following soft keys appear:



			COPY	PASTE			INSERT	ADD	CANCEL

Position the cursor at the "REGISTERED NAME" item, from the MDI keyboard enter the name of the fixed form sentence you want to register, then press the **INPUT** key to input the name.
Then, position the cursor at the "REGISTERED SENTENCE" item, enter a fixed form sentence from the MDI keyboard, press the **INPUT** key, then press [INSERT] or [ADD]. This can register the new fixed form sentence.

[COPY] : Select all registered names or fixed form sentences and copy them to the clipboard.
[PASTE] : Paste the clip board contents. You can copy a part of a machining program being edited to the clipboard in advance, then you can use it later to, for example, copy it in "REGISTERED SENTENCE."
[INSERT] : Add the name of the new fixed form sentence to be registered to a location immediately before the item positioned by the cursor when [NEW] is pressed. The menu numbers of the subsequent fixed form sentences are all incremented by one.
[ADD] : Add the name of the new fixed form sentence to be registered to the end of the menu already registered.
[CANCEL] : Cancel a registration operation.

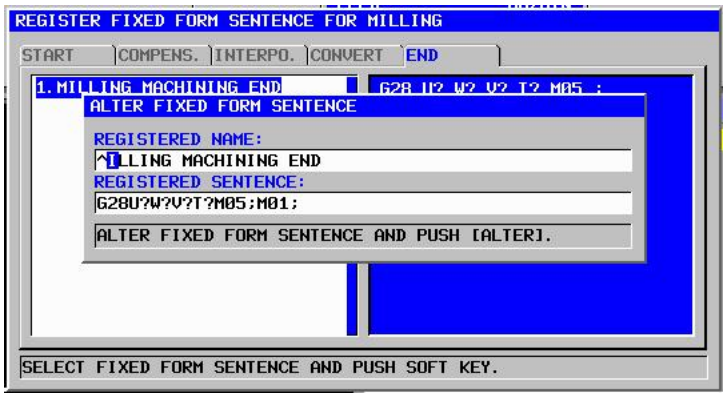
NOTE

1 About the number of fixed form sentences per tab and the maximum characters per fixed form sentence, the following settings can be selected.
  <1> The number of fixed form sentences per tab is 10 and the maximum characters per fixed form sentence is 128.
  <2> The number of fixed form sentences per tab is 5 and the maximum characters per fixed form sentence is 256.
2 The characters per fixed form sentence increase to 256 when bit 4 of parameter No.14852 = 1.
3 Please power on again when the above parameter is changed. And after that, fixed form sentences will be initialized on starting the machine again.

10.3.2 Modifying a Fixed Form Sentence

NEW	ALTER	DELETE			STAND.				TO MNU

Position the cursor to the name of the fixed form sentence you want to modify, and press [ALTER]. A window for modifying a fixed form sentence then appears.

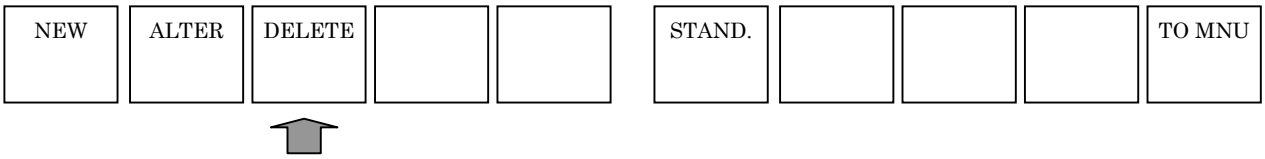


When the above window is displayed, the following soft keys appear:

			COPY	PASTE				ALTER	CANCEL

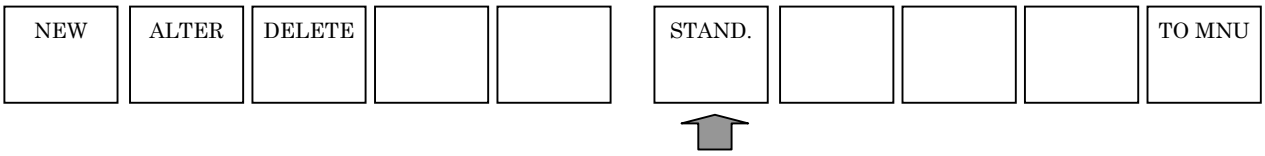
In the "REGISTERED NAME" and "REGISTERED SENTENCE" items, the contents of the selected fixed form sentence are displayed.
Select an item by using the ↑ and ↓ cursor keys, select the part you want to modify by using the ← and → cursor keys, then enter a new character string from the MDI keyboard. This character string is inserted immediately before the cursor position.
With the **CAN** key, you can delete the previous character string one character at a time.
In the same way as in registration, [COPY] and [PASTE] can also be used.
When you have modified "REGISTERED NAME" and "REGISTERED SENTENCE," you must press the **INPUT** key at the end of the modification.
Pressing [ALTER] replaces the original fixed form sentence with the new fixed form sentence.

10.3.3 Deleting a Fixed Form Sentence



Position the cursor at the name of the fixed form sentence you want to delete, and press [DELETE]. Then a message for confirming a deletion operation is displayed. If you press [YES], the fixed form sentence is deleted.
Pressing [NO] cancels the deletion operation.

10.3.4 Initialization to Standard Fixed Form Sentences



The machine tool builder can factory-set particular fixed form sentences as standard fixed form sentences. For details, refer to the relevant manual issued by the machine tool builder.
In this case, the fixed form sentences are stored as initial data in a memory area in which data cannot be deleted.
You can use [STAND.] to restore the initial state factory-set by the machine tool builder.

NOTE
1 When fixed form sentences are initialized, the sentences that have been entered or modified so far are all deleted; so, care is necessary.
2 Also when the machine tool builder reads standard fixed form sentences into memory, initialization using [STAND.] is required first.

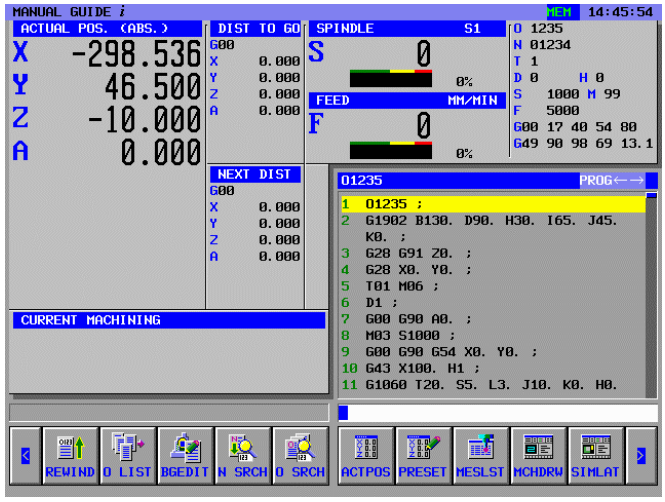
When [STAND.] is pressed, a message for confirming initialization is displayed. By pressing [YES], initialization is performed.
Selecting [NO] cancels the initialization.

11 BACKGROUND EDITING

During actual machining on the machine, contents of the other part program can be edited.

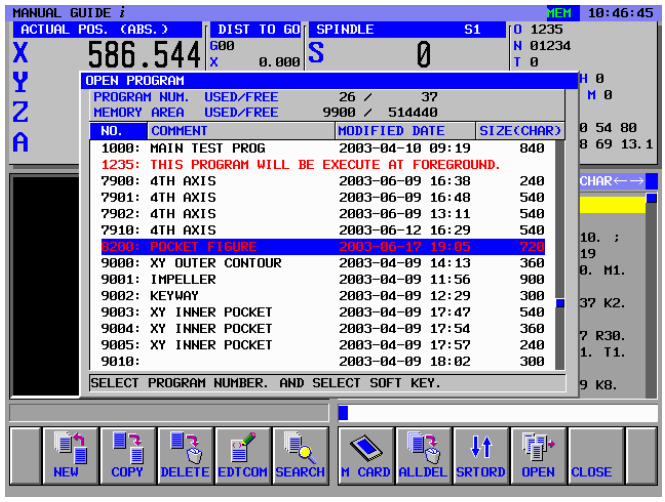
11.1 STARTING BACKGROUND EDITING

When MEM mode is selected on the machine operator's panel, The following program screen is displayed whether the actual machining is executing or not.



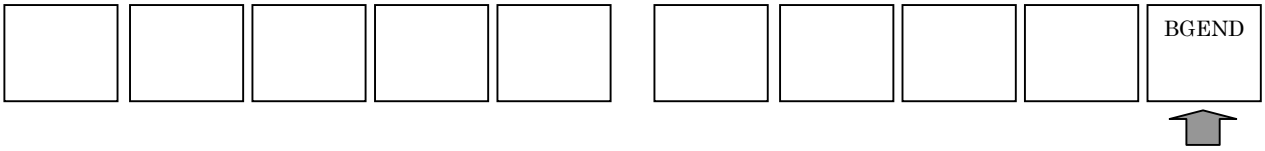
REWIND	O LIST	BGEDIT	N SRCH	O SRCH	ACTPOS.	PRESET	MESLST	MCHDRW	SIMLAT

Pressing the [BGEDIT] soft key switches the screen display to the background editing screen and displays the program list screen.
Move the cursor to a machining program to be edited, with the cursor keys, then select the machining program by pressing [OPEN].



11.2 ENDING BACKGROUND EDITING

During background editing, pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the soft-keys including [BGEND]



Pressing the [BGEND] soft key ends the background editing screen and returns the screen display to the automatic operation screen.

11.3 OPERATIONS DURING BACKGROUND EDITING

Operations during background editing are basically the same as editing operations in the EDIT mode. Note, however, that the use of some operations is restricted.

12 NC PROGRAM CONVERSION FUNCTION

Pressing the [NC CNV] soft key starts the NC program conversion function.
With the NC program conversion function, a 4-digit G cycle machining command can be dissolved into a single move command and stored in the part program storage area of the NC.

* When the NC program conversion function is used, the milling cycle option or turning cycle option is required.

12.1 BASIC SPECIFICATIONS

(1) With the NC program conversion function, only a 4-digit G cycle machining command can be dissolved into a single move command. Any other types of commands are output without modification.
(2) The NC program conversion function can be used in the MEM mode only.
(3) Only an executed block becomes an NC program conversion target.
(4) A block containing an M98, M99, or custom macro program is not output to the conversion destination program.
(5) In the case of multiple paths, NC program conversion is performed for each path.
(6) If there is a loop or conditional branch program of a custom macro program, executed blocks only are output. A block containing a conditional branch program of a custom macro is not output to the conversion destination program.
(7) If a loop or conditional branch program of a custom macro program includes a 4-digit G cycle, the 4-digit G cycle is expanded as many times as the number of repeats. A block containing a conditional branch program of a custom macro is not output to the conversion destination program.
(8) Whether to output a 4-digit G cycle machining command before expansion as a comment can be chosen using bit 5 of parameter No. 14703.

Bit 5 of parameter No. 14703 = 0:

Outputs a 4-digit G cycle machining command before expansion as a comment in NC program conversion.

Bit 5 of parameter No. 14703 = 1:

Does not output a 4-digit G cycle machining command before expansion as a comment in NC program conversion.

NOTE
In Series 30*i*, a 4-digit G cycle machining command before expansion as a comment is not outputted.

(9) In the case of a subprogram call, see the examples below. A block containing M98 or M99 is not output to the conversion destination program.

(Example 1)

(Before conversion)
```
O0001
M98 P0002;
M30;
%
```

→

```
O0002
G0 X100. ;
G0 X200. ;
G0 X300. ;
M99;
%
```

(After conversion)
```
O0001
G0 X100. ;
G0 X200. ;
G0 X300. ;
M30;
%
```

(Example 2)

(Before conversion)
```
O0001
G1128I1.R0.8A95.B80.J3.P3.L3.M0.F0.5X1.Y1.Z10. ;
M98 P0002;
M30;
%
```

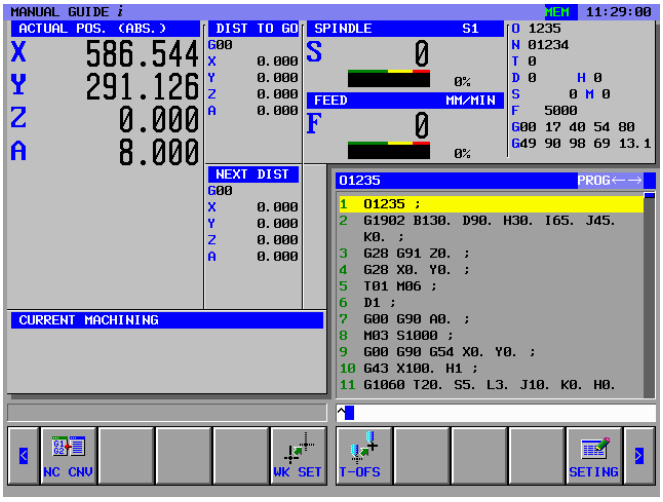
→

```
O0002
G1450H0.V75. ;
G1451H0.V0.K7.D0.L0.M0.T1. ;
G1451H5.V0.K1.C5.L0.M0.T1. ;
G1451H5.V75.K3.D75.L0.M0.T2. ;
G1456;
M99;
%
```

(After conversion)
```
O0001
(NC PROGRAM CONVERSION-START);
G01X963Z1616;
G01X896 Z1654;
 :
(NC PROGRAM CONVERSION-END);
M30;
%
```

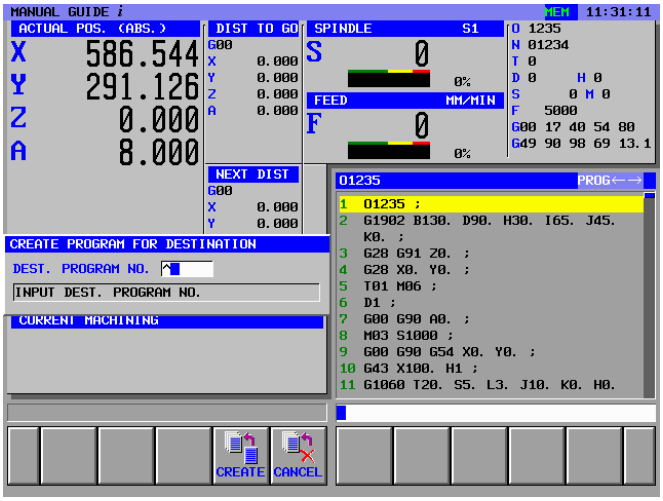
12.2 OPERATING THE NC PROGRAM CONVERSION FUNCTION

Selecting MEM mode on the machine operator's panel, and pressing the leftmost soft key [<] or rightmost soft key [>] several times displays the soft-keys including [NC CNV]



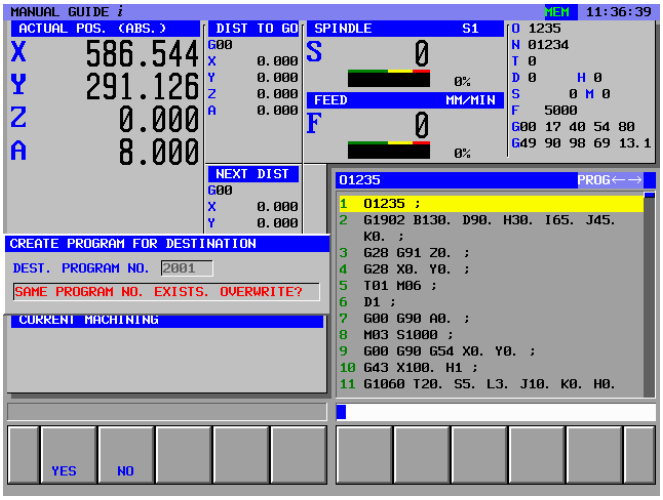
NC CNV				WK SET		T-OFS				SETING

<1> The [CREATE PROGRAM FOR DESTINATION] screen appears. Enter the number of a conversion destination program, then press the [CREATE] soft key.

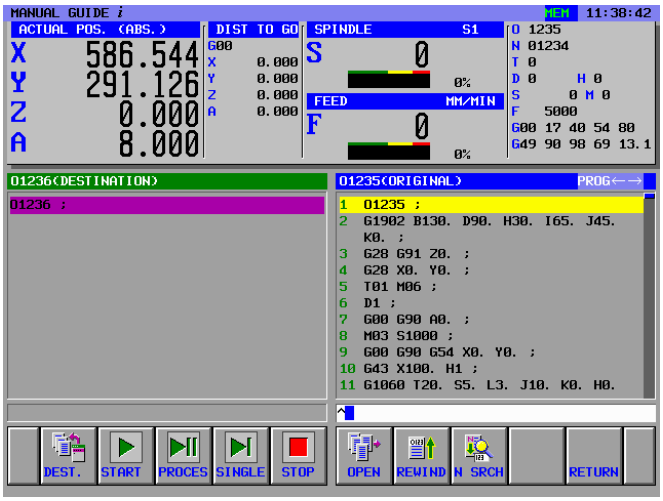


			CREATE	CANCEL					

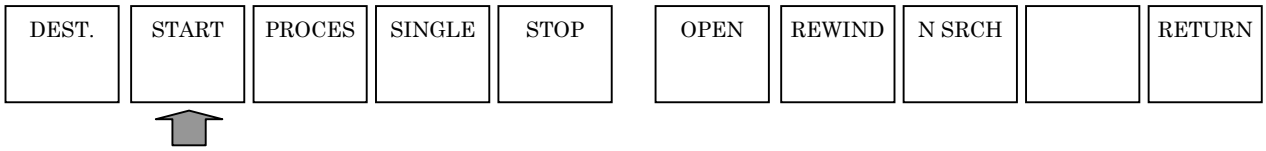
<2> If the program already exists, a message for checking if the program may be overwritten is displayed. If the program may be overwritten, press [YES]. If you select [NO], the screen goes back to the memory program screen, so press [NC CNV] again and enter other program number.



<3> Press the [CREATE] soft key. A new program with the entered number is created. The NC program conversion function screen shown below appears to display a conversion source program window and a post-conversion program window at the same time. Pressing [CANCEL] switches the screen display from the NC program conversion function screen to the program screen.



<4> The following soft keys appear on the NC program conversion function screen. Press [START] to start NC program conversion.

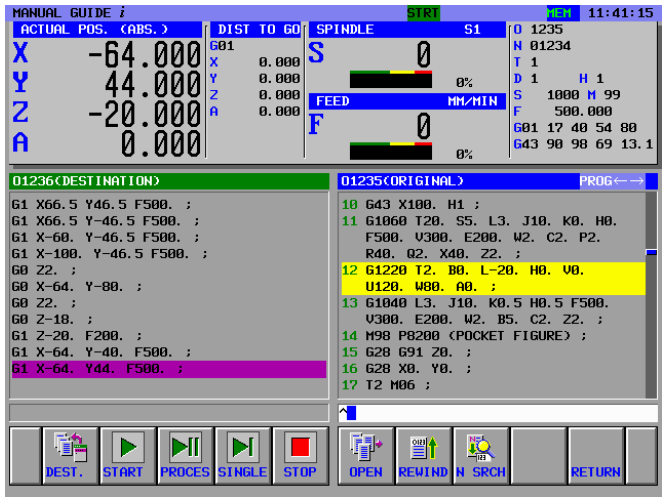


[DEST.] : Creates a new conversion destination program.
[START] : Starts NC program conversion execution.
[PAUSE] : Stops NC program conversion execution temporarily.
[SINGLE] : Executes NC program conversion block by block.
[STOP] : Stops NC program conversion execution.
[OPEN] : Selects a conversion source program from the program list screen.
[REWIND] : Searches for a conversion source program.
[N SRCH] : Makes an N search for a conversion source program.
[CHPATH] : Displayed only when the multi-path option is specified. This soft key switches between paths.
[RETURN] : Ends the NC program conversion function and returns the screen display to the ordinary MEM mode screen.

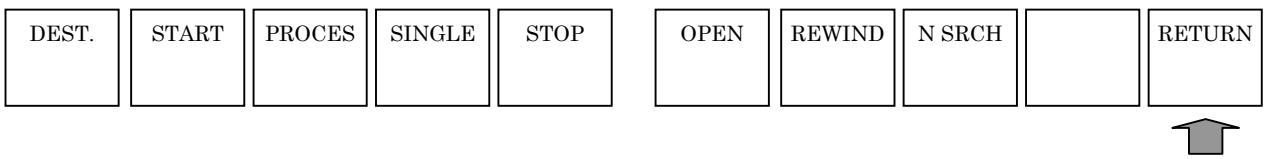
* When bit 0 of parameter No. 27310 = 1, the [PAUSE] soft key is replaced with the [PROCES] soft key.

[PROCES] : Executes NC program conversion process by process.

<5> During executing NC program conversion, the converted part program blocks are displayed in the destination window.



<6> Confirm that the NC program conversion ends, then press [RETURN].



12.3 RESTRICTIONS

(1) The NC program conversion function cannot be used during background editing.

(2) Blocks containing the following words are not output to the conversion destination program:
- M98
- M99
- Custom macro conditional branch program
  <1> GOTO
  <2> IF
  <3> THEN
  <4> WHILE
  <5> END
- Custom macro variable assignment program
  #?=~
- Custom macro external output command program
  <1> POPEN
  <2> PCLOS
  <3> BPRNT[~]
  <4> DPRNT[~]

(3) If an alarm is issued during NC program conversion, the results of conversion at that time are output to the conversion destination program.

(4) During NC program conversion, the following soft keys cannot be used:
[DEST.], [START], [OPEN], [REWIND], [SRCH], [CHPATH], [RETURN]

(5) If an execution mode other than MEM mode is set during NC program conversion, the execution of NC program conversion is terminated forcibly.

(6) During NC program conversion, the screen display cannot be switched to the NC screen.

(7) If only one block is inserted between WHILE programs, blocks as many as the loop count are not output but only one block is output.
(Before conversion)
WHILE [#1 EQ #2]DO1;
G0 X0. ;
END1 ;
(After conversion) Only one block is output even if three loop operations are performed.
G0 X0. ;

(8) The NC program conversion function is designed to expand a 4-digit G cycle machining code. So, NC program conversion is not performed as expected in cases other than the cases indicated below.

(Example 1) When both the machining command and figure command of a cycle are present on the main program

```
O0001
G1128I1.R0.8A95.B80.J3.P3.L3.M0.F0.5X1.Y1.Z10. ;
G1450H0.V75. ;
G1451H0.V0.K7.D0.L0.M0.T1. ;
G1451H5.V0.K1.C5.L0.M0.T1. ;
G1451H5.V75.K3.D75.L0.M0.T2. ;
G1456;
M30;
%
```

(Example 2) When the figure command of a cycle is present on a subprogram

```
O0001
G1128I1.R0.8A95.B80.J3.P3.L3.M0.F0.5X1.Y1.Z10. ;
M98 P0002;          →   O0002
M30;                    G1450H0.V75. ;
%                       G1451H0.V0.K7.D0.L0.M0.T1. ;
                        G1451H5.V0.K1.C5.L0.M0.T1. ;
                        G1451H5.V75.K3.D75.L0.M0.T2. ;
                        G1456;
                        M99;
                        %
```

* It is assumed that the subprogram includes only the figure command of a cycle.

(9) Executed program blocks are the target of NC statement conversion.
(10) Feedrate command is converted with decimal point
(11) When custom macro block exit before executing stop command (M00, M01, M02, M30, M99), executing stop command (M00, M01, M02, M30, M99) is not converted.
(12) When M98 block exit before executing stop command (M00, M01, M02, M30, M99), executing stop command (M00, M01, M02, M30, M99) is not converted.
(13) A block including M198 is not converted.

13 TOOL DATA BASE FUNCTION

13.1 SETTING OF TOOL OFFSET DATA

For a compound machine, the following data items are displayed for the T mode:

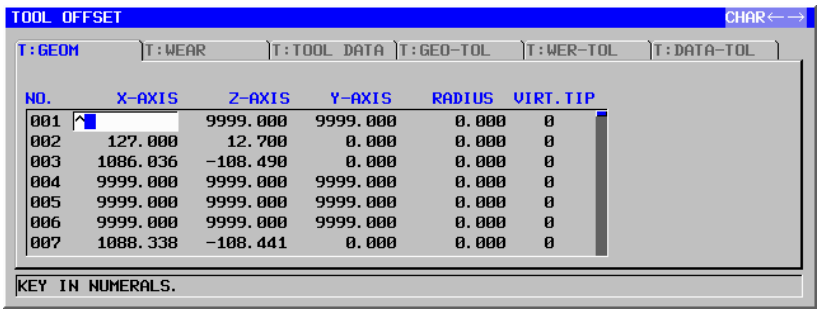
(1) T : GEOMETRY OFFSET
(2) T : WEAR OFFSET
(3) T : TOOL DATA
(4) T : GEOMETORY TOOL TYPE OFFSET
(5) T : GEOMETORY WEAR TYPE OFFSET
(6) T : GEOMETORY DATA TYPE OFFSET

The following data items are displayed for the M mode:

(7) M : TOOL OFFSET (TOOL LENGTH COMP. / CUTTER COMPENSATION)
(8) M : TOOL DATA
(9) M : TOOL TYPE OFFSET (TOOL LENGTH COMP. / CUTTER COMPENSATION)
(10) M : TOOL TYPE DATA

Each mode can be selected with [M←→T] soft key.

For lathes, data items (1) to (6) are displayed. For machining centers, data items (7) to (10) are displayed.



The data to be set and displayed in (1), (2) and (7) is common to the corresponding data in CNC. For details, refer to the operator's manual of the CNC.
And for lathes, inputting tip radius value of milling tools in "radius" is necessary. If not, sometimes alarm is given.
For data items (4), (5) and (9), which are related to the tool management function, see the description of "Tool Management Function" in Appendix.
For data items (3), (6), (8) and (10), which is about tool form data, see next section.

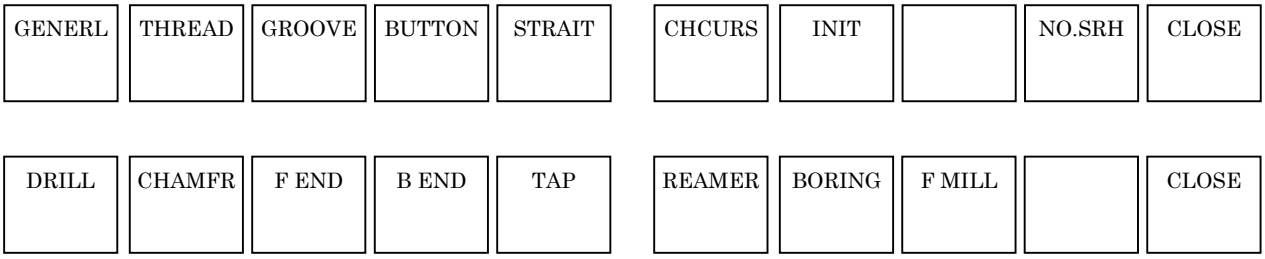
13.2 SETTING OF TOOL DATA

By selecting “tool data” tab in tool offset window, “tool data” setting window is displayed. Tool data is the data that is necessary for executing animation or cycle, and their items are tool radius, kind of tool, name, setting and tool form data. Of all items, tool radius is inputted into radius (for lathes) or cutter radius compensation (for milling) in tool offset table. The rest is inputted in “TOOL DATA” tab.
These data are reserved in SRAM, so once set, they aren’t deleted though the power supply is cut. But more than 300 tools can’t be inputted.
Besides, by parameter No.14850#0, you can decide whether “tool data” tab is displayed or not.

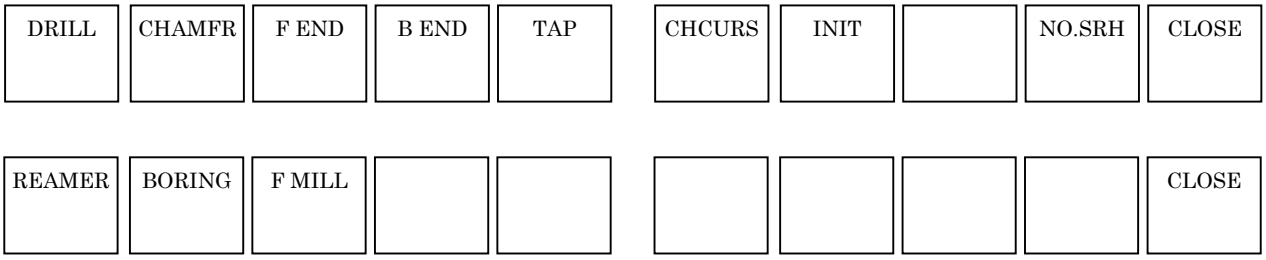
13.2.1 Setting of Tool Type

By placing cursor on an item for selecting kind of tool, the following soft keys appear. When appropriate soft key is pushed, a type of tool is selected and its icon is displayed. Also tool name is displayed on the right of the icon.

Soft keys for selecting kind of tool for lathes or T mode of combined machine



Soft keys for selecting kind of tool for machining centers or M mode of combined machine



13.2.2 Editing of Tool Name

To edit tool name, place cursor on tool name, change mode into character, input alphabets or numerals, and push **INPUT**.
This function is useful to distinguish similar tools.

13.2.3 Setting of Tool Set

When a cursor is placed on tool setting number, a guidance window is automatically displayed in the right of the screen. By inputting tool setting number and pushing **INPUT**, tool setting can be set.

13.2.4 Entering of Tool Data

By placing cursor on items of tool data, guidance window is automatically displayed. By inputting tool data value and pushing **INPUT**, tool data can be set. Then three-whole and one-decimal number can be inputted when unit is [deg.]. When [inch] or [mm], eight-figure number can be inputted. But numbers that is inputted to 7 decimal places are rounded off to 6 decimal places.
The names of item and the numbers of item depend on kind of tool. See below about the detail. In this table, tools that don't have to set tool data are omitted.
Tool data isn't necessary to execute milling cycle. So milling cycle can be executed though tool data isn't set.

Tool form data for turning

KIND OF TOOL	GENERAL	THREAD	GROOVE
Data1	Cutting edge angle	Nose angle	Tip width
Data2	Nose angle		Tip length (*)

KIND OF TOOL	BUTTON	STRAIGHT
Data1	Tip length (*)	Cutting edge angle
Data2		Nose angle

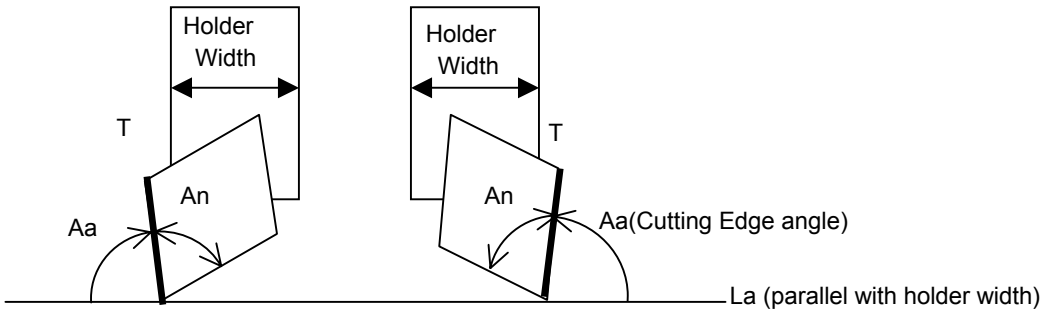
Tool form data for milling

KIND OF TOOL	DRILL	CHAMFER
Data	Nose angle (*)	Cutter diameter (*)

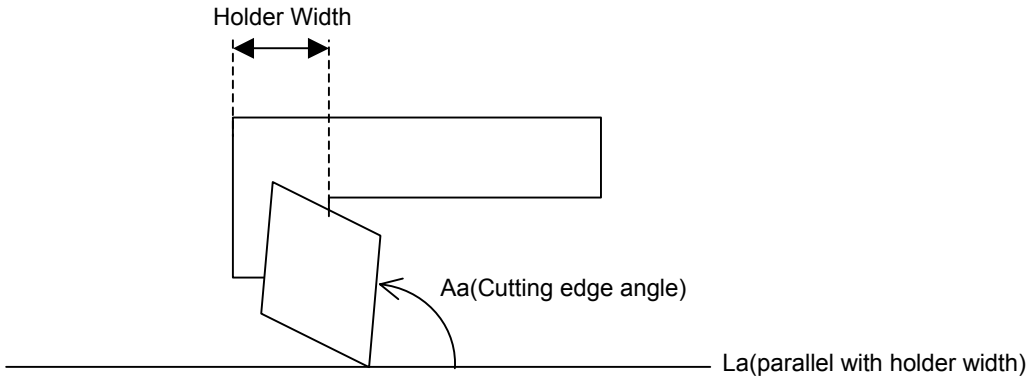
(*) : Cycle can be executed if not set

13.2.5 Cutting Edge angle of Tool Data Base Function

An angle that is made by a line parallel with holder width and cutting edge is defined as a cutting edge angle.

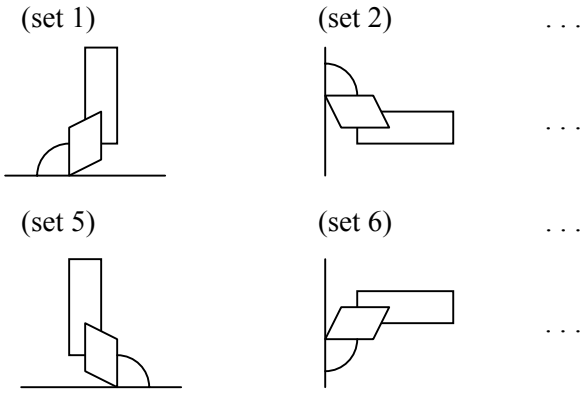


1. Draw a straight line that is parallel with holder width
2. An angle that is made by a straight line La and a straight line T is defined as a cutting edge angle

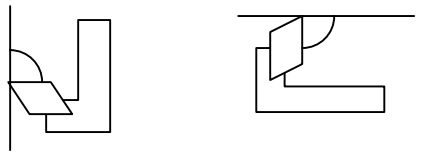


3. In case of L form holder, the width of a part that is installed tip is defined as holder width.

Actually MGi defines cutting edge angle from tool setting.
Show following concrete examples.

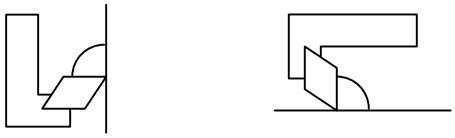


(set 9) (set 10) . . .



. . .

(set 13) (set 14) . . .



. . .

13.2.6 Initializing of Tool Data

Tool data can be initialized by [INIT] soft key. When [INIT] is pushed, a message for confirming initialization is displayed. By pressing [YES], initialization is performed.
But items to be initialized are only items placed in tool data tab and [INIT] has no connection with offset value.

13.3 SELECTING TOOL DATA AT PROGRAM ENTERING

To select dada number that is set in “TOOL DATA” tab, T code or D code is used to work machines.
For lathes, T code is used to specify numbers of three types, tool number, geometry tool offset number and wear tool offset number. In these numbers, the geometry offset number will be used for actual machine operations. There are some ways to specify geometry tool offset, for example setting parameter No.5002#1. But in all case, the geometry offset number will be used. With tool management function, D code is used to select offset number. But in this case, the geometry tool offset number will be used too.
For machining centers, the number of the cutter compensation that is specified by D code will be used for actual machine operation.
For compound machines, when T mode, the way to appoint tool data number is the same as lathes. So the same number of geometry tool offset and runs machine.
When M mode, the way to appoint tool data number is the same as machining centers. So the number of cutter compensation for actual machine operation.
In animated simulation, when tool data number is selected by T code or D code, a tool is drawn.

NOTE
There are 2 ways to draw tools. One is the way to select tool data number above. The other is the way to use G code. Place T (or D) code and G code 2 blocks apart when G code follows T (or D code). If T (or D) code and G code aren’t placed 2 blocks apart, order of G code sometimes doesn’t work correctly.

The relation between the specified tool kind and machining cycle will be checked. For example, if you try to use chamfer tool in drill cycle, some corresponding alarm will occur. But when kind of tool isn’t inputted, this checking will not done.

13.4 SETTING OF TOOL GRAPHIC DATA

Several items are needed to execute machine simulation in addition to items that explained up to here. These items are called Graphic Data. Graphic Data is showed below.

13.4.1 Tool Graphic Data

Tool graphic data are defined by parameters, from No.27350 to No.27383. When these parameters aren't defined, appropriate value will be inserted automatically. For details, refer to the description of "Parameter" in APPENDIX.

Tool graphic data for turning tools

Kind of tool	General	Threading	Grooving
Data 1	Tip position	Tip position	Tip position
Data 2	Tip length	Tip width	Holder length
Data 3	Holder length	Holder length	Holder width
Data 4	Holder width	Holder width	
Data 5	Holder length 2		
Data 6	Holder width 2		

Kind of tool	Button turning	Straight
Data 1	Tip position	Tip position
Data 2	Holder length	Tip length
Data 3	Holder width	Holder length
Data 4		Holder width
Data 5		Holder length2
Data 6		Holder width 2

Tool graphic data for milling tools

Kind of tool	Drill	Chamfer	Flat endmill	Ball endmill
Data 1	Tip length	Tip length	Tip length	Tip length
Data 2		Cutter length		
Data 3		Shank length		
Data 4		Shank diameter		

Kind of tool	Tap	Reamer	Boring	Face mill
Data 1	Tip length	Tip length	Tip length	Tip length

13.5 ACCESSING TOOL DATA BASE FUNCTION

Accessing tool data base function is the function that tool data registered in Manual Guide *i* are read or written from custom macro. So, it is possible that tool data are accessed from a program. And restoring the initial or copying tool data can be available.

13.5.1 Basic Specifications

The following data can be read and written.

1. Kind of tool
2. Setting
3. Tool data 1 (ex. cutting edge angle)
4. Tool data 2 (ex. nose angle)

NOTE

1 Tool name can't be read and written.
2 The option of custom macro B is necessary to use this function.
3 Only from custom macro or execution macro, tool data can be read and written.
4 To enable this function, bit 6 of parameter No. 14852 must be set to 1.

In addition to reading and writing, the following functions are available.

1. Copying tool data
   All tool data, including tool name, can be copied if source offset number and destination offset number are designated.
2. Initialization of tool data
   Tool data per tool or all tool data can be restored the initial.

13.5.2 System Variables

Tool data can be inputted or outputted from custom macro through #5750 - #5756 system variables. Input adequate value to the system variables when you want to access to tool data.
And when Manual Guide *i* find that adequate value have been set to it, tool data is outputted or inputted.

The following system variables whose numbers are used for the purpose of accessing tool data.

#5750 : The variable that is used to decide the working of accessing tool data base function. Tool data will be inputted or outputted from system variables when Manual Guide *i* have detected the value of #5750. The meanings are following.
- 0 : Do nothing
- 1 : Read tool data
- 2 : Write tool data
- 3 : Copy tool data from source to destination
- 4 : Restore the init tool data of designated offset number
- 5 : Restore the init of all tool data

#5751 : Result
The meanings are following.
- 0 : Idle
- 1 : Normal finish
- 2 : Registered tool data number exceeds maximum
- 3 : Designating the working is wrong
- 4 : Designating offset number is wrong
- 5 : Designating kind of tool is wrong (only when writing)
- 6 : Designating setting is wrong (only when writing)
- 7 : Designating data 1 is wrong (only when writing)
- 8 : Designating data 2 is wrong (only when writing)

#5752 : Designating offset number to be read or written. And in case of copying tool data, Designating offset number of the destination.
If wrong value is inputted, 4 will be returned to result in reading or writing.

#5753 : Gotten kind of tool in reading or designated kind of tool in writing. And in case of copying tool data, Designating offset number of the source.
If wrong value is inputted, 5 will be returned to result in writing.
10 : General tool
11 : Threading tool
12 : Grooving tool
13 : Button turning tool
14 : Straight tool
20 : Drill tool
21 : Chamfering tool
22 : Flat end mill tool
23 : Ball end mill tool
24 : Tap tool
25 : Reamer tool
26 : Boring tool
27 : Face mill tool

#5754 : Gotten setting of tool in reading and designated setting of tool in writing. If wrong value is inputted, 6 will be returned to result in writing.

#5755 : Gotten data 1 in reading and designated data 1 in writing. If wrong value is inputted, 7 will be returned to result in writing.

#5756 : Gotten data 2 in reading and designated data 2 in writing. If wrong value is inputted, 8 will be returned to result in writing.

13.5.3 Reading

In reading tool data, set offset number of tool data that should be gotten to #5752 and set 1 to #5750. Each data will be outputted to #5753, #5754, #5755 and #5756.

13.5.4 Writing

In writing tool data, set offset number of tool data that should be written to #5752. And set tool data to #5753, #5754, #5755 and #5756. Finally, set 2 to #5750. Each data will be reserved to memory field of Manual Guide *i* .

13.5.5 Copying

In copying, set offset number of destination to #5752, set offset number of source to #5753 and set 3 to #5750. Manual Guide *i* will copy tool data of source to destination.

13.5.6 Initialization

In initialization, set offset number of tool data that should be restored to the init to #5752 and set 4 to #5750. This tool data of designated offset number will be restored to the init.

13.5.7 Initialization of All Tool Data

In initialization of all tool data, set 5 to #5750. All tool data will be restored to the init. But in case of multi path system, the data that will be restored to the init are only the data of the path that custom macro executes.
And in case of compound machines, the data that will be restored to the init are only the data of the mode that custom macro executes (milling mode or turning mode).

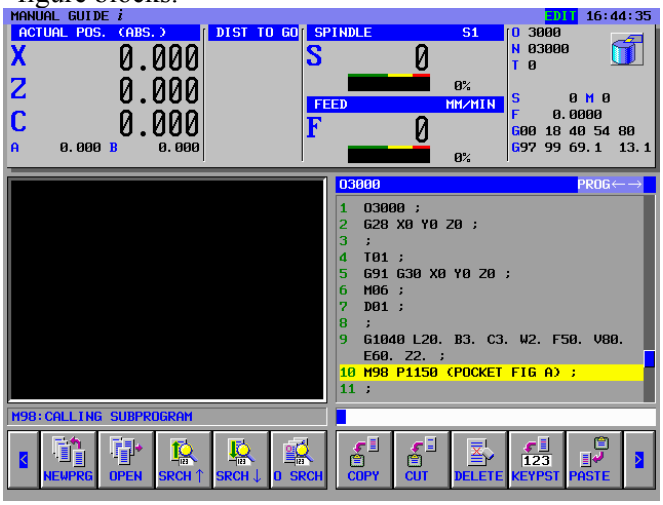
14 EDITING OF FREE FIGURE AND FIXED FORM FIGURE OF SUBPROGRAM FORM

On the program editing screen, after moving the cursor on the sub program call command (M98 P****) which is composed by the free figure blocks or just one fixed form figure block, pressing the [INPUT] key displays the window for editing the figures to edit directly.

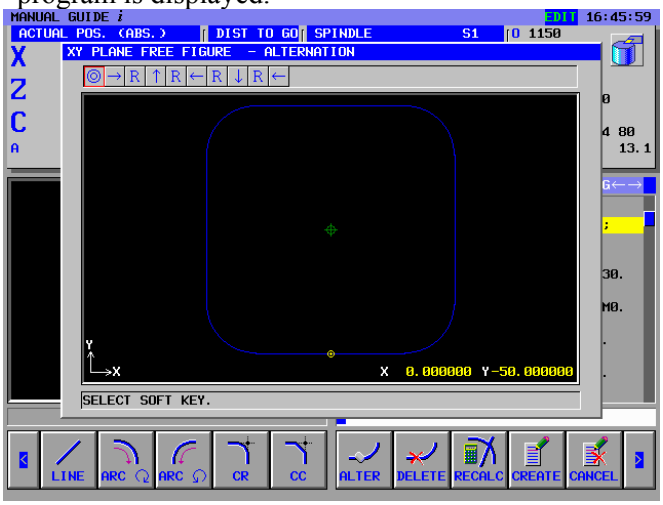
14.1 EDITING A FREE FORM FIGURE SUBPROGRAM

The operations are as follows.

(1) On the program editing screen, press the [INPUT] key or the [ALTER] soft key after moving the cursor on the sub program call command (M98 P****) which is composed by the free figure blocks.

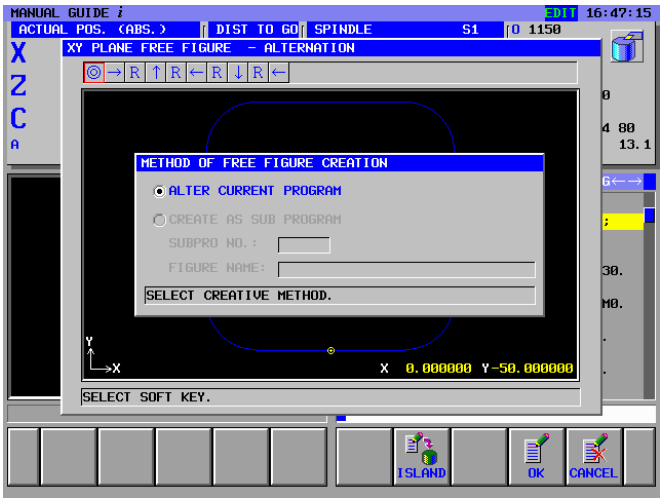


(2) The following window for editing the free figures included sub program is displayed.



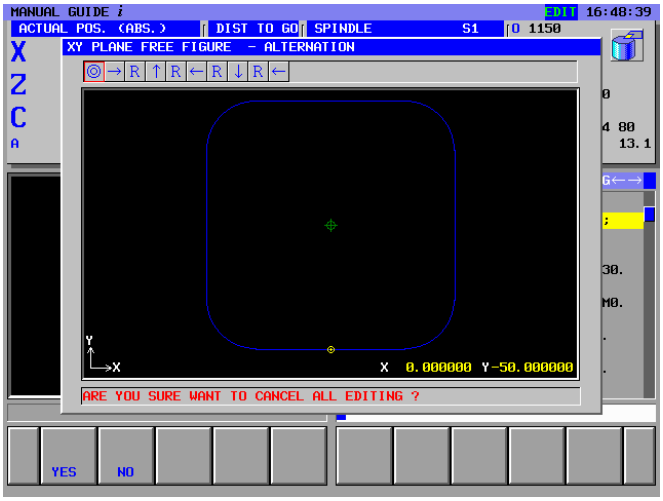
(3) On the window to finish editing the free figures, the operations are as follows.

- Pressing the [OK] soft key alters the existing blocks into the editing figures in the machining program and returns to the main program displaying screen.
  But, at the case that the another free figures exist after the editing figures, for example the island figures exist after the pocket outer free figures, the editing window of the next free figures is displayed.
- At the case of editing the free figures for pocketing, pressing the [ISLAND] soft key displays the window for editing the free figures (“START POINT”) to make the new island figures after altering the editing figures.



NOTE
On the editing sub program window, the free figures cannot be output into the another sub program.

(4) On the window of editing the free figures, pressing the [CANCEL] soft key displays a window for the confirmation to interrupt editing. From this window, pressing the [YES] soft key cancels the editing operations and returns to the main program displaying screen.



NOTE

1 During editing the subprogram of free figure, displaying free figure editing window, this editing mode is canceled and the editing window is closed by the following operations. The selected program is kept to the edited subprogram.
   - Mode is changed to other than EDIT mode when foreground editing is done.
   - Change to CNC screen.
   - Power turned off.

2 In this case, the free figure editing window is closed. The program that is then displayed in the program display area is a subprogram.

14.2 WARNING MESSAGE

The following warning messages are displayed at the editing of the subprogram.

- "SUB PROGRAM IS NOT FOUND"
  When the [INPUT] key or the [ALTER] soft key is pressed after moving the cursor on the sub program call command, this warning message is displayed at the case that the program number specified by address 'P' doesn't exists.
- "PROGRAM IS PROTECTED."
  When the [INPUT] key or the [ALTER] soft key is pressed after moving the cursor on the sub program call command, this warning message is displayed at the case that the program number specified by address 'P' is protected.
- "PROGRAM IS NOT FREE FIGURE"
  When the [INPUT] key or the [ALTER] soft key is pressed after moving the cursor on the sub program call command, this warning message is displayed at the case that the program number specified by address 'P' doesn't include the free figure blocks.
- "PROGRAM IS SELECTED FOREGROUND
  When the screen changes to the main program displaying screen after the [OK] soft key or the [CANCEL] soft key is pressed at background editing, this warning message is displayed at the case that the main program is already selected at foreground.

14.3 EDITING A FIXED FORM FIGURE SUBPROGRAM

When bit 1 of parameter No. 14851 is set to 1, a fixed form figure in subprogram form can be edited directly by placing the cursor on the subprogram call command of the main program then pressing [ALTER].

The procedure is as follows:

<1> Place the cursor on the block of a subprogram call command (M98P****) of the main program, then press the [ALTER] soft key or the [INPUT] key.
<2> The window for input of fixed form figure data on the subprogram is displayed. Enter data, then press the [ALTER] soft key.
<3> The modified data is output to the machining program, and the fixed form figure data input window is closed. The main program is then displayed again.

NOTE
When the cursor is placed on a subprogram call command of the main program, fixed form figure drawing is not performed on the graphic window.

15 SHORTCUT KEY OPERATIONS

On MANUAL GUIDE *i*, almost all the operations excepting numerical data entering are done by soft-keys. However, if you are well experienced in those operations, you can operate more quickly by using other key instead of the assigned soft-key. This other key operation is called shortcut key.

Pressing HELP key on the MDI panel displays the window in which explanations of shortcut key are displayed. Into details, refer to II 16. HELP SCREEN.

NOTE
If the small MDI key board is attached to the CNC, shortcut key operations cannot be used.

15.1 SHORTCUTS FOR VARIOUS CONFIRMATION OPERATIONS

Soft key	Shortcut key
[YES]	[INPUT]
[NO]	[CAN]

15.2 SHORTCUTS FOR RANGE SELECTION

Soft key	Shortcut key
[SELECT]	[INPUT]
[CANCEL]	[CAN]

15.3 SHORTCUTS FOR COPY OPERATION

Soft key	Shortcut key
[COPY]	[INPUT]
[CANCEL]	[CAN]

15.4 SHORTCUTS FOR CUT OPERATION

Soft key	Shortcut key
[CUT]	[INPUT]
[CANCEL]	[CAN]

15.5 SHORTCUTS FOR THE BASE SCREEN SOFT KEYS

Soft key	Shortcut key
[<]	[-]+[INPUT]
[SF1]	[1]+[INPUT]
[SF2]	[2]+[INPUT]
[SF3]	[3]+[INPUT]
[SF4]	[4]+[INPUT]
[SF5]	[5]+[INPUT]
[SF6]	[6]+[INPUT]
[SF7]	[7]+[INPUT]
[SF8]	[8]+[INPUT]
[SF9]	[9]+[INPUT]
[SF10]	[0]+[INPUT]
[>]	[.]+[INPUT]

* By setting bit 1 of parameter No. 14703 to 1, a number for shortcut operation can be displayed under each soft key.

15.6 SHORTCUT FOR STARTING THE CYCLE CHANGE SCREEN

Soft key	Shortcut key
[ALTER]	[INPUT]

15.7 SHORTCUTS FOR THE MENU SELECTION SCREEN

Soft key	Shortcut key
[SELECT]	[INPUT] or numeral +[INPUT]
[CANCEL]	[CAN]

15.8 SHORTCUTS FOR THE REGULAR PROGRAM INSERTION SCREEN

Soft key	Shortcut key
[INSERT]	[INPUT] or numeral +[INPUT]
[CLOSE]	[CAN]

15.9 SHORTCUTS FOR THE M CODE INSERTION SCREEN

Soft key	Shortcut key
[INSERT]	[INPUT]
[CLOSE]	[CAN]

15.10 SHORTCUTS FOR THE PROGRAM LIST SCREEN

Soft key	Shortcut key
[OPEN]	[INPUT] or [9]
[CLOSE]	[CAN] or [0]
[DELETE]	[DELETE] or [3]
[EDTCOM]	[ALTER] or [4]
[NEW]	[1]
[COPY]	[2]
[SEARCH]	[5]
[M CARD]	[6]
[ALLDEL]	[7]
[SRTORD]	[8]

15.11 SHORTCUTS FOR THE PROGRAM CREATION SCREEN

Soft key	Shortcut key
[CREATE]	[INPUT]
[CANCEL]	[EOB]

15.12 SHORTCUTS FOR THE COMMENT EDITING SCREEN

Soft key	Shortcut key
[CREATE]	[INPUT]
[CANCEL]	[EOB]

15.13 SHORTCUTS FOR THE SEARCH SCREEN

Soft key	Shortcut key
[SEARCH]	[INPUT]
[CANCEL]	[EOB]

15.14 SHORTCUTS FOR THE CYCLE INPUT SCREEN

Soft key	Shortcut key
[INSERT]	[INSERT]
[ALTER]	[ALTER]
[CANCEL]	[EOB]
[SF1]	[1]
[SF2]	[2]
[SF3]	[3]
[SF4]	[4]
[SF5]	[5]
[SF6]	[6]
[SF7]	[7]
[SF8]	[8]
[SF9]	[9]
[SF0]	[0]
[>]	[.]+[INPUT]

15.15 SHORTCUT FOR THE WORKPIECE COORDINATE SYSTEM SETTING SCREEN

Soft key	Shortcut key
[CLOSE]	[EOB]

15.16 SHORTCUT FOR THE TOOL OFFSET SETTING SCREEN

Soft key	Shortcut key
[CLOSE]	[EOB]

15.17 SHORTCUTS FOR THE REGULAR PROGRAM REGISTRATION SCREEN

Soft key	Shortcut key
[CLOSE]	[INPUT]
[NEW]	[INSERT]
[DELETE]	[DELETE]
[ALTER]	[ALTER]

15.18 SHORTCUTS FOR THE CREATION SCREEN FOR REGULAR PROGRAM REGISTRATION

Soft key	Shortcut key
[INSERT]	[INSERT]
[ADD]	[ALTER]
[CANCEL]	[EOB]

15.19 SHORTCUTS FOR THE ALTER SCREEN FOR REGULAR PROGRAM REGISTRATION

Soft key	Shortcut key
[ALTER]	[ALTER]
[CANCEL]	[EOB]

15.20 SHORTCUTS FOR THE PRESET SCREEN

Soft key	Shortcut key
[ALTER]	[ALTER]
[CANCEL]	[EOB]

15.21 SHORTCUT FEEDRATE THE MEASUREMENT RESULT SCREEN

Soft key	Shortcut key
[CLOSE]	[CAN]

15.22 SHORTCUT FOR THE MANUAL MEASUREMENT SCREEN

Soft key	Shortcut key
[CLOSE]	[EOB]

15.23 SHORTCUT FOR VARIOUS SETTING SCREENS

Soft key	Shortcut key
[CLOSE]	[EOB]

15.24 SHORTCUTS FOR THE FREE FIGURE MAIN SCREEN

Soft key	Shortcut key
[CREATE]	[INPUT]
[CANCEL]	[CAN]
[DELETE]	[DELETE]
[ALTER]	[ALTER]
[<]	[-]
[SF1]	[1]
[SF2]	[2]
[SF3]	[3]
[SF4]	[4]
[SF5]	[5]
[SF6]	[6]
[SF7]	[7]
[SF8]	[8]
[SF9]	[9]
[SF10]	[0]
[>]	[.]

15.25 SHORTCUTS FOR THE FREE FIGURE INPUT SCREEN

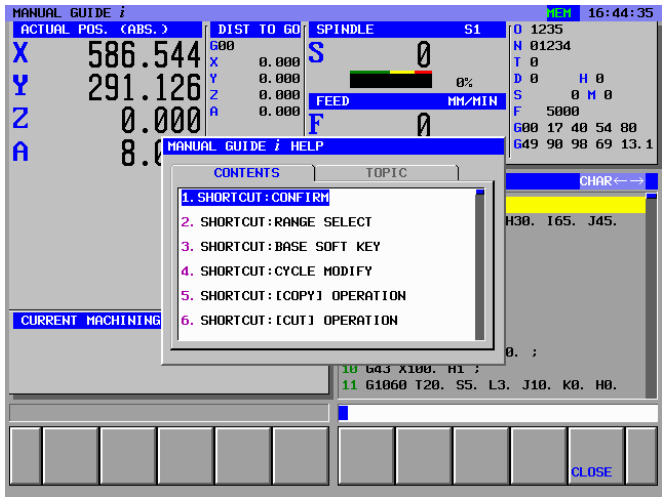
Soft key	Shortcut key
[OK]	[INSERT]
[CANCEL]	[EOB]

15.26 SHORTCUTS FOR THE FREE FIGURE CREATION SCREEN

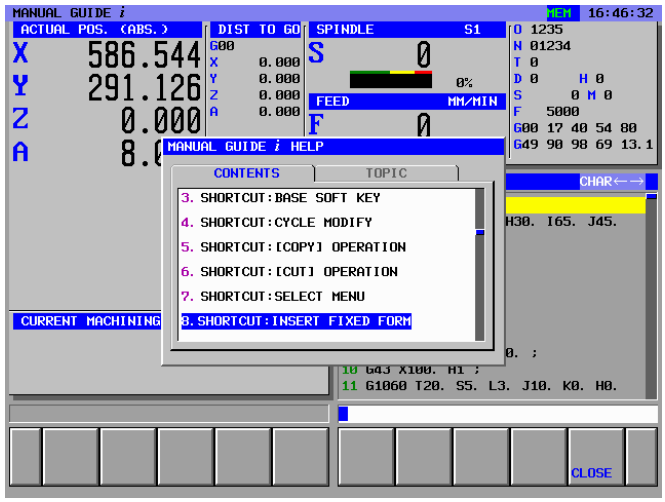
Soft key	Shortcut key
[OK]	[INSERT]
[CANCEL]	[EOB]

16 HELP SCREEN

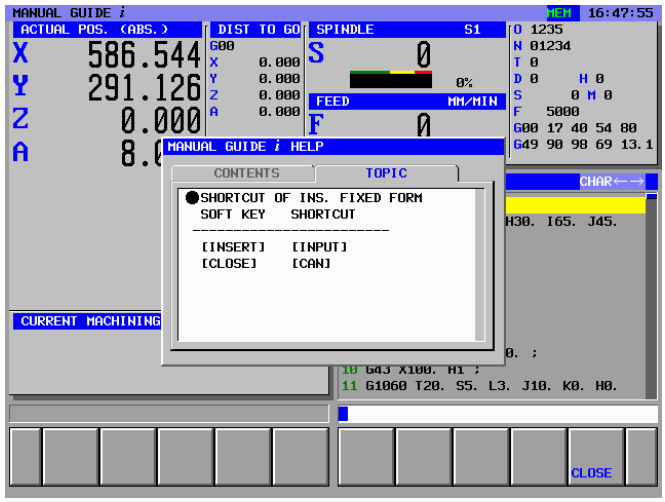
Pressing the [HELP] key on the MDI keyboard displays the HELP window, in which explanations for shortcut key operation are displayed.
In the window, “CONTENTS” and “TOPIC” tabs are displayed.



Moving the cursor by ↑ or ↓, place the cursor to the item of shortcut key to display the explanation.



Pressing the cursor key → displays the tab “TOPIC” tab and explanation of the selected shortcut key.



Pressing the cursor key ← returns to “CONTENTS” tab.

Pressing [CLOSE] closes the HELP window.

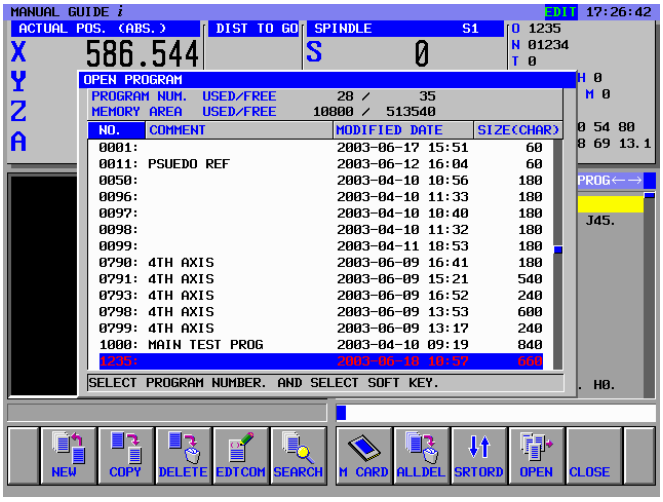
17 MEMORY CARD INPUT/OUTPUT FUNCTION

17.1 MEMORY CARD INPUT/OUTPUT OF PART PROGRAM

17.1.1 Memory Card Input/Output Screen of Part Program

NEWPRG	O LIST	SRCH↑	SRCH↓	O SRCH		COPY	CUT	DELETE	KEYPST	PASTE

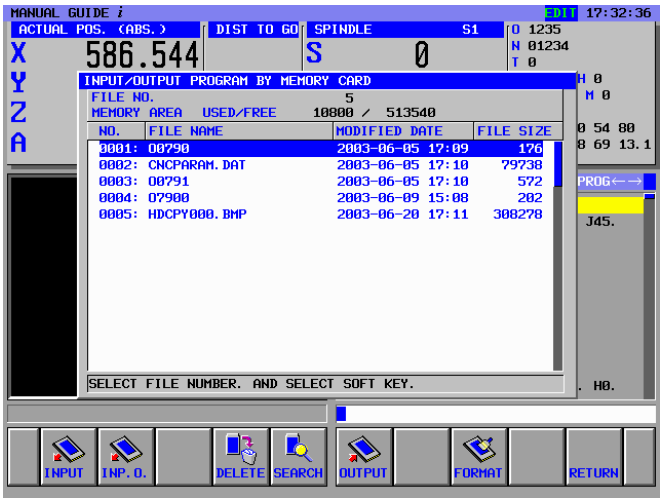
Select EDIT mode on the machine operator's panel. Pressing [O LIST] displays the program list window, programs registered in the CNC.



Following soft-keys are displayed.

NEW	COPY	DELETE	EDTCOM	SEARCH		M CARD	ALL DEL	SRTORD	OPEN	CLOSE

Pressing the [M CARD] soft key on the program list screen displays the INPUT/OUTPUT PROGRAM BY MEMORY CARD]screen.



Following soft-keys are displayed.

INPUT	INP.O		DELETE	SEARCH		OUTPUT		FORMAT		RETURN

[INPUT] : Inputs a program from the memory card.
[INP.O] : Inputs a program from the memory card (by changing the O number).
[DELETE] : Deletes files on the memory card.
[SEARCH] : Searches for a file on the memory card.
[OUTPUT] : Displays the screen for output to the memory card.
[FORMAT] : Formats the memory card.
[RETURN] : Returns the screen display to the program list screen.

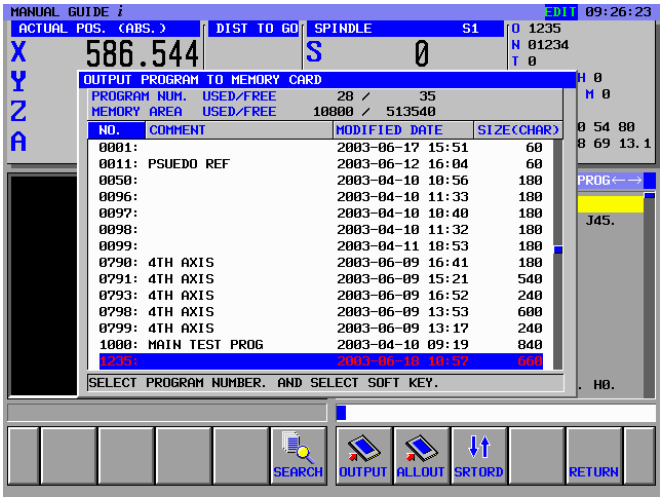
In order to delete a file in the memory card, place the cursor on the file name in the above window and press [DELETE]. This soft key displays a message for checking if selected file may be deleted. Pressing [YES] deletes the file in the memory card. Pressing [NO] cancels the deletion of all programs.

YES	NO									

In case of initializing the memory card such as deleting all files in the memory card, press [FORMAT]. This soft key displays a message for checking if the memory card may be initialized. Pressing [YES] initializes the memory card and all files in the memory card are deleted.. Pressing [NO] cancels the deletion of all programs.

17.1.2 Memory Card Output Operation for Part Program

Pressing the [OUTPUT] soft key on the INPUT/OUTPUT PROGRAM BY MEMORY CARD screen displays the OUTPUT PROGRAM TO MEMORY CARD screen.

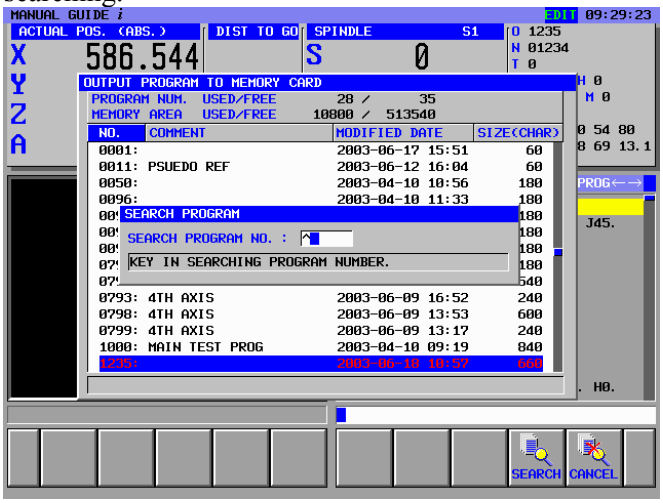


The following soft-keys are displayed.

				SEARCH		OUTPUT	ALLOUT	SRTORD		RETURN

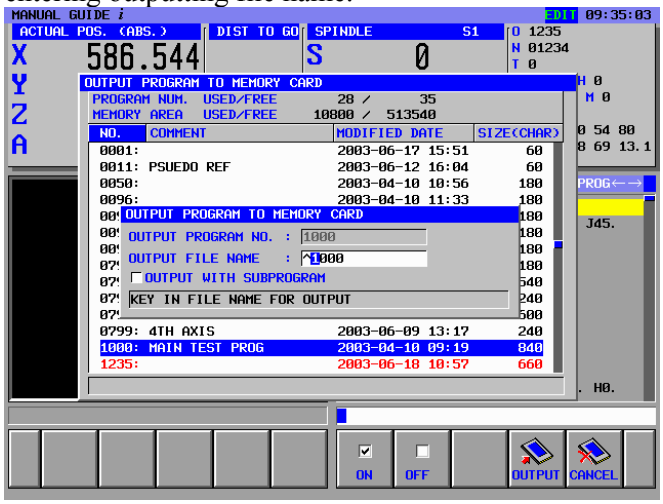
[SEARCH] : Searches for a program.
[OUTPUT] : Output a selected program to the memory card.
[ALLOUT] : Output all programs to the memory card.
[SRTORD] : Switches the sort order for displaying a program list between ascending order and descending order.
[RETURN] : Returns the screen display to the [INPUT/OUTPUT PROGRAM BY MEMORY CARD] screen.

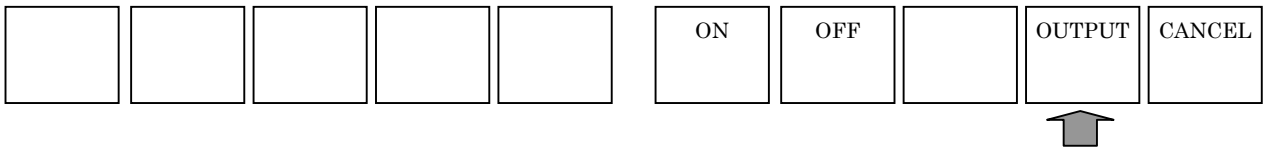
Pressing [SEARCH] displays the following window of program searching.



Enter the program number to be searched, then press [SEARCH].

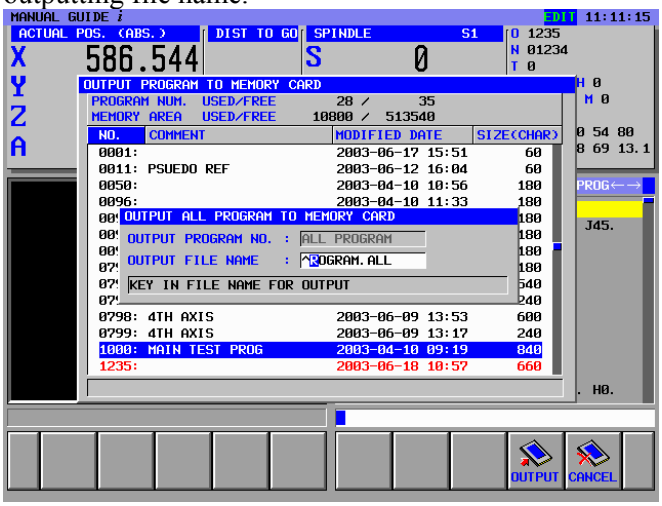
1. Output single part program
   Select the part program to be outputted by placing the cursor on it. Pressing [OUTPUT] displays the following window for entering outputting file name.





   If the program number can be used as the outputted file name, press [OUTPUT] without entering file name.
   In case that the outputted file name should be changed, enter the file name to OUTPUT FILE NAME and press [OUTPUT].
   In order to output the selected program together with subprogram called from the program, press [ON] for OUTPUT WITH SUBPROGRAM item. If not, press [OFF].

2. Output all part programs
   Pressing [ALLOUT] displays the following window for entering outputting file name.



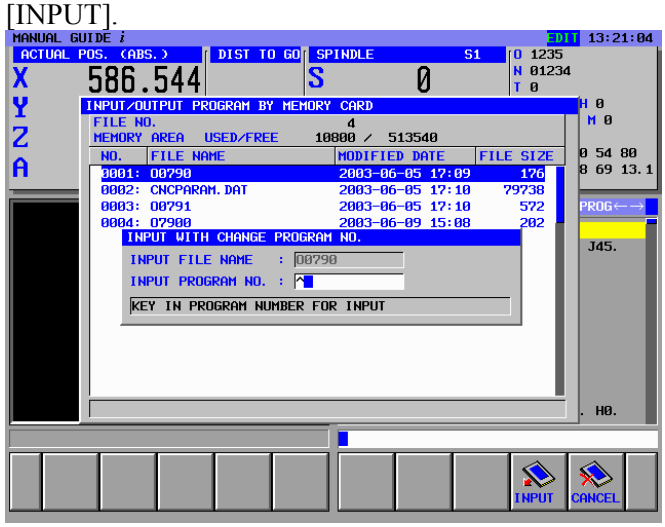
   If the file name “PROGRAM ALL” can be used as it is, press [OUTPUT] without entering file name and all part programs stored

in the CNC, the currently selected path when multi-path lathe, are outputted to the memory card with this name.
In case that the outputted file name should be changed, enter the file name to OUTPUT FILE NAME and press [OUTPUT].

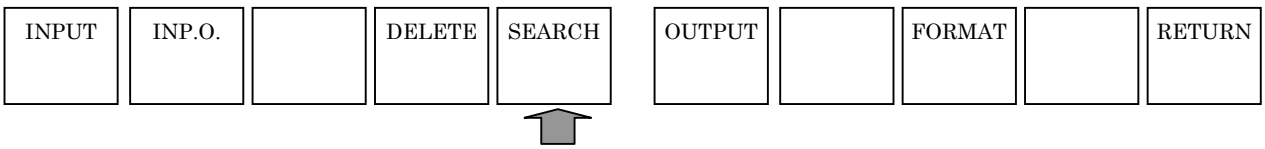
17.1.3 Memory Card Input Operation for Part Program

INPUT	INP.O.		DELETE	SEARCH	OUTPUT		FORMAT		RETURN

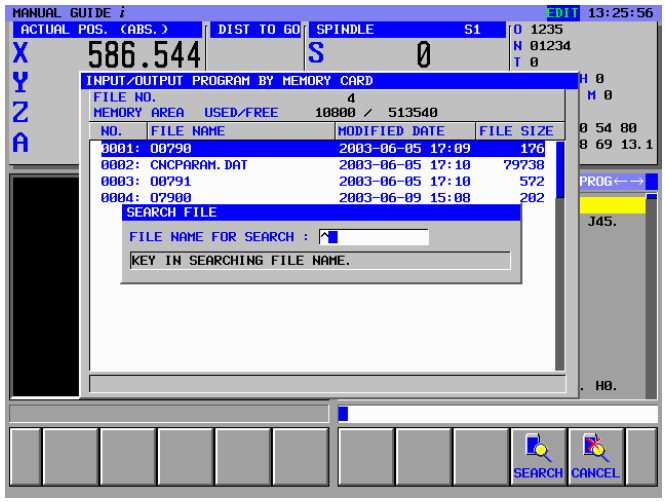
Select the file to be inputted by placing the cursor on it on the program input/output window.
Pressing [INPUT] begins to read the part program in the file from memory card to CNC.
Pressing [INP.O.] displays the following window for changing the reading program number, so enter the new program number and press [INPUT].



When the file, in which all part program was outputted with file name of “PROGRAM ALL”, is read to CNC with changing the program number by [INP.O.], the program number of the 1st program is changed to the new number.



In order to search the file to be inputted to CNC, press [SEARCH] and the following file searching window is displayed.



Enter the file name to be searched, and press [SEARCH], then the file is searched if the file is stored in the memory card.

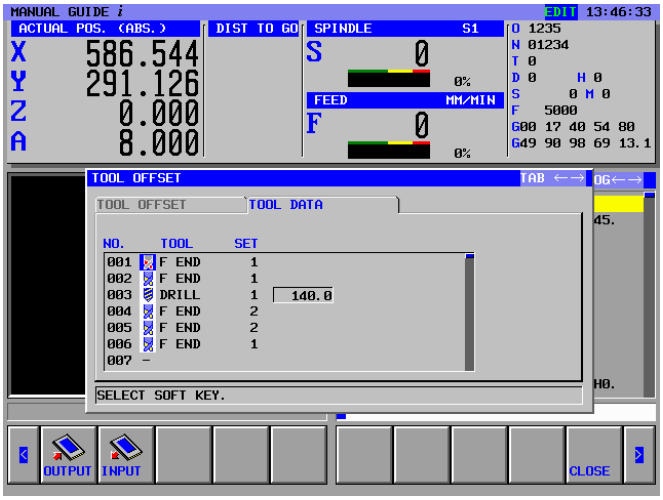
17.1.4 File Format Allowed for Memory Card Input/Output

Only text files can be input to and output from the memory card. The file format described below must be observed.

<1> A file must start with "%" and "LF".
<2> A file must end with "%".
<3> For input, data read operation is skipped after the first "%" is detected until an "LF" is detected.
<4> A block must end not with a semicolon (;) but with an "LF". ("LF" is 0A of ASCII code.)
<5> When a file containing lowercase alphabetic characters, kana characters, and some special characters (such as $, \, and !) is input, those characteristics are ignored.
<6> ASCII code is used as input/output code, regardless of the setting parameter (ISO/EIA).
<7> Whether to output an "LF" only or an "LF, CR, CR" as an EOB can be chosen using bit 3 (NCR) of parameter No. 0100.
<8> Characters usable for a file name
Alphabetic characters : A to Z
Numeric characters : 0 to 9
Special characters : $ & # % ' ( ) - @ ^ { } ~ ` ! _

17.2 MEMORY CARD INPUT/OUTPUT OF TOOL DATA

17.2.1 Memory Card Input/Output Screen of Tool Data

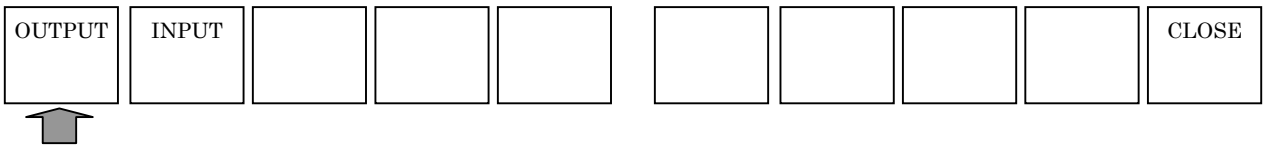


During displaying TOOL DATA window, the following soft-keys are displayed by pressing the leftmost soft key [<] or rightmost soft key [>] several times.

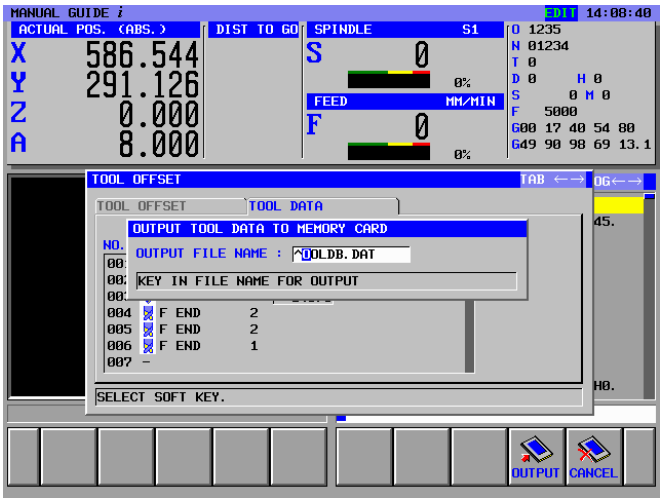
OUTPUT	INPUT								CLOSE

In order to input or output the tool data between memory card, select EDIT mode on the machine operator's panel.
Insert the memory card into the memory card slot on the LCD/MDI panel.

17.2.2 Memory Card Output Operation for Tool Data

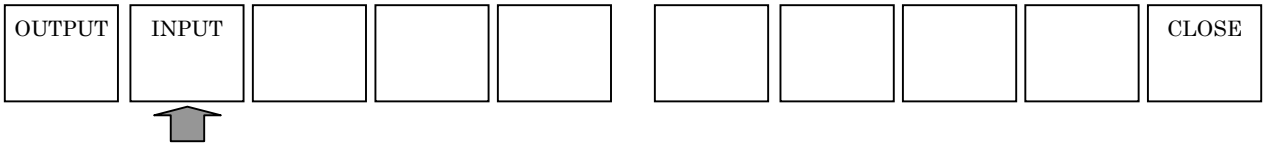


Pressing [OUTPUT] displays the following window for entering the output file name.

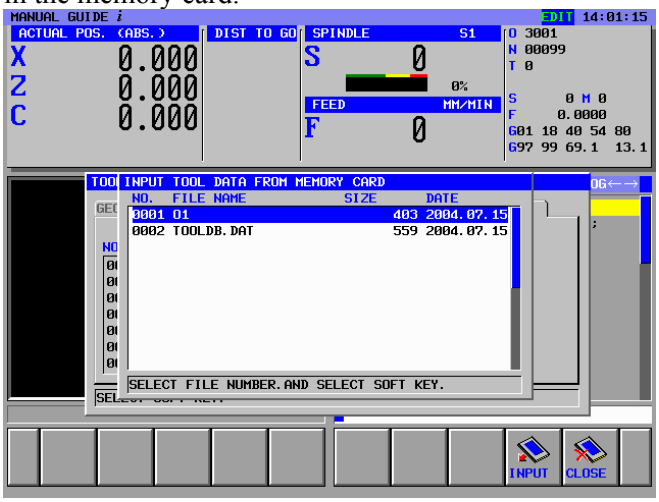


Pressing [OUTPUT] without entering the file name outputs the tool data with the file name “TOOLDAB.DAT].
In order to change the file name, enter the file name and press [OUTPUT].

17.2.3 Memory Card Input Operation for Tool Data



Pressing [INPUT] displays the following window of f the file list store in the memory card.



Select the file in which tool data are stored and to be read to CNC by placing the cursor by the cursor key.
Pressing [INPUT] begins reading the tool data to CNC form the memory card.

17.2.4 Data Format

The following format can be inputted or outputted.

1. Machining Center

G1980 P_ K_ T_ S_ A_ ;

P : Offset Number (1 → 999)
K : Kind of Tool
T : Name of Tool
S : Setting
A : Tool Data

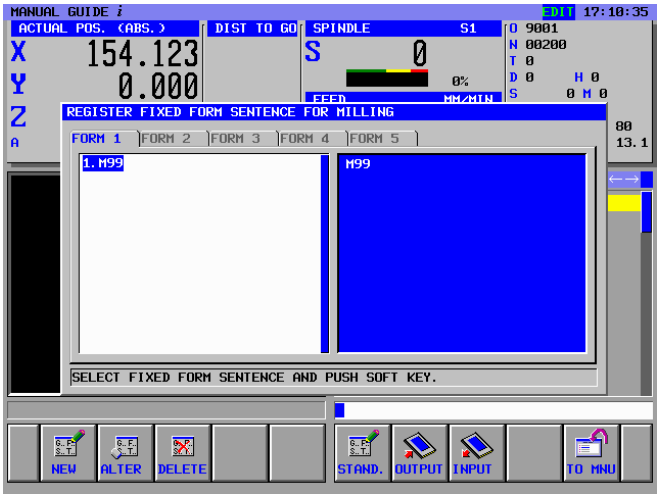
2. Lathe

G1981 P_ J_ K_ Q_ S_ A_ B_ ;

P : Offset Number (1 → 999)
J : Path Number (only multiple paths)
K : Kind of Tool
T : Name of Tool
S : Setting
A : Tool Data1
B : Tool Data2

17.3 MEMORY CARD INPUT/OUTPUT OF FIXED FORM SENTENCES

17.3.1 Memory Card Input/Output Screen of Fixed Form Sentences



The following soft key is displayed after [SETING] is pushed and “REGISTER FIXED FORM SENTENCES FOR MILLING” or “REGISTER FIXED FORM SENTENCES FOR TURNING” is selected.

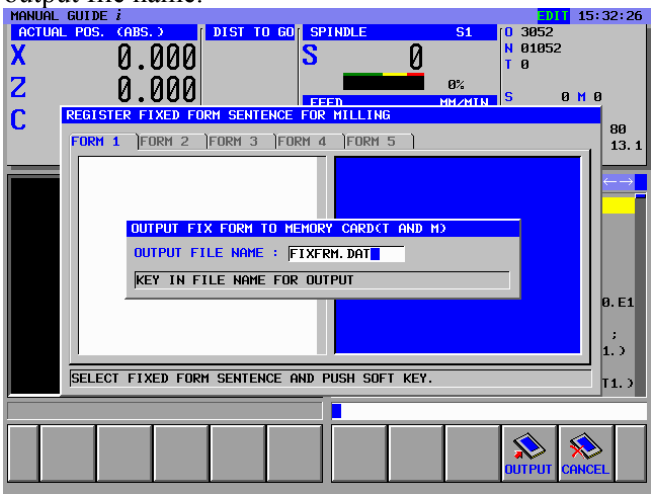
NEW	ALTER	DELETE			STAND.	OUTPUT	INPUT		TO MNU

Select EDIT mode on the machine operator’s panel.
Insert the memory card into the memory card slot on the LCD/MDI panel.

17.3.2 Output Fixed Form Sentences

NEW	ALTER	DELETE			STAND.	OUTPUT	INPUT		TO MNU

Pressing [OUTPUT] displays the following window for entering the output file name.

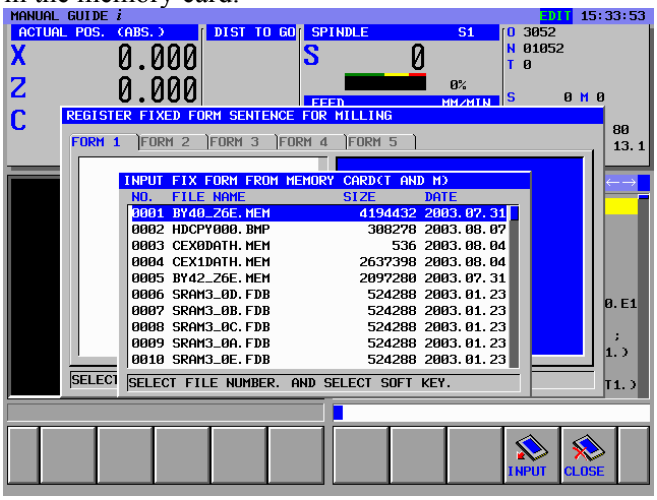


Pressing [OUTPUT] without entering the file name outputs the fixed form sentences with the file name “FIXFRM.DAT” (Output fixed form sentences for milling and turning in a lump.)
In order to change the file name, enter the file name and press [OUTPUT].

17.3.3 Input Fixed Form Sentences

NEW	ALTER	DELETE			STAND.	OUTPUT	INPUT		TO MNU

Pressing [INPUT] displays the following window of the file list store in the memory card.



Select the file in which fixed form sentences are stored and to be read to CNC by placing the cursor by the cursor key.
Pressing [INPUT] begins reading the tool data to CNC form the memory card.

17.3.4 Format of Fixed Form Sentences

The following format can be inputted or outputted for fixed form sentences.

<Mode>=_,<Tab>=_,<Title>=_,<Code>=_

Mode : Setting the mode of Fixed form sentences (Milling or Turning).

1:Fixed Form Sentence for turning mode

2:Fixed Form Sentence for milling mode

3: Fixed Form Sentence for milling and turning mode

Tab : The number of tab(1 - 5)

Title : Setting the name of Fixed Form Sentence

Code : Setting Fixed Form Sentence

4 words, <Mode>=, <Tab>=, <Title>=, <Code>=, can't be inputted as the name of fixed form sentence or fixed form sentence.

18 HANDLING LARGE PROGRAMS

18.1 SETTING A MAXIMUM PROGRAM SIZE THAT CAN BE HANDLED

In parameter No. 14795, specify a maximum allowable memory size to be used for program management.

Parameter

<1> No.14795#4 = 0 & No.14795#5 = 0
Set the maximum allowable program size to 250K bytes.

<2> No.14795#4 = 1 & No.14795#5 = 0
Set the maximum allowable program size to 500K bytes.

<3> No.14795#4 = 0 & No.14795#5 = 1
Set the maximum allowable program size to 1M bytes.

<4> No.14795#4 = 1 & No.14795#5 = 1
Set the maximum allowable program size to 2M bytes.

* A size of 200K bytes represents about 100,000 characters (4,000 blocks) when one block consists of 25 characters on average.

> ⚠ **CAUTION**
> To specify a maximum allowable memory size greater than 250K bytes in parameter No. 14795, set an appropriate value in parameter No. 8781 (DRAM size that can be used by a C language application).
> To increase the DRAM size, the custom capacity option is separately required.
> * If the DRAM size is increased by 1M bytes by using parameter No. 8781, about 500,000 characters (about 20,000 blocks) can be increased as a guideline when one block consists of 25 characters on average.

Restrictions

<1> If a large program is selected, the time required for switching from the NC screen to MGi screen increases.

<2> If a large program is selected, a longer time is required to move the program cursor as the program number increases.

18.2 HANDLING A PROGRAM LARGER THAN THE MAXIMUM ALLOWABLE SIZE

If the size of a program calculated according to the formula below exceeds the maximum allowable memory size set in parameter No. 14795, the program cannot be handled on MANUAL GUIDE *i*.

Calculated size = (18 bytes)×(total number of blocks)+((number of program characters)×1.1)

A program larger than the maximum allowable size is handled as described below.

(1) If the screen display is switched from the NC screen to the MGi screen
If the screen display is switched from the NC screen to the MGi screen when a program larger than the maximum allowable memory size is selected, the screen described below appears.
All MGi operations are disabled. Only switching to the NC screen with the function key is enabled.

(2) If a selection is made on the program list screen
If a program larger than the maximum allowable memory size is selected with the cursor and the [OPEN] soft key is pressed on the program list screen, the [PROGRAM EXCEED MAXIMUM SIZE.] message is displayed in the message display field on the program list screen and the selection of the program is disabled.

> ⚠ **CAUTION**
> If the program list screen contains only those programs that are larger than the maximum allowable memory size, the program list screen cannot be closed. In this case, create a program to close the program list screen.

(3) If an O search is made
<1> If the program number of a program larger than the maximum allowable memory size is entered into the key-in buffer, and the [O SRCH] soft key is pressed, the [PROGRAM EXCEED MAXIMUM SIZE.] message is displayed in the message display field on the base screen.
<2> If the program number of a program larger than the maximum allowable memory size is not entered into the key-in buffer, but the [O SRCH] soft key is pressed, the program is not searched for.

(4) If a program larger than the maximum allowable memory size is called by a subprogram call during operation or animated simulation

If a program larger than the maximum allowable memory size is called by a subprogram call during operation or animated simulation, the [PROGRAM EXCEED MAXIMUM SIZE.] message is displayed in the program display field, and the program is not displayed.

19 CALCULATOR FUNCTION

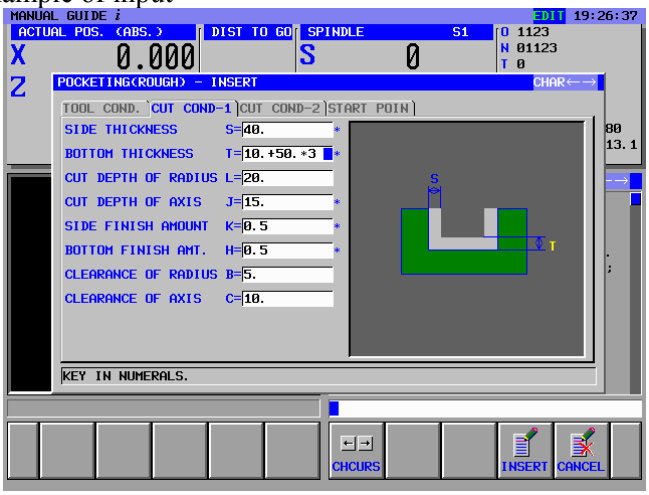
19.1 CALCULATOR FUNCTION

When numeric data is input, expressions for arithmetic operations, trigonometric functions, square root calculations, and so forth can be input for calculation.

1) Applications
The fixed-point format calculation function can be used for cycle input, arbitrary figure input, contour program input, setting of various data items (basis setting, measurement condition setting, calibration setting), relative coordinate preset input, and the key-in buffer(*).

2) Calculation methods
- Arithmetic operations (addition, subtraction, multiplication, and division)
  Arithmetic operations are performed using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Addition : 100.+200. [INPUT]
  (2) Subtraction : 100.-200. [INPUT]
  (3) Multiplication : 100.*200. [INPUT]
  (4) Division : 100./200. [INPUT]

Example of input



- Trigonometric functions (sine, cosine, tangent, arcsine, arccosine, arctangent)
  Trigonometric function calculations are made using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Sin : SIN(45) [INPUT]
  (2) Cosine : COS(45) [INPUT]
  (3) Tangent : TAN(45) [INPUT]
  (4) Arcsine : ASIN(0.5) [INPUT]
  (5) Arccosine : ACOS(0.5) [INPUT]
  (6) Arctangent : ATAN(20,2) [INPUT]
  (Note that for an arctangent calculation, a special format using two arguments is required. Enter data according to the format ATAN(a,b). arctan(a/b) is calculated.)

  For a calculation, () is required at all times.

- Square root
  A square root calculation is made using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Square root : SQRT(45) [INPUT]

  For a calculation, () is required at all times.

- Exponential functions
  Exponential function calculations are made using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Exponential function 1 (An exponential function of e = 2.718... can be calculated.) :
  EXP(4) [INPUT]
  (2) Exponential function 2 ("a" raised to the power of "b" can be calculated.) :
  PWR(4,3) [INPUT]
  (Note that for a calculation of exponential function 2, a special format using two arguments is required. Enter data according to the format PWR(a,b). "a" raised to the power of "b" is calculated.)

  For a calculation, () is required at all times.

- Logarithmic functions (common logarithm, natural logarithm)
  Logarithmic function calculations are made using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Common logarithm : LOG(45) [INPUT]
  (2) Natural logarithm : LN(45) [INPUT]

  For a calculation, () is required at all times.

- Absolute value
  An absolute value calculation is made using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Absolute value : ABS(-45) [INPUT]

  For a calculation, () is required at all times.

- Rounding
  Rounding operations are performed using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Rounding 1 (rounding off to an integer) :
  RND(1.234) [INPUT]
  (2) Rounding 2 (rounding off "a" to the decimal places specified by "b") :
  RND2(1.267,0.01) [INPUT]
  (Note that for a calculation of rounding 2, a special format using two arguments is required. Enter data according to the format RND(a,b). The value of "a" is rounded off to the decimal places specified by "b". As "b", do not specify a value other than 1, 0.1, 0.01, and so forth.)

  For a calculation, () is required at all times.

- Discarding
  This operation discards all decimal places. A discarding operation is performed using the key operations described below. The result of a calculation is displayed at the cursor position for input data.
  (1) Discarding : FIX(1.234) [INPUT]

  For a calculation, () is required at all times.

- Circle ratio
  A circle ratio calculation is made using the key operations described below. The circle ratio 3.14... is indicated.
  (1) Circle ratio : PAI [INPUT]

(*) In the case of key-in buffer input, no decimal point is assigned when an integer is produced as the result of a calculation. In cases other than key-in buffer input, the respective input formats are to be followed.

20 AUTOMATIC SETTING OF INITIAL VALUE DATA

20.1 AUTOMATIC SETTING OF INITIAL VALUES ON THE INPUT DATA SCREEN

Data previously entered on the data input screen of the cycle menu or drawing definition menu (blank figure block and tool definition block) is automatically set as initial input value data.

Accordingly, the operator needs to enter cycle menu and drawing definition menu data just once at the beginning. Then, the previously entered data is set as initial values.

The input data items other than the [TOOL COND] tab of the cycle menu are automatically set. The input data of the [TOOL COND] tab is automatically set according to Section 20.2, "AUTOMATIC TOOL DEFINITION BLOCK COPY".

For the figure menu, input data depends on the drawing data, so that no initial values are automatically set.

20.2 AUTOMATIC TOOL DEFINITION BLOCK COPY

If the user does not use Tool Data Base function, the user needs to specify the tool definition block of Drawing Definition menu before Milling cycles or Turning cycles.
In that case, the user specifies the same data as the tool condition data of each cycle menu.
This time, in each cycle menu, the tool condition data is set automatically by copying the data of tool definition block.

That is, when data is input in the tool definition block of the drawing definition menu, the data is once saved as internal data. Then, when data is to be input according to the cycle menu, the tool definition block data saved internally is copied as tool condition input data.

Accordingly, the operator just needs to enter a tool definition block initially. Then, the operator need not enter tool condition data in subsequent cycle menus.

The following tool definition block data is saved as internal data:

- Milling tool
  1) TOOL DIAMETER (D)

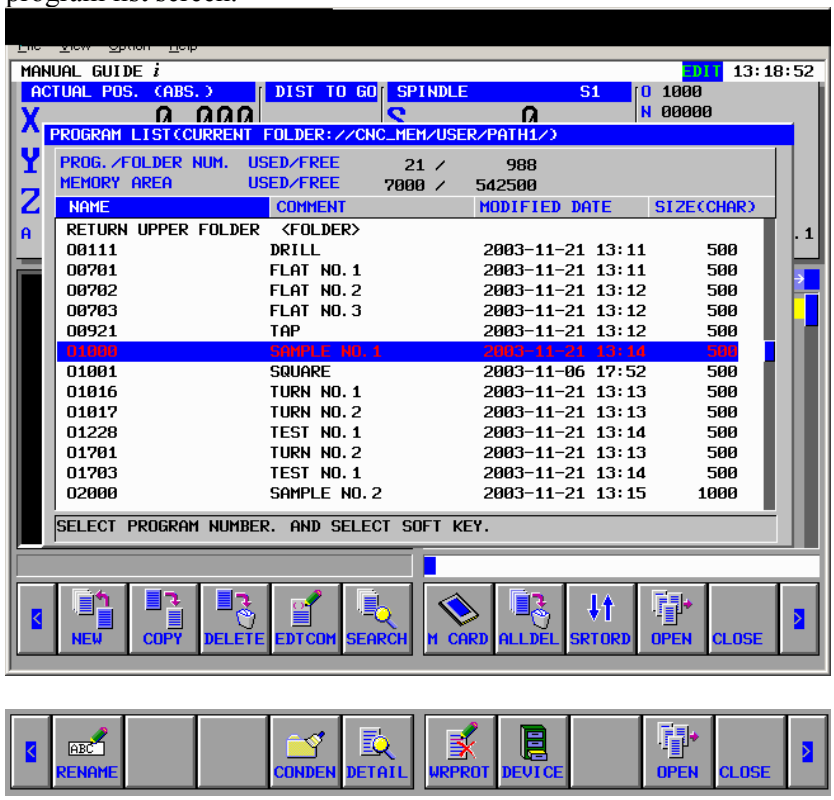
- Turning tool
  1) NOSE RADIUS (R)
  2) CUT EDGE ANGLE (A)
  3) NOSE ANGLE (B)

21 SUPPORT FOR FOLDER MANAGEMENT (FOR Series 30*i* ONLY)

This function is supported only for the Series 30*i*.

21.1 PROGRAM LIST SCREEN

This section describes the specifications of folder management on the program list screen.



21.1.1 Data Displayed in the Program List

(1) Program number
This program number is equivalent to a conventional program number.

(2) Comment
If a comment is longer than the displayable range, "..." is indicated at the end.

(3) Program update time (Year/month/day/hours/minutes)

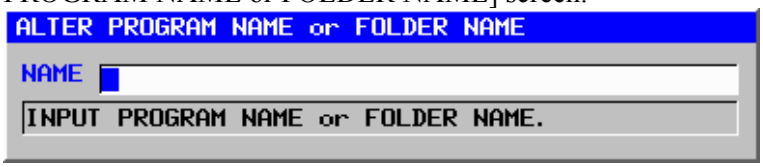
(4) Program size (number of characters)

(5) [RETURN UPPER FOLDER]

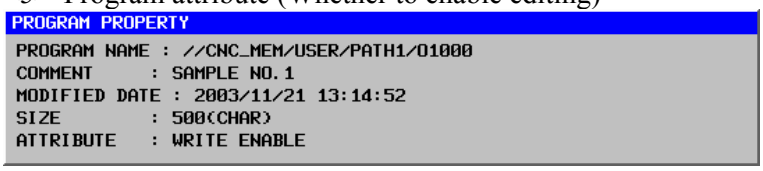
(6) Program attribute (whether to enable editing)
The character "R" is indicated at the right end of a program that must not be edited.

21.1.2 Operations Added for the Program List Screen

(1) Changing the program name/folder name
Pressing the [RENAME] soft key displays the [ALTER PROGRAM NAME or FOLDER NAME] screen.



(2) Program detail information
Pressing the [DETAIL] soft key displays the [PROGRAM PROPERTY] screen. This screen displays the following information:
<1> Program name (Character string: 32 characters max.) (NOTE)
<2> Comment (Character string: 48 characters max.) (NOTE)
<3> Date and time of program modification
(Year/month/day/hours/minutes/seconds)
<4> Program size (Number of characters)
<5> Program attribute (Whether to enable editing)

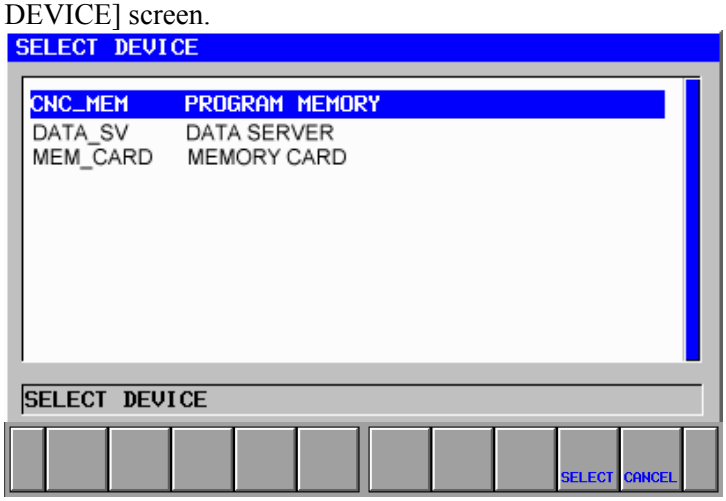


NOTE
With the Series 30*i*, lowercase letters can be used for a program name and comment.

(3) Program condensation
Pressing the [CONDEN] soft key condenses programs.

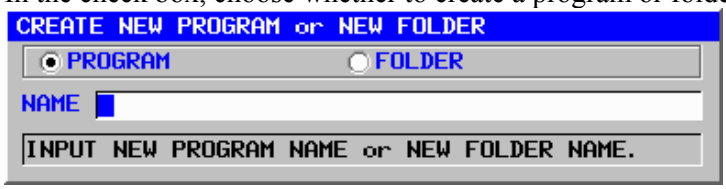
(4) Changing the program attribute (write protection)/folder attribute (write protection)
Pressing the [WRPROT] soft key changes the attribute cyclically.
The character "R" is indicated at the right end of a program that must not be edited.

(5) Device selection
Pressing the [DEVICE] soft key displays the [SELECT DEVICE] screen.



When you select a device then press the [SELECT] soft key, the list of programs on the device is displayed.

(6) Creation of a new program
In the check box, choose whether to create a program or folder.



21.1.3 Support for the Data Server

(1) MGi supports the editing and operation of a program on the data server.
By switching the device on the program list screen, a program on the data server can be handled in the same way as a program in the program memory.
However, there are differences in operation as indicated below. The differences comply with the operation specifications of the NC screen.

Operation	Program memory	Data Server
Creating a new program	Possible	Impossible
Copying a program	Possible	The program currently selected cannot be copied.
Deleting a program	Possible	The program currently selected cannot be deleted.
Renaming a program	Possible	The program currently selected cannot be renamed.
Memory card input/output	Possible	Impossible

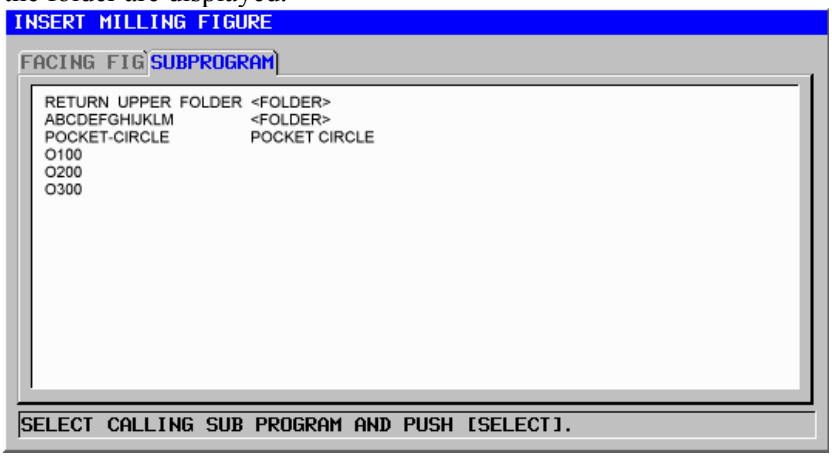
(2) The other functions on the data server (such as setting and FTP transfer) are to be performed on the NC screen (program list screen).

21.2 MEMORY CARD I/O SCREEN

The current folder is input/output.

21.3 SUBPROGRAM TAB ON THE CYCLE FIGURE SELECTION SCREEN

The folder containing the program currently selected as the main program is displayed as the current folder. When a folder is selected using the same operation as for the list of programs, the programs in the folder are displayed.

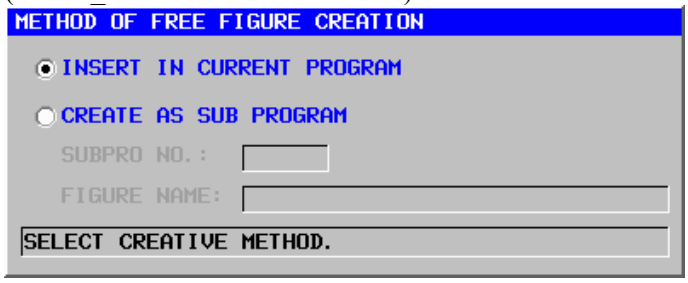


21.4 OUTPUT OF AN ARBITRARY FIGURE AS A SUBPROGRAM

As a folder for subprogram output, one of the following can be selected:

<1> Current folder

<2> Common program folder
(//CNC_MEM/USER/LIBRARY/)



21.5 DISPLAY OF ARBITRARY FIGURES OF M98 SUBPROGRAMS

The following folders are searched in this order, and the program first found is displayed:

<1> Folder containing the main program
<2> Common program folder
(//CNC_MEM/USER/LIBRARY/)

22 SCREEN HARD COPY

In order to make a copy to memory card of the screen of MANUAL GUIDE *i*, you need to operate as follows.

1. Setting of parameters
   In addition to the parameter for hard copy of standard CNC screen, No.3301#7HDC = 1, setting of the parameter for C executor, No.8650#4CKM = 1, is necessary.
   And furthermore, you need to set the parameter for memory card, No.20 = 4.

2. Operations
   Mount a memory card and display a necessary screen, then push "SHIFT" key more than 5 seconds.
   Release "SHIFT" key after checking the clock display on the screen stops. The clock display will move again when copying the screen completes.

3. Created File
   By the above operations, new file with name of "Hdcpy***.bmp" is created in the memory card. *** is the serial number and numbered as 001, 002. However, once the CNC power is turned off, this number is initialized to 000 from the next time of creating hard copy file.

23 DISPLAYING MACHINING TIME (FOR Series 16*i*/18*i*/21*i* ONLY)

During simulation, the logical machining time of each block is calculated from feedrate and distance for movement. And the result is displayed.

23.1 FORMAT OF MACHINING TIME DATA

Machining time data is inserted in program and it is conserved.
The place when the data are inserted is in the comment that is next to O number.
The format is "**,T_,A_**". "**,T_**" is cutting time data and "**,A_**" is non-cutting time data.

Ex. Cutting time is 1h 34' 38"(= 5678 s) and non-cutting time is 20' 34"(= 1234 s)

```
O2000                                    CHAR←→
1  O2000 (HORN-UPPER,T5678,A1234);
2  (RK382215-B-001-A) ;
3  (2003.08.22) ;
4  ;
5  G28 U0 ;
6  G28 W0 V0 M706 ;
7  G341 T23. ;
8  M01 ;
9  ;
10 G10 L2 P1 X-1220.0 Z-401.160 Y0 ;
11 G10 L2 P2 X-1220.0 Z463.38 Y0 ;
12 ;
```

23.2 OPERATION FOR INSERTING MACHINING TIME

The following soft keys are displayed on MEM mode after [SIMLAT] soft key is pushed.

REWIND	START	PAUSE	SINGLE	STOP	INIT	CUTDSP	INTERF	TLPATH	GRPOFF
LARGE	SMALL	AUTO	REVERS	ROTATE	←MOVE	MOVE→	↑ MOVE	↓ MOVE	CENTER
				WK SET	T-OFS				SETING
REWIND	O LIST	CHGDSP	N SRCH	O SRCH	ACTPOS	PRESET		INSERT	

After [START] is pushed and simulation is started, the machining time is calcurated and it is kept in the memory temporarily. After that, the color of [INSERT] soft key is changed. If [INSERT] soft key is pushed in this state, the message “MACHINE TIME DATA IS INSERTED. OK?” is displayed. Then, if “YES” is selected, the machining time data are inserted to the end of comment.

NOTE

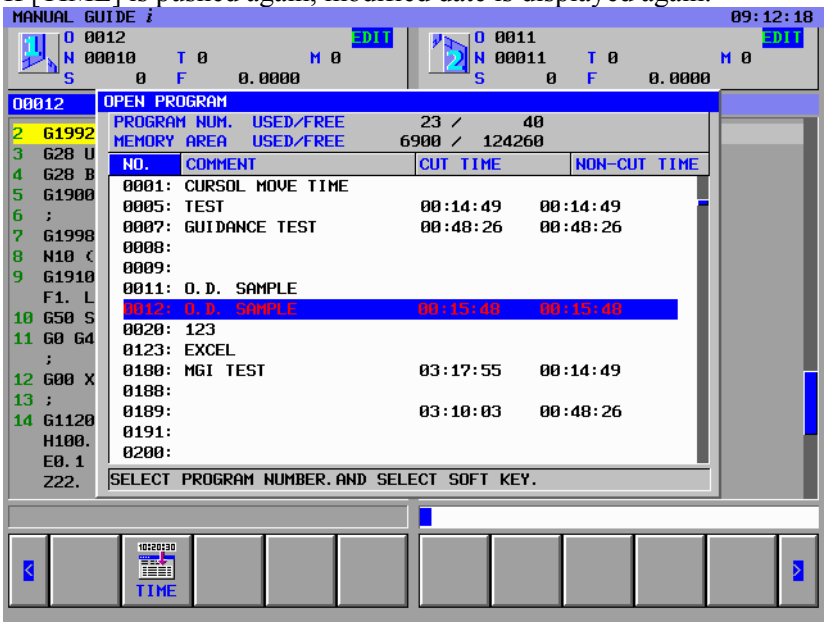
1 Don’t operate the machine during inserting machining time data.
2 When the machining time isn’t kept in the memory, the machining time can’t be inserted.
3 When the program has been protected, the machining time can’t be inserted.
4 The machining time can’t be inserted during operation.
5 All paths must be on MEM mode to insert machining time data. When at least one path isn’t on MEM mode, The machining time can’t be inserted.
6 Even if all path is on MEM mode, if at least one path is on background editing, machining time data can’t be inserted.
7 When the machining time data are inserted on one path, the machining time data are inserted on other paths too. But when simulation for each path is executed, the machining time data are inserted on only the path in which the simulation is executed.

23.3 DISPLAY MACHINING TIME

The following soft keys are displayed after [O LIST] soft key is pushed.

NEW	COPY	DELETE	EDTCOM	SEARCH	M CARD	ALLDEL	SRTORD	OPEN	CLOSE
	TIME								

If [TIME] soft key is pushed, the indication of modified date and program size are changed to machining time as follows.
If [TIME] is pushed again, modified date is displayed again.



NOTE
The parameters about displaying machining time are from 27390 to 27392.

24 PROGRAM COORDINATE SYSTEM CHANGING FUNCTION AND TOOL OFFSET MEMORY CHANGING FUNCTION

Programming, machining simulation and input/output of data which are fit for changing coordinate by "Program Coordinate System Changing Function" and "Tool Offset Memory Changing Function" can be executed in MANUAL GUIDE *i*.

.

NOTE

1 About the details of these functions, refer to the document of "FANUC Series 16*i* /18*i* /21*i* -TB Program Coordinate System Changing Function and Tool Offset Memory Changing Function" Specifications.

2 The Program Coordinate System Changing Functions of MANUAL GUIDE *i* can be used under the following case.

1) The axes, which are possible to reverse the direction by the Program Coordinate System Changing, must be Z-axis and Y-axis.

2) The right-handed coordinate system must be still used after changing the program coordinate system.

24.1 PROGRAM COORDINATE SYSTEM CHANGING FUNCTION

In this paragraph, the way to select program coordinate during operation, executing simulation and making arbitrary figures is explained.

NOTE
1 The turning cycle option is essential to use this function.
2 By setting the prameter No.14851#4=1, this function becomes available

24.1.1 COORDINATE SYSTEM SELECTION COMMAND

The program coordinate system is selected by Process Start Block G1992.

Insert Coordinate System Selection Command

<1> G1992 Block

When the cell is inserted on the first spindle side, the following block is inserted.
(Please refer to the section of process list editing function about cell.)
And when this block is executed, the program coordinate system changes to the coordinate system-1.

G1992 S1 W1 (COMMENT) ;

When the cell is inserted on the second spindle side, the following block is inserted. And when this block is executed, the program coordinate system changes to the coordinate system-2.

G1992 S2 W2 (COMMENT) ;

When the cell is copied or moved between the process list of the different spindles, ‘W1’ or ‘W2’ are automatically changed according to the spindle which the cell is moved to.

Executing Program

When G1992 block is executed, the program coordinate system can be changed by the followings.

<1> Change by M-code specified in the parameter

Please input M code number to parameter No. 27180 for changing to the coordinate system-1 and to parameter No. 27181 for changing to the coordinate system-2.

When G1992 S** W** block is executed, the M-code which is specified in the each parameter is outputted. Thus the program coordinate system changes.

<2> Change by executing P-code macro sub-program specified in the parameter

Please input P-code program number to parameter No. 27184 for changing to the coordinate system-1, and to parameter No. 27185 for changing to the coordinate system-2.

When G1992 S** W** block is executed, the P-code program which is specified in the each parameter is outputted. Thus the program coordinate system changes.

If both parameters the above <1> and <2> are specified, the P-code program is called first. And next, the M-code is output.

24.1.2 COORDINATE OF ARBITRARY FIGURES (XZ, ZC, ZY PLANE)

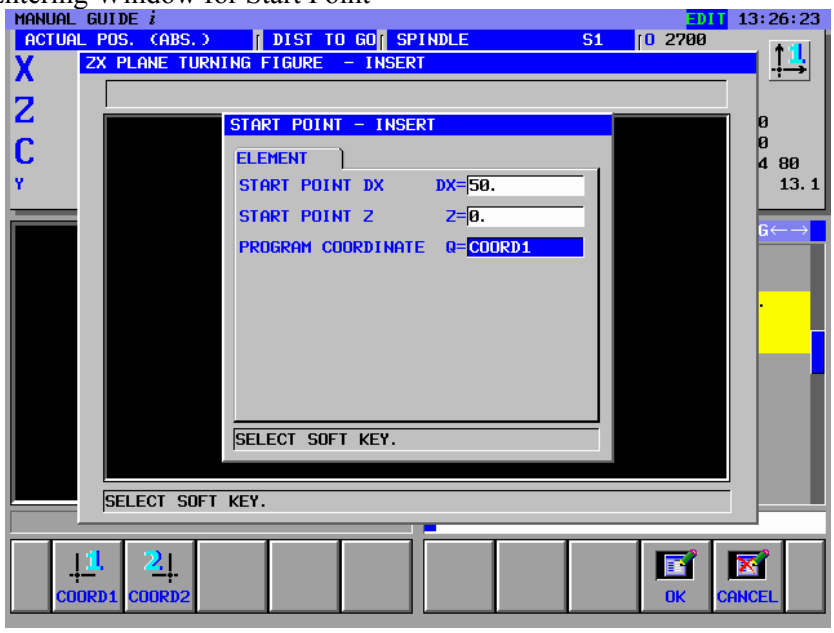
On the following arbitrary figures entering window, the programming figures are displayed according to the selected program coordinate system.

<1> Arbitrary Figures for the XZ Plane
<2> Arbitrary Figures for the ZC Plane
<3> Arbitrary Figures for the ZY Plane

Select Display Coordinate System

The display of coordinate system can be selected on the entering window for Start Point.
If "COORD1" is selected, the display of coordinate system changes to the program coordinate system-1. And if "COORD2" is selected, the display of coordinate system changes to the program coordinate system-2.

Entering Window for Start Point



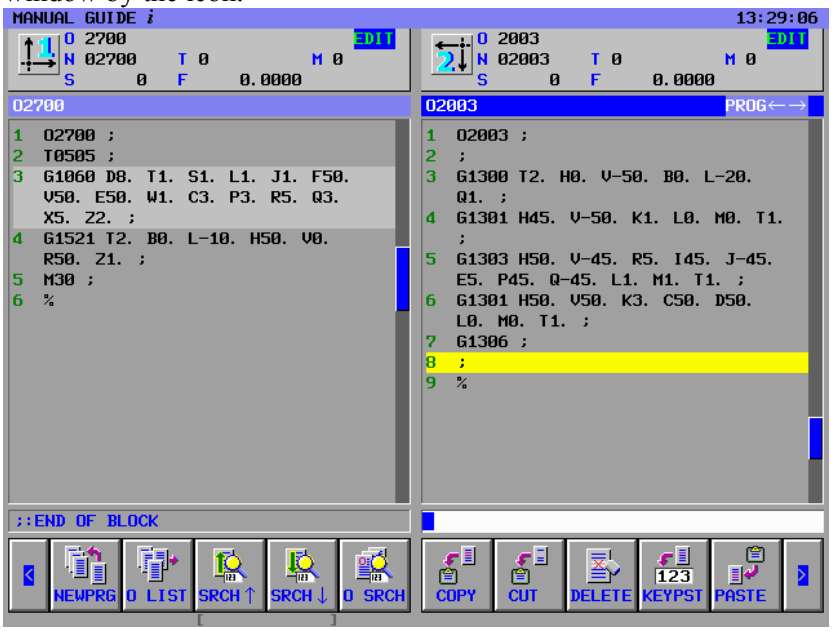
24.1.3 MACHINING SIMULATION

In the case of executing the machining simulation (Tool Path and Animated), the program coordinate system is changed by the address W1 and W2 of G1992 block.

NOTE
If the machining operation is finished at the status of selecting the coordinate system-2 and the machining operation is started again, the program is executed on the coordinate system-2.

24.1.4 STATUS DIPLAY

The current program coordinate system displays in the status display window by the icon.



The display icon, which is described the selected program coordinate system, is specified by the parameters No.27188 and 27189.

24.2 TOOL OFFSET MEMORY CHANGING FUNCTION

Tool offset, tool data and work shift for program coordinate system 1 and 2 can be inputted separately.

NOTE
The following functions can be used when the Tool Offset Memory Changing Function is enabled.

24.2.1 TOOL OFFSET DATA WINDOW

It is possible to set the tool offset data for each program coordinate system 1 and 2.

Select Coordinate System

The display of the data for each coordinate system changes as following by the parameter GCC(No.14851#6).

<1> In the case parameter GCC is 0
It is possible to change the display of the tool offset data for each coordinate system by pressing the [1←→2] soft-key.

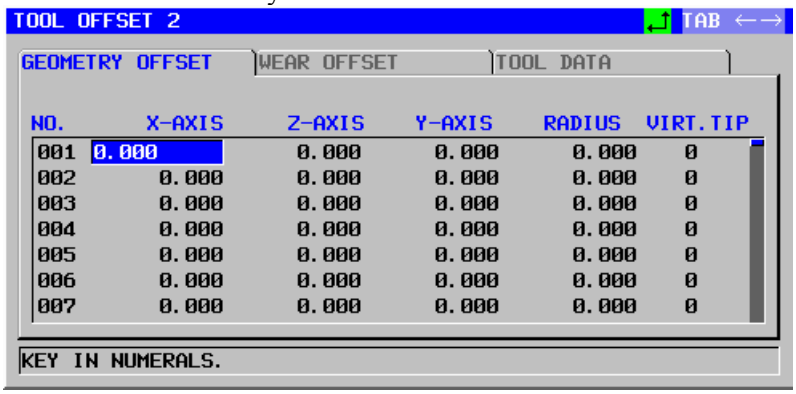
ACTPOS	MEASUR	+INPUT		INP.C.	CHCURS		1←→2	NO.SRH	CLOSE

<2> In the case parameter GCC is 1
The display of the tool offset data for each coordinate system changes according to DO-signal(F0347#GCO) dynamically.

Display Selected Coordinate System

The symbol for the selected coordinate system is displayed in the title of the window. The symbol is displayed according to the parameters No.27188 and No.27189.

When the coordinate system-2 is selected.



Setting of Tool Data

When the Tool Offset Memory Changing Function is effective, the maximum tool number which is possible to set to the tool data base is as follows.

for Program Coordinate System-1 : 150
for Program Coordinate System-2 : 150

The tool data for each program coordinate system are possible to input from and output to the memory card.

24.2.2 WORKPIECE SHIFT OFFSET DATA WINDOW

It is possible to set the workpiece shift offset data for each program coordinate system 1 and 2.

Select Coordinate System

The display of the data for each coordinate system changes as follows by the parameter GCC(No.14851#6).

<1> In the case parameter GCC is 0
It is possible to change the display of workpiece shift data for each coordinate system by pressing the [1←→2] soft-key.

ACTPOS	MEASUR	+INPUT			CHCURS		1←→2	NO.SRH	CLOSE

<2> In the case parameter GCC is 1
The display of the data for each coordinate system changes according to DO-signal(F0347#GCO) dynamically.

24.3 SET-UP GUIDANCE FUNCTIONS

There is not improvement in Set-Up Guidance Function.
So, even if the Tool Offset Memory Changing Function is effective, the exclusive measurement condition data and the calibration data for each program coordinate system are not provided. So it had better to distinguish these data between for the measurement in the program coordinate system-1 and 2.

24.3.1 MANUAL MEASUREMENT

When Tool Measurement or Measure is executed, the measurement data is set to the tool offset data of which program coordinate system is selected.

24.3.2 MEASUREMENT CYCLE

When Tool Measurement or Measure is executed, the measurement data is set to the tool offset data of which program coordinate system is selected.

24.3.3 MEASUREMENT RESULT DISPLAY SCREEN

In the case that the measurement data is set to the tool offset data, whether the offset data for program coordinate system-1 or for program coordinate system-2 is described after 'G' or 'W' character as follows.

(Example)
Z 0.973 →Z 0.000 T-Z10-G**1** -0.973
Geometry offset data for program coordinate system-1 was set.
Z 0.973 →Z 0.000 T-Z10-G**2** -0.973
Geometry offset data for program coordinate system-2 was set.
Z 0.973 →Z 0.000 T-Z10-W**1** -0.973
Wear offset data for program coordinate system-1 was set.
Z 0.973 →Z 0.000 T-Z10-W**2** -0.973
Wear offset data for program coordinate system- 2 was set.

24.4 CAUTIONS

If the machining operation is finished at the status of selecting the coordinate system-2 and the machining operation is started again, the program is executed on the coordinate system-2.
So be sure to insert the coordinate system selection commands to the top of the machining program to avoid executing machining under the wrong coordinate system.

There are the following restrictions.

- The Program Coordinate System Changing Functions and the Tool Offset Memory Changing Functions can not be used on turnin mode of compound machine.

III. CYCLE MACHINING TYPES

1 MILLING

MANUAL GUIDE *i* supports the following types of milling.

Hole machining (with the tool rotated)				
	Machining type block	Milling (Note)	Turning (Note)	
		G1000	G1110	Center drilling
		G1001	G1111	Drilling
		G1002	G1112	Tapping
		G1003	G1113	Reaming
		G1004	G1114	Boring
		G1005	-	Fine boring
		G1006	-	Back boring
	Hole position block (XY plane)	G1210	Random points	
		G1211	Linear points (same interval)	
		G1212	Linear points (different interval)	
		G1213	Grid points	
		G1214	Rectangle points	
		G1215	Circle points	
		G1216	Arc points (same interval)	
		G1217	Arc points (different interval)	
	Hole position block (YZ plane)	G1310	Random points	
		G1311	Linear points (same interval)	
		G1312	Linear points (different interval)	
		G1313	Grid points	
		G1314	Rectangle points	
		G1315	Circle points	
		G1316	Arc points (same interval)	
		G1317	Arc points (different interval)	
	Hole position block (XC plane, end face)	G1572	Circle points	
		G1573	Random points	
	Hole position block (ZC plane, cylindrical surface)	G1672	Circle points	
		G1673	Random points	
	Hole position block (XA plane, cylindrical surface)	G1772	Circle points	
		G1773	Random points	

NOTE

Hole Machining with the tool rotated has 2 type for Milling machine and Turning machine (Lathe). Therefore, please set the parameter No.27000 #1 in order to use suitable one for your machine.

No.27000#1=0 : For Milling machine
No.27000#1=1 : For Lathe

Facing			
	Machining type block	G1020	Roughing
		G1021	Finishing
	Fixed-figure block (XY plane)	G1220	Rectangle
		G1221	Circle
		G1222	Track
	Arbitrary-figure block (XY plane)	G1200	Start point
		G1201	Straight line
		G1202	Arc (CW)
		G1203	Arc (CCW)
		G1204	Chamfering
		G1205	Corner rounding
		G1206	End
	Fixed-figure block (YZ plane)	G1320	Rectangle
		G1321	Circle
		G1322	Track
	Arbitrary-figure block (YZ plane)	G1300	Start point
		G1301	Straight line
		G1302	Arc (CW)
		G1303	Arc (CCW)
		G1304	Chamfering
		G1305	Corner rounding
		G1306	End
	Fixed figure block (XC plane, end face)	G1520	Rectangle
		G1521	Circle
		G1522	Track
	Arbitrary-figure block (XC plane, end face)	G1500	Start point
		G1501	Straight line
		G1502	Arc (CW)
		G1503	Arc (CCW)
		G1504	Chamfering
		G1505	Corner rounding
		G1506	End
	Arbitrary-figure block (ZC plane, cylindrical surface)	G1600	Start point
		G1601	Straight line
		G1602	Arc (CW)
		G1603	Arc (CCW)
		G1604	Chamfering
		G1605	Corner rounding
		G1606	End
	Arbitrary-figure block (XA plane, cylindrical surface)	G1700	Start point
		G1701	Straight line
		G1702	Arc (CW)
		G1703	Arc (CCW)
		G1704	Chamfering
		G1705	Corner rounding
		G1706	End

Contouring			
	Machining type block	G1060	Outer Wall Roughing
		G1061	Outer Wall Bottom finishing
		G1062	Outer Wall Side finishing
		G1063	Outer Wall Chamfering
		G1064	Inner Wall Roughing
		G1065	Inner Wall Bottom finishing
		G1066	Inner Wall Side finishing
		G1067	Inner Wall Chamfering
		G1068	Partial Roughing
		G1069	Partial Bottom finishing
		G1070	Partial Side finishing
		G1071	Partial Chamfering
	Fixed-figure block (XY plane)	G1220	Rectangle
		G1221	Circle
		G1222	Track
	Arbitrary-figure block (XY plane)	G1200	Start point
		G1201	Straight line
		G1202	Arc (CW)
		G1203	Arc (CCW)
		G1204	Chamfering
		G1205	Corner rounding
		G1206	End
	Fixed-figure block (YZ plane)	G1320	Rectangle
		G1321	Circle
		G1322	Track
	Arbitrary-figure block (YZ plane)	G1300	Start point
		G1301	Straight line
		G1302	Arc (CW)
		G1303	Arc (CCW)
		G1304	Chamfering
		G1305	Corner rounding
		G1306	End
	Fixed-figure block (XC plane, end face)	G1520	Rectangle
		G1521	Circle
		G1522	Track
	Arbitrary-figure block (XC plane, end face)	G1500	Start point
		G1501	Straight line
		G1502	Arc (CW)
		G1503	Arc (CCW)
		G1504	Chamfering
		G1505	Corner rounding
		G1506	End
	Arbitrary-figure block (ZC plane, cylindrical surface)	G1600	Start point
		G1601	Straight line
		G1602	Arc (CW)
		G1603	Arc (CCW)
		G1604	Chamfering
		G1605	Corner rounding
		G1606	End

	Arbitrary-figure block (XA plane, cylindrical surface)	G1700	Start point
		G1701	Straight line
		G1702	Arc (CW)
		G1703	Arc (CCW)
		G1704	Chamfering
		G1705	Corner rounding
		G1706	End

Emboss machining			
	Machining type block	G1080	Roughing
		G1081	Bottom finishing
		G1082	Side finishing
		G1083	Chamfering
	Arbitrary-figure block (XY plane)	G1200	Start point
		G1201	Straight line
		G1202	Arc (CW)
		G1203	Arc (CCW)
		G1204	Chamfering
		G1205	Corner rounding
		G1206	End
	Arbitrary-figure block (YZ plane)	G1300	Start point
		G1301	Straight line
		G1302	Arc (CW)
		G1303	Arc (CCW)
		G1304	Chamfering
		G1305	Corner rounding
		G1306	End
	Arbitrary-figure block (XC plane, end face)	G1500	Start point
		G1501	Straight line
		G1502	Arc (CW)
		G1503	Arc (CCW)
		G1504	Chamfering
		G1505	Corner rounding
		G1506	End
	Arbitrary-figure block (ZC plane, cylindrical surface)	G1600	Start point
		G1601	Straight line
		G1602	Arc (CW)
		G1603	Arc (CCW)
		G1604	Chamfering
		G1605	Corner rounding
		G1606	End
	Arbitrary-figure block (XA plane, cylindrical surface)	G1700	Start point
		G1701	Straight line
		G1702	Arc (CW)
		G1703	Arc (CCW)
		G1704	Chamfering
		G1705	Corner rounding
		G1706	End

Pocketing			
	Machining type block	G1040	Roughing
		G1041	Bottom finishing
		G1042	Side finishing
		G1043	Chamfering
	Fixed-figure block (XY plane)	G1220	Rectangle
		G1221	Circle
		G1222	Track
	Arbitrary-figure block (XY plane)	G1200	Start point
		G1201	Straight line
		G1202	Arc (CW)
		G1203	Arc (CCW)
		G1204	Chamfering
		G1205	Corner rounding
		G1206	End
	Fixed-figure block (YZ plane)	G1320	Rectangle
		G1321	Circle
		G1322	Track
	Arbitrary-figure block (YZ plane)	G1300	Start point
		G1301	Straight line
		G1302	Arc (CW)
		G1303	Arc (CCW)
		G1304	Chamfering
		G1305	Corner rounding
		G1306	End
	Fixed-figure block (XC plane, end face)	G1520	Rectangle
		G1521	Circle
		G1522	Track
	Arbitrary-figure block (XC plane, end face)	G1500	Start point
		G1501	Straight line
		G1502	Arc (CW)
		G1503	Arc (CCW)
		G1504	Chamfering
		G1505	Corner rounding
		G1506	End
	Arbitrary-figure block (ZC plane, cylindrical surface)	G1600	Start point
		G1601	Straight line
		G1602	Arc (CW)
		G1603	Arc (CCW)
		G1604	Chamfering
		G1605	Corner rounding
		G1606	End
	Arbitrary-figure block (XA plane, cylindrical surface)	G1700	Start point
		G1701	Straight line
		G1702	Arc (CW)
		G1703	Arc (CCW)
		G1704	Chamfering
		G1705	Corner rounding
		G1706	End

Grooving			
	Machining process block	G1050	Roughing
		G1051	Bottom finishing
		G1052	Side finishing
		G1053	Chamfering
	Fixed-figure block (XY plane)	G1220	Rectangle
		G1221	Circle
		G1222	Track
		G1223	Radial groove
	Arbitrary-figure block (XY plane)	G1200	Start point
		G1201	Straight line
		G1202	Arc (CW)
		G1203	Arc (CCW)
		G1204	Chamfering
		G1205	Corner rounding
		G1206	End
	Fixed-figure block (YZ plane)	G1320	Rectangle
		G1321	Circle
		G1322	Track
		G1323	Radial groove
	Arbitrary-figure block (YZ plane)	G1300	Start point
		G1301	Straight line
		G1302	Arc (CW)
		G1303	Arc (CCW)
		G1304	Chamfering
		G1305	Corner rounding
		G1306	End
	Fixed-figure block (XC plane, end face)	G1520	Rectangle
		G1521	Circle
		G1522	Track
		G1523	Radial groove
	Arbitrary-figure block (XC plane, end face)	G1500	Start point
		G1501	Straight line
		G1502	Arc (CW)
		G1503	Arc (CCW)
		G1504	Chamfering
		G1505	Corner rounding
		G1506	End
	Arbitrary-figure block (ZC plane, cylindrical surface)	G1600	Start point
		G1601	Straight line
		G1602	Arc (CW)
		G1603	Arc (CCW)
		G1604	Chamfering
		G1605	Corner rounding
		G1606	End
	Arbitrary-figure block (XA plane, cylindrical surface)	G1700	Start point
		G1701	Straight line
		G1702	Arc (CW)
		G1703	Arc (CCW)
		G1704	Chamfering
		G1705	Corner rounding
		G1706	End

C axis grooving			
	Machining process block	G1056	C axis grooving
	Fixed-figure block (XC plane, end face)	G1570	C axis groove
		G1571	X axis groove
	Fixed-figure block (ZC plane, cylindrical surface)	G1670	C axis groove
		G1671	Z axis groove

A axis grooving			
	Fixed-figure block (XA plane, cylindrical surface)	G1770	A axis groove
		G1771	X axis groove

NOTE

1 MANUAL GUIDE *i* supports three types of hole machining, that is, hole machining by milling, hole machining by turning (with the tool rotated), and hole machining by turning (with the workpiece rotated).
On the CNC for milling, only hole machining by milling is usable. On the CNC for lathe turning, hole machining by turning (with the tool rotated) and hole machining by turning (with the workpiece rotated) are usable.
On the CNC for complex machining, all types of machining are usable, and what machining type to use can be selected using parameters.
Refer to manuals from respective machine tool builders for explanations about what type is actually usable.

2 When MANUAL GUIDE *i* is used to enter cycle machining data, combinations of a machining type block and a figure block must be entered in succession for all machining types except hole machining by turning.
More than one figure block can be entered for one machining type block.
For arbitrary figures, however, more than one set of arbitrary figures can be entered if one set is assumed to consist of up to 90 figures enclosed between start and end points.

3 A data item whose address is indicated with [*] in the following tables is automatically set with a typical value unless a value is entered for it. You need to enter no data if you accept the typical value.

4 On the screen, [*] is displayed in the right end of the data item.

5 In some of the input data items, the system sets the last inputted data as an initial data. As to these input data items, the mark of “(COPY)” was described.

6 In the input data item of “APROCH MOTION”, when the machine is 2 axes of Maximum simultaneously controlled axes, please sure not to specify the [3 AXES] soft-key.
(If [3 AXES] is specified, the alarm 15 occurred during execution of Cycles.)

NOTE

7 In the input data item of “CUT ANGLE” of Pocketing Rough (G1040) and Bottom Finish (G1041), when the machine is 2 axes of Maximum simultaneously controlled axes, please sure not to set the data.
(If the data is set, the alarm 15 occurred during execution of Cycles.)

8 The motions go back to the start point where the machining cycle started after completing the all cycle motions.
(It is possible not to go back to the start point when the parameter No. 27002#7 is set to 1.)

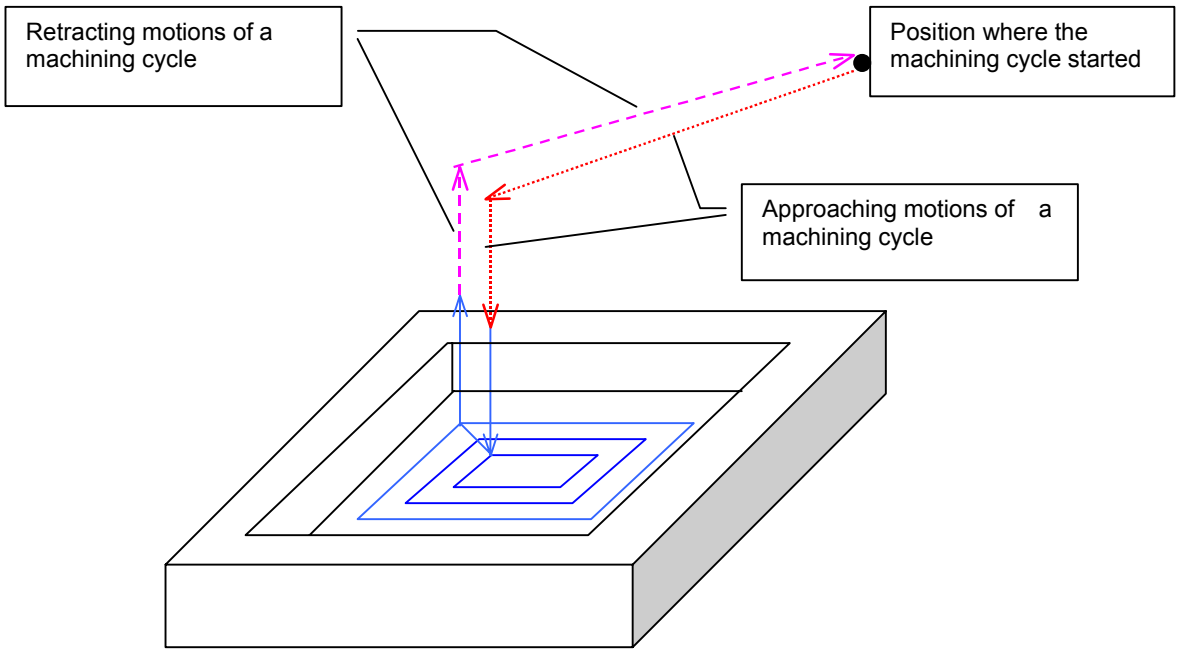
9 Cycle machining cannot be performed in any of control modes of the high-speed and high-precision functions (commands for advanced preview control, AI advanced preview control, AI contour control, AI nano contour control, high-precision contour control, AI high-precision contour control, and AI nano high-precision contour control). To perform cycle machining, control mode must be canceled.

10 On the CNC for lathe turning with G-code system B or C, when cycle machining is specified, G90 must be specified in advance to make a switchover to the absolute coordinate system. For hole machining by turning (with the tool rotated), G98 (initial level return) or G99 (R position level return) must be specified in advance.

11 On the CNC for milling, when cycle machining is specified, a switchover to G90 (absolute command) occurs internally. Therefore, G91 must be specified if incremental commands are used after cycle machining.

Remarks) Cycle retract motions

In case of No.27002#7=0, Retracting motions indicated as broken lines in the following drawing will be outputted. The order of motion axis will be opposite to the approached motions

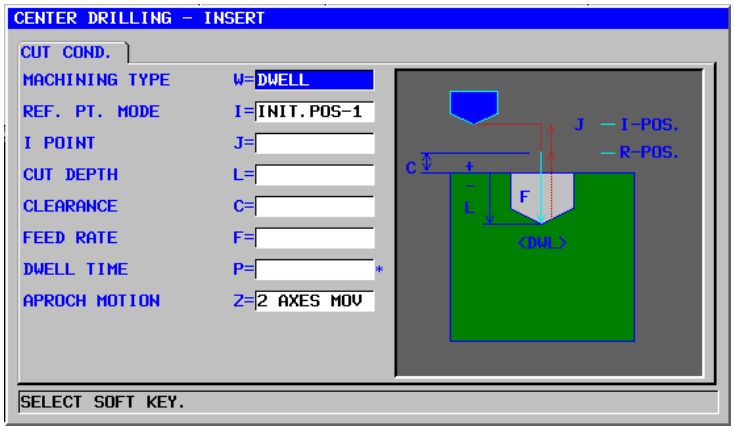


In case of carrying out plural figures machining by one G4 digit cycle, such like drilling or C-axis grooving, this retracting motions will be available at the last figure machining.

1.1 HOLE MACHINING BY MILLING

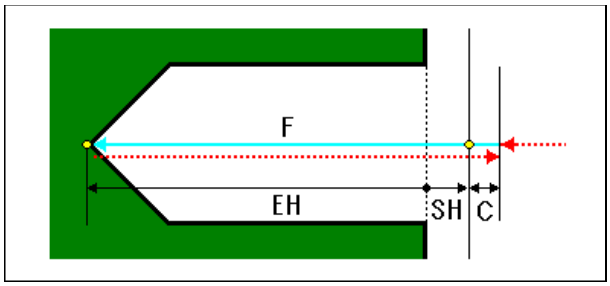
1.1.1 Hole Machining Type Block

Center Drilling: G1000



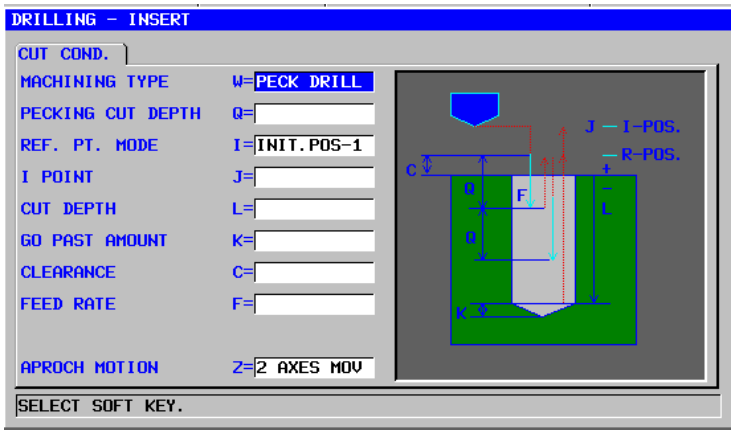
CUT COND.		
Data item		**Meaning**
W	MACHINING TYPE	[NORMAL] : No dwelling is performed. (initial value) [DWELL] : Dwelling is performed.
I	REF. PT. MODE	[INIT-1]: An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2]: All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.

Drilling: G1001



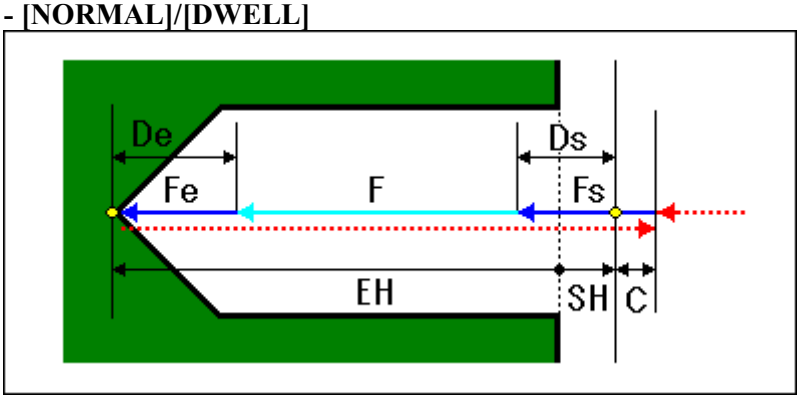
CUT COND.		
Data item		**Meaning**
W	MACHINING TYPE	[NORMAL] : One cut with no dwelling performed (initial value) [DWELL] : One cut with dwelling performed [PECK] : Peck drilling (Note 1) [H SPED] : High-speed peck drilling (Note 2)
Q	PECKING CUT DEPTH	Depth of cut made by one cut (radius value, positive value) (COPY)
I	REF. PT. MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
K	GO PAST AMOUNT	Length of the incomplete hole portion at the tool tip. If omitted, 0 is assumed. (Radius value, positive value) (COPY)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

NOTE
1 In the case of 'MACHINING TYPE' = 'PECK', the system refers to the parameter No.5115 as the return amount. Therefore, please set No.5115 to suitable value before machining.
2 In the case of 'MACHINING TYPE' = 'H SPED', the system refers to the parameter No.5114 as the return amount. Therefore, please set No.5114 to suitable value before machining.

- **Tool path**

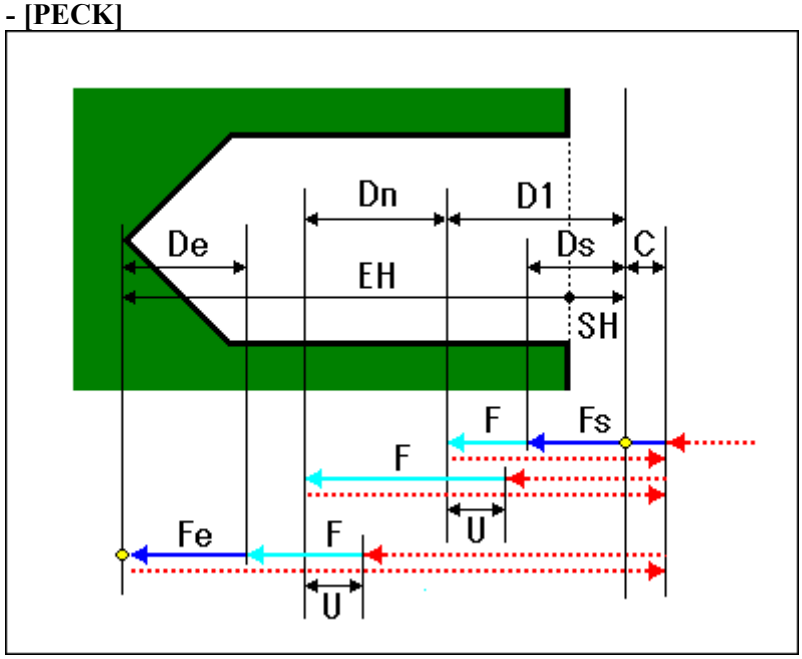
A drilling tool path can be selected from the following:

- [NORMAL]/[DWELL]



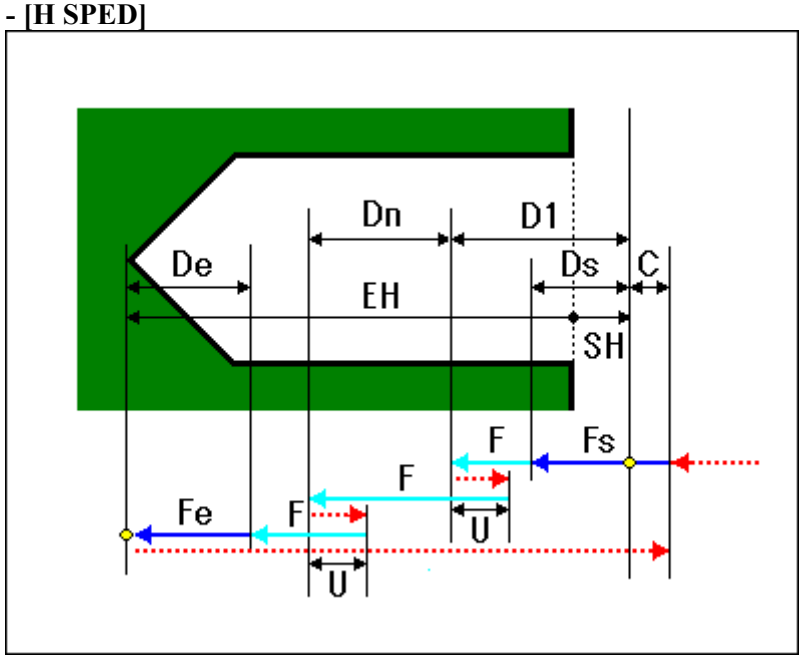
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.

- [PECK]



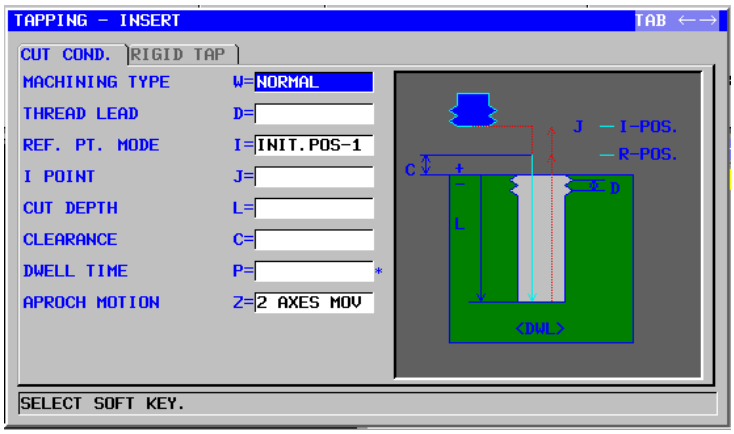
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the position "cutting start position - primary cut depth (D1)" at the cutting feedrate (F).
<3> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<4> Move the tool to the position "previous cutting end position + return clearance (U)" in rapid traverse.
<5> Move the tool to the position "previous cutting end position - compensation cut depth (Dn)" at the cutting feedrate (F).
<6> Repeat steps <3> to <5> until the last cutting end position is reached.
<7> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.

- [H SPED]



<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the position "cutting start position - primary cut depth (D1)" at the cutting feedrate (F).
<3> Move the tool to the position "current position + return clearance (U)" in rapid traverse.
<4> Move the tool to cut to the position "previous cutting end position - compensation cut depth (Dn)" at the cutting feedrate (F).
<5> Repeat steps <3> and <4> until the cutting end position is reached.
<6> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.

Tapping: G1002

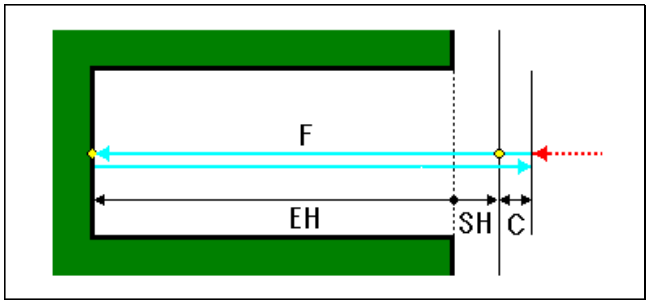


CUT COND.		
Data item		**Meaning**
W	MACHINING TYPE	[NORMAL] : CW tapping (initial value) [REVERS] : CCW tapping
D	THREAD LEAD	Tapping tool lead (radius value, positive value) (COPY)
I	REF. PT. MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

RIGID TAP		
Data item		**Meaning**
R	TAP TYPE	[FLOAT]: Specifies the float tapping. (initial value) [RIGID]: Specifies the rigid tapping. (Note)
S	SPINDLE SPEED	Spindle speed (min^{-1})

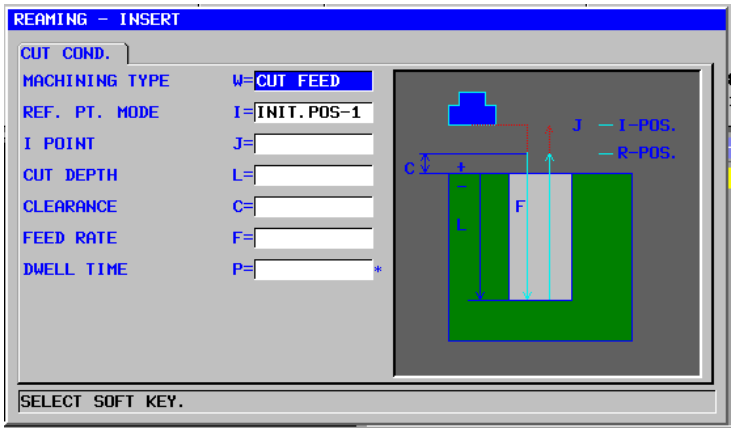
NOTE
1 When you use rigid tapping mode M code command (No.5200#0=0), the system refers to No.5210 or No.5212 as the value of M code. Therefore, please set No.5210 or No.5212 to suitable value before machining.

- **Tool path**



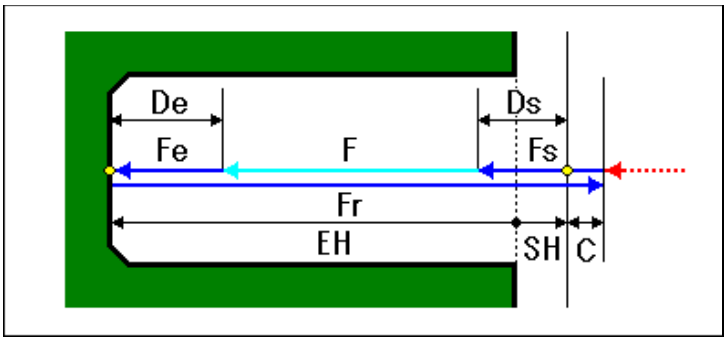
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> Stop the spindle.
<4> Rotate the spindle in reverse.
<5> Move the tool to the position "cutting start position + clearance (C)" at the cutting feedrate (F).
<6> Cause the spindle to start rotating normally.

Reaming: G1003



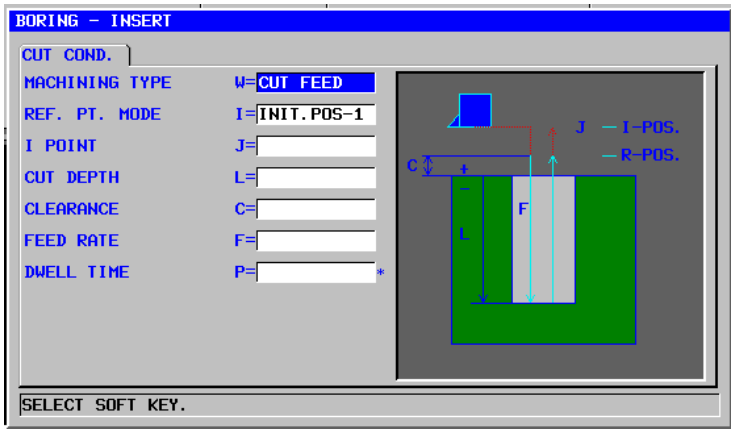
CUT COND.		
Data item		**Meaning**
W	MACHINING TYPE	[CUT] : The tool retracts from the hole bottom in cutting feed. (initial value) [RAPID] : The tool retracts from the hole bottom in rapid traverse. [DWELL] : After dwelling at the hole bottom, the tool retracts in cutting feed.
I	REF. PT. MODE	[INIT-1]: An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2]: All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



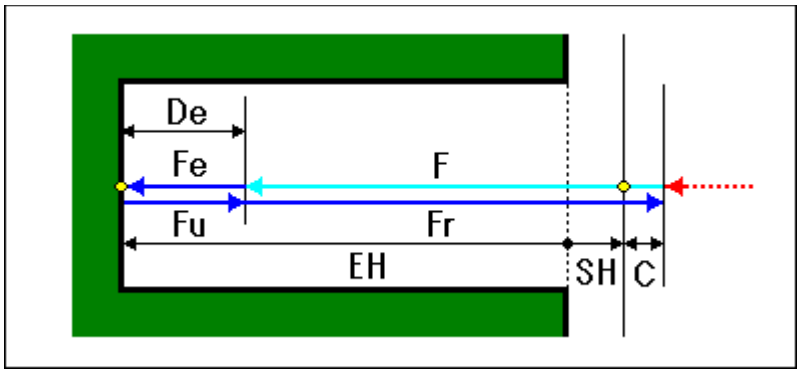
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> Move the tool to the position "cutting start position + clearance (C)" at the returning feedrate (Fr).

Boring: G1004



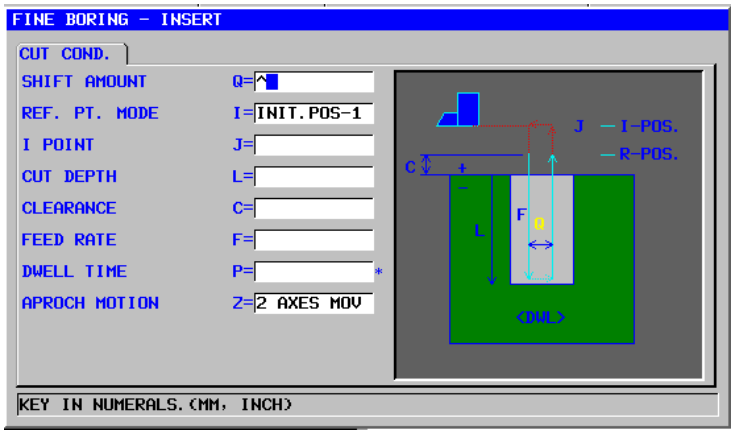
CUT COND.		
Data item		**Meaning**
W	MACHINING TYPE	[CUT] : The tool retracts from the hole bottom in cutting feed. (initial value) [RAPID] : The tool retracts from the hole bottom in rapid traverse. [DWELL] : After dwelling at the hole bottom, the tool retracts in cutting feed.
I	REF. PT. MODE	[INIT-1]: An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2]: All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



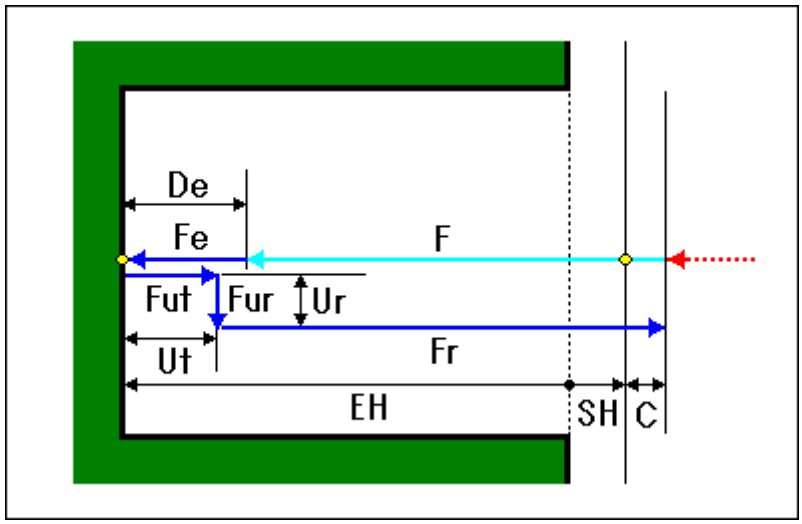
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> Move the tool to the position "cutting start position + clearance (C)" at the returning feedrate (Fr).

Fine Boring: G1005



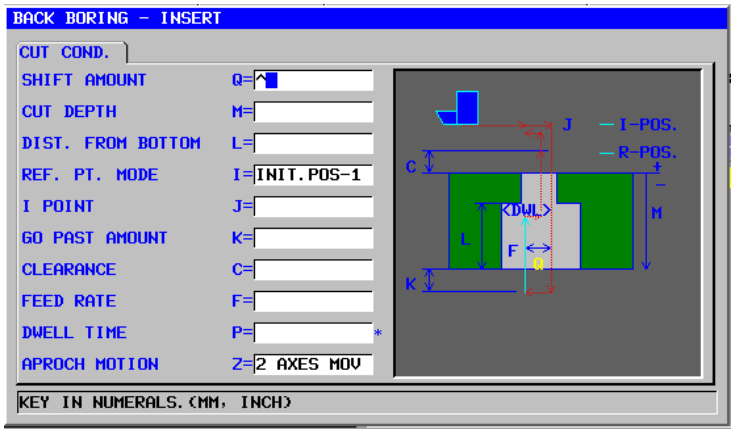
CUT COND.		
Data item		**Meaning**
Q	SHIFT AMOUNT	Shift amount (radius value) at the hole bottom after spindle orientation (COPY)
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REAF.]: All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



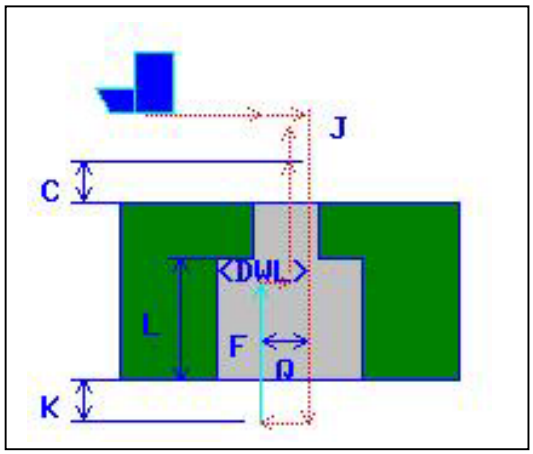
<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> Move the tool to the cutting end position at the cutting feedrate (F).
<3> The tool retracts to the position "cut end position + clearance (Ut) along the tool axis" at the feedrate (Fut) specified for clearance along the tool axis.
<4> Spindle orientation is performed.
<5> The tool retracts to the position "current position + clearance (Ur) in the tool radius direction" at the feedrate (Fur) specified for clearance in the tool radius direction.
<6> Move the tool to the position "cutting start position + clearance (C)" at the returning feedrate (Fr).

Back Boring: G1006



CUT COND.		
Data item		**Meaning**
Q	SHIFT AMOUNT	Shift amount (radius value) at the hole bottom after spindle orientation (COPY)
M	CUT DEPTH	Cut depth (radius value, negative value)
L	DIST. FROM BOTTOM	Distance (radius value) at the hole bottom in the lifting direction
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
K*	GO PAST AMOUNT	Go-past amount at the hole bottom. If omitted, 0 is assumed. (radius value, positive value) (COPY)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom after lift machining. If omitted, 0 is assumed. (units of seconds, positive value) (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

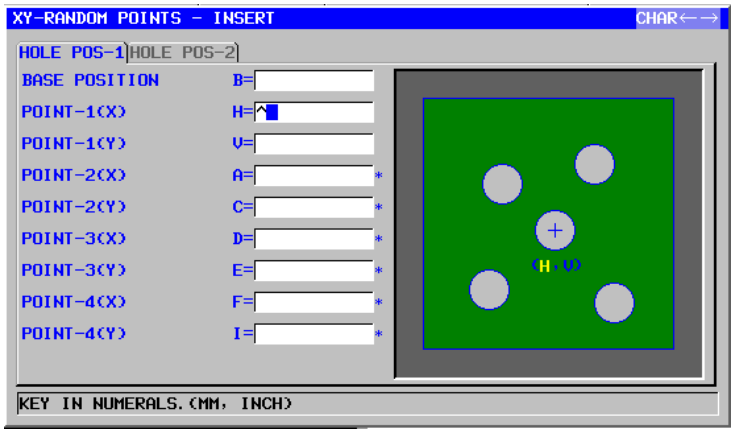
- **Tool path**



<1> Move the tool to the position "cutting start position + clearance (C)" in rapid traverse.
<2> The tool is shifted away from the tool tip.
<3> The tool moves to the bottom of the hole (R point) by rapid traverse.
<4> The tool returns by a shift amount toward the tool tip.
<5> The spindle is turned in the normal direction to cut in to the position "K + L" along the tool axis at the feedrate (F) specified for cut-in.
<6> The spindle is stopped.
<7> After being shifted away from the tool tip, the tool is pulled out from the hole.

1.1.2 Hole Position Block (XY Plane)

Random Points: G1210

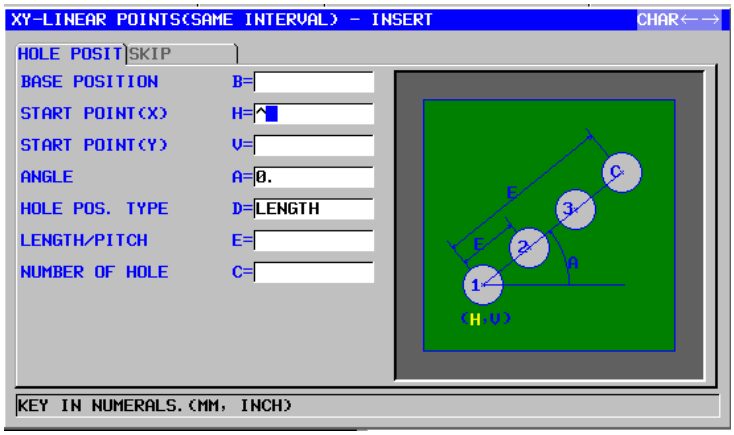


HOLE POS-1		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	POINT-1 (X)	X coordinate of the first hole
V	POINT-1 (Y)	Y coordinate of the first hole
A*	POINT-2 (X)	X coordinate of the second hole
C*	POINT-2 (Y)	Y coordinate of the second hole
D*	POINT-3 (X)	X coordinate of the third hole
E*	POINT-3 (Y)	Y coordinate of the third hole
F*	POINT-4 (X)	X coordinate of the fourth hole
I*	POINT-4 (Y)	Y coordinate of the fourth hole

HOLE POS-2		
Data item		**Meaning**
J*	POINT-5 (X)	X coordinate of the fifth hole
K*	POINT-5 (Y)	Y coordinate of the fifth hole
M*	POINT-6 (X)	X coordinate of the sixth hole
P*	POINT-6 (Y)	Y coordinate of the sixth hole
Q*	POINT-7 (X)	X coordinate of the seventh hole
R*	POINT-7 (Y)	Y coordinate of the seventh hole
S*	POINT-8 (X)	X coordinate of the eighth hole
T*	POINT-8 (Y)	Y coordinate of the eighth hole

NOTE
It is unnecessary to enter values for every hole position. If entered, however, both X and Y coordinates in a pair must be entered for a hole position.

Linear Points (Same Interval): G1211

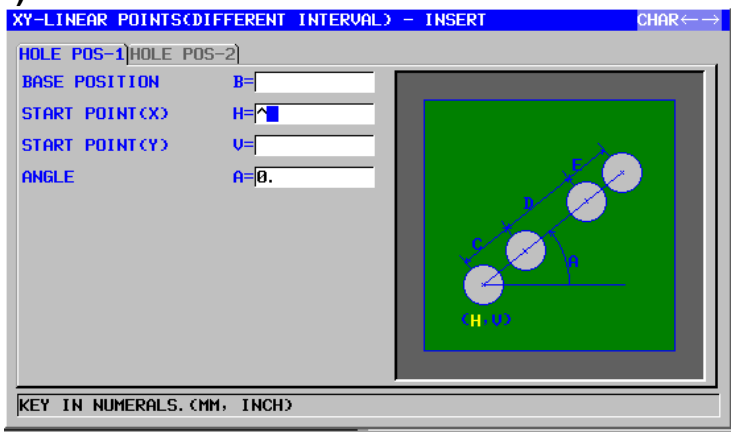


HOLE POSIT		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	START POINT (X)	X coordinate of the start point (first hole) of a straight line
V	START POINT (Y)	Y coordinate of the start point (first hole) of a straight line
A	ANGLE	Angle of a straight line from the X-axis (initial value = 0)
D	HOLE POS.TYPE	[LENGTH] : Specified with the distance between the first and last holes, and the number of holes. [PITCH] : Specified with the distance between two adjacent holes, and the number of holes.
E	LENGTH / PITCH	Length : Distance between the first and last holes (if [LENGTH] is selected for item D) Pitch : Distance between two adjacent holes (if [PITCH] is selected for item D)
C	NUMBER OF HOLE	Number of holes

SKIP		
Data item		**Meaning**
F*	OMITTING POINT 1	Point where no hole is to be made (1)
I*	OMITTING POINT 2	Point where no hole is to be made (2)
J*	OMITTING POINT 3	Point where no hole is to be made (3)
K*	OMITTING POINT 4	Point where no hole is to be made (4)

NOTE
It is unnecessary to enter a value for any item of an omitting point.

Linear Points (Different Interval): G1212

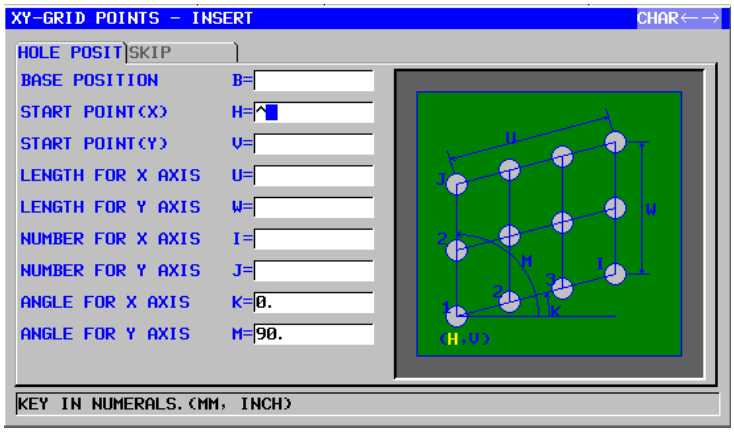


HOLE POS-1		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	START POINT (X)	X coordinate of the start point (first hole) of a straight line
V	START POINT (Y)	Y coordinate of the start point (first hole) of a straight line
A	ANGLE	Angle of a straight line from the X-axis (initial value = 0)

HOLE POS-2		
Data item		**Meaning**
C	PITCH WIDTH-1	Distance between the first and second holes (positive or negative value)
D	PITCH WIDTH-2	Distance between the second and third holes (positive or negative value)
E*	PITCH WIDTH-3	Distance between the third and fourth holes (positive or negative value)
F*	PITCH WIDTH-4	Distance between the fourth and fifth holes (positive or negative value)
I*	PITCH WIDTH-5	Distance between the fifth and sixth holes (positive or negative value)
J*	PITCH WIDTH-6	Distance between the sixth and seventh holes (positive or negative value)
K*	PITCH WIDTH-7	Distance between the seventh and eighth holes (positive or negative value)
M*	PITCH WIDTH-8	Distance between the eighth and ninth holes (positive or negative value)
P*	PITCH WIDTH-9	Distance between the ninth and tenth holes (positive or negative value)
Q*	PITCH WIDTH-10	Distance between the tenth and eleventh holes (positive or negative value)

NOTE
It is unnecessary to enter a value for every pitch width data item.

Grid Points: G1213

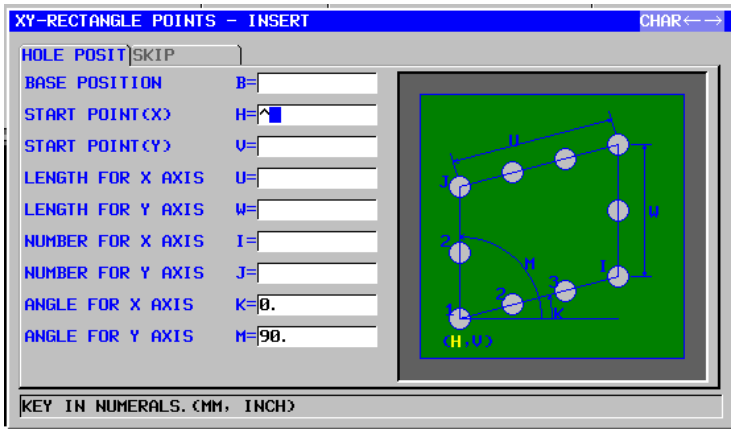


HOLE POSIT		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	START POINT (X)	X coordinate of the start point (first hole) of a straight line
V	START POINT (Y)	Y coordinate of the start point (first hole) of a straight line
U	LENGTH FOR X AXIS	Length of the first side of a grid (positive value)
W	LENGTH FOR Y AXIS	Length of the second side of a grid (positive value)
I	NUMBER FOR X AXIS	Number of holes on the first side of a grid (positive value)
J	NUMBER FOR Y AXIS	Number of holes on the second side of a grid (positive value)
K	ANGLE FOR X AXIS	Angle of the first side of a grid from the X-axis (initial value = 0)
M	ANGLE FOR Y AXIS	Angle of the second side of a grid from the X-axis (initial value = 90)

SKIP		
Data item		**Meaning**
A*	OMITTING POINT 1	Point where no hole is to be made (1)
C*	OMITTING POINT 2	Point where no hole is to be made (2)
D*	OMITTING POINT 3	Point where no hole is to be made (3)
E*	OMITTING POINT 4	Point where no hole is to be made (4)

NOTE
It is unnecessary to enter a value for any item of an omitting point.

Rectangle Points: G1214

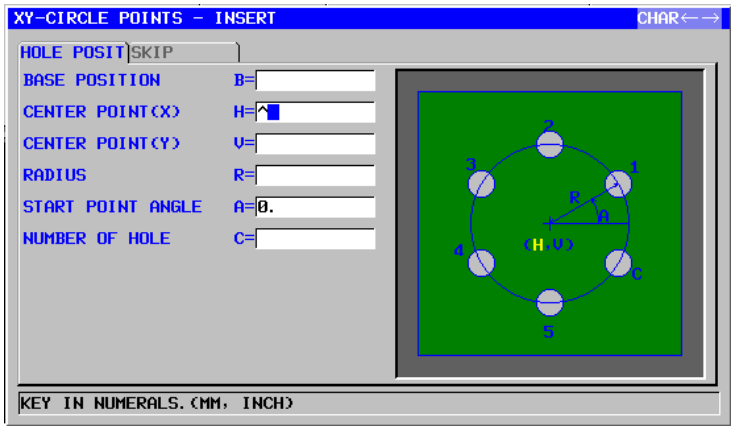


HOLE POSIT		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	START POINT (X)	X coordinate of the start point (first hole) of a straight line
V	START POINT (Y)	Y coordinate of the start point (first hole) of a straight line
U	LENGTH FOR X AXIS	Length of the first side of a rectangle (positive value)
W	LENGTH FOR Y AXIS	Length of the second side of a rectangle (positive value)
I	NUMBER FOR X AXIS	Number of holes on the first side of a rectangle (positive value)
J	NUMBER FOR Y AXIS	Number of holes on the second side of a rectangle (positive value)
K	ANGLE FOR X AXIS	Angle of the first side of a rectangle from the X-axis (initial value = 0)
M	ANGLE FOR Y AXIS	Angle of the second side of a rectangle from the X-axis (initial value = 90)

SKIP		
Data item		**Meaning**
A*	OMITTING POINT 1	Point where no hole is to be made (1)
C*	OMITTING POINT 2	Point where no hole is to be made (2)
D*	OMITTING POINT 3	Point where no hole is to be made (3)
E*	OMITTING POINT 4	Point where no hole is to be made (4)

NOTE
It is unnecessary to enter a value for any item of an omitting point.

Circle Points: G1215

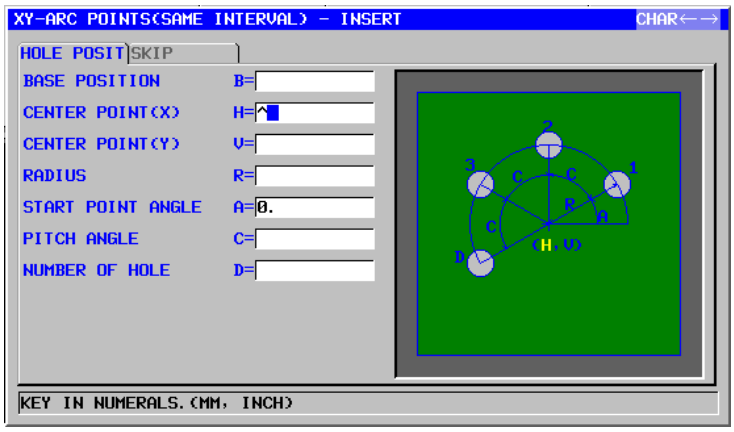


HOLE POINTS		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	CENTER POINT (X)	X coordinate of the center of a circle
V	CENTER POINT (Y)	Y coordinate of the center of a circle
R	RADIUS	Radius of a circle (positive value)
A	START POINT ANGLE	Central angle of the first hole from the X-axis (positive or negative value) (initial value = 0)
C	NUMBER OF HOLE	Number of holes to be made (positive value)

SKIP		
Data item		**Meaning**
D*	OMITTING POINT 1	Point where no hole is to be made (1)
E*	OMITTING POINT 2	Point where no hole is to be made (2)
F*	OMITTING POINT 3	Point where no hole is to be made (3)
I*	OMITTING POINT 4	Point where no hole is to be made (4)

NOTE
It is unnecessary to enter a value for any item of an omitting point.

Arc Points (Same Interval): G1216

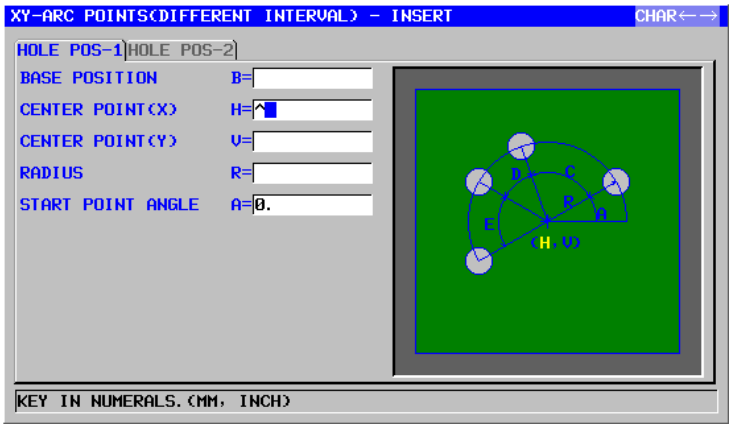


HOLE POSIT		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	CENTER POINT (X)	X coordinate of the center of an arc
V	CENTER POINT (Y)	Y coordinate of the center of an arc
R	RADIUS	Radius of an arc (positive value)
A	START POINT ANGLE	Central angle of the first hole from the X-axis (positive or negative value) (initial value = 0)
C	PITCH ANGLE	Central angle between two adjacent holes (positive or negative value)
D	NUMBER OF HOLE	Number of holes to be made (positive value)

SKIP		
Data item		**Meaning**
E*	OMITTING POINT 1	Point where no hole is to be made (1)
F*	OMITTING POINT 2	Point where no hole is to be made (2)
I*	OMITTING POINT 3	Point where no hole is to be made (3)
J*	OMITTING POINT 4	Point where no hole is to be made (4)

NOTE
It is unnecessary to enter a value for any item of an omitting point.

Arc Points (Different Interval): G1217



HOLE POS-1		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	CENTER POINT (X)	X coordinate of the center of an arc
V	CENTER POINT (Y)	Y coordinate of the center of an arc
R	RADIUS	Radius of an arc (positive value)
A	START POINT ANGLE	Central angle of the first hole from the X-axis (positive or negative value) (initial value = 0)

HOLE POS-2		
Data item		**Meaning**
C*	PITCH ANGLE-1	Central angle between the first and second holes (positive or negative value)
D*	PITCH ANGLE-2	Central angle between the second and third holes (positive or negative value)
E*	PITCH ANGLE-3	Central angle between the third and fourth holes (positive or negative value)
F*	PITCH ANGLE-4	Central angle between the fourth and fifth holes (positive or negative value)
I*	PITCH ANGLE-5	Central angle between the fifth and sixth holes (positive or negative value)
J*	PITCH ANGLE-6	Central angle between the sixth and seventh holes (positive or negative value)
K*	PITCH ANGLE-7	Central angle between the seventh and eighth holes (positive or negative value)
M*	PITCH ANGLE-8	Central angle between the eighth and ninth holes (positive or negative value)
P*	PITCH ANGLE-9	Central angle between the ninth and tenth holes (positive or negative value)
Q*	PITCH ANGLE-10	Central angle between the tenth and eleventh holes (positive or negative value)

1.1.3 Hole Position Block (YZ Plane)

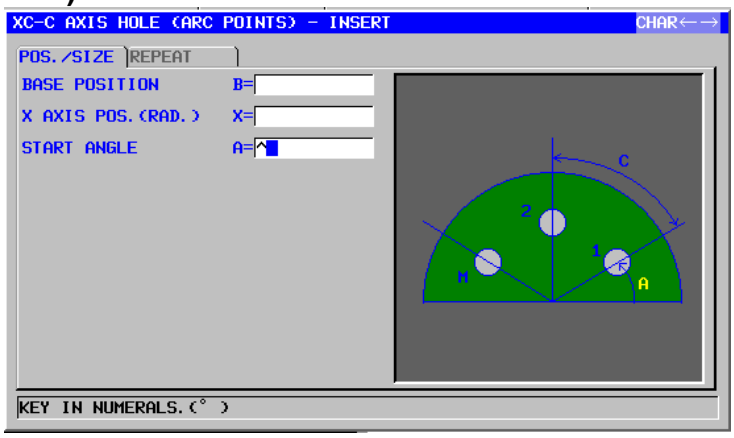
The same hole position block types as for the XY plane explained in the previous subsection are available for the YZ plane. They are provided with the following G codes.
The data to be set for the YZ plane is the same as for the XY plane except that the XY plane is changed to the YZ plane and that the direction in which the tool moves to cut is changed from the Z-axis to the X-axis.

Random Points	: G1310
Linear Points (Same Interval)	: G1311
Linear Points (Different Interval)	: G1312
Grid Points	: G1313
Rectangle Points	: G1314
Circle Points	: G1315
Arc Points (Same Interval)	: G1316
Arc Points (Different Interval)	: G1317

1.1.4 Hole Position Block (XC Plane and End Face)

A menu for selecting a hole position block in which the C-axis is used in making holes is displayed by selecting the "C-axis Figure" tab from the milling figure menu, using the ← and → cursor keys.

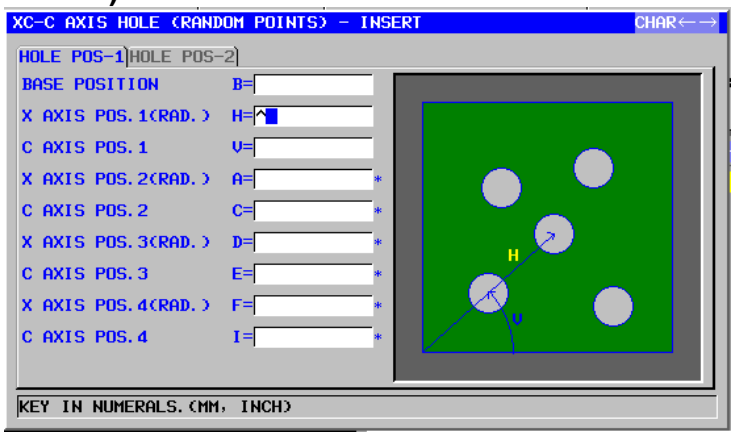
C Axis Hole on Face (Circle Points): G1572



POS / SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
X	X AXIS POS.(RAD.)	X coordinate of a hole position (common to all holes)
A	START ANGLE	Central angle of the first hole form the C0 axis (positive or negative value)

REPEAT		
Data item		**Meaning**
C	PITCH ANGLE	Amount of movement between two holes along the C-axis (positive or negative value)
M	NUMBER OF HOLE	Number of holes to be made (positive value)

C Axis Hole on Face (Random Points): G1573



HOLE POS-1		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the workpiece surface
H	X AXIS POS.1(RAD.)	X coordinate of the first hole (radius value)
V	C AXIS POS.1	C coordinate of the first hole
A*	X AXIS POS.2(RAD.)	X coordinate of the second hole (radius value)
C*	C AXIS POS.2	C coordinate of the second hole
D*	X AXIS POS.3(RAD.)	X coordinate of the third hole (radius value)
E*	C AXIS POS.3	C coordinate of the third hole
F*	X AXIS POS.4(RAD.)	X coordinate of the fourth hole (radius value)
I*	C AXIS POS.4	C coordinate of the fourth hole

HOLE POS-2		
Data item		**Meaning**
J*	X AXIS POS.5(RAD.)	X coordinate of the fifth hole (radius value)
K*	C AXIS POS.5	C coordinate of the fifth hole
M*	X AXIS POS.6(RAD.)	X coordinate of the sixth hole (radius value)
P*	C AXIS POS.6	C coordinate of the sixth hole
Q*	X AXIS POS.7(RAD.)	X coordinate of the seventh hole (radius value)
R*	C AXIS POS.7	C coordinate of the seventh hole
S*	X AXIS POS.8(RAD.)	X coordinate of the eighth hole (radius value)
T*	C AXIS POS.8	C coordinate of the eighth hole

1.1.5 Hole Position Block (ZC Plane and Cylindrical Surface)

The same hole position block types as for the XC plane explained in the previous subsection are available for the ZC plane. They are provided with the following G codes.
The data to be set for the ZC plane is the same as for the XC plane except that the XC plane (blank end face) is changed to the ZC plane (blank cylindrical surface) and that the direction in which the tool moves to cut is changed from the Z-axis to the X-axis.

Holes on a cylindrical surface (ZC) figure along the C-axis - Circle Points: G1672
Holes on a cylindrical surface (ZC) figure along the C-axis - Random Points: G1673

1.1.6 Hole Position Block (XA Plane and Cylindrical Surface)

The same hole position block types as for the ZC plane explained in the previous subsection are available for the XA plane. They are provided with the following G codes.
The data to be set for the XA plane is the same as for the ZC plane except that the ZC plane (blank end face) is changed to the XA plane (blank cylindrical surface) and that the direction in which the tool moves to cut is changed from the X-axis to the Z-axis.

Holes on a cylindrical surface (XA) figure along the A-axis - Circle Points: G1772
Holes on a cylindrical surface (XA) figure along the A-axis - Random Points: G1773

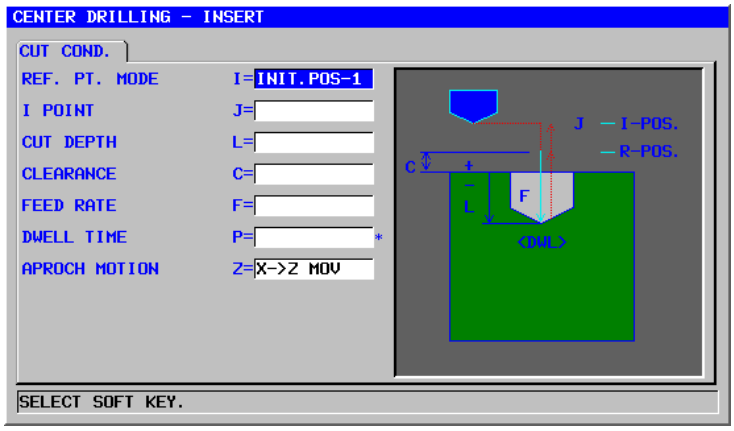
NOTE
The following parameter setting is necessary to execute XA plane cycle.
- No.27003#2=1, 27003#1=0 and 27003#0=0

1.2 HOLE MACHINING BY TURNING (WITH THE TOOL ROTATED)

1.2.1 Machining Type Blocks for Hole Machining by Turning (with the Tool Rotated)

NOTE
1 Hole machining by turning (with the tool rotated) is enabled when bit 1 of parameter No. 27000 = 1.
2 The hole position blocks for hole machining by turning (with the tool rotated) are the same as those for hole machining by milling. See the previous section (Hole Position Block).

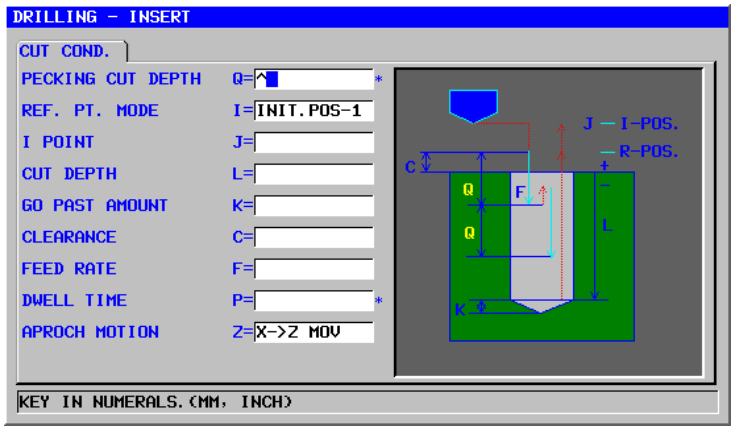
Center drilling: G1110



Cutting condition		
Data item		**Meaning**
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INITI-2]: All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom (units of seconds, positive value). (COPY)

Cutting condition		
Data item		**Meaning**
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

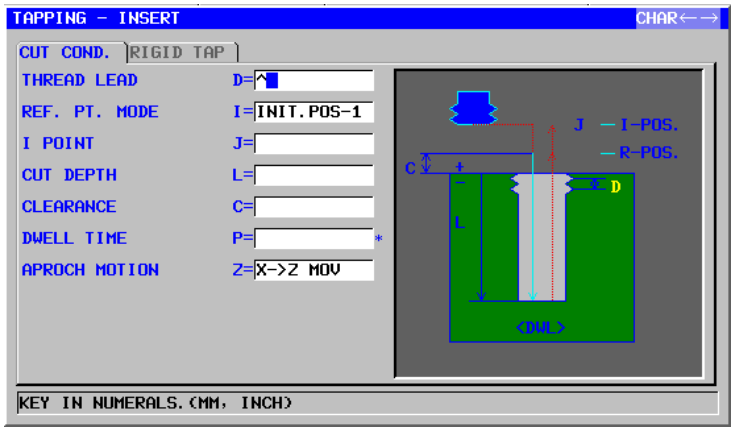
Drilling: G1111



Cutting condition		
Data item		**Meaning**
Q*	PECKING CUT DEPTH	Depth of cut made by one cut (radius value, positive value) (COPY) (Note)
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
K	GO PAST AMOUNT	Length of the incomplete hole portion at the tool tip (radius value, positive value) (COPY)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom (units of seconds, positive value). (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

NOTE
The system refers to the parameter No.5114 as the return amount. Therefore, please set No.5114 to suitable value before machining.

Tapping: G1112



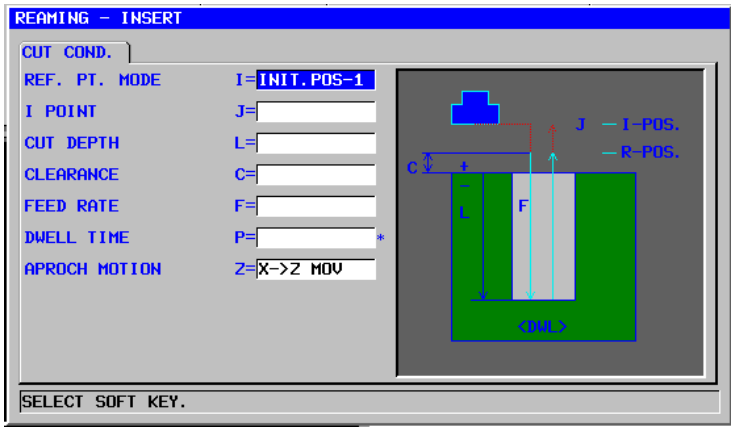
Cutting condition		
Data item		**Meaning**
D	THREAD LEAD	Tapping tool lead (radius value, positive value) (COPY)
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom (units of seconds, positive value). (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

Rigid tapping		
Data item		**Meaning**
R	TAP TYPE	[FLOAT] : Specifies the float tapping. (initial value) [RIGID] : Specifies the rigid tapping.
S	SPINDLE SPEED	Spindle speed (min^{-1}) (Note)

NOTE

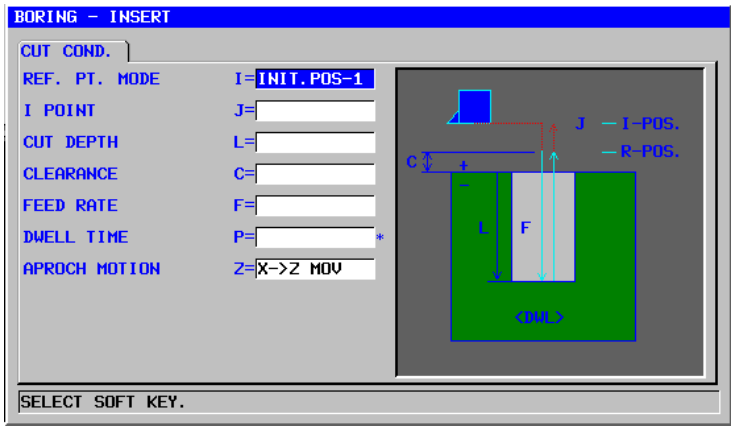
1 When you use rigid tapping mode M code command (No.5200#0=0), the system refers to No.5210 or No.5212 as the value of M code. Therefore, please set No.5210 or No.5212 to suitable value before machining.

Reaming: G1113



Cutting condition		
Data item		**Meaning**
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom (units of seconds, positive value). (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

Boring: G1114

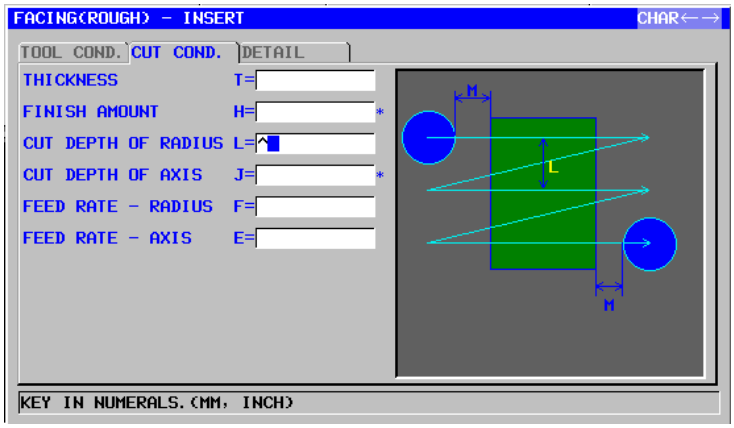


Cutting condition		
Data item		**Meaning**
I	REF.PT.MODE	[INIT-1] : An R position return is made in moving between holes. Finally, a return is made to the I point. (initial value) [INIT-2] : All movements between holes, including the last return, are made as an I point return. [REF.] : All movements between holes, including the last return, are made as an R position return.
J	I POINT	Coordinate of the I point (COPY)
L	CUT DEPTH	Cut depth (radius value, negative value)
C	CLEARANCE	Distance between the workpiece surface and R position (radius value, positive value)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the hole bottom (units of seconds, positive value). (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

1.3 FACING

1.3.1 Machining Type Blocks for Facing

Rough: G1020



TOOL COND.		
Data item		Meaning
D	TOOL DIAMETER	Face mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

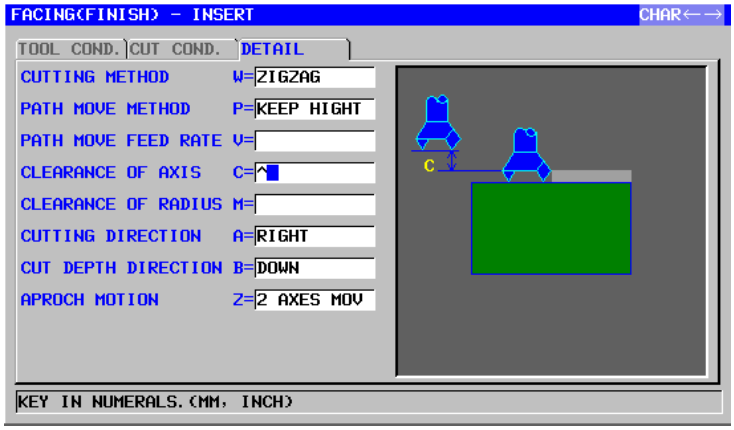
CUT COND.		
Data item		Meaning
T*	THICKNESS	Face mill diameter
H*	FINISH AMOUNT	Finishing allowance in facing
L	CUT DEPTH OF RADIUS	Depth of cut in the tool radius direction to the next cutting path
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation
F	FEED RATE – RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE – AXIS	Feedrate applicable when cutting is performed in the tool axis direction

NOTE
If the parameter 27030#1=1, only the menu of perpendicular directions to "CUTTING DIRECTION" can have been displayed during in case of selecting "CUT SHIFT DIRECTION". Thus, the extra selection can be removed.

DETAIL		
Data item		**Meaning**
I	INITIAL FEED OVERRIDE	Feedrate override value for the first cutting. The initial value is 100 (1 to 200, positive value).
W	CUTTING METHOD	[SINGLE] : Cutting in the tool radius direction is always performed in the same direction. [ZIGZAG] : Cutting in the tool radius direction is performed back and forth. (COPY)
P	PATH MOVE METHOD	[PULL] : Retracts to point R before moving to the start point of the next cutting path (in the tool axis direction). [KEEP] : Moves to the start point of the next cutting path directly without retracting to point R. Remark1) This data item is indicated only when [ZIGZAG] is selected for CUTTING METHOD. Remark2) This data item is enabled when the parameter **No.27030#0(FC0) = 0**. **Remark3) In case that the movement is done by 1 axis, tool is kept even if [PULL] is specified.** **Remark4) If [KEEP] is specified, an end point of previous cutting motion is adjusted in order to enable a tool to move with 1 axis to next cutting start point** (COPY)
V	PATH MOVE FEED RATE	Feedrate applicable when the tool moves to the start point of the next cutting path. **In case that the feedrate is set to 0, the tool moves at a rapid traverse rate**. Remark1) This data item is indicated only when [ZIGZAG] is selected for CUTTING METHOD. Remark2) This data item is enabled when the parameter **No.27030#0(FC0) = 0**. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
M	CLERANCE OF RADIUS	Distance between the end of a blank being machined and the end of the tool placed at the retract position (radius value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

DETAIL		
Data item		**Meaning**
A	CUTTING DIRECTION	[RIGHT] : Performs cutting rightward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed rightward. [LEFT] : Performs cutting leftward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed leftward. [UP] : Performs cutting upward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed upward. [DOWN] : Performs cutting downward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed downward. Remark) The actual cutting direction is determined by the coordinate axis indicated in the illustration. (COPY)
B	CUT DEPTH DIRECTION	[RIGHT] : Performs cutting while shifting the cutting path rightward as indicated in the illustration. [LEFT] : Performs cutting while shifting the cutting path leftward as indicated in the illustration. [UP] : Performs cutting while shifting the cutting path upward as indicated in the illustration. [DOWN] : Performs cutting while shifting the cutting path downward as indicated in the illustration. Remark) The actual cutting direction is determined by the coordinate axis indicated in the illustration. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

Finish: G1021



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	Face mill diameter

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CUT COND.		
Data item		**Meaning**
L	CUT DEPTH OF RADIUS	Depth of cut in the tool radius direction to the next cutting path
F	FEED RATE – RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE – AXIS	Feedrate applicable when cutting is performed in the tool axis direction

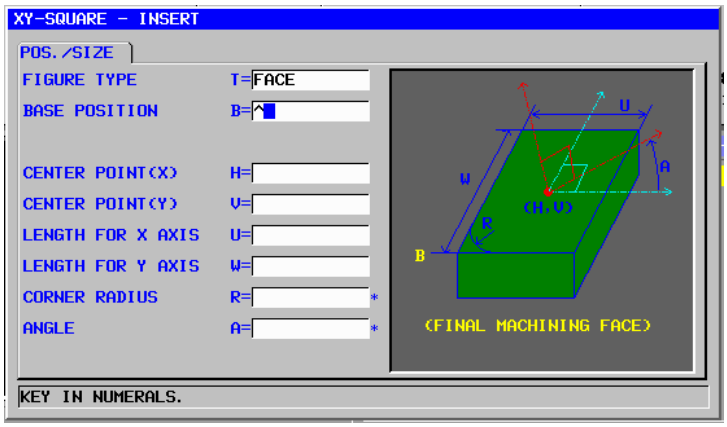
DETAIL		
Data item		**Meaning**
W	CUTTING METHOD	[SINGLE] : Cutting in the tool radius direction is always performed in the same direction. [ZIGZAG] : Cutting in the tool radius direction is performed back and forth. (COPY)
P	PATH MOVE METHOD	[PULL] : Retracts to point R before moving to the start point of the next cutting path (in the tool axis direction). [KEEP] : Moves to the start point of the next cutting path directly without retracting to point R. Remark1) This data item is indicated only when [ZIGZAG] is selected for CUTTING METHOD, and [KEEP] is set automatically. Remark2) This data item is enabled when the parameter No.27030#0(FC0) = 1. (COPY)
V	PATH MOVE FEED RATE	Feedrate applicable when the tool moves to the start point of the next cutting path. Initially, the feedrate is set to 0, allowing the tool to move at a rapid traverse rate. (COPY) Remark1) This data item is indicated only when [ZIGZAG] is selected for CUTTING METHOD. Remark2) This data item is enabled when the parameter No.27030#0(FC0) = 1.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
M	CLERANCE OF RADIUS	Distance between the end of a blank being machined and the end of the tool placed at the retract position (radius value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

DETAIL		
Data item		**Meaning**
A	CUTTING DIRECTION	[RIGHT] : Performs cutting rightward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed rightward. [LEFT] : Performs cutting leftward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed leftward. [UP] : Performs cutting upward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed upward. [DOWN] : Performs cutting downward as indicated in the illustration. When both directions are selected, cutting for the first cutting path is performed downward. Remark) The actual cutting direction is determined by the coordinate axis indicated in the illustration. (COPY)
B	CUT DEPTH DIRECTION	[RIGHT] : Performs cutting while shifting the cutting path rightward as indicated in the illustration. [LEFT] : Performs cutting while shifting the cutting path leftward as indicated in the illustration. [UP] : Performs cutting while shifting the cutting path upward as indicated in the illustration. [DOWN] : Performs cutting while shifting the cutting path downward as indicated in the illustration. Remark) The actual cutting direction is determined by the coordinate axis indicated in the illustration. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

NOTE
When a cutting direction is selected, only those menu items that are perpendicular to the cutting direction can be displayed to delete unnecessary options by setting bit 1 of parameter No. 27030 to 1.

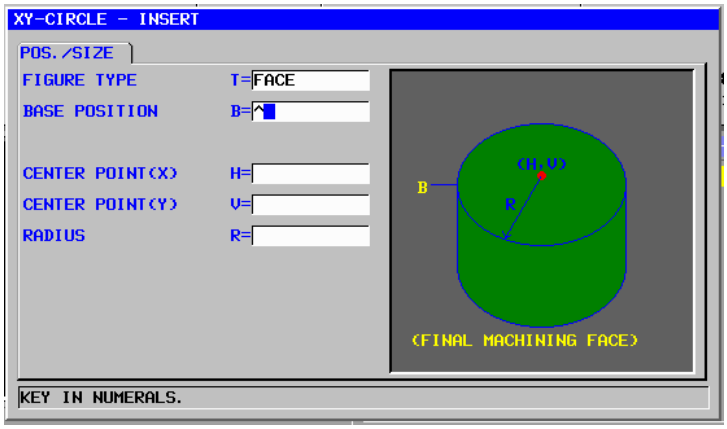
1.3.2 Fixed Form Figure Blocks for Facing (XY Plane)

Square: G1220 (XY plane)



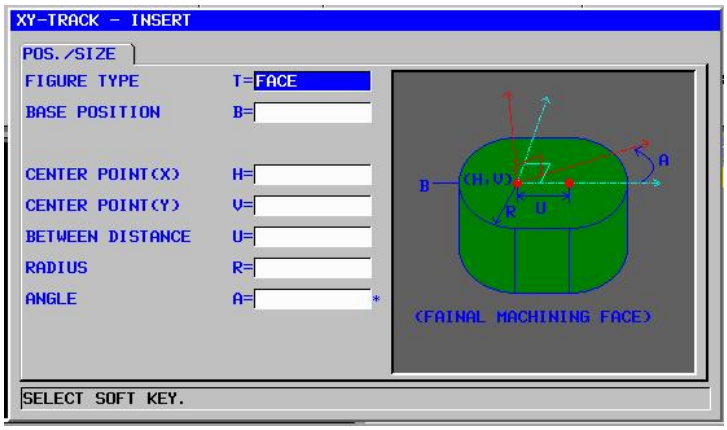
POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When facing is selected as machining type, be sure to select [FACE].
B	BASE POSITION	Z coordinate of the final surface for facing (in the tool axis direction)
H	CENTER POINT (X)	X coordinate of the center position of a rectangular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a rectangular figure
U	LEBGTH FOR X AXIS	Length of the side in the X-axis direction (radius value, positive value)
W	LENGTH FOR Y AXIS	Length of the side in the Y-axis direction (radius value, positive value)
R*	CORNER RADIUS	Radius for corner rounding (positive value)
A*	ANGLE	Inclination angle of a rectangular figure relative to the X-axis (positive or negative value)

Circle: G1221 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When facing is selected as machining type, be sure to select [FACE].
B	BASE POSITION	Z coordinate of the final surface for facing (in the tool axis direction)
H	CENTER POINT (X)	X coordinate of the center position of a circular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a circular figure
R	RADIUS	Radius of a circular figure (radius value, positive value)

Track: G1222 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When facing is selected as machining type, be sure to select [FACE].
B	BASE POSITION	Z coordinate of the final surface for facing (in the tool axis direction)
H	CENTER POINT (X)	X coordinate of the center position of the left semicircle
V	CENTER POINT (Y)	Y coordinate of the center position of the left semicircle
U	BETWEEN DISTANCE	Distance between the centers of the right and left semicircles (radius value, positive value)
R	RADIUS	Radius of the left and right semicircles (radius value, positive value)
A*	ANGLE	Inclination angle of a track figure relative to the X-axis. The blank is regarded as 0 degrees. (positive or negative value)

1.3.3 Fixed Form Figure Blocks for Facing (YZ Plane, XC Plane)

The same fixed-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane and the XC plane (polar coordinate interpolation plane). They are provided with the following G codes.
The data to be set for the YZ and XC planes is the same as for the XY plane except that the XY plane is changed to the YZ or XC plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ plane) or the Z-axis (XC plane).

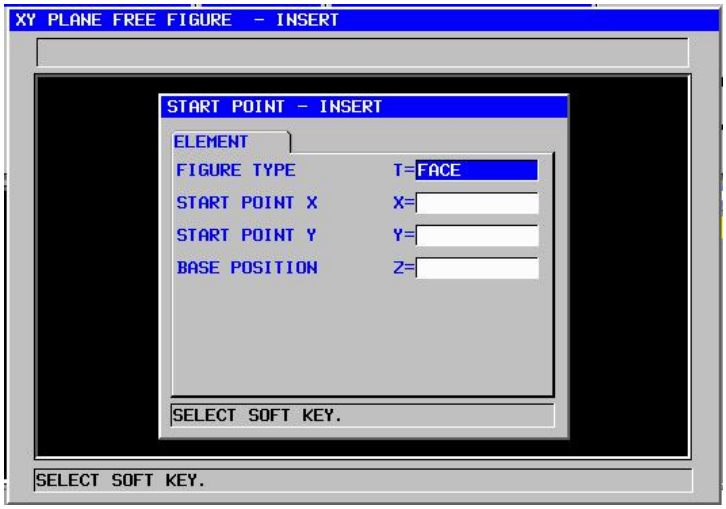
Square : G1320 (YZ plane)
Circle : G1321 (YZ plane)
Track : G1322 (YZ plane)
Square : G1520 (XC plane, polar coordinate interpolation plane)
Circle : G1521 (XC plane, polar coordinate interpolation plane)
Track : G1522 (XC plane, polar coordinate interpolation plane)

NOTE
When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.

1.3.4 Arbitrary Figure Blocks for Facing (XY Plane)

When an arbitrary figure for facing is input, data such as a figure type and machining reference position is specified in the start point block. Other data items to be input such as a straight line and arc are exactly the same as for arbitrary figures of other machining types.
So, this subsection describes only the start point block of an arbitrary figure for facing.
For other arbitrary figures, see Chapter 5, "DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES," in Part II.

Start point: G1200 (XY plane)



ELEMENT		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring and emboss machining or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When facing is selected as machining type, be sure to select [FACE].
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
Z	BASE POSITION	Z coordinate of the final surface for facing (in the tool axis direction)

1.3.5 Arbitrary Figure Blocks for Facing (YZ Plane, XC Plane, ZC Plane, XA Plane)

The same arbitrary-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane, the XC plane (polar coordinate interpolation plane), ZC plane and XA plane (cylindrical surface). Their start points are specified with the following G codes.
The data to be set for the YZ, XC, ZC, and XA planes is the same as for the XY plane except that the XY plane is changed to the YZ, XC, ZC or XA plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ and ZC planes) or the Z-axis (XC and XA plane).

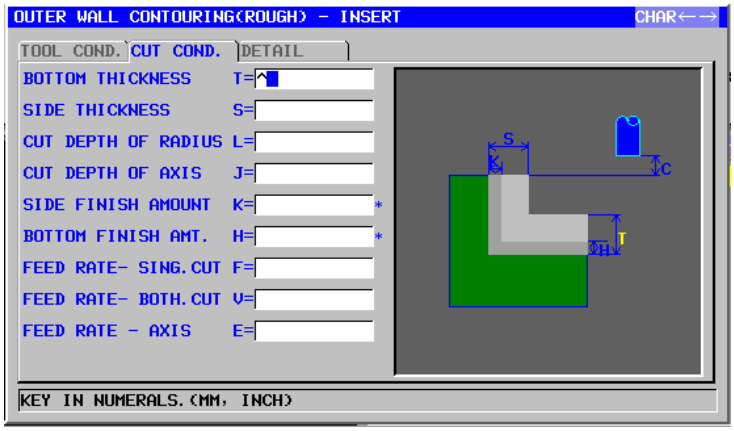
Start point : G1300 (YZ plane)
Start point : G1500 (XC plane, polar coordinate interpolation plane)
Start point : G1600 (ZC plane, plane)
Start point : G1700 (XA plane, plane)

NOTE
1 When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.
2 When performing machining (cylindrical interpolation) on the ZC and XA plane, note the following:
(When bit 3 of parameter No. 27000 = 0)
The mode needs to be switched to the cylindrical coordinate interpolation mode beforehand. Specifically, enter G07.1C (cylinder radius) before the machining type.
Enter G07.1C0 for canceling cylindrical interpolation as required.
(When bit 3 of parameter No. 27000 = 1)
G07.1C (cylinder radius) and G07.1C0 are automatically output before and after cycle machining, respectively.
3 The following parameter setting is necessary to execute XA plane cycle.
- No.27003#2=1, 27003#1=0 and 27003#0=0

1.4 CONTOURING

1.4.1 Machining Type Blocks for Contouring

Outer Wall Rough: G1060
Inner Wall Rough: G1054
Partial Rough: G1068



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

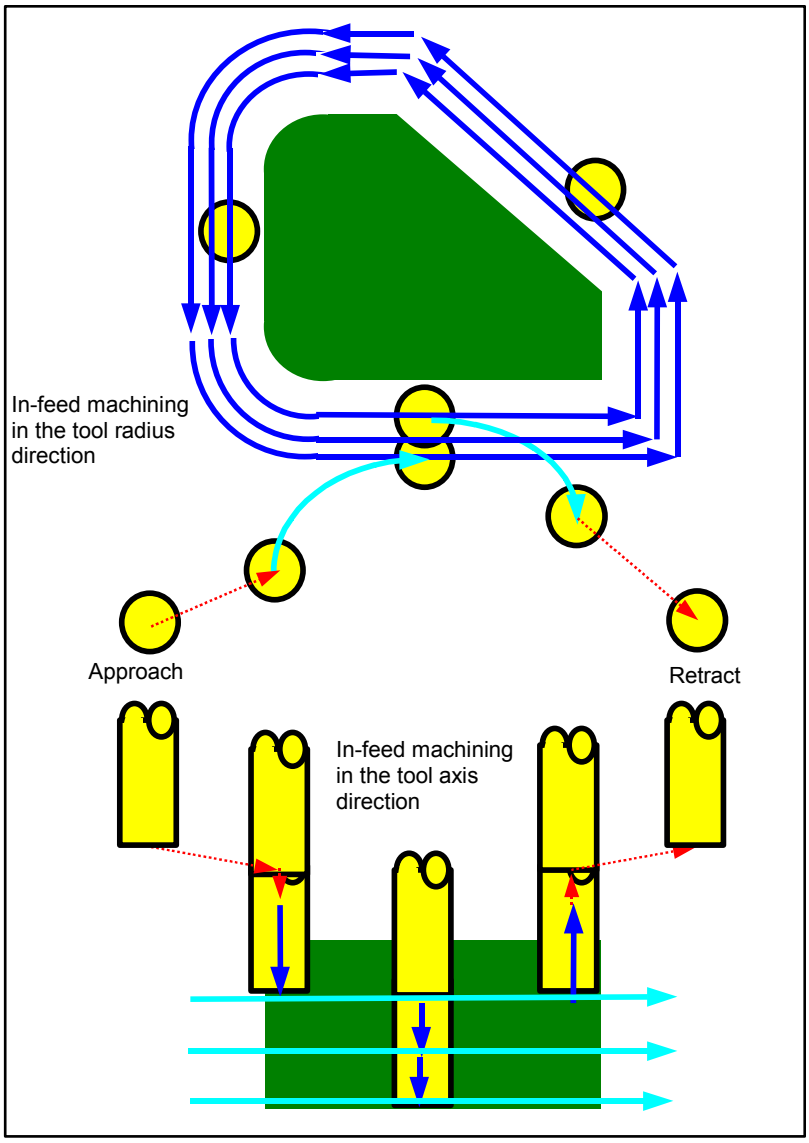
CUT COND		
Data item		**Meaning**
T	BOTTOM THICKNESS	Cutting allowance of the bottom in side face machining (radius value, positive value)
S	SIDE THICKNESS	Cutting allowance of the side face (radius value, positive value)
L	CUT DEPTH OF RADIUS	Depth of cut per side face machining operation (in the tool radius direction) (radius value, positive value)
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation (radius value, positive value) The default is (bottom surplus thickness - bottom finishing allowance).
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)

CUT COND		
Data item		**Meaning**
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE – AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
M	INITIAL FEED OVERRIDE	Feedrate override value for the first cutting. The initial value is 100 (1 to 200, positive value).
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)

DETAIL		
Data item		**Meaning**
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

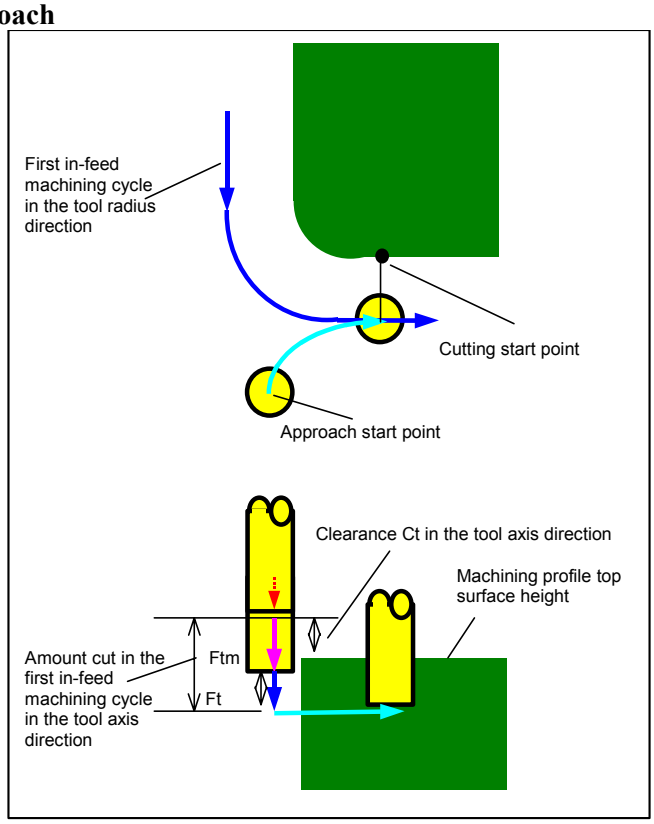
- **Tool path**



The side-face contour of a machining profile is cut off.
The following tool path is created.

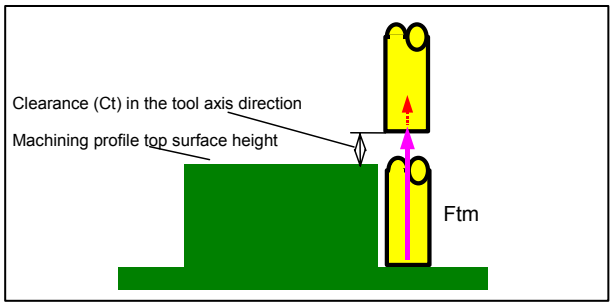
<1> The tool moves to above the approach start point.
<2> The tool moves to the height of the cutting surface.
<3> The tool cuts along the side-face contour of the machining profile.
The tool performs in-feed machining in the tool radius direction until the cutting allowance in the tool radius direction is removed.
<4> Steps <2> and <3> are repeated until the cutting allowance in the tool axis direction is removed.
<5> The tool retracts.

- Approach



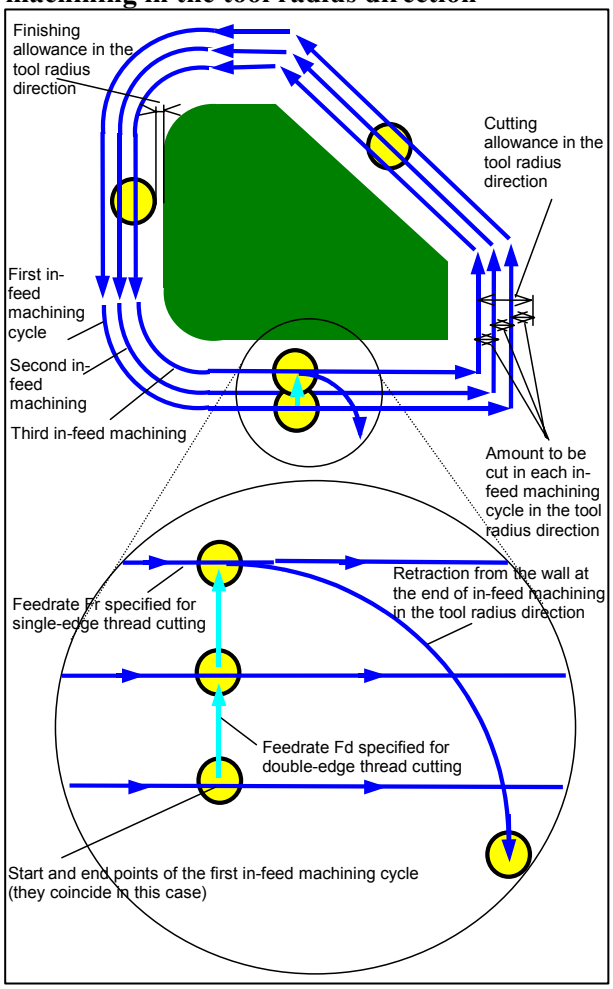
<1> The tool moves to the position "machining profile top surface height + clearance (Ct) in the tool axis direction" by rapid traverse.
<2> The tool moves to the position "amount to be cut in the first in-feed machining cycle in the tool axis direction - clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.
<3> The tool cut in to the position "amount to be cut in the first in-feed machining cycle in the tool axis direction" at the feedrate (Ft) specified for cutting in the tool axis direction.
<4> The tool approaches, in the tool radius direction, the start point of the first in-feed machining cycle in the tool radius direction.

- Retraction



<1> The tool moves from the approach end point to the position "machining profile top surface height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.

- In-feed machining in the tool radius direction



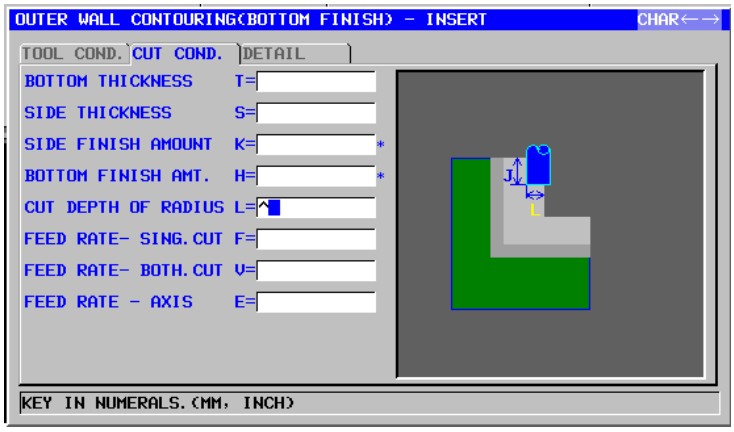
<1> The tool moves to cut along the contour from the first in-feed machining cycle start point to the end point at the feedrate (Fs) specified for single-edge thread cutting.
<2> The tool approaches, using the following method.

When the in-feed machining start point coincides with the in-feed machining end point:
The tool directly approaches the next in-feed machining start point in the normal direction at the feedrate (Fd) specified for double-edge thread cutting.

When the in-feed machining start point does not coincide with the in-feed machining end point:
The tool approaches the second in-feed machining start point.

<3> The tool moves to cut along the contour of the machining profile at the feedrate (Fs) specified for single-edge thread cutting.
<4> Steps <2> and <3> are repeated until the cutting allowance (cutting allowance in the tool radius direction - finishing allowance) is removed.
<5> The tool retracts.

Outer Wall Bottom finish : G1061
Inner Wall Bottom finish : G1065
Partial Bottom finish : G1069



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND		
Data item		**Meaning**
T	BOTTOM THICKNESS	Cutting allowance of the bottom in side face machining (radius value, positive value)
S	SIDE THICKNESS	Cutting allowance of the side face (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value) Remark) This data item is used when machining is to be performed with a small amount of cutting allowance left.
L	CUT DEPTH OF RADIUS	Depth of cut per side face machining operation (in the tool radius direction) (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.

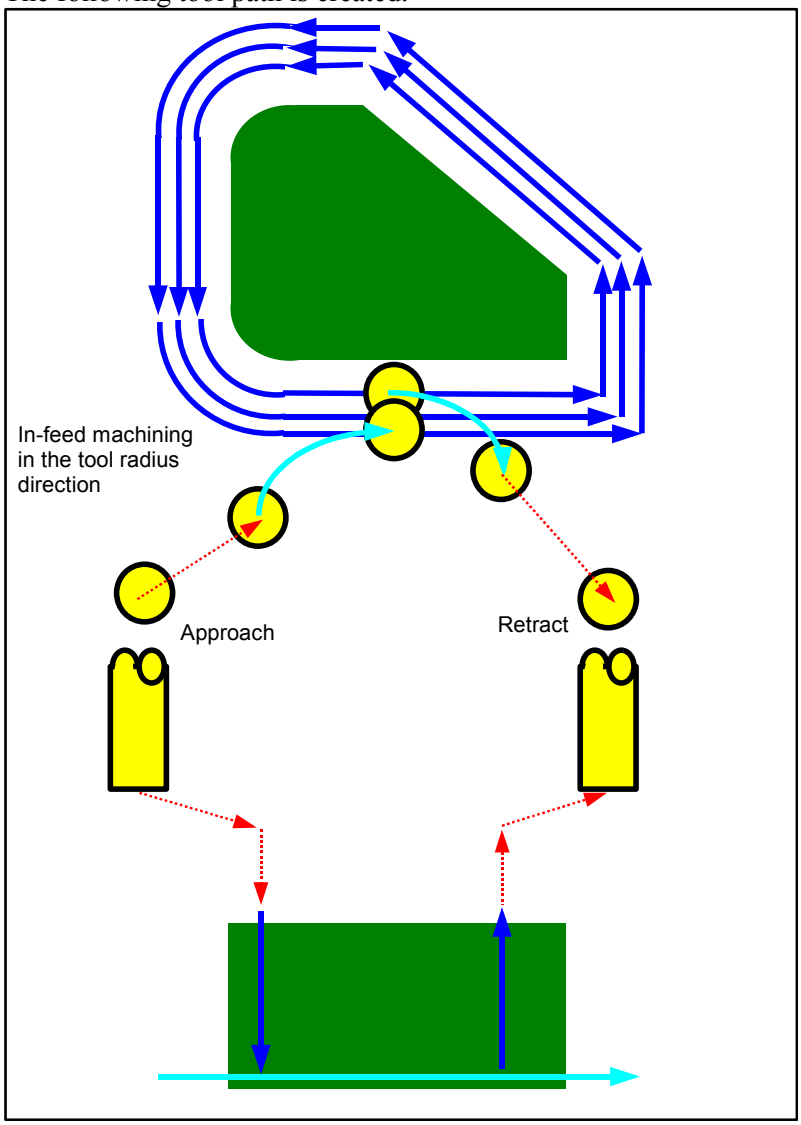
CUT COND		
Data item		**Meaning**
V	FEED RATE- BOTH CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)

DETAIL		
Data item		**Meaning**
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

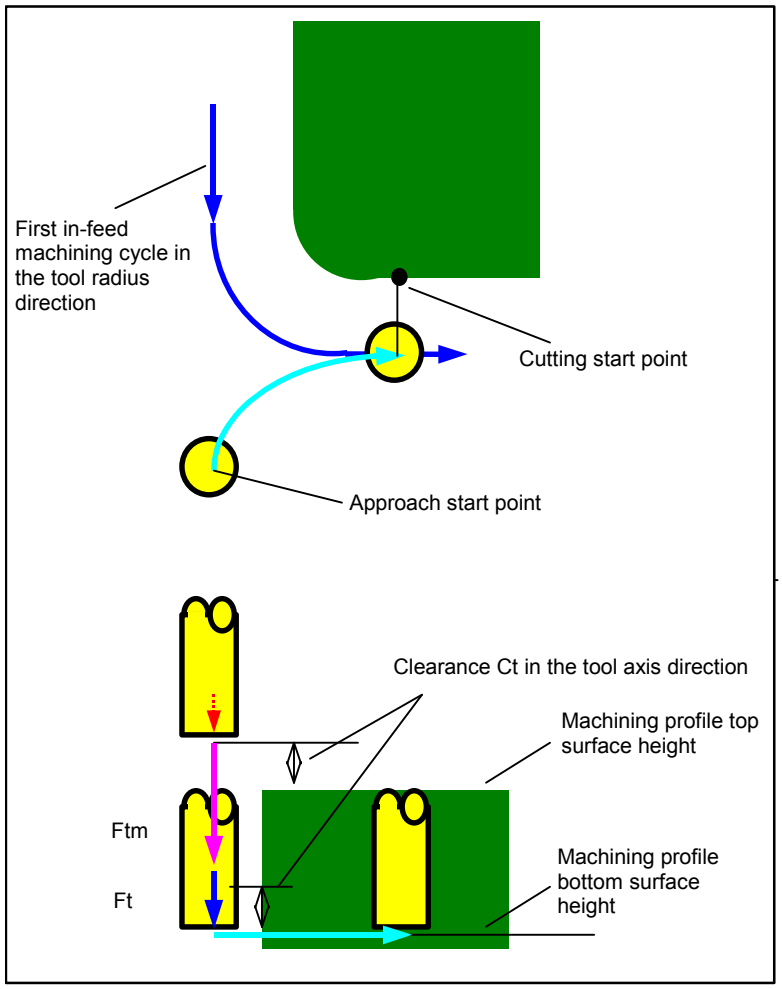
- **Tool path**

The bottom surface of the side-face contour of the machining profile is finished.
The following tool path is created.



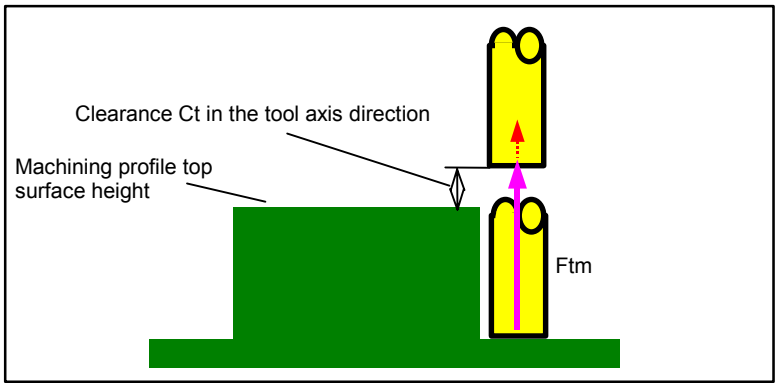
<1> The tool approaches the approach start point of the machining profile.
<2> The tool moves to the bottom surface height of the machining profile.
<3> The tool moves to cut along the side-face contour of the machining profile.
In-feed machining in the tool radius direction is performed until the cutting allowance in the tool radius direction is removed.
<4> The tool retracts.

- Approach



<1> The tool moves to the position "machining profile top surface height + clearance (Ct) in the tool axis direction" by rapid traverse.
<2> The tool moves to the position "machining profile bottom surface + cutting allowance (Vt) in the tool axis direction + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.
<3> The tool moves to the machining profile bottom surface at the feedrate (Ft) specified for cutting in the tool axis direction.
<4> The tool approaches, in the tool radius direction, the in-feed machining start point in the tool radius direction.

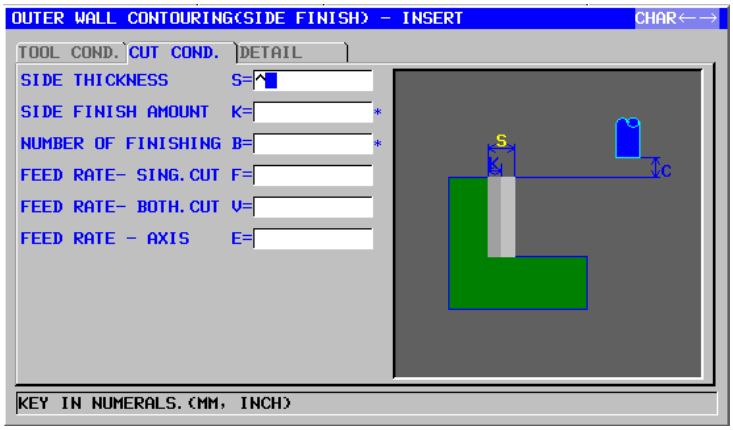
- Retraction



<1> The tool moves from the approach end point to the position "machining profile top height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.

- In-feed machining in the tool radius direction
  This movement is the same as for contouring (roughing).
  See descriptions about contouring (roughing) for details.

Outer Wall Side finish: G1062
Inner Wall Side finish : G1066
Partial Side finish : G1070



TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Inputs a cutter compensation value directly. [REF.] : Inputs a cutter compensation number to read a cutter compensation value by that number.
D	TOOL DIAMETER	End mill diameter (positive value) Remark) This item is indicated only when [INPUT] is selected for INPUT TYPE.
M	CUTTER COMP.NO.	Cutter compensation number of an end mill (positive value) Remark) This item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'
3 Tab 'TOOL COND.' is not enabled when bit 3 (CN3) of parameter No. 27040 = 1.

CUT COND.		
Data item		**Meaning**
S	SIDE THICKNESS	Cutting allowance in side face finishing (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance of the side face. The blank is regarded as 0. (radius value, positive value) Remark) This data item is used when machining is to be performed with a small amount of cutting allowance left.

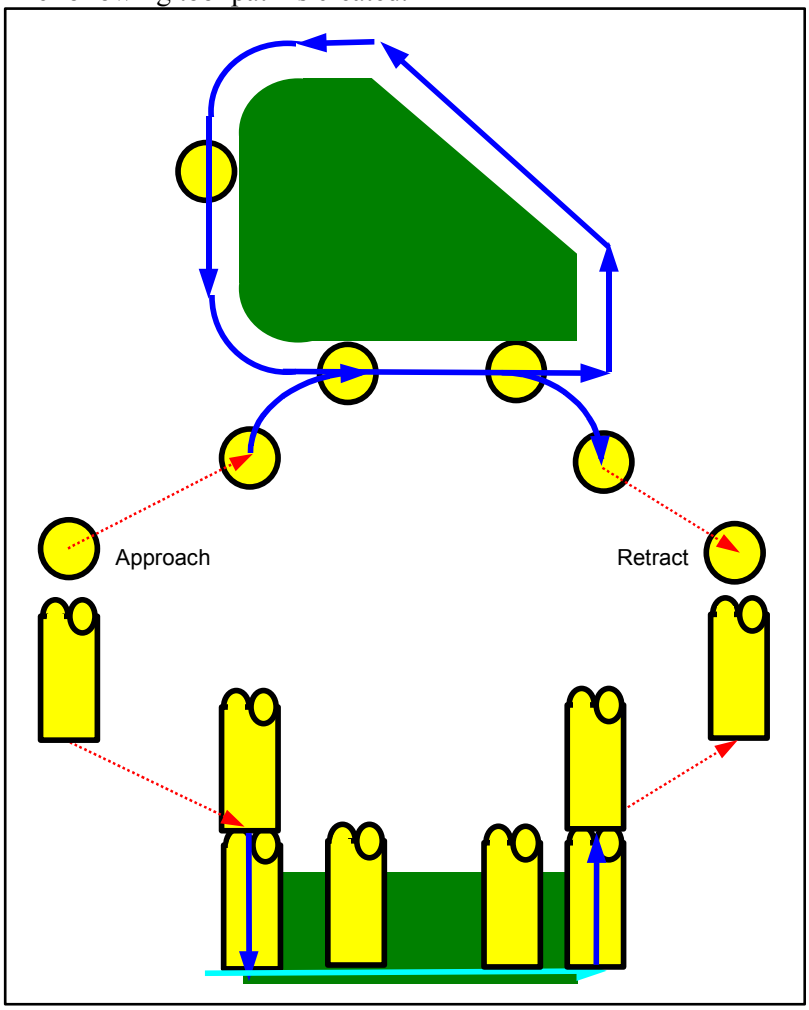
CUT COND.		
Data item		**Meaning**
B	NUMBER OF FINISHING	Number of cuts for finishing (positive value) Remark) Depth of each cut = (side surplus thickness)/(number of finishing cuts)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)

DETAIL		
Data item		**Meaning**
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

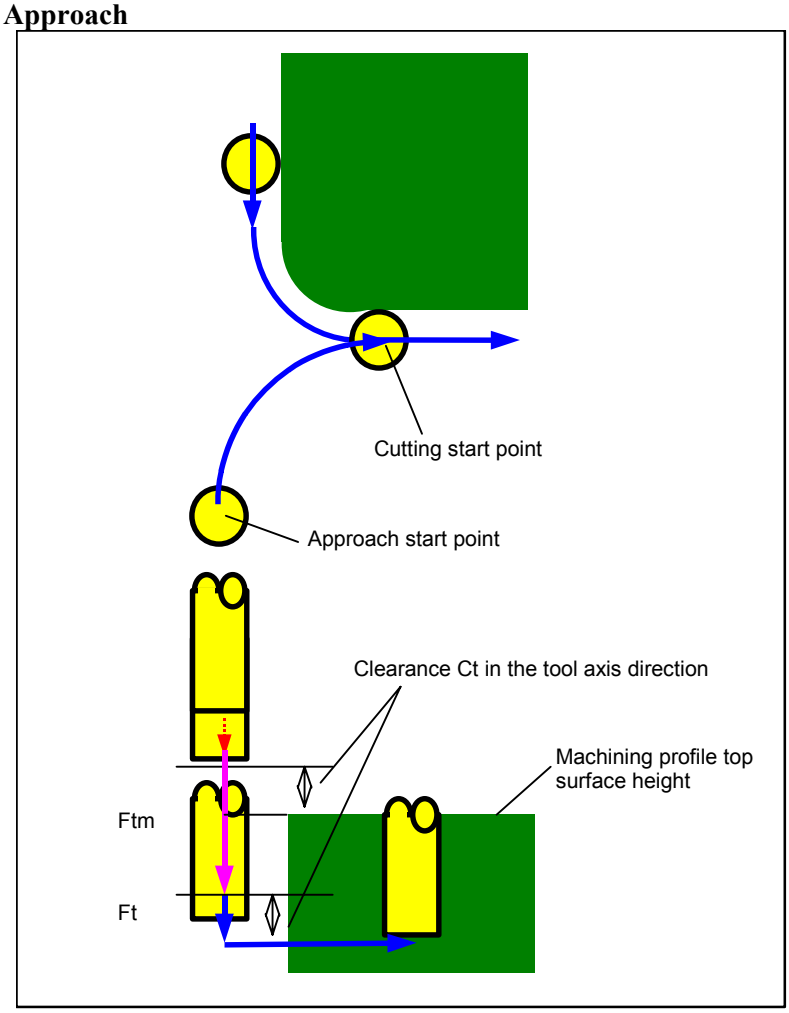
- **Tool path**

The side-face contour of the machining profile is finished.
The following tool path is created.



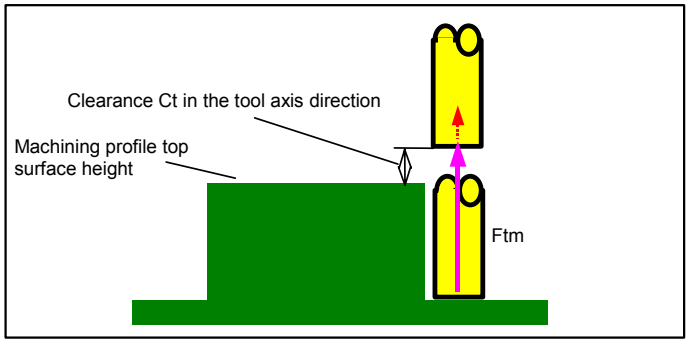
<1> The tool approaches a point above the approach start point.
<2> The tool moves to the bottom surface of the machining profile.
<3> The tool moves to cut along the side-face contour of the machining profile.
In-feed machining is performed for the cutting allowance (Vt) in the tool radius direction a specified number of finishing cycles.
<4> The tool retracts.

- Approach



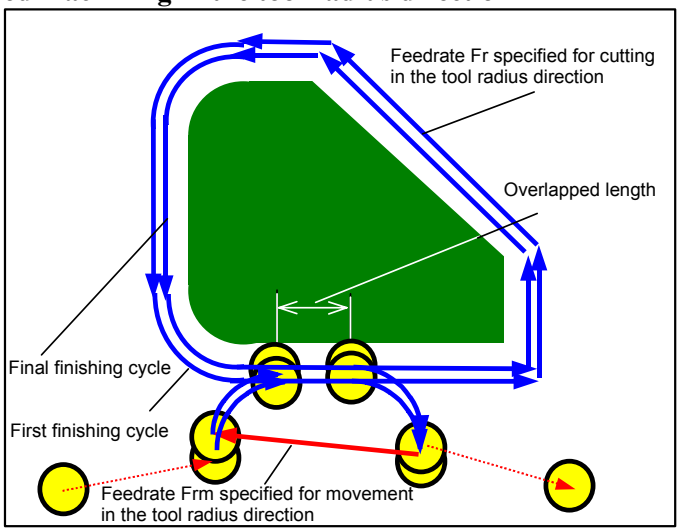
<1> The tool moves to the position "machining profile top surface height + clearance (Ct) in the tool axis direction" by rapid traverse.
<2> The tool moves to the position "machining profile bottom surface + finishing allowance (Tt) in the tool axis direction + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.
<3> The tool moves to the position "machining profile bottom surface + finishing allowance (Tt) in the tool axis direction" at the feedrate (Ft) specified for cutting in the machining profile.
<4> The tool approaches, in the tool radius direction, the in-feed machining start point in the tool radius direction.

- Retraction



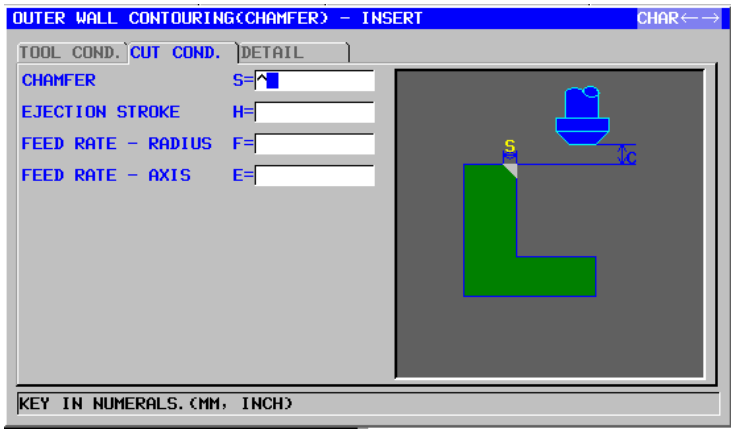
<1> The tool moves from the approach end point to the position "machining profile top surface height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.

- In-feed machining in the tool radius direction



<1> The tool approaches the cutting start point in the tool radius direction at the feedrate (Fr) specified for cutting in the tool radius direction.
<2> The tool moves to cut along the contour from the first in-feed machining cycle start point to the first in-feed machining cycle end point at the feedrate (Fr) specified for cutting in the tool radius direction.
<3> The tool retracts from the cutting end point in the tool radius direction at the feedrate (Fr) specified for cutting in the tool radius direction.
<4> The tool approaches the next in-feed machining cycle start point, using the specified [movement method for in-feed machining].
<5> Steps <2> to <4> are repeated as many times as the required number of finishing cycles.

Outer Wall Chamfer : G1063
Inner Wall Chamfer : G1067
Partial Chamfer : G1071



TOOL COND.		
Data item		**Meaning**
K	TOOL SMALL DIAMETER	Diameter of the tip of a chamfering tool (positive value)

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

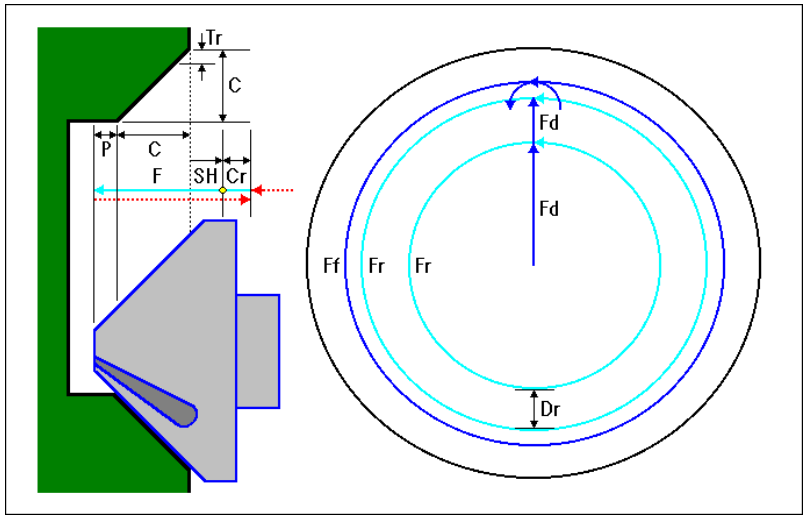
CUT COND.		
Data item		**Meaning**
S	CHAMFER	Chamfer length (radius value, positive value)
H	EJECTION STROKE	Distance between the tip of a chamfering tool and an actually cut position in the tool axis direction (radius value, positive value)
F	FEED RATE- RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)

DETAIL		
Data item		**Meaning**
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)

DETAIL		
Data item		**Meaning**
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

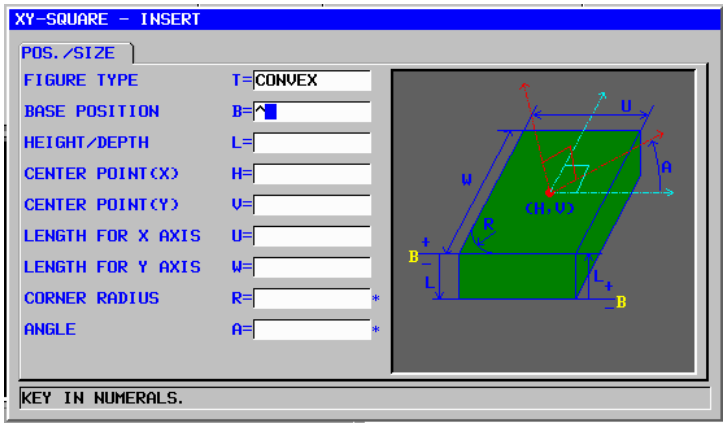
- **Tool path**



<1> The tool moves to the position "cutting start position + clearance (Cr)" by rapid traverse.
<2> The tool moves to the position "amount to be chamfered (C) + ejection stroke (P)" at the feedrate (F) specified for cutting.
<3> The tool cuts in to the portion to be chamfered by the cut depth (Dr) in the tool radius direction at the feedrate (Fd) specified for both-edge thread cutting.
<4> The tool cuts along the contour of the hole machining profile at the feedrate (Fr) specified for single-edge thread cutting.
<5> Steps <3> and <4> are repeated until only the finishing allowance (Tr) in the tool radius direction is left uncut.
<6> The tool turns around the finishing allowance (Tr) in the tool radius direction to perform finishing at the feedrate (Ff) specified for finishing.
<7> The tool moves to the position "cut-in start position + clearance (Cr)" by rapid traverse.

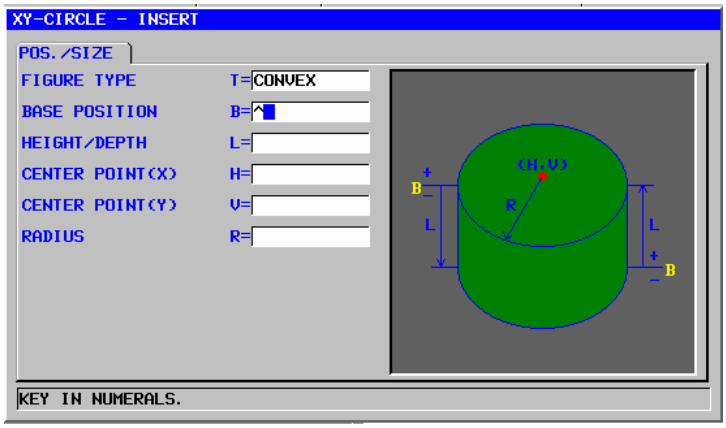
1.4.2 Fixed Form Figure Blocks for Contouring (XY Plane)

Square: G1220 (XY plane)



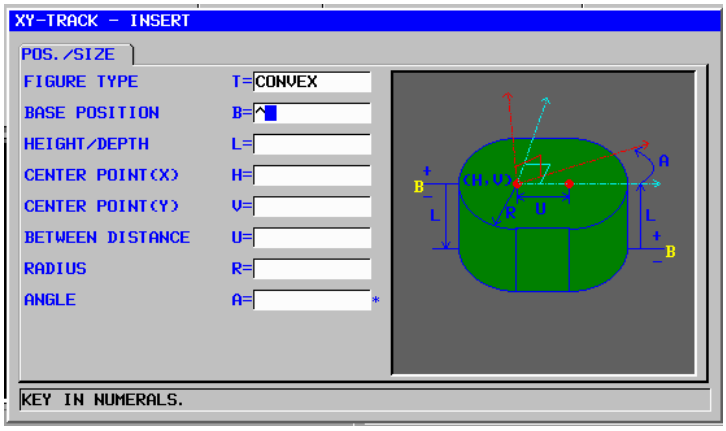
POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When contouring is selected as machining type, select [CONVEX] if an outer figure is to be machined; if an inner figure is to be machined, select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface or the bottom of the side face of a part to be subject to contouring (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of a side face is selected as the BASE POSITION, specify the distance to the top surface of the workpiece by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a rectangular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a rectangular figure
U	LENGTH FOR X AXIS	Length of the side in the X-axis direction (radius value, positive value)
W	LENGTH FOR Y AXIS	Length of the side in the Y-axis direction (radius value, positive value)
R*	CORNER RADIUS	Radius for corner rounding (positive value)
A*	ANGLE	Inclination angle of a rectangular figure relative to the X-axis (positive or negative value)

Circle: G1221 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When contouring is selected as machining type, select [CONVEX] if an outer figure is to be machined; if an inner figure is to be machined, select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface or the bottom of the side face of a part to be subject to contouring (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of a side face is selected as the BASE POSITION, specify the distance to the top surface of the workpiece by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a circular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a circular figure
R	RADIUS	Radius of a circular figure (radius value, positive value)

Track: G1222 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When contouring is selected as machining type, select [CONVEX] if an outer figure is to be machined; if an inner figure is to be machined, select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface or the bottom of the side face of a part to be subject to contouring(in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of a side face is selected as the BASE POSITION, specify the distance to the top surface of the workpiece by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of the left semicircle
V	CENTER POINT (Y)	Y coordinate of the center position of the left semicircle
U	BETWEEN DISTANCE	Distance between the centers of the right and left semicircles (radius value, positive value)
R	RADIUS	Radius of the left and right semicircles (radius value, positive value)
A*	ANGLE	Inclination angle of a track figure relative to the X-axis (positive or negative value)

1.4.3 Fixed Form Figure Blocks for Contouring (YZ Plane, XC Plane)

The same fixed-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane and the XC plane (polar coordinate interpolation plane). They are provided with the following G codes.
The data to be set for the YZ and XC planes is the same as for the XY plane except that the XY plane is changed to the YZ or XC plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ plane) or the Z-axis (XC plane).

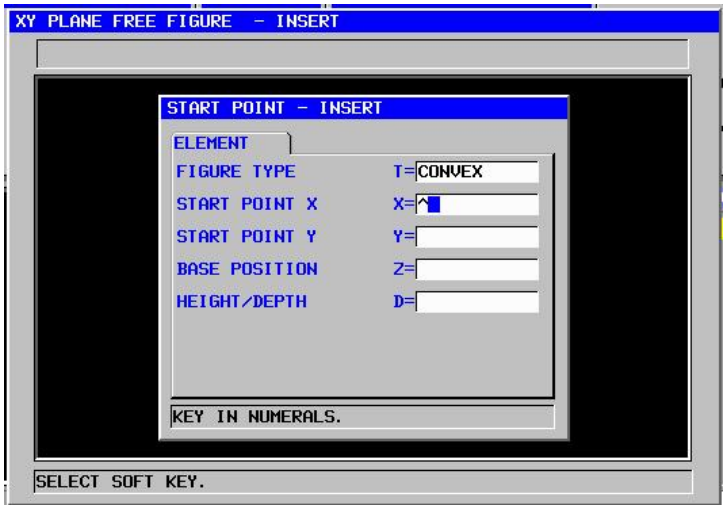
Square : G1320 (YZ plane)
Square : G1520 (XC plane), polar coordinate interpolation plane)
Circle : G1321 (YZ plane)
Circle : G1521 (XC plane), polar coordinate interpolation plane)
Track : G1322 (YZ plane)
Track : G1522 (XC plane), polar coordinate interpolation plane)

NOTE
When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.

1.4.4 Arbitrary Figure Blocks for Contouring (XY Plane)

When an arbitrary figure for contouring is input, data such as a figure type and machining reference position is specified in the start point block. Other data items to be input such as a straight line and arc are exactly the same as for arbitrary figures of other machining types.
So, this subsection describes only the start point block of an arbitrary figure for contouring.
For other arbitrary figures, see Chapter 5, "DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES," in Part II.

Start point: G1200 (XY plane)



ELEMENT		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring and emboss machining or as a figure for pocketing [GROOVE] : Used as a figure for grooving [OPEN] : Used when only one side is cut during contouring Remark) When contouring is specified as a machining type, one of [CONVEX], [CONCAVE], and [OPEN] must be selected.
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
Z	BASE POSITION	Z coordinate of the top surface or the bottom of the side face of a part to be subject to contouring (tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of a side face is selected as the BASE POSITION, specify the distance to the top surface of the workpiece by using a positive value (radius value). → Height

ELEMENT		
Data item		**Meaning**
P	FIGURE ATTRIBUTE	[RIGHT] : The right side of an entered figure as viewed with respect to the direction of movement is cut. (initial value) [LEFT] : The left side of an entered figure as viewed with respect to the direction of movement is cut. (initial value) Remark) These items are displayed only when [OPEN] is selected as a figure type.

1.4.5 Arbitrary Figure Blocks for Contouring (YZ Plane, XC Plane, ZC Plane, XA Plane)

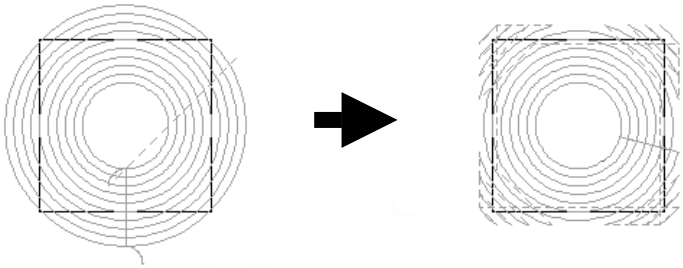
The same arbitrary-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane, the XC plane (polar coordinate interpolation plane), ZC plane and XA plane (cylindrical surface). Their start points are specified with the following G codes.
The data to be set for the YZ, XC, ZC planes and XA plane is the same as for the XY plane except that the XY plane is changed to the YZ, XC, ZC XA plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ and ZC planes) or the Z-axis (XC and XA plane).

Start point : G1300 (YZ plane)
Start point : G1500 (XC plane, polar coordinate interpolation plane)
Start point : G1600 (ZC plane, cylindrical surface)
Start point : G1700 (XA plane, cylindrical surface)

NOTE
1 When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.
2 When performing machining (cylindrical interpolation) on the ZC and XA plane, note the following:
(When bit 3 of parameter No. 27000 = 0)
The mode needs to be switched to the cylindrical coordinate interpolation mode beforehand. Specifically, enter G07.1C (cylinder radius) before the machining type.
Enter G07.1C0 for canceling cylindrical interpolation as required.
(When bit 3 of parameter No. 27000 = 1)
G07.1C (cylinder radius) and G07.1C0 are automatically output before and after cycle machining, respectively.
3 The following parameter setting is necessary to execute XA plane cycle.
- No.27003#2=1, 27003#1=0 and 27003#0=0

1.5 EMBOSS MACHINING

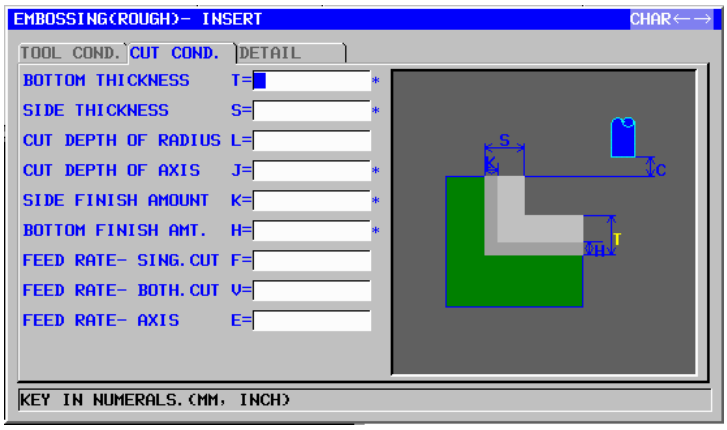
In the case of the contouring, the tool cuts along the side-face contour of the machining profile and performs in-feed machining in the tool radius direction. These tool passes sometimes generate many air-cut movement as the following left figure.
So, the machining called “EMBOSS MACHINING CYCLE” that can reduce the air-cut passes as the following right figure is prepared.



NOTE
1 This function is available when parameter No.27000#7=1.
2 The parameters for the machining conditions for embossing cycles are same with the parameters for pocketing cycles. (Ex. No. 27066)

1.5.1 Machining Type Blocks for Emboss machining

Rough: G1080



TOOL COND.		
Data item		Meaning
D	TOOL DIAMETER	End mill diameter

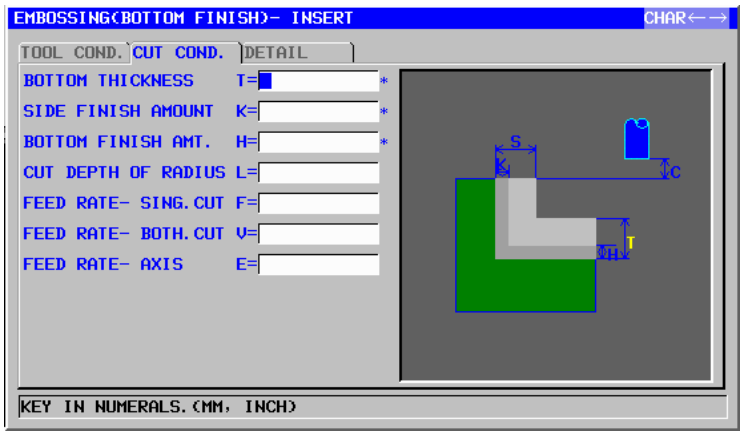
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
T*	BOTTOM THICKNESS	Cutting allowance of the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
S*	SIDE THICKNESS	Cutting allowance of the side face. The blank is regarded as 0. (radius value, positive value) Remark) When both the side surplus thickness and bottom surplus thickness are omitted, an entire pocket area is cut.
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)

DETAIL		
Data item		**Meaning**
B	CLEARANCE OF RADIUS	Distance between the side face and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one pocket cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the pocket by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

Bottom Finish: G1081



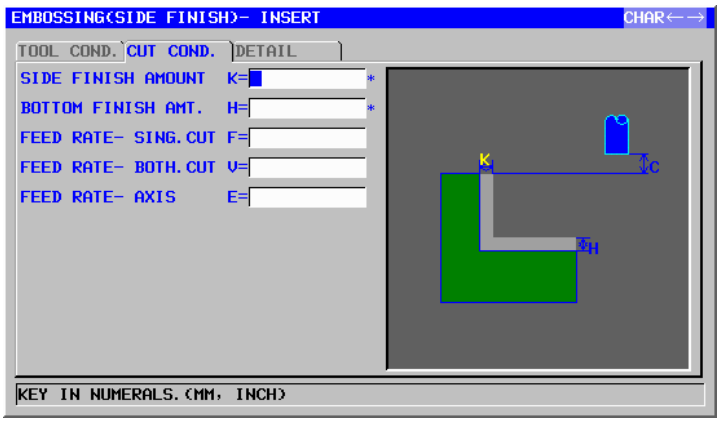
TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
T*	BOTTOM THICKNESS	Cutting allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance of the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
B	CLEARANCE OF RADIUS	Distance between the side face and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one pocket cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the pocket by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

Side face finish: G1082



TOOL COND.		
Data item		**Meaning**
I	INPUT	[INPUT] : Inputs a cutter compensation value directly. [REF.] : Inputs a cutter compensation number to read a cutter compensation value by that number.
D	TOOL DIAMETER	End mill diameter (positive value) Remark) This item is indicated only when [INPUT] is selected for INPUT.
M	CUTTER COMP.NO.	Cutter compensation number of an end mill (positive value) Remark) This item is indicated only when [REF.] is selected for INPUT.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'
3 Tab 'TOOL COND.' is not enabled when bit 3 (PF3) of parameter No. 27061 = 1.

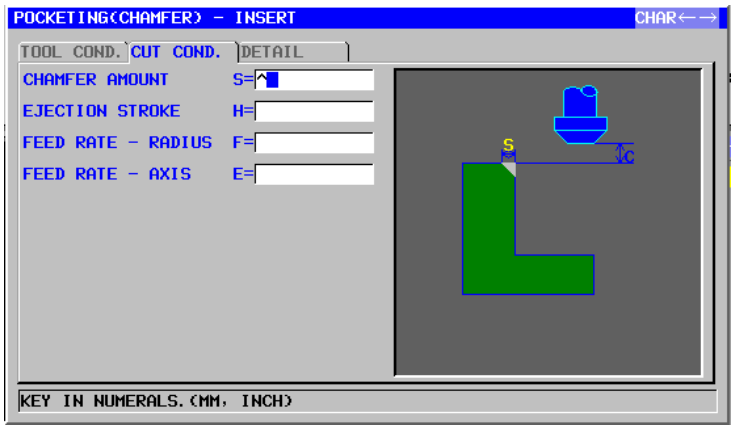
CUT COND.		
Data item		**Meaning**
K*	SIDE FINISH AMOUNT	Finishing allowance of the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.

CUT COND.		
Data item		**Meaning**
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)

DETAIL		
Data item		**Meaning**
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

Chamfer: G1083



TOOL COND.		
Data item		**Meaning**
K	TOOL SMALL DIAMETER	Diameter of the tip of a chamfering tool (positive value)

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
S	CHAMFER AMOUNT	Chamfer length (radius value, positive value)
H	EJECTION STROKE	Distance between the tip of a chamfering tool and an actually cut position in the tool axis direction (radius value, positive value)
F	FEED RATE - RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE - AXIS	Feedrate applicable when cutting is performed in the tool axis direction

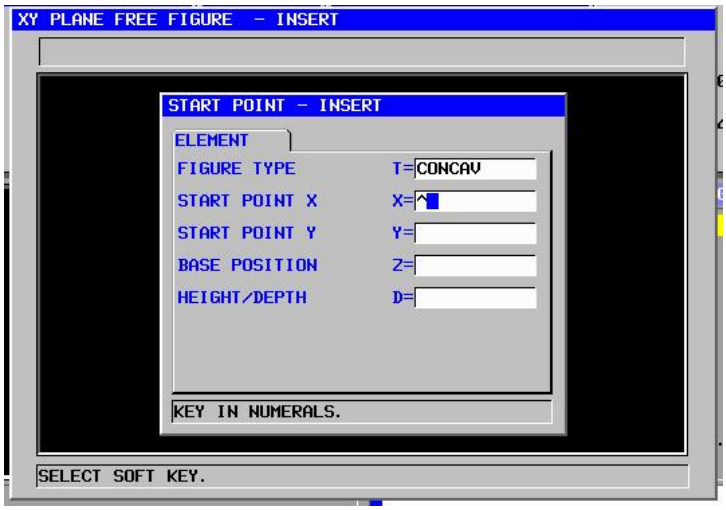
DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (initial value)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

DETAIL		
Data item		**Meaning**
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

1.5.2 Arbitrary Figure Blocks for Emboss machining (XY Plane)

When an arbitrary figure for emboss machining is input, data such as a figure type and machining reference position is specified in the start point block. Other data items to be input such as a straight line and arc are exactly the same as for arbitrary figures of other machining types.
So, this subsection describes only the start point block of an arbitrary figure for pocketing.
For other arbitrary figures, see Chapter 5, "DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES," in Part II.
In creating the arbitrary figure for the emboss machining, create the arbitrary figures which are the outside boundary of the cutting area at fast. After finishing the outside boundary figures, press [ISLAND] soft key and create the arbitrary figures which are the contour product figure.

Start point: G1200 (XY plane)



ELEMENT		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring and emboss machining or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When emboss machining is selected as machining type, be sure to select [CONCAVE].
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
B	BASE POSITION	Z coordinate of the top surface of a workpiece subject to pocketing or the bottom of a pocket (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of the side face is selected as the BASE POSITION, specify the distance to the top surface of the pocket by using a positive value (radius value). → Height

1.5.3 Arbitrary Figure Blocks for Emboss machining (YZ Plane, XC Plane, ZC Plane, XA plane)

The same arbitrary-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane, the XC plane (polar coordinate interpolation plane), ZC plane and XA plane (cylindrical surface). Their start points are specified with the following G codes.
The data to be set for the YZ, XC, ZC and XA planes is the same as for the XY plane except that the XY plane is changed to the YZ, XC, ZC or XA plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ and ZC planes) or the Z-axis (XC and XA plane).

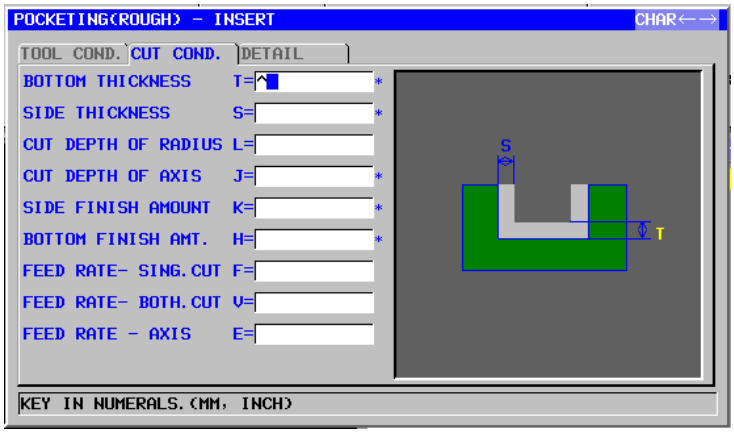
Start point : G1300 (YZ plane)
Start point : G1500 (XC plane, polar coordinate interpolation plane)
Start point : G1600 (ZC plane, cylindrical surface)
Start point : G1700 (XA plane, cylindrical surface)

NOTE
1 When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.
2 When performing machining (cylindrical interpolation) on the ZC and XA plane, note the following:
(When bit 3 of parameter No. 27000 = 0)
The mode needs to be switched to the cylindrical coordinate interpolation mode beforehand. Specifically, enter G07.1C (cylinder radius) before the machining type.
Enter G07.1C0 for canceling cylindrical interpolation as required.
(When bit 3 of parameter No. 27000 = 1)
G07.1C (cylinder radius) and G07.1C0 are automatically output before and after cycle machining, respectively.
3 The following parameter setting is necessary to execute XA plane cycle.
No.27003#2=1, 27003#1=0 and 27003#0=0

1.6 POCKETING

1.6.1 Machining Type Blocks for Pocketing

Rough: G1040



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
T*	BOTTOM THICKNESS	Cutting allowance of the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
S*	SIDE THICKNESS	Cutting allowance of the side face. The blank is regarded as 0. (radius value, positive value) Remark) When both the side surplus thickness and bottom surplus thickness are omitted, an entire pocket area is cut.
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)

CUT COND.		
Data item		Meaning
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

NOTE

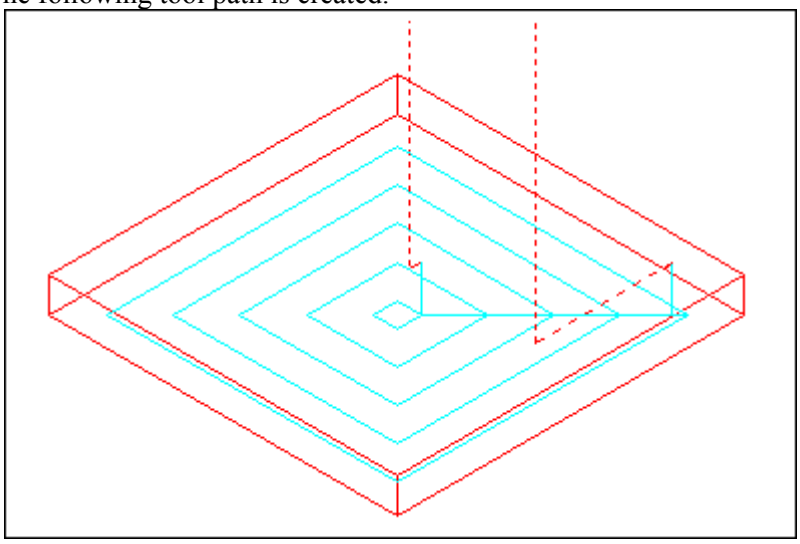
When a non-zero value is input for just one of the data items, side surplus thickness and bottom surplus thickness, cutting is performed for the input surplus thickness, and for the omitted surplus thickness, cutting operation is not performed on the assumption that the omitted surplus thickness is 0. When non-zero values are input for both the side surplus thickness and bottom surplus thickness, cutting is performed for each surplus thickness.

DETAIL		
Data item		Meaning
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
B	CLEARANCE OF RADIUS	Distance between the wall of a pocket and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one pocket cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the pocket by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

DETAIL		
Data item		**Meaning**
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.
A*	CUT ANGLE	Angle at which the tool cuts into the pocket obliquely. The blank is regarded as 0 degrees. (in one-degree increments, positive value) Remark) By performing cutting also using the side face of an end mill, the load on the tool can be reduced. (COPY)
P*	START PT.(1ST AXIS)	1st-axis coordinate of the cutting start point of pocketing. When omitting this item, also omit the 2nd-axis coordinate. In this case, the coordinates of the start point are determined automatically. Remark1) The 1st axis is the X-axis on the XY plane, the Y-axis on the YZ plane, the X-axis on the XC plane, the Z-axis on the ZC plane, or the X-axis on the XA plane. Remark2) This item is enable when the parameter No. 27060#7(PR7) = 1.
Q*	START PT.(2ND AXIS)	2nd-axis coordinate of the cutting start point of pocketing. When this item is omitted, the coordinate is determined automatically. Remark1) The 2nd axis is the Y-axis on the XY plane, the Z-axis on the YZ plane, the C-axis on the XC plane, the C-axis on the ZC plane, or the A-axis on the XA plane. Remark2) This item is enable when the parameter No. 27060#7(PR7) = 1.

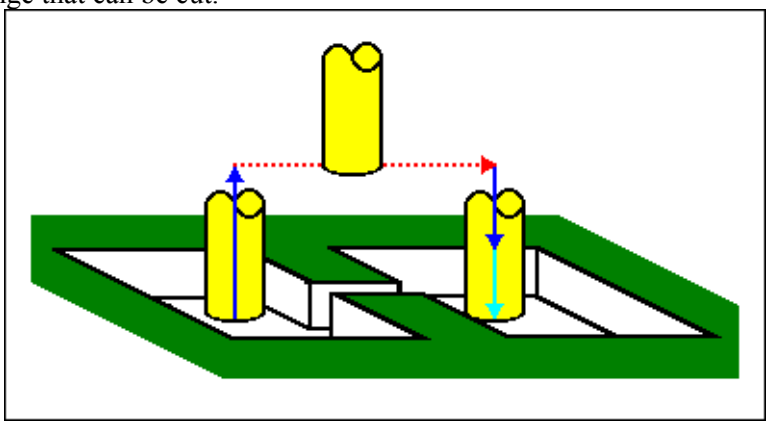
- **Tool path**

The inside of a pocket machining profile is cut off in a spiral manner. The following tool path is created.

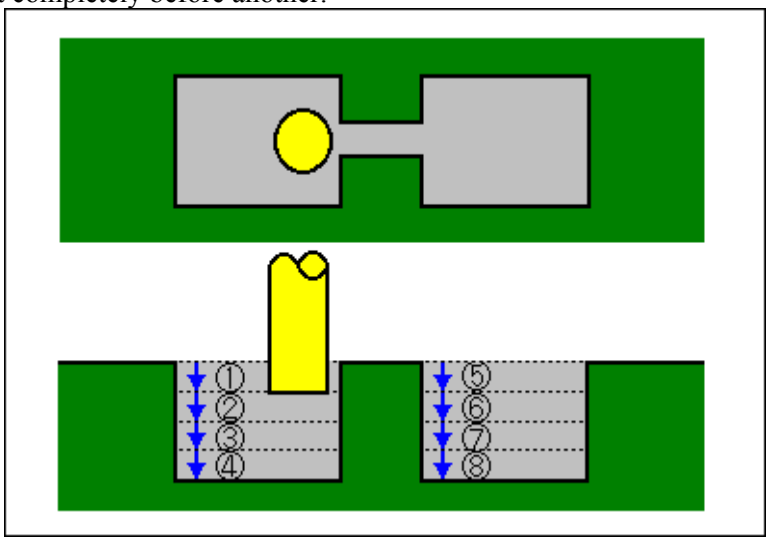


More than one island machining profile and more than one cavity machining profile can be defined for a pocket machining profile. The island machining profiles are left uncut.
The cavity machining profiles are detoured so that they will not be cut.
The tool path is created in such a way that an anticipated interference with pocket machining profiles or island machining profiles can be avoided.
The created tool path is effective because retraction in the tool axis direction is avoided as much as possible.
For this tool path, in-feed machining in the tool axis direction is possible.
Only specified cutting allowances can be cut off.
The direction of cutting can be either up-cut or down-cut. The direction of cutting is controlled automatically around islands.
It is possible to cut in from the inside and outside of the machining profile.
If there is a portion left uncut at a corner, it is possible to automatically discriminate the uncut portion and cut it off.
It is possible to cut in at an arbitrary angle in the tool axis direction.
It is possible to specify an arbitrary position as a cutting start point.
It is possible to determine a cutting start point automatically.
It is possible to select a movement method for tool movement.
It is possible to control automatically a cut-in depth on the island machining profile top surface automatically.
It is possible to set up cutting conditions, such as finishing allowance for island machining profiles, for each island machining profile.

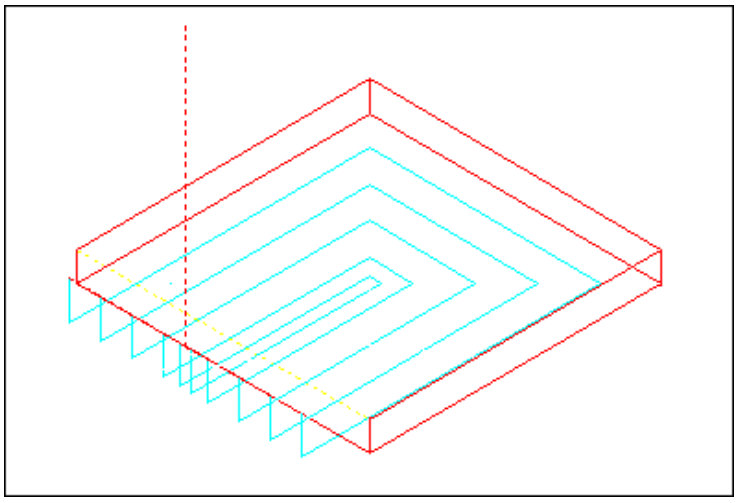
In the following pocket machining profile, which has a pocket through which the tool can pass, the tool is lifted automatically to cut only a range that can be cut.



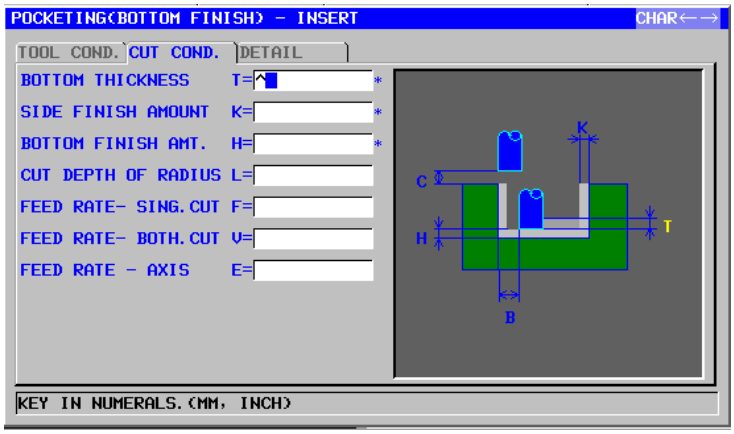
If there is more than one cut in the tool axis direction, each range is cut completely before another.



A contour specified as an open element of a pocket machining profile is cut off as shown below.



Bottom Finish: G1041



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

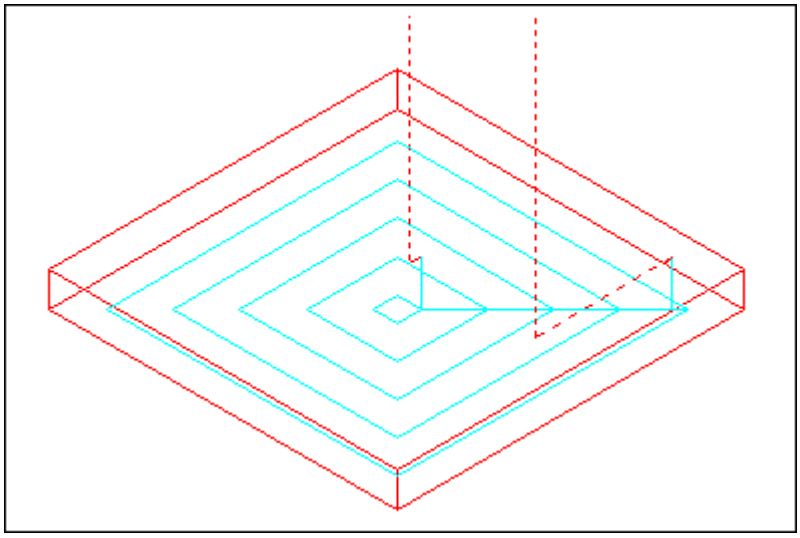
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
T*	BOTTOM THICKNESS	Cutting allowance at the bottom of a pocket. The blank is regarded as 0. (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face of a pocket. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom of a pocket. The blank is regarded as 0. (radius value, positive value)
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		Meaning
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
B	CLEARANCE OF RADIUS	Distance between the wall of a pocket and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one pocket cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the pocket by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.
A*	CUT ANGLE	Angle at which the tool cuts into the pocket obliquely. The blank is regarded as 0 degrees. (in one-degree increments, positive value) Remark) By performing cutting also using the side face of an end mill, the load on the tool can be reduced. (COPY)

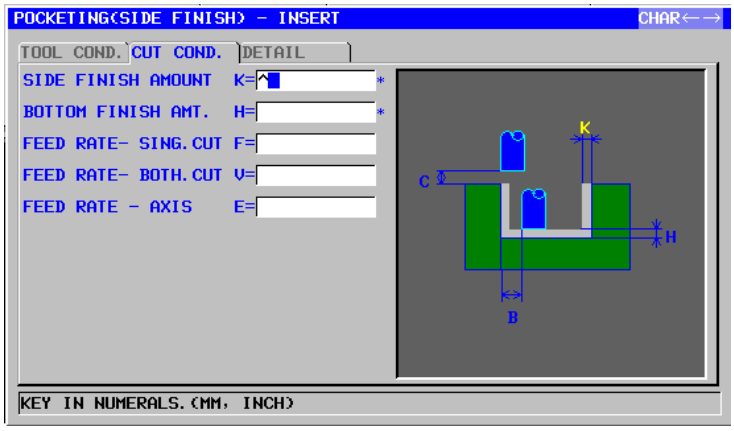
DETAIL		
Data item		**Meaning**
P*	START PT.(1ST AXIS)	1st-axis coordinate of the cutting start point of pocketing. When omitting this item, also omit the 2nd-axis coordinate. In this case, the coordinates of the start point are determined automatically. Remark1) The 1st axis is the X-axis on the XY plane, the Y-axis on the YZ plane, the X-axis on the XC plane, the Z-axis on the ZC plane, or the X-axis on the XA plane. Remark2) This item is enable when the parameter No. 27060#7(PR7) = 1.
Q*	START PT.(2ND AXIS)	2nd-axis coordinate of the cutting start point of pocketing. When this item is omitted, the coordinate is determined automatically. Remark1) The 2nd axis is the Y-axis on the XY plane, the Z-axis on the YZ plane, the C-axis on the XC plane, the C-axis on the ZC plane, or the A-axis on the XA plane. Remark2) This item is enable when the parameter No. 27060#7(PR7) = 1.

- **Tool path**



The bottom surface of a pocket machining profile is finished in a spiral manner.
The tool path for it is the same as for pocketing (roughing).
See descriptions about pocketing (roughing) for details.
No in-feed machining in the tool axis direction is performed, though.
The top surface of island machining profiles is not cut off either.

Side face finish: G1042



TOOL COND.		
Data item		**Meaning**
I	INPUT	[INPUT] : Inputs a cutter compensation value directly. [REF.] : Inputs a cutter compensation number to read a cutter compensation value by that number.
D	TOOL DIAMETER	End mill diameter (positive value) Remark) This item is indicated only when [INPUT] is selected for INPUT.
M	CUTTER COMP.NO.	Cutter compensation number of an end mill (positive value) Remark) This item is indicated only when [REF.] is selected for INPUT.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'
3 Tab 'TOOL COND.' is not enabled when bit 3 (PF3) of parameter No. 27061 = 1.

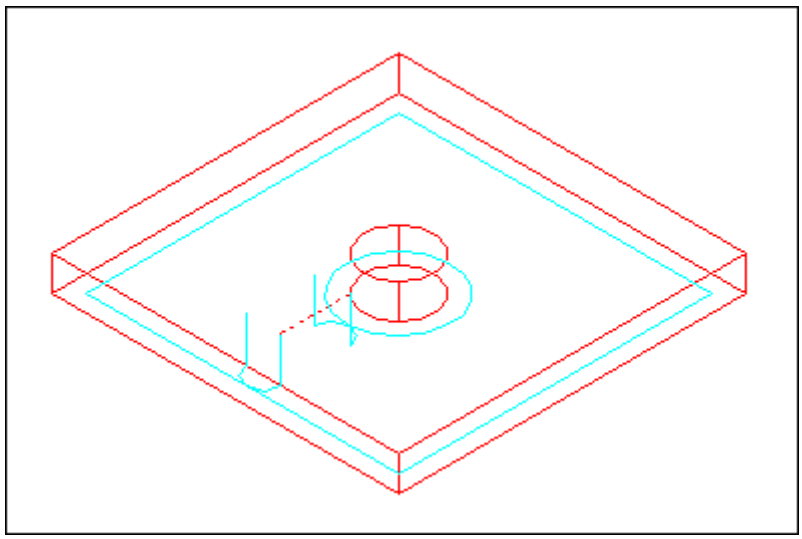
CUT COND.		
Data item		**Meaning**
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face of a pocket. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance at the bottom of a pocket. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.

CUT COND.		
Data item		**Meaning**
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)

DETAIL		
Data item		Meaning
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

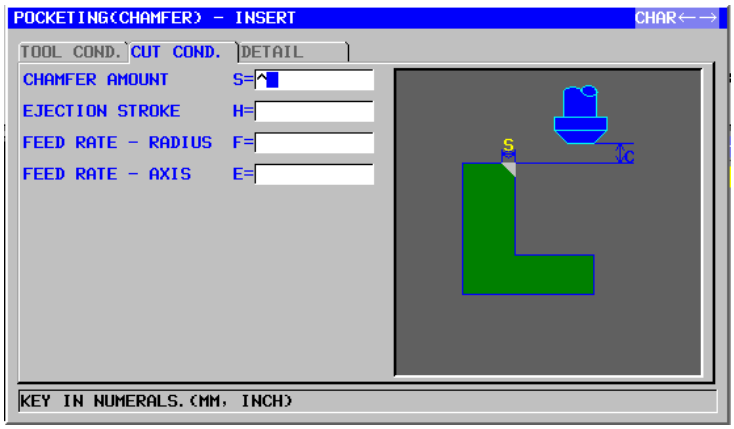
- **Tool path**



The side-face contour of pocket and island machining profiles is finished. The tool path for it is the same as for contouring (side-face finishing).
See descriptions about contouring (side-face finishing) for details.

The specifications for the following points differ in part, though.
No in-feed machining is performed in the tool radius direction or tool axis direction.
Even if it is anticipated that the tool may interfere with a pocket or island machining profile during finishing, no tool path that can avoid interference is created.

Chamfer: G1043



TOOL COND.		
Data item		**Meaning**
K	TOOL SMALL DIAMETER	Diameter of the tip of a chamfering tool (positive value)

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CUT COND.		
Data item		**Meaning**
S	CHAMFER AMOUNT	Chamfer length (radius value, positive value)
H	EJECTION STROKE	Distance between the tip of a chamfering tool and an actually cut position in the tool axis direction (radius value, positive value)
F	FEED RATE - RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE - AXIS	Feedrate applicable when cutting is performed in the tool axis direction

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cut mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cut mode, assuming that the tool is rotating clockwise. (initial value)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

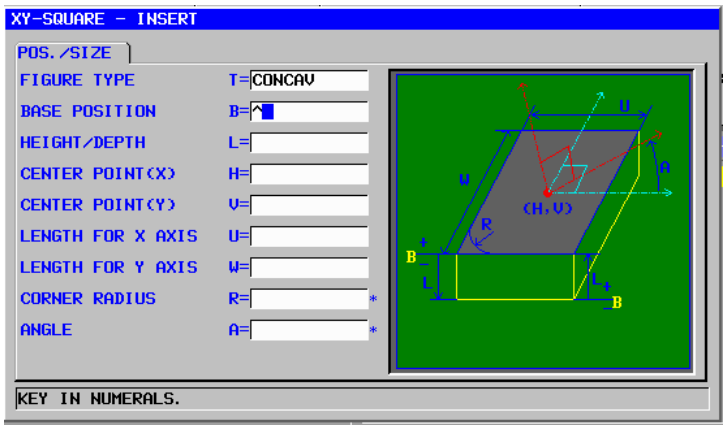
DETAIL		
Data item		**Meaning**
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**

The top surface of a wall of a pocket is chamfered. The tool path for it is the same as for contouring (chamfering).
See descriptions about contouring (chamfering) for details.

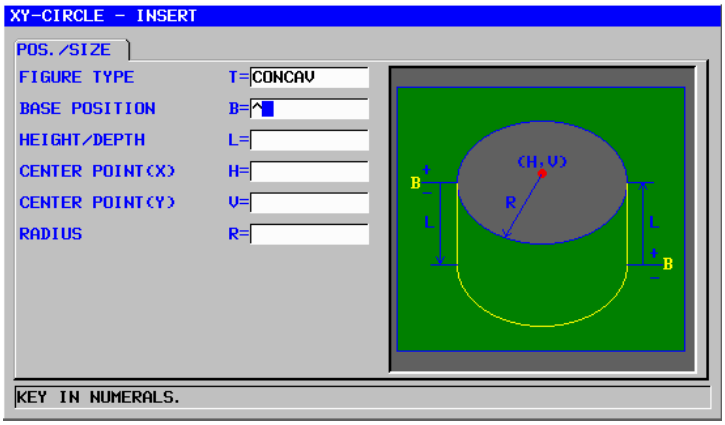
1.6.2 Fixed Form Figure Blocks for Pocketing (XY Plane)

Square: G1220 (XY plane)



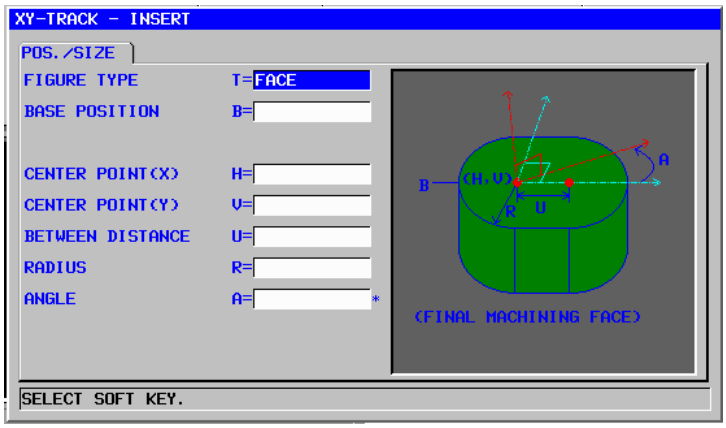
POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When pocketing is selected as machining type, be sure to select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface of a workpiece subject to pocketing or the bottom of a pocket (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of a pocket by using a negative value (radius value). → Depth When the bottom of a pocket is selected as the BASE POSITION, specify the distance to the top surface of the pocket by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a rectangular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a rectangular figure
U	LENGTH FOR X AXIS	Length of the side in the X-axis direction (radius value, positive value)
W	LENGTH FOR Y AXIS	Length of the side in the Y-axis direction (radius value, positive value)
R*	CORNER RADIUS	Radius for corner rounding (positive value)
A*	ANGLE	Inclination angle of a rectangular figure relative to the X-axis (positive or negative value)

Circle: G1221 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When pocketing is selected as machining type, be sure to select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface of a workpiece subject to pocketing or the bottom of a pocket (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of a pocket by using a negative value (radius value). → Depth When the bottom of a pocket is selected as the BASE POSITION, specify the distance to the top surface of the pocket by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a circular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a circular figure
R	RADIUS	Radius of a circular figure (radius value, positive value)

Track: G1222 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When pocketing is selected as machining type, be sure to select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface of a workpiece subject to pocketing or the bottom of a pocket (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of a pocket by using a negative value (radius value). → Depth When the bottom of a pocket is selected as the BASE POSITION, specify the distance to the top surface of the pocket by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of the left semicircle
V	CENTER POINT (Y)	Y coordinate of the center position of the left semicircle
U	BETWEEN DISTANCE	Distance between the centers of the right and left semicircles (radius value, positive value)
R	RADIUS	Radius of the left and right semicircles (radius value, positive value)
A*	ANGLE	Inclination angle of a track figure relative to the X-axis (positive or negative value)

1.6.3 Fixed Form Figure Blocks for Pocketing (YZ Plane, XC Plane)

The same fixed-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane and the XC plane (polar coordinate interpolation plane). They are provided with the following G codes.
The data to be set for the YZ and XC planes is the same as for the XY plane except that the XY plane is changed to the YZ or XC plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ plane) or the Z-axis (XC plane).

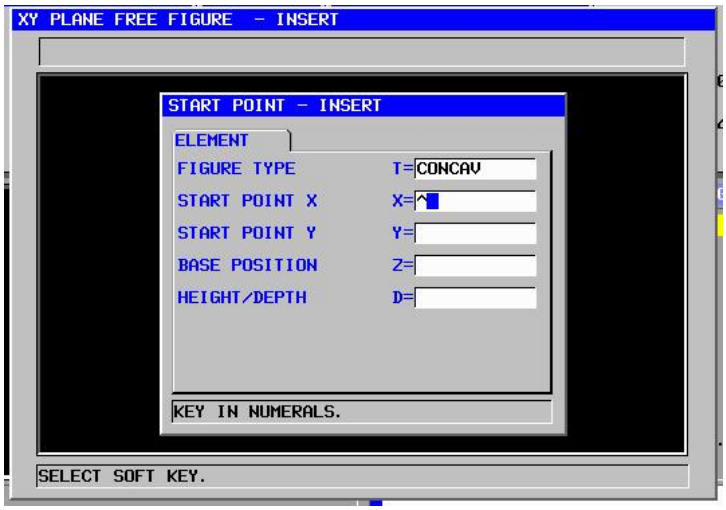
Square : G1320 (YZ plane)
Circle : G1321 (YZ plane)
Track : G1322 (YZ plane)
Square : G1520 (XC plane, polar coordinate interpolation plane)
Circle : G1521 (XC plane, polar coordinate interpolation plane)
Track : G1522 (XC plane, polar coordinate interpolation plane)

NOTE
When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.

1.6.4 Arbitrary Figure Blocks for Pocketing (XY Plane)

When an arbitrary figure for pocketing is input, data such as a figure type and machining reference position is specified in the start point block. Other data items to be input such as a straight line and arc are exactly the same as for arbitrary figures of other machining types.
So, this subsection describes only the start point block of an arbitrary figure for pocketing.
For other arbitrary figures, see Chapter 5, "DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES," in Part II.

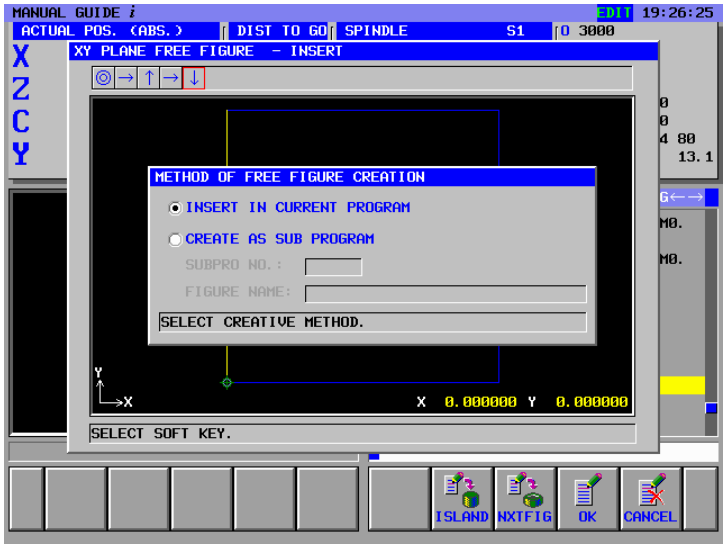
Start point: G1200 (XY plane)



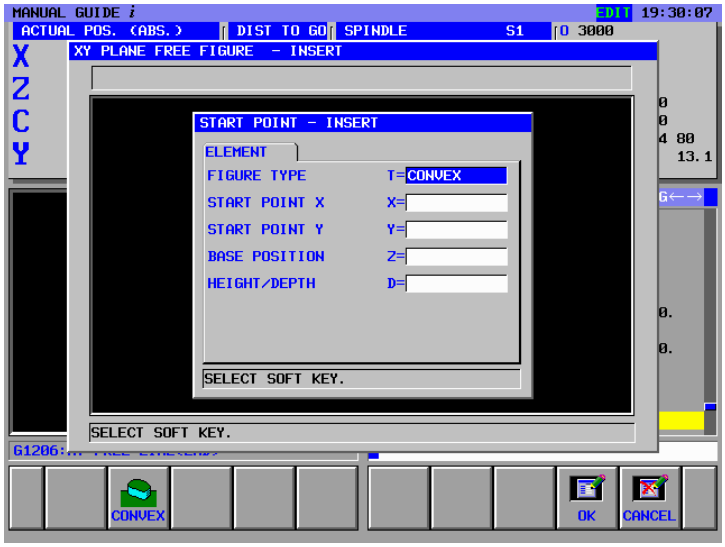
ELEMENT		
Data item		Meaning
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When pocketing is selected as machining type, be sure to select [CONCAVE].
B	BASE POSITION	Z coordinate of the top surface of a workpiece subject to pocketing or the bottom of a pocket (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of a pocket by using a negative value (radius value). → Depth When the bottom of a pocket is selected as the BASE POSITION, specify the distance to the top surface of the pocket by using a positive value (radius value). → Height

Input of Island :

After inputting the outer wall figure of Pocket, the following screen is displayed by pushing the soft-key [CREATE].



If there is a island, push the soft-key [ISLAND] in order to input the island figure. The following START POINT screen is displayed.
As to the input data item [FIGURE TYPE] is set “CONVEX” automatically. (When a outer wall, it is set “CONCAV”.)
After that, input the island figure as the same of a outer wall figure.



NOTE
As to the soft-key [NXTFIG], it is used in the case of inputting the other pocket figure one after another.

1.6.5 Arbitrary Figure Blocks for Pocketing (YZ Plane, XC Plane, ZC Plane, XA Plane)

The same arbitrary-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane, the XC plane (polar coordinate interpolation plane), ZC plane and XA plane (cylindrical surface). Their start points are specified with the following G codes.
The data to be set for the YZ, XC, ZC and XA planes is the same as for the XY plane except that the XY plane is changed to the YZ, XC, ZC, or XA plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ and ZC planes) or the Z-axis (XC and XA plane).

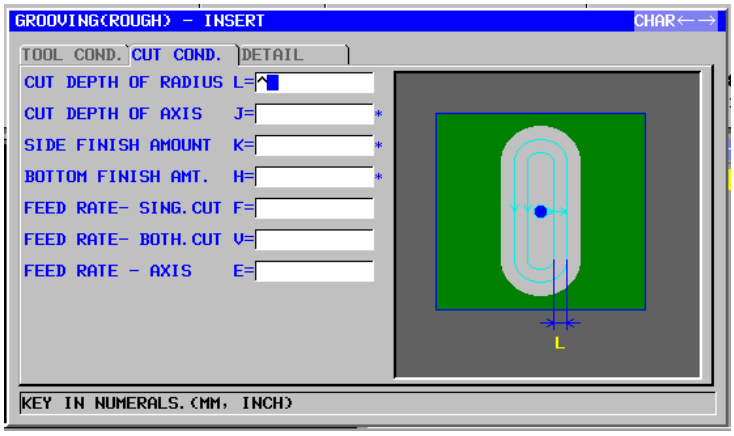
Start point : G1300 (YZ plane)
Start point : G1500 (XC plane, polar coordinate interpolation plane)
Start point : G1600 (ZC plane, cylindrical surface)
Start point : G1700 (XA plane, cylindrical surface)

NOTE
1 When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.
2 When performing machining (cylindrical interpolation) on the ZC and XA plane, note the following:
(When bit 3 of parameter No. 27000 = 0)
The mode needs to be switched to the cylindrical coordinate interpolation mode beforehand. Specifically, enter G07.1C (cylinder radius) before the machining type.
Enter G07.1C0 for canceling cylindrical interpolation as required.
(When bit 3 of parameter No. 27000 = 1)
G07.1C (cylinder radius) and G07.1C0 are automatically output before and after cycle machining, respectively.
3 The following parameter setting is necessary to execute XA plane cycle.
- No.27003#2=1, 27003#1=0 and 27003#0=0

1.7 GROOVING

1.7.1 Machining Type Blocks for Grooving

Roughing: G1050



TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

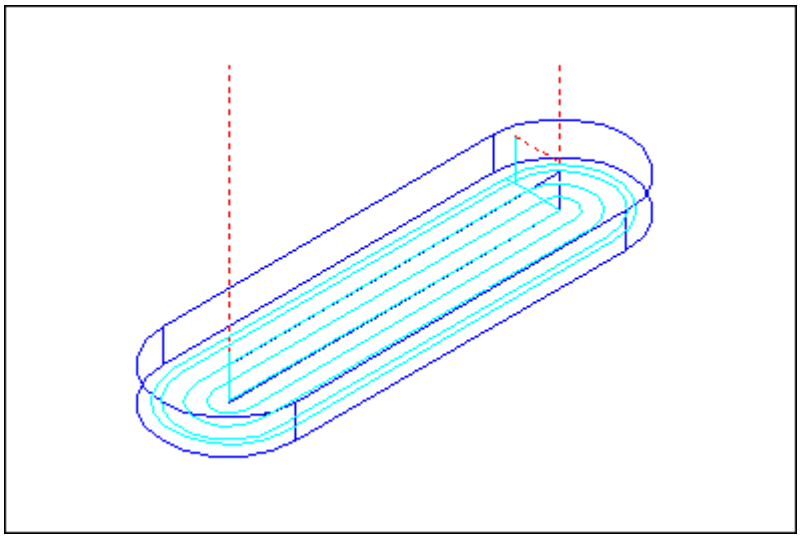
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISH AMT.	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)

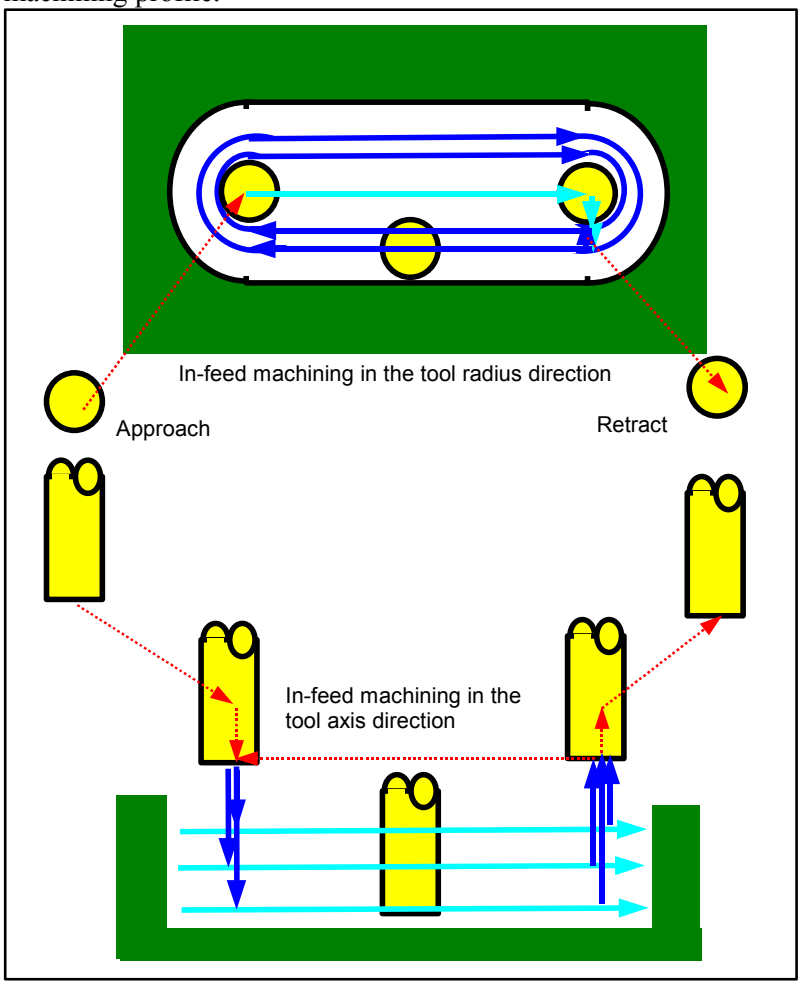
CUT COND.		
Data item		**Meaning**
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cutting mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cutting mode, assuming that the tool is rotating clockwise. (COPY)
B	CLEARANCE OF RADIUS	Distance between the wall of a groove and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one groove cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the groove by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



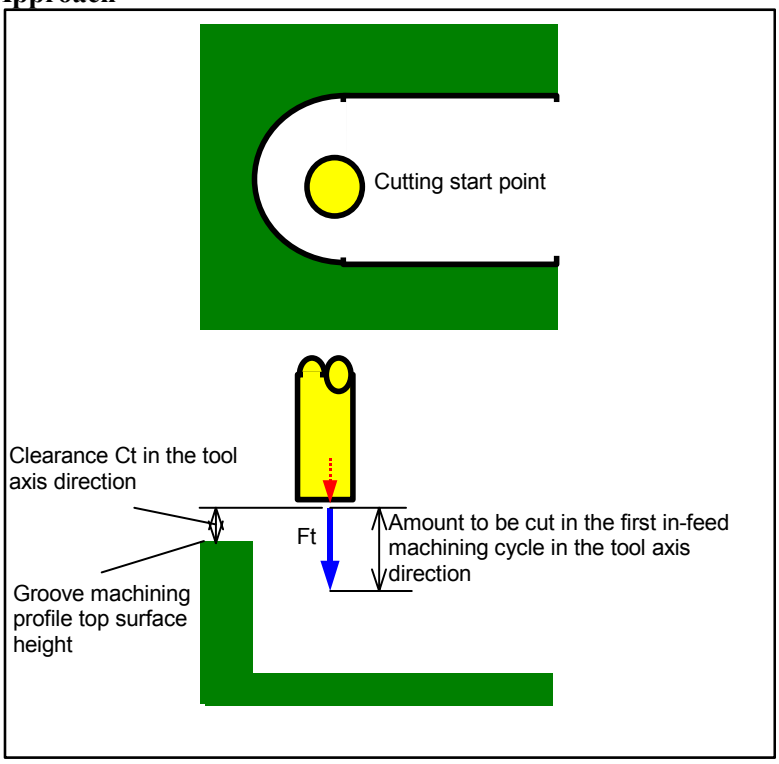
The following tool path is created to cut off the inside of a groove machining profile.



<1> The tool approaches a point above the cutting start point of a groove machining profile.

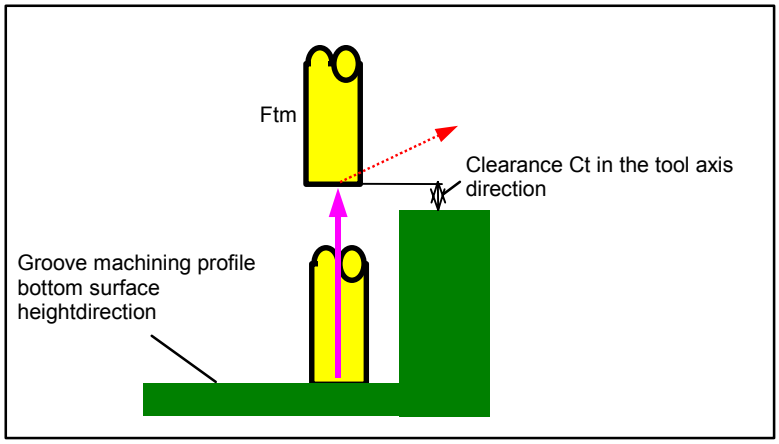
<2> The tool cuts in the groove machining profile in the tool radius direction.
<3> The tool cuts in the groove machining profile in the tool axis direction.
<4> Step <2> and <3> are repeated until the cutting allowance is removed.
<5> The tool retracts.

- Approach



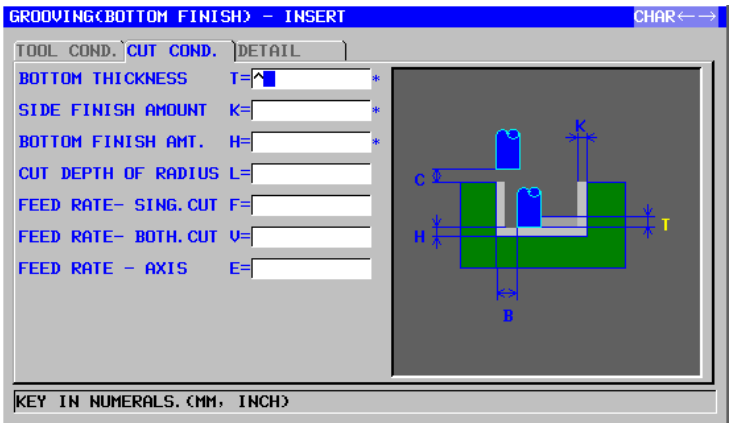
<1> The tool moves to the position "groove machining profile top surface height + clearance (Ct) in the tool axis direction" by rapid traverse.
<2> The tool cuts in by the amount to be cut in the first in-feed machining cycle in the tool axis direction at the feedrate (Ft) specified for cutting in the tool axis direction.

- Retract



<1> The tool retracts from the groove machining profile bottom surface height to the position "groove machining profile top surface height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.

Bottom face finishing: G1051



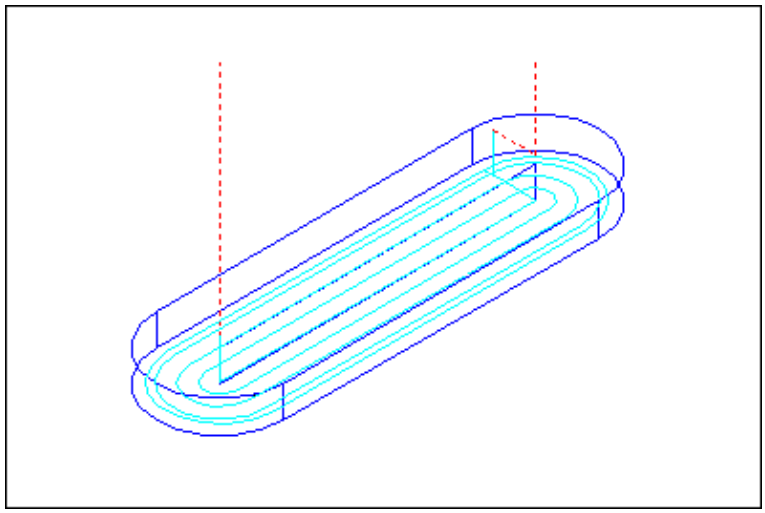
TOOL COND.		
Data item		**Meaning**
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

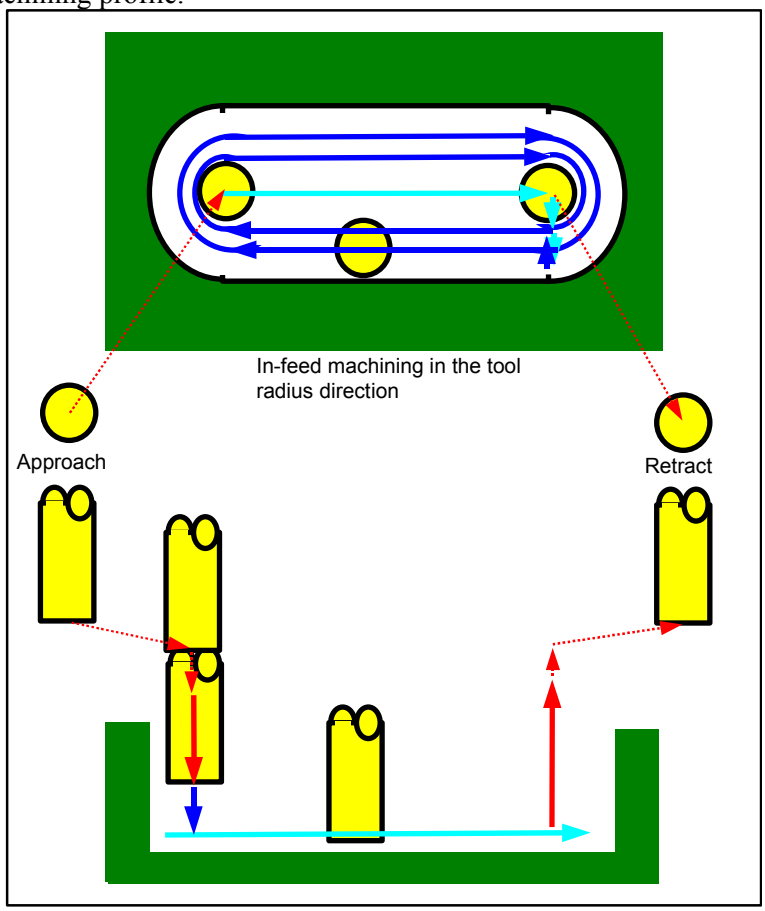
CUT COND.		
Data item		**Meaning**
T*	BOTTOM THICKNESS	Cutting allowance of the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance of the side face. The blank is regarded as 0. (radius value, positive value)
H*	BOTTOM FINISHI AMT.	Finishing allowance of the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cutting mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cutting mode, assuming that the tool is rotating clockwise. (COPY)
B	CLEARANCE OF RADIUS	Distance between the wall of a groove and a tool retract position in the tool radius direction (radius value, positive value) Remark1) When one groove cutting operation is completed, the tool performs a retract operation in the tool axis direction from the side face of the groove by this clearance amount. Remark2) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**

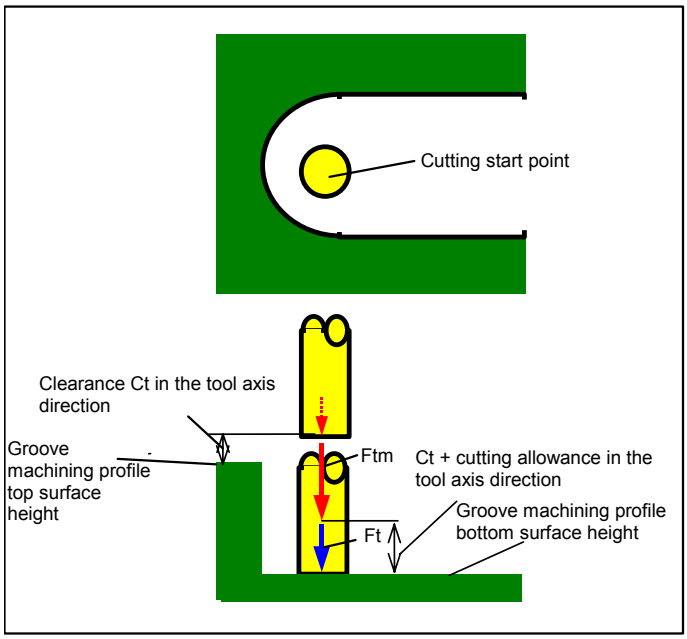


The following tool path is created to cut off the inside of a groove machining profile.



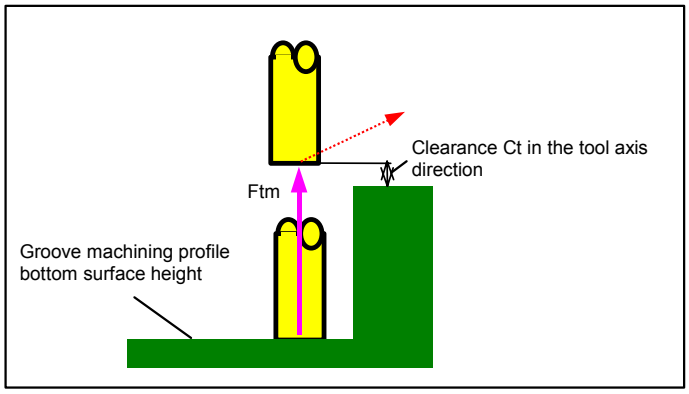
<1> The tool approaches a point above the cutting start point of a groove machining profile.
<2> The tool cuts in the groove machining profile in the tool radius direction.
<3> The tool retracts.

- Approach



<1> The tool moves to the position "groove machining profile top height + clearance (Ct) in the tool axis direction" by rapid traverse.
<2> The tool moves to the position "groove machining profile bottom surface height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for cutting in the tool axis direction.
<3> The tool cuts in to the height of the groove machining profile bottom surface at the feedrate (Ft) specified for cutting in the tool axis direction.

- Retract

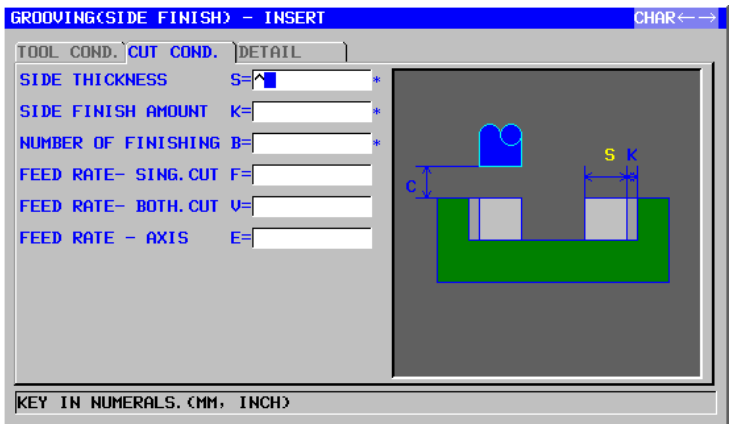


<1> The tool retracts from the height of the groove machining profile bottom surface to the position "groove machining profile top surface height + clearance (Ct) in the tool axis direction" at the feedrate (Ftm) specified for movement in the tool axis direction.

- In-feed machining in the tool radius direction

This operation is the same as grooving (roughing).
See descriptions about grooving (roughing) for details.

Side face finishing: G1052



TOOL COND.		
Data item		**Meaning**
I	INPUT	[INPUT] : Inputs a cutter compensation value directly. [REF.] : Inputs a cutter compensation number to read a cutter compensation value by that number.
D	TOOL DIAMETER	End mill diameter (positive value) Remark) This item is indicated only when [INPUT] is selected for INPUT.
M	CUTTER COMP.NO.	Cutter compensation number of an end mill (positive value) Remark) This item is indicated only when [REF.] is selected for INPUT.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'
3 Tab 'TOOL COND.' is not enabled when bit 3 (GF3) of parameter No. 27081 = 1.

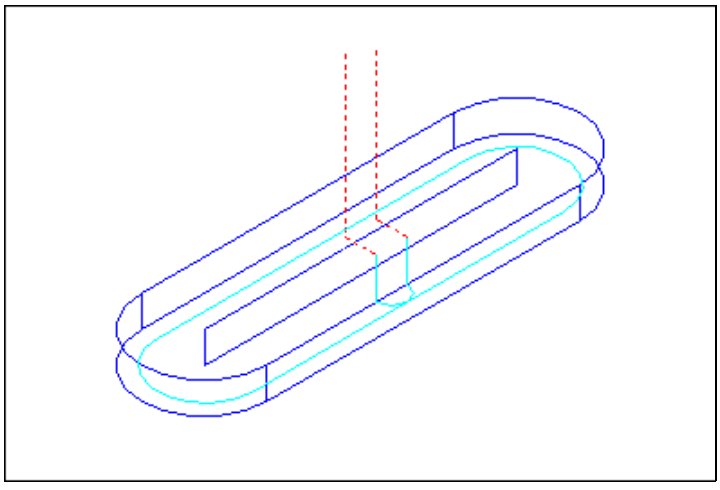
CUT COND.		
Data item		**Meaning**
S*	SIDE THICKNESS	Cutting allowance of the side face. The blank is regarded as 0. (radius value, positive value)
K*	SIDE FINISH AMOUNT	Finishing allowance of the side face. The blank is regarded as 0. (radius value, positive value)
B	NUMBER OF FINISHING	Number of cuts for finishing (positive value) Remark) Depth of each cut = (side surplus thickness)/(number of finishing cuts)

CUT COND.		
Data item		**Meaning**
F	FEED RATE-SING.CUT	Feedrate applicable when only the one-side cutter portion of an end mill is used for cutting. This feedrate is used for cutting in retract operation and on the side face other than initial cutting.
V	FEED RATE- BOTH.CUT	Feedrate applicable when the entire front side of an end mill is used for cutting. This feedrate is used for initial cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cutting mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cutting mode, assuming that the tool is rotating clockwise. (COPY)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)

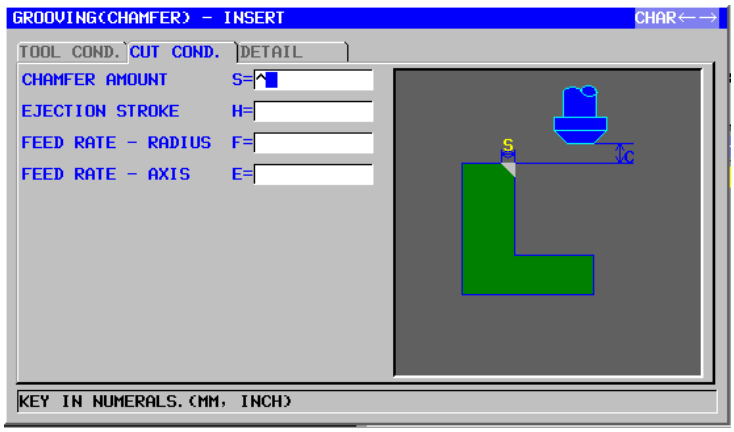
DETAIL		
Data item		**Meaning**
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**



The side-face contour of a groove machining profile is finished.
The tool path for it is the same as for contouring (side-face finishing).
See descriptions about contouring (side-face finishing) for details.

Chamfer: G1053



TOOL COND.		
Data item		**Meaning**
K	TOOL SMALL DIAMETER	Diameter of the tip of a chamfering tool (positive value)

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CUT COND.		
Data item		**Meaning**
S	CHAMFER AMOUNT	Chamfer length (radius value, positive value)
H	EJECTION STROKE	Distance between the tip of a chamfering tool and an actually cut position in the tool axis direction (radius value, positive value)
F	FEED RATE - RADIUS	Feedrate applicable when cutting is performed in the tool radius direction
E	FEED RATE - AXIS	Feedrate applicable when cutting is performed in the tool axis direction

DETAIL		
Data item		**Meaning**
W	UP CUT/DOWN CUT	[UP CUT] : Performs machining in up-cutting mode, assuming that the tool is rotating clockwise. [DWNCUT] : Performs machining in down-cutting mode, assuming that the tool is rotating clockwise. (COPY)

DETAIL		
Data item		**Meaning**
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.
P	APPROACH TYPE	[ARC] : Approaches a side face along an arc. [TANGEN]: Approaches a side face along the straight line tangent to the first figure in side face cutting. [VERTIC] : Approaches a side face along the straight line normal to the first figure in side face cutting. (COPY)
R	APPROACH RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
A*	APPROACH ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for APPROACH TYPE. (COPY)
Q	ESCAPE TYPE	[ARC] : Retracts from a side face along an arc. [TANGEN]: Retracts from a side face along the straight line tangent to the last figure in side face cutting. [VERTIC] : Retracts from a side face along the straight line normal to the last figure in side face cutting. (COPY)
X	ESCAPE RAD./DIST.	Radius when [ARC] is specified. Straight line length when [TANGEN] or [VERTIC] is specified. (radius value, positive value) Remark) By referring to the parameter No.27010 (minimum clamp value), the system sets the data as Initial value.
Y*	ESCAPE ANGLE	Center angle of the arc when [ARC] is specified. The default is 90 degrees. (positive value) Remark) This data item is indicated only when [ARC] is selected for ESCAPE TYPE. (COPY)
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

- **Tool path**

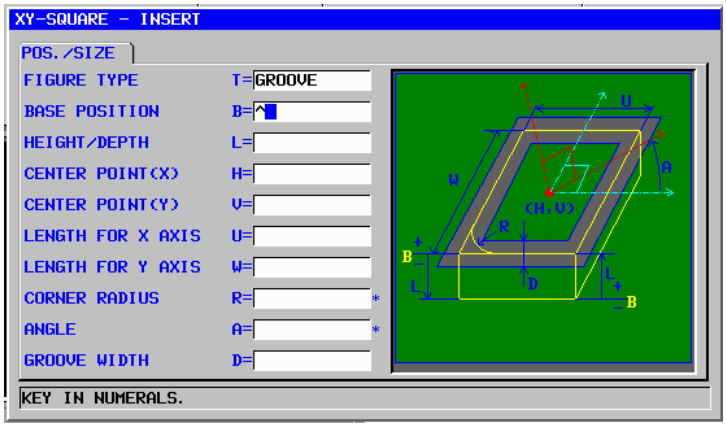
The top surface of a wall of a groove is chamfered. The tool path for it is the same as for contouring (chamfering).
See descriptions abut contouring (chamfering) for details.

1.7.2 Fixed Form Figure Blocks for Grooving (XY Plane)

As fixed form figures for grooving, a "square", "circle", "track", and "radial grooves" are available. When any of these pattern figures is specified, a groove with a specified width is cut along the contour.

NOTE
Each of a "square", "circle", and "track" is a closed figure. In actual grooving, sets of a start point and end point are determined, and cutting from the start point to the end point of a set is repeated.

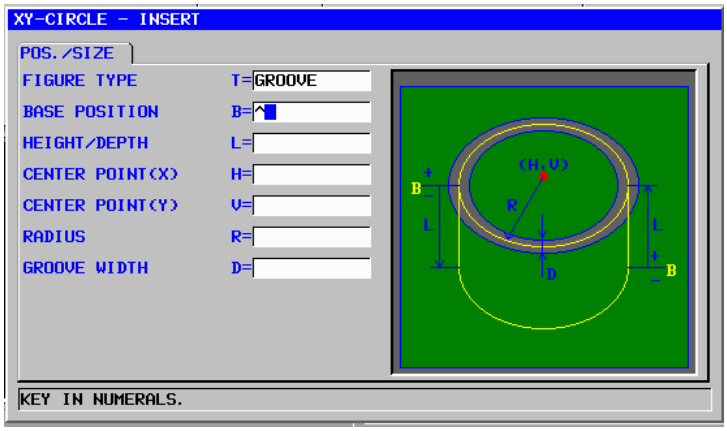
Square: G1220 (XY plane)



POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When grooving is selected as machining type, be sure to select [GROOVE].
B	BASE POSITION	Z coordinate of the top surface or the bottom of the side face of a part to be subject to contouring (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the side face by using a negative value (radius value). → Depth When the bottom of a side face is selected as the BASE POSITION, specify the distance to the top surface of the workpiece by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a rectangular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a rectangular figure
U	LENGTH FOR X AXIS	Length of the side in the X-axis direction (radius value, positive value)

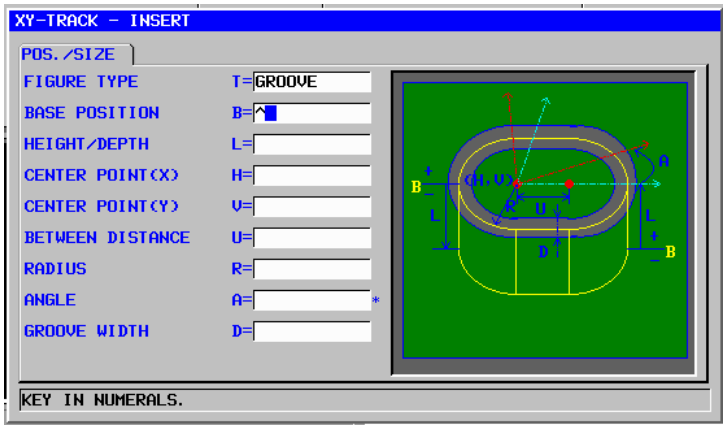
POS./SIZE		
Data item		Meaning
W	LENGTH FOR Y AXIS	Length of the side in the Y-axis direction (radius value, positive value)
R*	CORNER RADIUS	Radius for corner rounding (positive value)
A*	ANGLE	Inclination angle of a rectangular figure relative to the X-axis (positive or negative value)
D	GROOVE WIDTH	Groove width (radius value, positive value)

Circle: G1221 (XY plane)



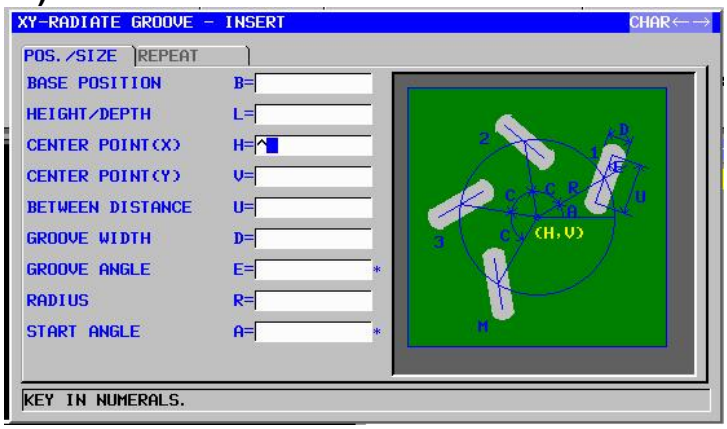
POS./SIZE		
Data item		**Meaning**
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When grooving is selected as machining type, be sure to select [GROOVE].
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a circular figure
V	CENTER POINT (Y)	Y coordinate of the center position of a circular figure
R	RADIUS	Radius of a circular figure (radius value, positive value)
D	GROOVE WIDTH	Groove width (radius value, positive value)

Track: G1222 (XY plane)



POS./SIZE		
Data item		Meaning
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When grooving is selected as machining type, be sure to select [GROOVE].
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a left semicircle
V	CENTER POINT (Y)	Y coordinate of the center position of a left semicircle
U	BETWEEN DISTANCE	Distance between the centers of the right and left semicircles (radius value, positive value)
R	RADIUS	Radius of the left and right semicircles (radius value, positive value)
A*	ANGLE	Inclination angle of a track figure relative to the X-axis (positive or negative value)
D	GROOVE WIDTH	Groove width (radius value, positive value)

Radial grooves: G1223 (XY plane)



POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
H	CENTER POINT (X)	X coordinate of the center position of a circle where radial grooves are placed
V	CENTER POINT (Y)	Y coordinate of the center position of a circle where radial grooves are placed
U	BETWEEN DISTANCE	Distance between the centers of two semicircles along a radial groove (in a track figure) (radius value, positive value)
D	GROOVE WIDTH	Width of a radial groove (radius value, positive value)
E*	GROOVE ANGLE	Groove inclination angle in an arc where radial grooves are placed, in the radial direction (positive or negative value)
R	RADIUS	Radius of an arc where radial grooves are placed (radius value, positive value)
A*	START ANGLE	Center angle of the first groove position relative to the X-axis (positive or negative value)

REPEAT		
Data item		**Meaning**
C*	PITCH ANGLE	Center angle formed by two adjacent grooves (positive or negative value)
M*	BROOVE NUMBER	Number of grooves to be cut (positive value)

1.7.3 Fixed Form Figure Blocks for Grooving (YZ Plane, XC Plane)

The same fixed-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane and the XC plane (polar coordinate interpolation plane). They are provided with the following G codes.
The data to be set for the YZ and XC planes is the same as for the XY plane except that the XY plane is changed to the YZ or XC plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ plane) or the Z-axis (XC plane).

Square : G1320 (YZ plane)
Circle : G1321 (YZ plane)
Track : G1322 (YZ plane)
Radial groove : G1323 (YZ plane)
Square : G1520 (XC plane, polar coordinate interpolation plane)
Circle : G1521 (XC plane, polar coordinate interpolation plane)
Track : G1522 (XC plane, polar coordinate interpolation plane)
Radial groove : G1523 (XC plane, polar coordinate interpolation plane)

NOTE
When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.

1.7.4 Arbitrary Figure Blocks for Grooving (XY Plane)

When an arbitrary figure for grooving is input, data such as a figure type and machining reference position is specified in the start point block. Other data items to be input such as a straight line and arc are exactly the same as for arbitrary figures of other machining types.
So, this subsection describes only the start point block of an arbitrary figure for pocketing.
For other arbitrary figures, see Chapter 5, "DETAILED DESCRIPTIONS ABOUT ENTERING ARBITRARY FIGURES", in Part II.
When an arbitrary figure is specified for grooving, a groove with a specified width is cut along the arbitrary figure line.

Start point: G1200 (XY plane)

ELEMENT		
Data item		**Meaning**
X	START POINT X	X coordinate of the start point of an arbitrary figure
Y	START POINT Y	Y coordinate of the start point of an arbitrary figure
T	FIGURE TYPE	[FACE] : Used as a figure for facing [CONVEX] : Used as an outer figure for contouring [CONCAVE]: Used as an inner figure for contouring or as a figure for pocketing [GROOVE] : Used as a figure for grooving Remark) When grooving is selected as machining type, be sure to select [GROOVE].
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
D	GROOVE WIDTH	Groove width (radius value, positive value)

1.7.5 Arbitrary Figure Blocks for Grooving (YZ Plane, XC Plane, ZC Plane, XA Plane)

The same arbitrary-figure block types as for the XY plane explained in the previous subsection are available for the YZ plane, the XC plane (polar coordinate interpolation plane), ZC plane and XA plane (cylindrical surface). Their start points are specified with the following G codes.
The data to be set for the YZ, XC, ZC, XA planes is the same as for the XY plane except that the XY plane is changed to the YZ, XC, ZC or XA plane and that the direction in which the tool moves to cut is changed to the X-axis (YZ and ZC planes) or the Z-axis (XC and XA plane).

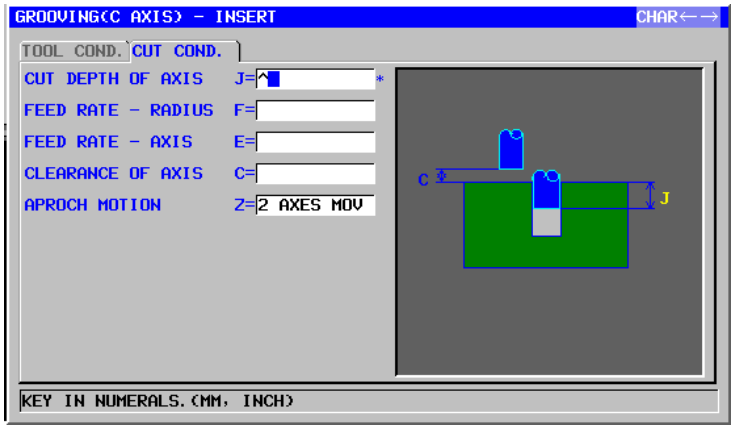
Start point : G1300 (YZ plane)
Start point : G1500 (XC plane, polar coordinate interpolation plane)
Start point : G1600 (ZC plane, cylindrical surface)
Start point : G1700 (XA plane, cylindrical surface)

NOTE
1 When performing machining (polar coordinate interpolation) on the XC plane, note the following:
(When bit 2 of parameter No. 27000 = 0)
The mode needs to be switched to the polar coordinate interpolation mode beforehand. Specifically, enter G12.1 before the machining type.
Enter G13.1 for canceling polar coordinate interpolation as required.
(When bit 2 of parameter No. 27000 = 1)
G12.1 and G13.1 are automatically output before and after cycle machining, respectively.
2 When performing machining (cylindrical interpolation) on the ZC and XA plane, note the following:
(When bit 3 of parameter No. 27000 = 0)
The mode needs to be switched to the cylindrical coordinate interpolation mode beforehand. Specifically, enter G07.1C (cylinder radius) before the machining type.
Enter G07.1C0 for canceling cylindrical interpolation as required.
(When bit 3 of parameter No. 27000 = 1)
G07.1C (cylinder radius) and G07.1C0 are automatically output before and after cycle machining, respectively.
3 The following parameter setting is necessary to execute XA plane cycle.
- No.27003#2=1, 27003#1=0 and 27003#0=0

1.8 C-AXIS GROOVING

1.8.1 Machining Type Blocks for C-axis Grooving

Roughing: G1056



TOOL COND.		
Data item		Meaning
D	TOOL DIAMETER	End mill diameter

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27002 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		Meaning
J*	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation. By default, one cutting operation is used. (radius value, positive value)
F	FEED RATE- RADIUS	Feedrate applicable when cutting is performed in the side face direction of the end mill
E	FEEDRATE - AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value) Remark) By referring to the parameter No.27009 (minimum clamp value), the system sets the data as Initial value.

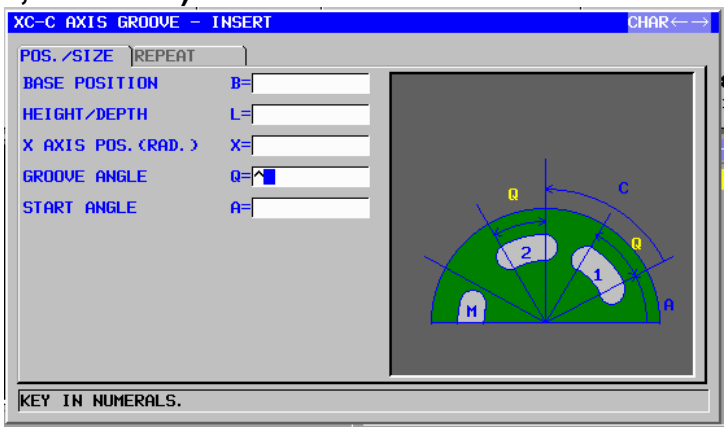
CUT COND.		
Data item		**Meaning**
Z	APROCH MOTION	[2 AXES] : When moving from the current position to the machining start point, the tool first moves in the machining plane in two-axis synchronous operation and then moves along the tool axis. (initial value) [3 AXES] : The tool moves from the current position to the machining start point in three-axis synchronous operation.

1.8.2 Figure Blocks for C-axis Grooving and A-axis Grooving

As C-axis grooving figures, a "C-axis groove on the polar coordinate plane (XC plane)", "X-axis groove on the polar coordinate plane (XC plane)", "C-axis groove on the cylindrical surface (ZC plane)", "A-axis groove on the cylindrical surface (XA plane)", "Z-axis groove on the cylindrical surface (ZC plane)" and "X-axis groove on the cylindrical surface (XA plane)" are available. When any of these figures is specified, a groove with the diameter as large as the end mill diameter is cut.
In C-axis grooving, in-feed cutting in the tool axis direction is performed, but in-feed cutting in the width direction is not performed.

NOTE
C-axis grooving uses neither polar coordinate interpolation nor cylinder interpolation. So, it is unnecessary to switch to the polar coordinate interpolation or cylinder interpolation mode.

C-axis groove: G1570 (XC plane, end face)

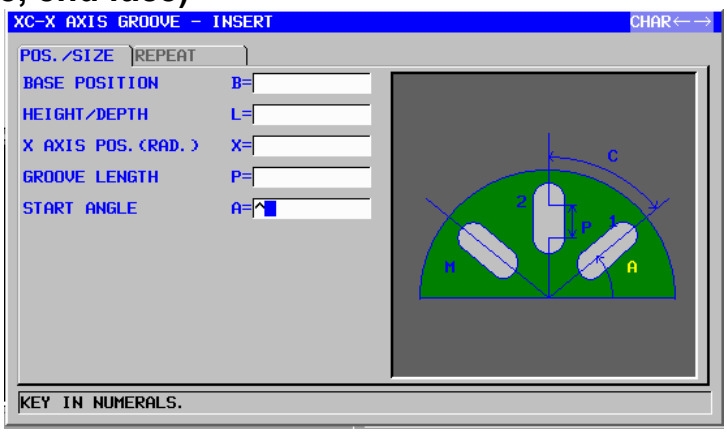


On the end face of a workpiece, circular grooves are cut by rotating the C-axis with the X-axis position of the tool fixed. Multiple grooves of the same figure can be cut.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
X	X AXIS POS.(RAD.)	X coordinate of the center line of a groove (radius value)
Q	GROOVE ANGLE	Center angle formed by the start point (tool center) and end point (tool center) of a groove (positive or negative value)
A	START ANGLE	C coordinate of the start point (tool center) of the first groove

REPEAT		
Data item		**Meaning**
C*	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves (positive or negative value)
M*	GROOVE NUMBER	Number of grooves to be cut (positive value)

X-axis groove: G1571 (XC plane, end face)

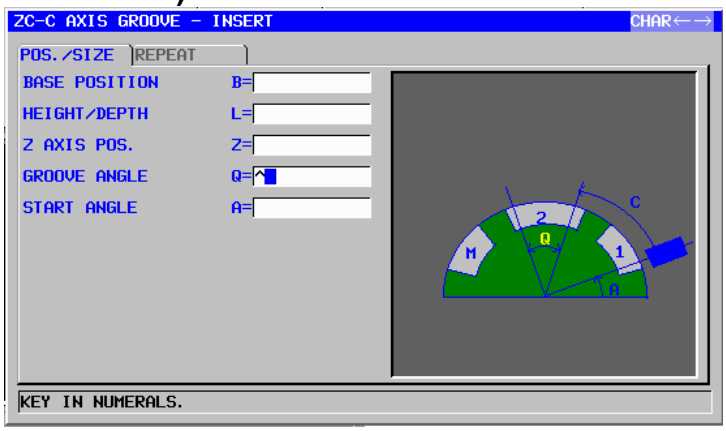


On the end face of a workpiece, radial grooves are cut by moving the tool in the X-axis direction with the C-axis position fixed. Multiple grooves of the same figure can be cut.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
X	X AXIS POS.(RAD.)	X coordinate of the start point of a groove (radius value)
P	GROOVE LENGTH	Distance between the start point (tool center) and end point (tool center) of a groove (radius value, positive/negative value)
A	START ANGLE	C coordinate of the start point (tool center) of the first groove

REPEAT		
Data item		**Meaning**
C*	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves (positive or negative value)
M*	GROOVE NUMBER	Number of grooves to be cut (positive value)

C-axis groove: G1670 (cylindrical surface)

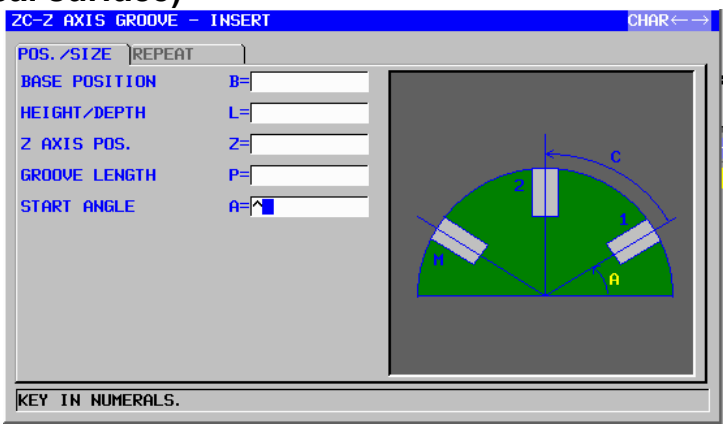


On the peripheral surface of a workpiece, grooves are cut by rotating the C-axis with the Z-axis position of the tool fixed. Multiple grooves of the same figure can be cut.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	X coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
Z	Z AXIS POS.	Z coordinate of the center line of a groove
Q	GROOVE ANGLE	Center angle formed by the start point (tool center) and end point (tool center) of a groove (positive or negative value)
A	START ANGLE	C coordinate of the start point (tool center) of the first groove

REPEAT		
Data item		**Meaning**
C*	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves (positive or negative value)
M*	GROOVE NUMBER	Number of grooves to be cut (positive value)

Z-axis groove: G1671 (cylindrical surface)



On the peripheral surface of a workpiece, straight grooves are cut by moving the tool in the Z-axis direction with the C-axis position fixed. Multiple grooves of the same figure can be cut.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	X coordinate of the bottom of a groove or the top surface of a workpiece subject to grooving (in the tool axis direction)
L	HEIGHT/DEPTH	When the top surface of a workpiece is selected as the BASE POSITION, specify the distance to the bottom of the groove by using a negative value (radius value). → Depth When the bottom of a groove is selected as the BASE POSITION, specify the distance to the top surface of the groove by using a positive value (radius value). → Height
Z	Z AXIS POS.	Z coordinate of the start point of a groove
P	GROOVE LENGTH	Distance between the start point (tool center) and end point (tool center) of a groove (radius value, positive/negative value)
A	START ANGLE	C coordinate of the start point (tool center) of the first groove

REPEAT		
Data item		**Meaning**
C*	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves (positive or negative value)
M*	GROOVE NUMBER	Number of grooves to be cut (positive value)

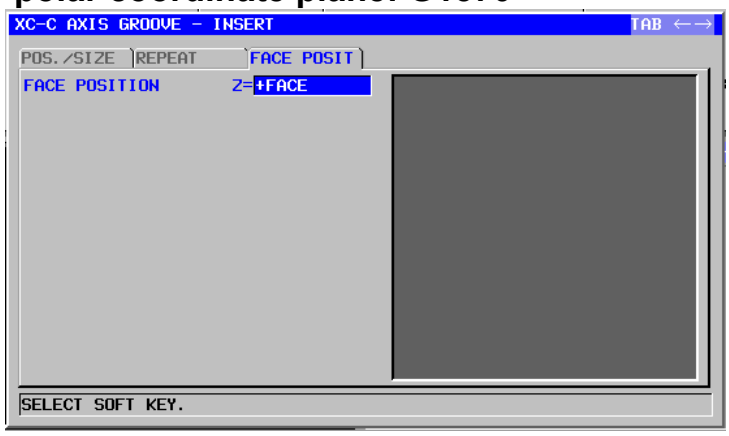
1.9 REAR END FACING BY MILLING

1.9.1 Rear End Facing

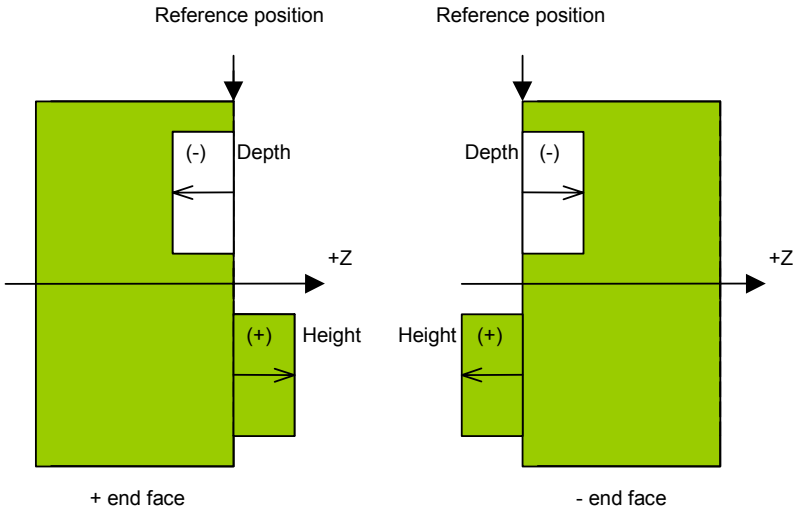
By setting bit 4 of parameter No. 27000 to 1, the input item "FACE POSITION" is displayed on the figure menu for milling below. By entering this data, rear end facing is enabled.

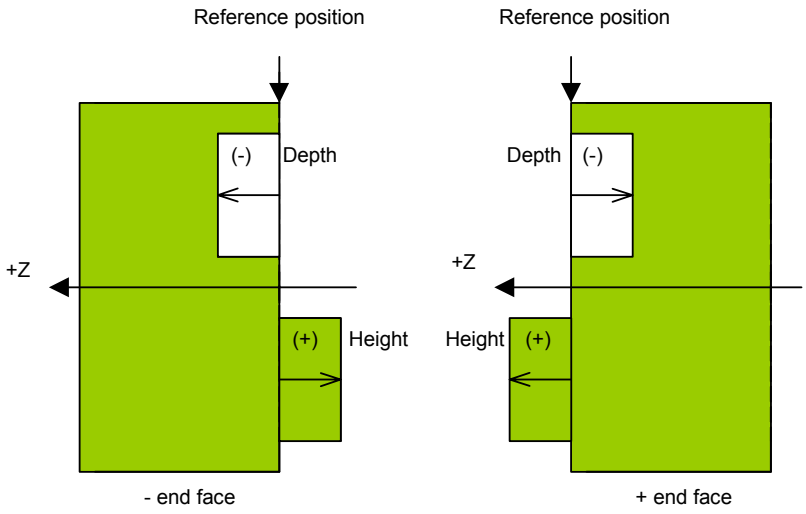
1. Arbitrary-figure (XY plane) - Start point : G1200
2. Hole position (XY plane) - Random points : G1210
3. Hole position (XY plane) - Linear points (same interval) : G1211
4. Hole position (XY plane) - Linear points (different interval) : G1212
5. Hole position (XY plane) - Grid points : G1213
6. Hole position (XY plane) - Rectangle points : G1214
7. Hole position (XY plane) - Circle points : G1215
8. Hole position (XY plane) - Arc points (same interval): G1216
9. Hole position (XY plane) - Arc points (different interval) : G1217
10. Fixed-figure (XY plane) - Rectangle : G1220
11. Fixed-figure (XY plane) - Circle : G1221
12. Fixed-figure (XY plane) - Track : G1222
13. Fixed-figure (XY plane) - Radial groove : G1223
14. Arbitrary-figure (XC plane) - Start point : G1500
15. Fixed-figure (XC plane) - Rectangle : G1520
16. Fixed-figure (XC plane) - Circle : G1521
17. Fixed-figure (XC plane) - Track : G1522
18. Fixed-figure (XC plane) - Radial groove : G1523
19. C-axis groove on the polar coordinate plane: G1570
20. X-axis groove on the polar coordinate plane : G1571
21. C-axis hole on the polar coordinate plane (circle points) : G1572
22. C-axis hole on the polar coordinate plane (random points) : G1573

Example) C-axis groove on the polar coordinate plane: G1570



FACE POSIT		
Data item		**Meaning**
Z	FACE POSITION	[+FACE]: References the figure below (+ end face). [-FACE]: References the figure below (- end face).





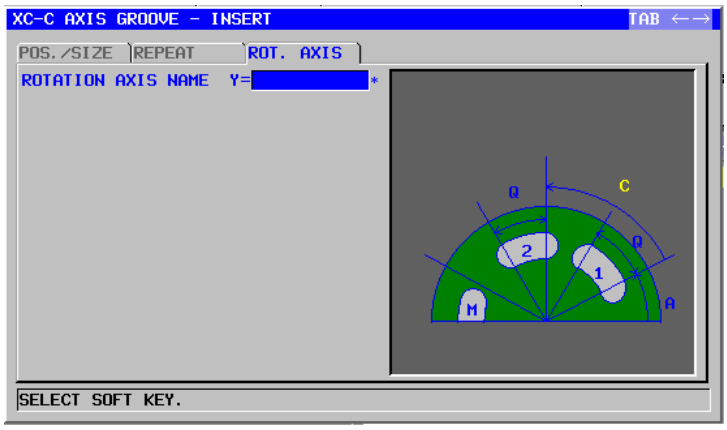
1.10 ADDRESS SETTING OF ROTATION AXIS

1.10.1 Support for C-Axis Machining with Rotation Axis

By setting bit 0 of parameter No. 27001 to 1, the input item "ROTATION AXIS NAME" is displayed on the figure menu for milling below. By entering this data, C-axis machining using a rotation axis is enabled.

1. Rectangle on the polar coordinate plane: G1520
2. Circle on the polar coordinate plane : G1521
3. Track on the polar coordinate plane : G1522
4. Radial groove on the polar coordinate plane : G1523
5. C-axis groove on the polar coordinate plane: G1570
6. X-axis groove on the polar coordinate plane: G1571
7. C-axis hole on the polar coordinate plane (circle points) : G1572
8. C-axis hole on the polar coordinate plane (random points) : G1573
9. C-axis groove on the cylindrical surface : G1670
10. X-axis groove on the cylindrical surface : G1671
11. C-axis hole on the cylindrical surface (circle points) : G1672
12. C-axis hole on the cylindrical surface (random points) : G1673
13. Arbitrary-figure on the end face : G1500
13. Arbitrary-figure on the cylindrical surface : G1600

Example) C-axis groove on the polar coordinate plane: G1570



ROT. AXIS		
Data item		**Meaning**
Y	ROTATION AXIS NAME	When bit 1 of parameter No. 27001 #1 = 1 [C]: The rotation axis is the C-axis. [A]: The rotation axis is the A-axis. When bit 2 of parameter No. 27001 #2 = 1 [C]: The rotation axis is the C-axis. [B]: The rotation axis is the B-axis. When bit 3 of parameter No. 27001 #3 = 1 [C]: The rotation axis is the C-axis. [E]: The rotation axis is the E-axis.

NOTE
This data item is enabled when the parameter No.27001#0 = 1.

1.11 C AXIS CLAMPING M CODE OUTPUT

1.11.1 Outline

C axis clamping and unclamping M codes are automatically output in C axis cycles as followings, which position C axis in the cycle motion.

* C axis represents a rotating axis around Z axis in this specifications.

(1) Hole machining cycles
In hole machining cycles combined with following figures, C axis clamping and unclamping M codes are automatically output.

a) C-axis hole on the end face : G1572
b) C-axis hole on the end face (arbitrary) : G1573
c) C-axis hole on the cylindrical surface (arbitrary) : G1672
d) C-axis hole on the cylindrical surface (arbitrary) : G1673

(2) Grooving cycles
In grooving cycles combined with following figures, C axis clamping and unclamping M codes are automatically output.

a) X-axis groove on the end face : G1571
b) Z-axis groove on the cylindrical surface : G1671

1.11.2 Value of M Code Output

M code set in following parameters are output
When zero is set, M code is not output.

(1) C axis clamping M code for main spindle : Parameter No.27005
(2) C axis unclamping M code for main spindle : Parameter No.27006
(3) C axis clamping M code for sub spindle : Parameter No.27011
(4) C axis unclamping M code for sub spindle : Parameter No.27012

1.11.3 Distinction between Main and Sub Spindle

When an axis name “C” is specified in figure command and no axis name is specified, M code set in parameter No.27005 or 27006 for main spindle is output.

When an axis name “A”, “B”, or “E” is specified in figure command, M code set in parameter No.27011 or 27012 for sub spindle is output.

1.11.4 Position of M Code Output

(1) Hole machining cycle

```
G90
G17
Mb.....*1
G00 X(x1) C(c1) Z(I point)
G99
G81 Z(depth) R(R point) F(feed) Ma .....*1
Mb
X(x2) C(c2) Ma .....*1
Mb
X(x3) C(c3) Ma .....*1
Mb
G80
G00 Z(I point)
```

*1 Ma means C axis clamping M code, Mb means C axis unclamping one.
a, b should be set in parameters No.27005, No.27006, No.27011 and No.27012.
When value of the parameter is zero, no M code is output.

(2) Grooving Cycle

a) X-axis groove on the end face : G1571

```
G1056J10.C2.F100.E100.Z2
            C axis grooving cycle command
G1571B0.L-10.X40.P20.A0.C120.M2.Z1.Y1.
            X-axis groove on the end face
```

In case above commands are specified, C axis unclamping M code is output at the beginning of a cycle, C axis clamping M code is output before cutting motion, and C axis unclamping M code is output after cutting motion during a cycle

```
Mβ .....* 1
G17
G0X80.C0.
```

```
G0Z2.
Mα .....* 1
G1Z-10.F100.
G1X40.F100.
G1Z2.F100.
Mβ .....* 1
G17
G0X80.C120.
G0Z2.
Mα .....* 1
G1Z-10.F100.
G1X40.F100.
G1Z2.F100.
Mb .....* 1
```

*1 Mα means C axis clamping M code, Mβ means C axis un clamping one.
α, β should be set in parameters No.27005, No.27006, No.27011 and No.27012.
When value of the parameter is zero, no M code is output.

b) Z-axis groove on the cylindrical surface : G1671

```
G1056J10.C2.F100.E100.Z2.
      C axis grooving command
G1671B50.L-10.Z-30.P20.A0.C120.M2.Y1.
      Z-axis groove on the cylindrical surface
```

In case above commands are specified, C axis unclamping M code is output at the beginning of a cycle, C axis clamping M code is output before cutting motion, and C axis unclamping M code is output after cutting motion during a cycle

```
Mβ .....*1
G19
G0Z-30.C0.
G0X104.
Mα.....* 1
G1X80.F100.
G1Z-50.F100.
G1X104.F100.
Mβ .....* 1
G19
G0Z-30.C120.
G0X104.
Mα.....* 1
G1X80.F100.
G1Z-50.F100.
G1X104.F100.
Mβ .....* 1
```

*1 Mα means C axis clamping M code, Mβ means C axis un clamping one.
α, β should be set in parameters No.27005, No.27006, No.27011 and No.27012. When value of the parameter is zero, no M code is output.

2 TURNING

With MANUAL GUIDE *i*, the cycles motions listed below are available for turning.

Hole machining (workpiece rotation)			
	Machining type block	G1100	Center drilling
		G1101	Drilling
		G1102	Tapping
		G1103	Reaming
		G1104	Boring

Turning			
	Machining type block	G1120	Outer surface roughing
		G1121	Inner surface roughing
		G1122	End face roughing
		G1123	Outer surface semifinishing
		G1124	Inner surface semifinishing
		G1125	End face semifinishing
		G1126	Outer surface finishing
		G1127	Inner surface finishing
		G1128	End face finishing
	Turning figure block (ZX plane)	G1450	Start point
		G1451	Straight line
		G1452	Arc (CW)
		G1453	Arc (CCW)
		G1454	Chamfer
		G1455	Corner rounding
		G1456	End

Turning groove			
	Machining type block	G1130	Outer surface roughing
		G1131	Inner surface roughing
		G1132	End face roughing
		G1133	Outer surface roughing and finishing
		G1134	Inner surface roughing and finishing
		G1135	End face roughing and finishing
		G1136	Outer surface finishing
		G1137	Inner surface finishing
		G1138	End face finishing
	Groove figure block (ZX plane)	G1470	Outer normal groove
		G1471	Outer trapezoidal groove
		G1472	Inner normal groove
		G1473	Inner trapezoidal groove
		G1474	End face normal groove
		G1475	End face trapezoidal groove

Threading			
	Machining type block	G1140	Outer surface
		G1141	Inner surface
	Thread figure block (ZX plane)	G1460	General-purpose thread
		G1461	Metric thread
		G1462	Unified thread
		G1463	PT thread
		G1464	PF thread

NOTE

1 When cycle motions are input with MANUAL GUIDE *i*, sets of a machining type block and figure blocks must be input in succession for all cycle motions except turning hole machining.
For one machining type block, multiple figure blocks can be input.
However, multiple sets of arbitrary figures can be input, with one set including up to 50 figures enclosed between a start point and end block.

2 For each item marked with [*] in the tables below, a standard value is automatically set if no value is input.
The screen displays [*] to the right of a data item.

3 In some of the input data items, the system sets the last inputed data as an initial data. As to these input data items, the mark of “(COPY)” was described.

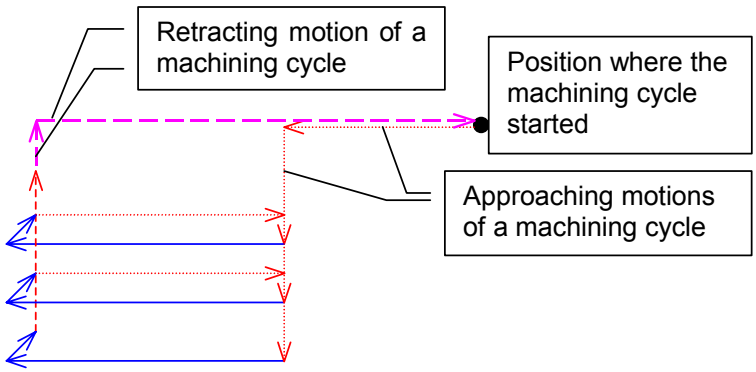
4 The motions go back to the start point where the machining cycle started after completing the all cycle motions.
(It is possible not to go back the satrt point when the parameter No.27102#7 is set to 1.)

5 Cycle machining cannot be performed in any of control modes of the high-speed and high-precision functions (commands for advanced preview control, AI advanced preview control, AI contour control, AI nano contour control, high-precision contour control, AI high-precision contour control, and AI nano high-precision contour control). To perform cycle machining, control mode must be canceled.

6 On the CNC for lathe turning with G-code system B or C, when cycle machining is specified, G90 must be specified in advance to make a switchover to the absolute coordinate system. For hole machining by turning (with the tool rotated), G98 (initial level return) or G99 (R position level return) must be specified in advance.

Remarks) Cycle retract motions

In case of No.27102#7=0, Retracting motions indicated as bloken lines in the following drawing will be outputted. The order of motion axis will be opposite to the approched motions



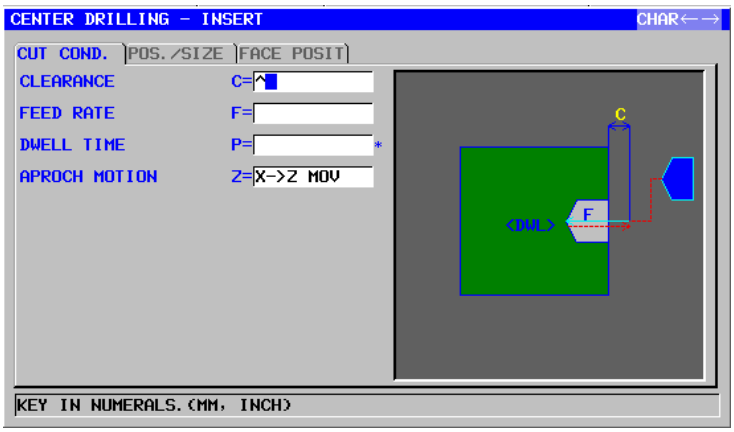
In case of carrying out plural figures machining by one G4 digit cycle, such like grooving, this retracting motions will be available at the last figure machining.

2.1 HOLE MACHINING (WORKPIECE ROTATION)

2.1.1 Machining Type Blocks for Hole Machining (Workpiece Rotation)

NOTE
Hole machining (workpiece rotation) is performed only at the center of a workpiece. So, unlike other cycle motions, figure blocks cannot be specified.

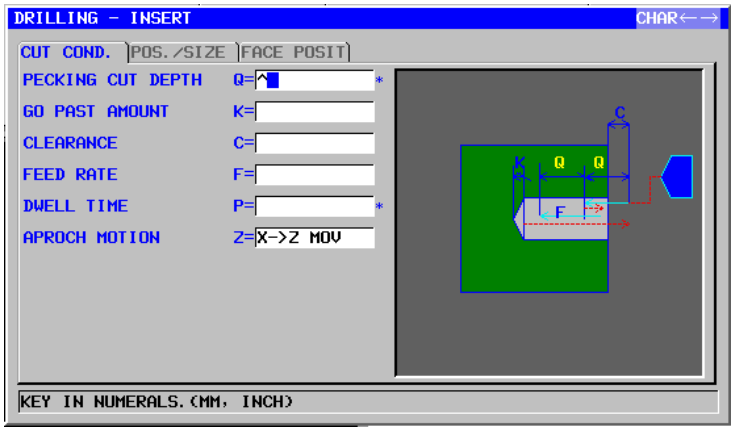
Center drilling: G1100



CUT COND.		
Data item		**Meaning**
C	CLEARANCE	Distance between the surface of a workpiece and point R (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the bottom of a hole (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the surface of a workpiece
L	CUT DEPTH	Hole depth (radius value, negative value)

Drilling: G1101

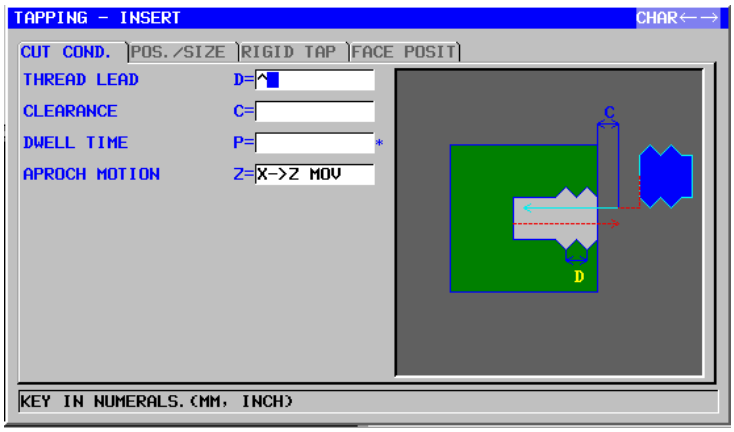


CUT COND.		
Data item		**Meaning**
Q*	PECKING CUT DEPT	Depth of cut per drilling operation (radius value, positive value) (COPY) (Note)
K*	GO PAST AMOUNT	Length of the incomplete hole at the tip of the tool (radius value, positive value) (COPY)
C	CLEARANCE	Distance between the surface of a workpiece and point R (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the bottom of a hole (in seconds) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the surface of a workpiece
L	CUT DEPTH	Hole depth (radius value, negative value)

NOTE
The system refers to the parameter No.5114 as the return amount. Therefore, please set No.5114 to the suitable value before machining.

Tapping: G1102



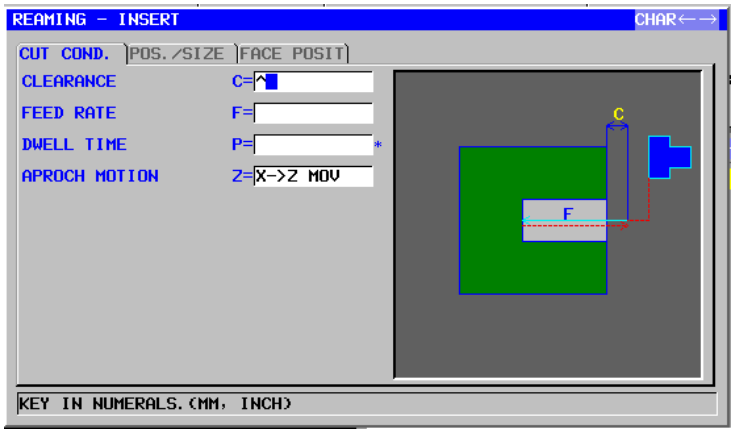
CUT COND.		
Data item		**Meaning**
D	THREAD LEAD	Lead of a tapping tool (radius value, positive value) (COPY)
C	CLEARANCE	Distance between the surface of a workpiece and point R (radius value, positive value) (COPY)
P*	DWELL TIME	Dwell time at the bottom of a hole (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the surface of a workpiece
L	CUT DEPTH	Hole depth (radius value, negative value)

RIGID TAP		
Data item		**Meaning**
R	TAP TYPE	[FLOAT] : Specifies the float tap. (initial value) [RIGID] : Specifies the rigid tap. (Note)
S	SPINDLE SPEED	Spindle speed (min^{-1})

NOTE
When you use the rigid tap M code command (No.5200#0=0), the system refers to the parameter No.5210 or No.5212 as M code. Therefore, please set No.5210 or No.5212 to the suitable value before machining.

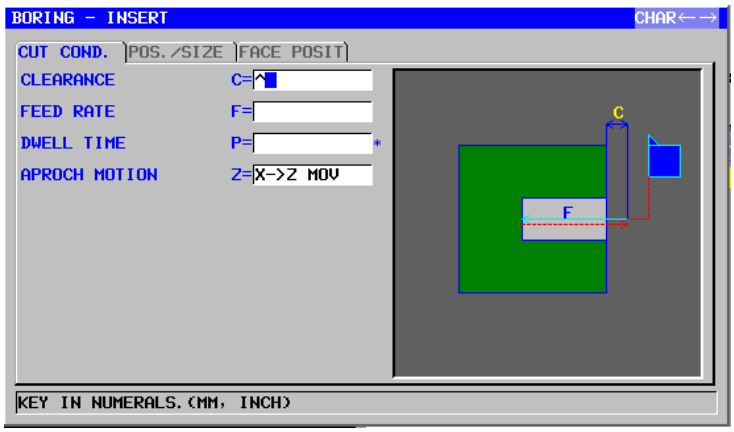
Reaming: G1103



CUT COND.		
Data item		**Meaning**
C	CLEARANCE	Distance between the surface of a workpiece and point R (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the bottom of a hole (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES]: From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the surface of a workpiece
L	CUT DEPTH	Hole depth (radius value, negative value)

Boring: G1104



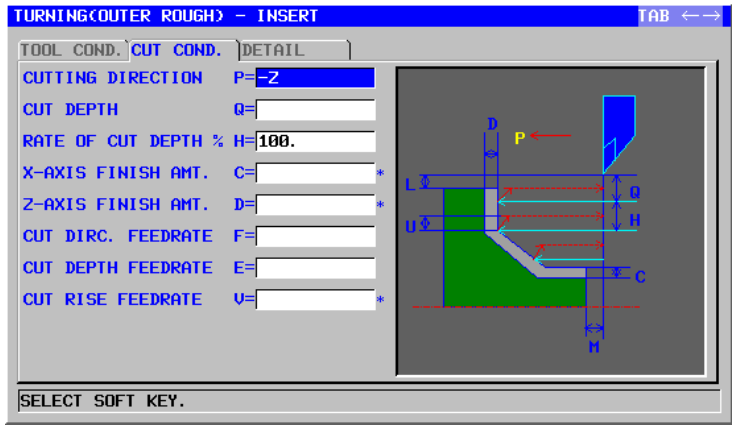
CUT COND.		
Data item		**Meaning**
C	CLEARANCE	Distance between the surface of a workpiece and point R (radius value, positive value) (COPY)
F	FEED RATE	Cutting feedrate (positive value) (COPY)
P*	DWELL TIME	Dwell time at the bottom of a hole (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

POS./SIZE		
Data item		**Meaning**
B	BASE POSITION	Z coordinate of the surface of a workpiece
L	CUT DEPTH	Hole depth (radius value, negative value)

2.2 TURNING

2.2.1 Machining Type Blocks for Turning

Outer surface roughing: G1120



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value)
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

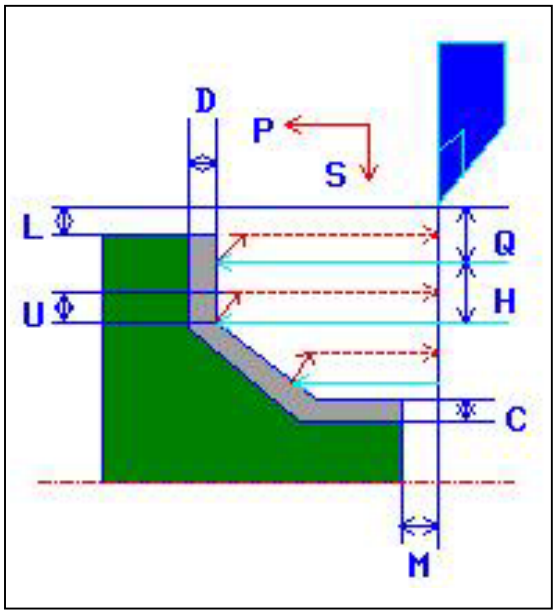
CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. Remark) [+Z] is used for cutting in the reverse direction or for machining with a subspindle. (COPY)
Q	CUT DEPTH	Depth of each cut (radius value, positive value)
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. The default is 100%, meaning that the depth of cut remains unchanged. (1 to 200, positive value)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)

CUT COND.		
Data item		**Meaning**
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)
F	CUT DIRC.FEEDRATE	Feedrate applicable when the tool cuts in the workpiece radius direction (positive value)
E	CUT DEPTH FEEDRATE	Feedrate applicable when the tool cuts in the Z-axis direction (positive value)
V	CUT RISE FEEDRATE	Feedrate applicable when the tool cuts up in the direction of retraction from the workpiece (positive value)

DETAIL		
Data item		**Meaning**
K	1ST OVERRIDE	Feedrate override value for the first cut. Specify an override value in steps of 1%. The default is 100%. (1 - 200, positive value) Remark) This data item is used, for example, to cut the black coating of a cast workpiece.
W	CUT RISE METHOD	[SPEED]: The tool retracts by a distance specified with ESCAPE AMOUNT in the XZ direction immediately after cutting. [CUT] : The tool retracts by a "retract" distance" after cutting along the figure. (COPY)
U	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut (radius value, positive value) Remark) By referring to the parameter No. 27128 (minimum clamp value), the system sets the data as an initial value.
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

DETAIL		
Data item		**Meaning**
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

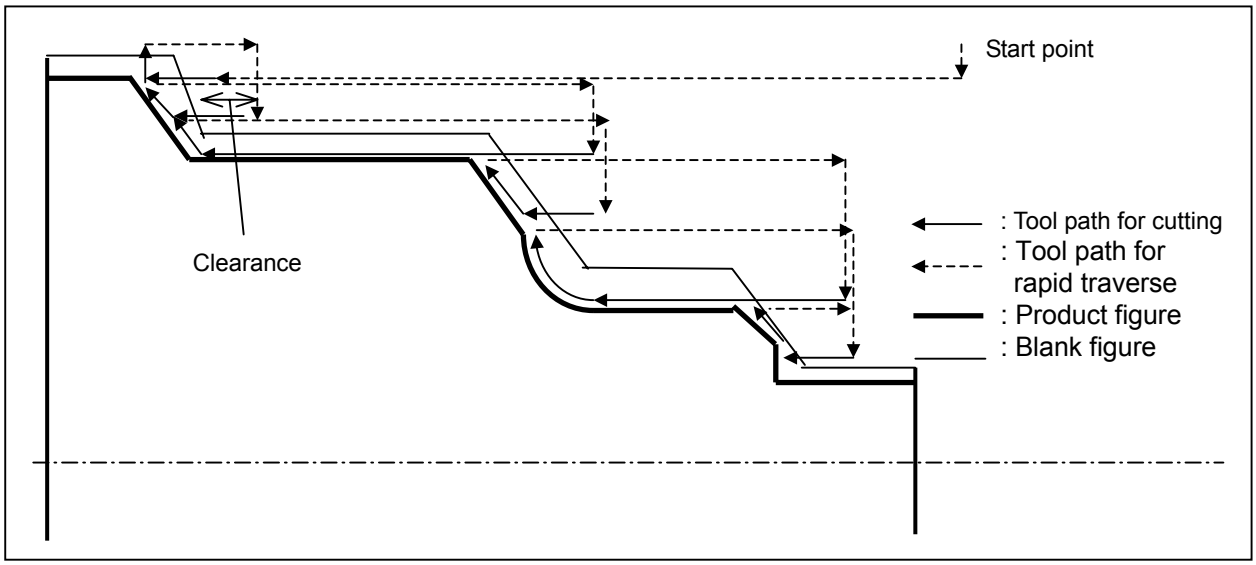
- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After cutting in in the X-axis direction at the feedrate (F) specified for the cutting direction, the tool cuts in to the entered-figure position where there is still a finishing allowance in the Z-axis direction at the feedrate (E) specified for the cut-in direction.
<3> If [SPEED] is selected, the tool immediately retracts by a return clearance (U) in the X- and Z-axis direction. If [CUT] is selected, the tool first cuts along the figure and then retracts by a return clearance (U) in the X- and Z-axis direction.
<4> The tool moves to the cut-in start position in the Z-axis direction by rapid traverse.
<5> Steps <2> to <4> are repeated until the lowest portion in the X-axis direction is reached.
<6> If there is another pocket, the tool is positioned on the pocket, and steps <2> to <4> are repeated.
<7> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (L)" in the X-axis direction at the rapid traverse rate.

NOTE
It is possible to specify "PART" and "BLANK" as "ELEMENT TYPE" for an individual arbitrary figure. This "BLANK" can be used to enter an arbitrary figure resembling a blank figure to be actually machined, so any portion other than the blank portion will not be cut. This way, optimum rough turning can be realized.

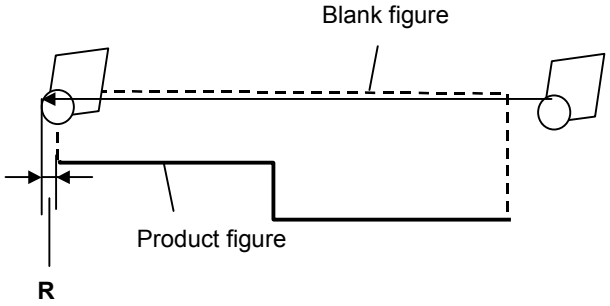
Example of outer-surface machining



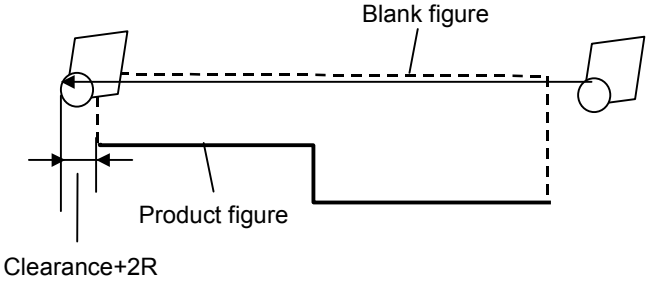
Motion of the tool in the cutting direction on the blank element portion

When the tool advances in the cutting direction, the excessive amount of travel of the tool is as describe below.

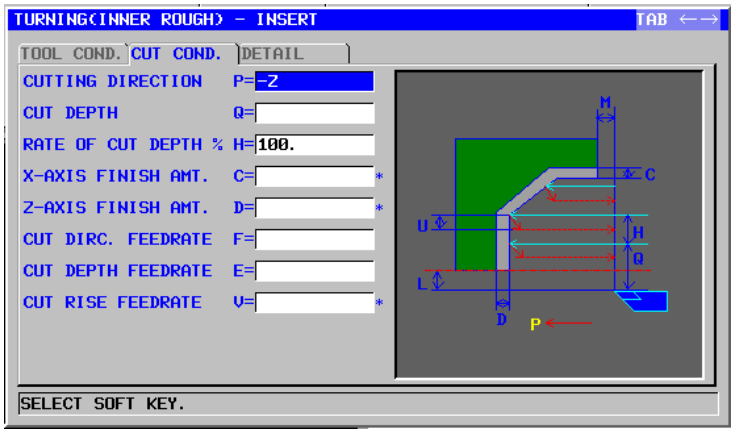
When bit 0 of parameter No. 27120 is set to 0, and the tool advances in the cutting direction, the excessive amount of travel of the tool is nose radius R if the attribute of the figure across which the tool moves is the blank element. In this case, the excessive amount of travel is the distance from the end point of the part figure.



When bit 0 of parameter No. 27120 is set to 1, and the tool advances in the cutting direction, the excessive amount of travel of the tool is [clearance + 2 × nose radius R] if the attribute of the figure across which the tool moves is the blank element. In this case, the excessive amount of travel is the distance from the blank element.



Inner surface roughing: G1121



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value)
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

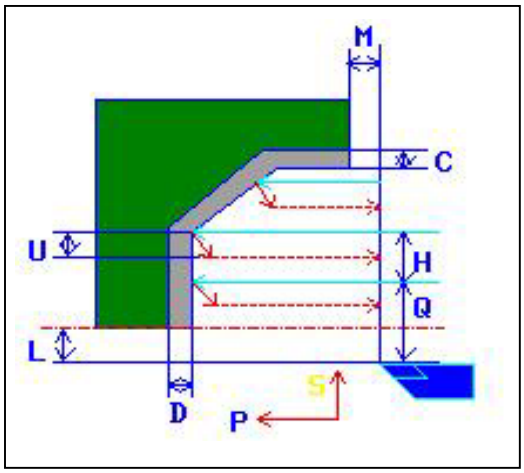
CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. Remark) [+Z] is used for cutting in the reverse direction or for machining with a subspindle. (COPY)
Q	CUT DEPTH	Depth of each cut (radius value, positive value)
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. The default is 100%, meaning that the depth of cut remains unchanged. (1 to 200, positive value)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)

CUT COND.		
Data item		**Meaning**
F	CUT DIRC.FEEDRATE	Feedrate applicable when the tool cuts in the workpiece radius direction (positive value)
E	CUT DEPTH FEEDRATE	Feedrate applicable when the tool cuts in the Z-axis direction (positive value)
V	CUT RISE FEEDRATE	Feedrate applicable when the tool cuts up in the direction of retraction from the workpiece (positive value)

DETAIL		
Data item		**Meaning**
K	1ST OVERRIDE	Feedrate override value for the first cut. Specify an override value in steps of 1%. The default is 100%. (1 - 200, positive value) Remark) This data item is used, for example, to cut the black coating of a cast workpiece.
W	CUT RISE METHOD	[SPEED]: The tool retracts by a distance specified with ESCAPE AMOUNT in the XZ direction immediately after cutting. [CUT] : The tool retracts by a "retract" distance" after cutting along the figure. (COPY)
U	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut (radius value, positive value) Remark) By referring to the parameter No. 27128 (minimum clamp value), the system sets the data as an initial value.
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

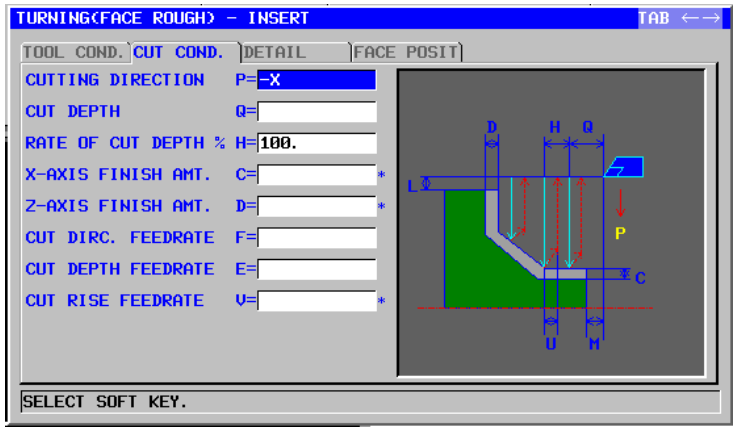
DETAIL		
Data item		**Meaning**
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After cutting in in the X-axis direction at the feedrate (F) specified for the cutting direction, the tool cuts in to the entered-figure position where there is still a finishing allowance in the Z-axis direction at the feedrate (E) specified for the cut-in direction.
<3> If [SPEED] is selected, the tool immediately retracts by a return clearance (U) in the X- and Z-axis direction. If [CUT] is selected, the tool first cuts along the figure and then retracts by a return clearance (U) in the X- and Z-axis direction.
<4> The tool moves to the cut-in start position in the Z-axis direction by rapid traverse.
<5> Steps <2> to <4> are repeated until the lowest portion in the X-axis direction is reached.
<6> If there is another pocket, the tool is positioned on the pocket, and steps <2> to <4> are repeated.
<7> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

End face roughing: G1122



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value)
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

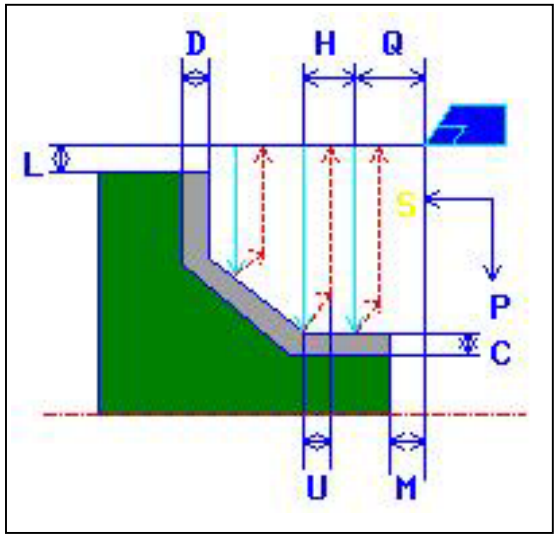
CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) [+X] is used for cutting in the reverse direction or for machining with a subspindle. (COPY)
Q	CUT DEPTH	Depth of each cut (radius value, positive value)
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. The default is 100%, meaning that the depth of cut remains unchanged. (1 to 200, positive value)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)

CUT COND.		
Data item		**Meaning**
F	CUT DIRC.FEEDRATE	Feedrate applicable when the tool cuts in the workpiece radius direction (positive value)
E	CUT DEPTH FEEDRATE	Feedrate applicable when the tool cuts in the Z-axis direction (positive value)
V	CUT RISE FEEDRATE	Feedrate applicable when the tool cuts up in the direction of retraction from the workpiece (positive value)

DETAIL		
Data item		**Meaning**
K	1ST OVERRIDE	Feedrate override value for the first cut. Specify an override value in steps of 1%. The default is 100%. (1 - 200, positive value) Remark) This data item is used, for example, to cut the black coating of a cast workpiece.
W	CUT RISE METHOD	[SPEED]: The tool retracts by a distance specified with ESCAPE AMOUNT in the XZ direction immediately after cutting. (initial value) [CUT] : The tool retracts by a "retract" distance" after cutting along the figure. (COPY)
U	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut (radius value, positive value) Remark) By referring to the parameter No. 27128 (minimum clamp value), the system sets the data as an initial value.
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.

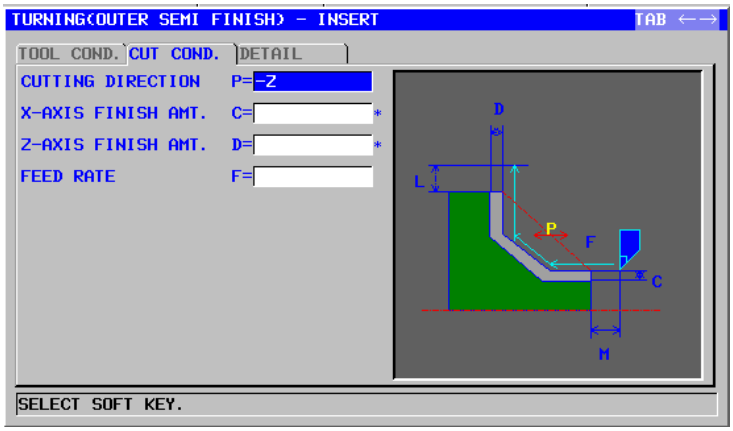
DETAIL		
Data item		Meaning
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After cutting in in the Z-axis direction at the feedrate (F) specified for the cutting direction, the tool cuts in to the entered-figure position where there is still a finishing allowance in the X-axis direction at the feedrate (E) specified for the cut-in direction.
<3> If [SPEED] is selected, the tool immediately retracts by a return clearance (U) in the X- and Z-axis direction. If [CUT] is selected, the tool first cuts along the figure and then retracts by a return clearance (U) in the X- and Z-axis direction.
<4> The tool moves to the cut-in start position in the X-axis direction by rapid traverse.
<5> Steps <2> to <4> are repeated until the lowest portion in the Z-axis direction is reached.
<6> If there is another pocket, the tool is positioned on the pocket, and steps <2> to <4> are repeated.
<7> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

Outer surface semifinishing: G1123



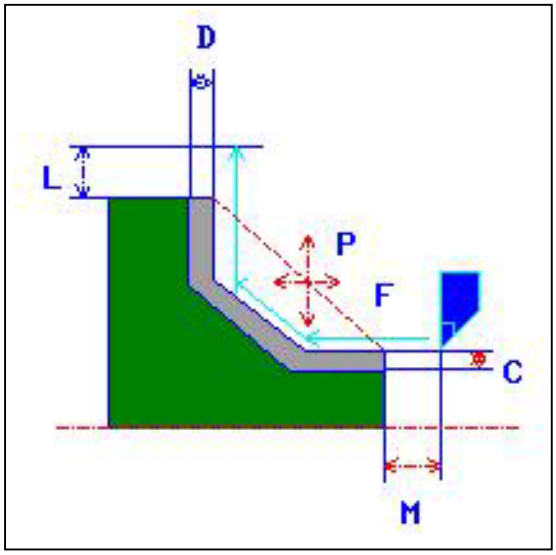
TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. (initial value) [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. (COPY)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Cutting feedrate for semifinishing (positive value)

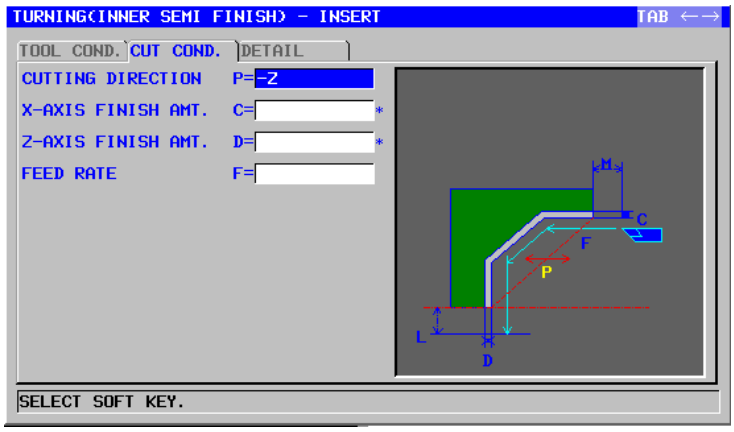
DETAIL		
Data item		**Meaning**
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129(minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> The tool cuts along the entered figure on which the finishing allowance is left uncut at the semifinishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (L)" in the X-axis direction at the rapid traverse rate.

Inner surface semifinishing: G1124



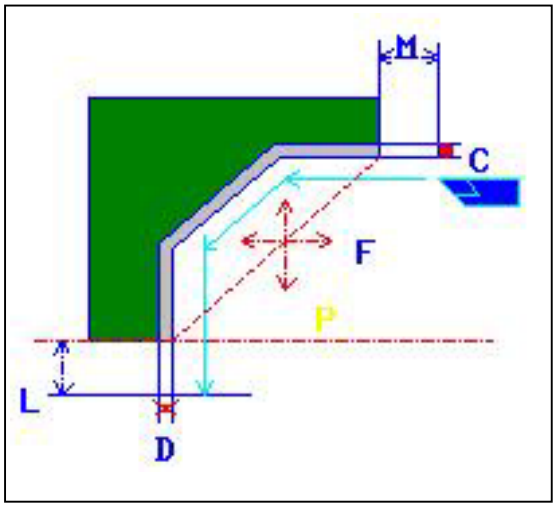
TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. (initial value) [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. (COPY)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Cutting feedrate for semifinishing (positive value)

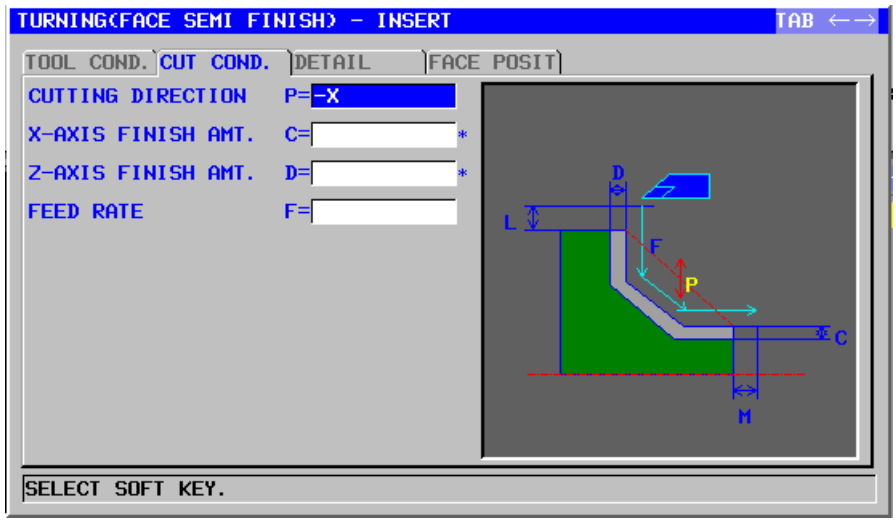
DETAIL		
Data item		**Meaning**
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> The tool cuts along the entered figure on which the finishing allowance is left uncut at the semifinishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

End face semifinishing: G1125



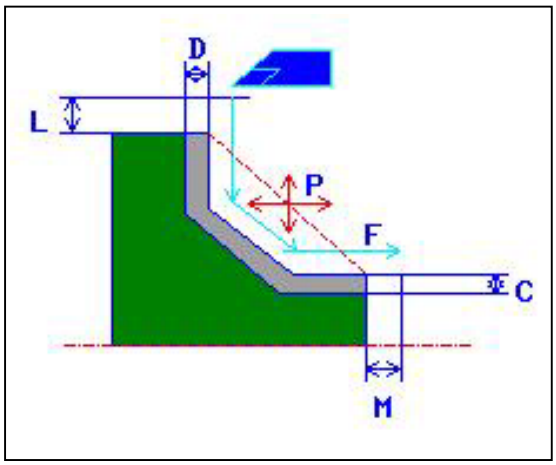
TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. (initial value) [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-X] : Cuts in the -X direction. [+X] : Cuts in the +X direction. (COPY)
C*	X-AXIS FINISH AMT.	Finishing allowance in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
D*	Z-AXIS FINISH AMT.	Finishing allowance in the Z-axis direction. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Cutting feedrate for semifinishing (positive value)

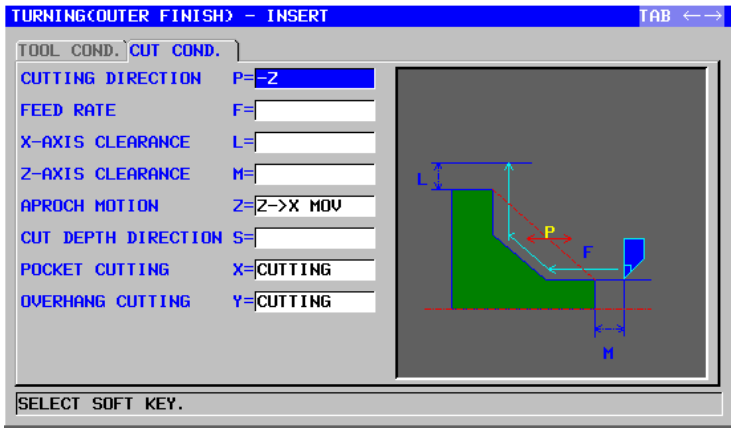
DETAIL		
Data item		**Meaning**
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> The tool cuts along the entered figure on which the finishing allowance is left uncut at the semifinishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

Outer surface finishing: G1126

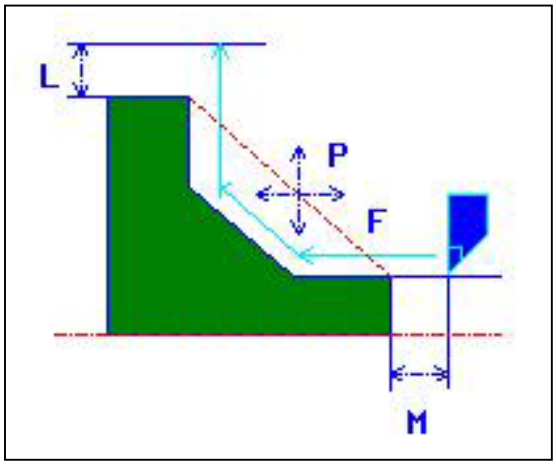


TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. (initial value) [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

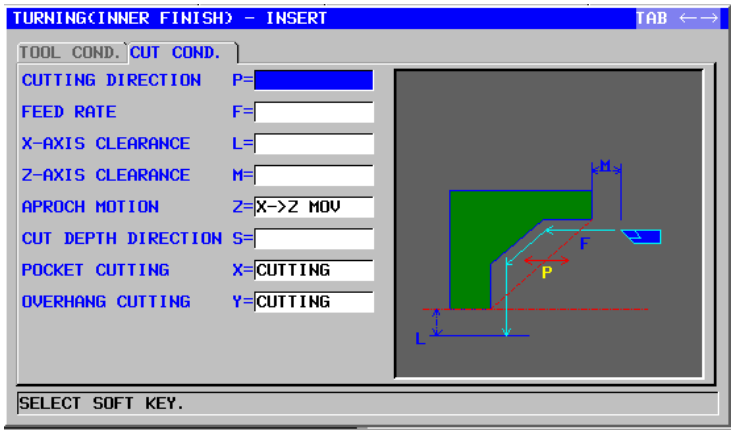
CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. (COPY)
F	FEED RATE	Cutting feedrate for finishing (positive value)
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> The tool cuts along the entered figure at the finishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (L)" in the X-axis direction at the rapid traverse rate.

Inner surface finishing: G1127

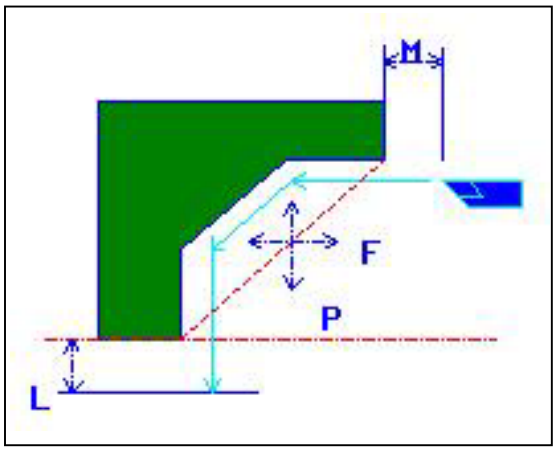


TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. (initial value) [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

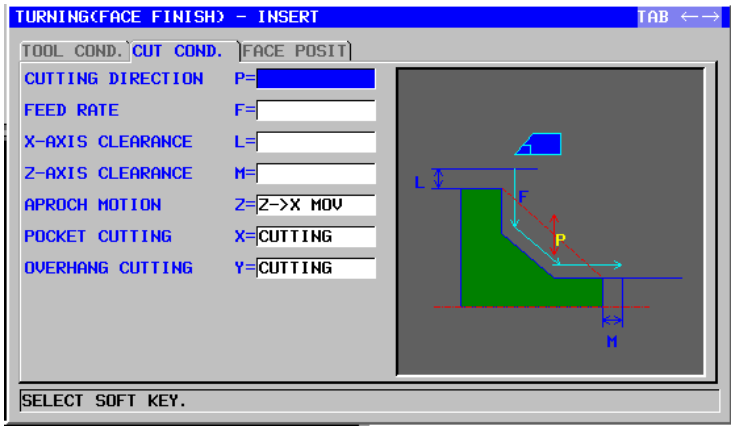
CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-Z] : Cuts in the -Z direction. [+Z] : Cuts in the +Z direction. (COPY)
F	FEED RATE	Cutting feedrate for finishing (positive value)
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> The tool cuts along the entered figure at the finishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

End face finishing: G1128



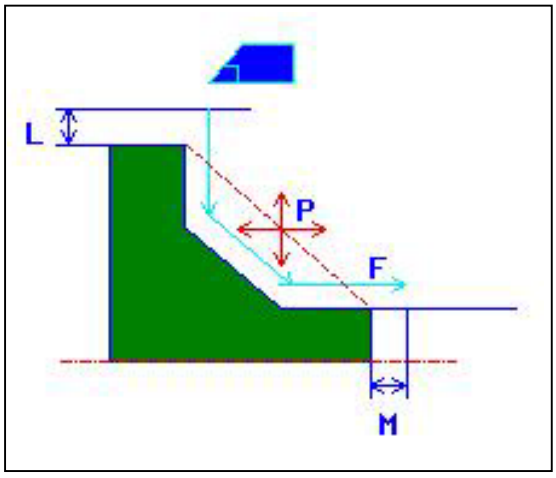
TOOL COND.		
Data item		**Meaning**
I	INPUT TYPE	[INPUT] : Directly inputs the tool nose radius of a tool used for semifinishing. [REF.] : Inputs the offset number of a tool used for semifinishing to read the offset value.
R	NOSE RADIUS	Tool nose radius of a roughing tool (positive value) Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
A	CUT EDGE ANGLE	Cutting edge angle of a roughing tool (positive value)
B	NOSE ANGLE	Tool angle of a roughing tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a roughing tool Remark) This data item is indicated only when [INPUT] is selected for INPUT TYPE.
T	OFFSET NO.	Offset number of a roughing tool (positive value) Remark) This data item is indicated only when [REF.] is selected for INPUT TYPE.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
P	CUTTING DIRECTION	[-X] : Cuts in the -X direction. [+X] : Cuts in the +X direction. (COPY)
F	FEED RATE	Cutting feedrate for finishing (positive value)

CUT COND.		
Data item		**Meaning**
L	X-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27129 (minimum clamp value), the system sets the data as an initial value.
M	Z-AXIS CLEARANCE	Distance between a blank and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27130 (minimum clamp value), the system sets the data as an initial value.
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
X	POCKET CUTTING	[CUT] : Cuts a pocket. (initial value) [NOTHIN] : Does not cut a pocket. Remark) This data item is enable when the parameter No.27100#1 = 1.
Y	OVERHANG CUTTING	[CUT] : Cuts an overhang. (initial value) [NOTHIN] : Does not cut an overhang. Remark) This data item is enable when the parameter No.27100#1 = 1.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.

<2> The tool cuts along the entered figure at the finishing feedrate until the final figure is obtained.
<3> Once all portions are cut, the tool retracts to the position "cut-in start position + clearance (M)" in the Z-axis direction at the rapid traverse rate.

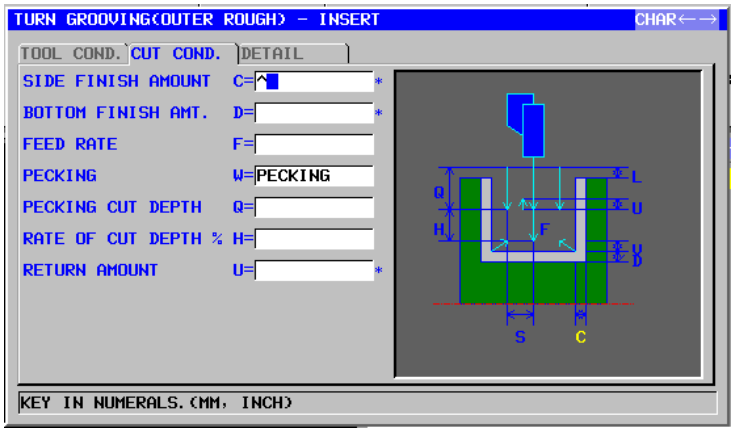
2.2.2 Arbitrary Figure Blocks for Turning

When an arbitrary figure is input, a function such as the automatic intersection calculation function can be used. For details, see Section 5.5, "ARBITRARY FIGURES FOR TURNING (ZX PLANE)," in Part II.

2.3 TURNING GROOVING

2.3.1 Machining Type Blocks for Turning Grooving

Outer surface roughing: G1130



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

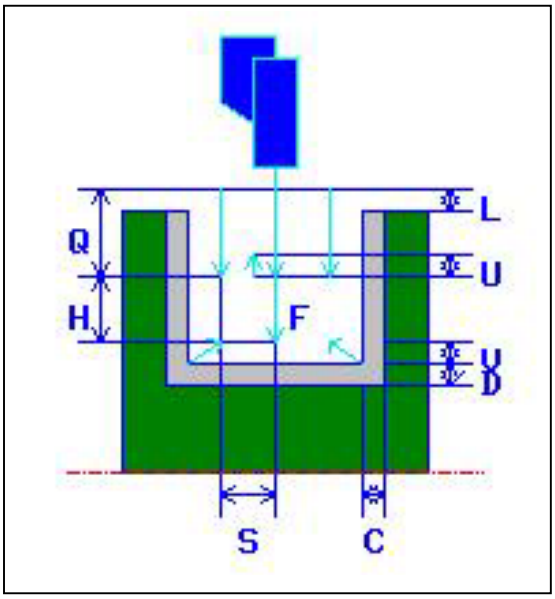
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Feedrate for cutting in the tool axis direction (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.

CUT COND.		
Data item		**Meaning**
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. (1 to 200, positive value) (COPY)
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

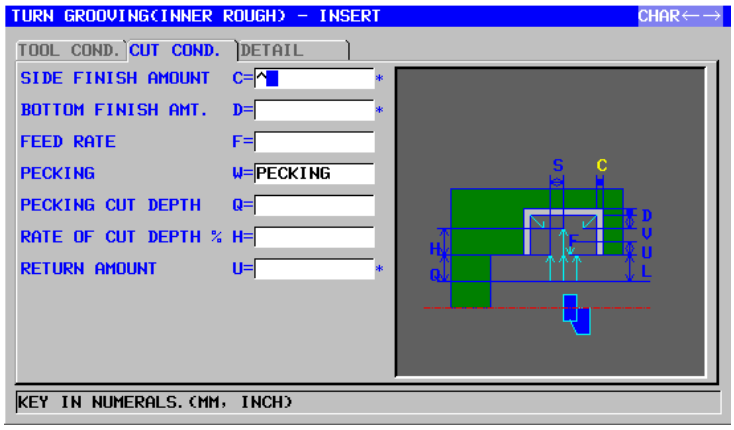
DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut. The blank is regarded as 0. (radius value, positive value) Remark) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After moving to the center of the groove (in the Z-axis direction) by rapid traverse, the tool cuts in in the X-axis direction at the feedrate (F) specified for the cutting direction. If pecking is specified, the tool cuts a specified pecking amount while retracting in the +X-axis direction between pecking cycles.
<3> When the tool reaches the groove bottom (but the finishing allowance) in the X-axis direction, the tool retracts in the +X-axis direction, shifts by a cut-in amount through the width in the -Z-axis direction, and then cuts in the X-axis direction.
<4> After step <3> is repeated until one groove wall is reached, the tool cuts another wall.
<5> After all portions are cut, the tool moves to the position "cut-in start position + clearance (L)" in the X-axis direction by rapid traverse.

Inner surface roughing: G1131



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

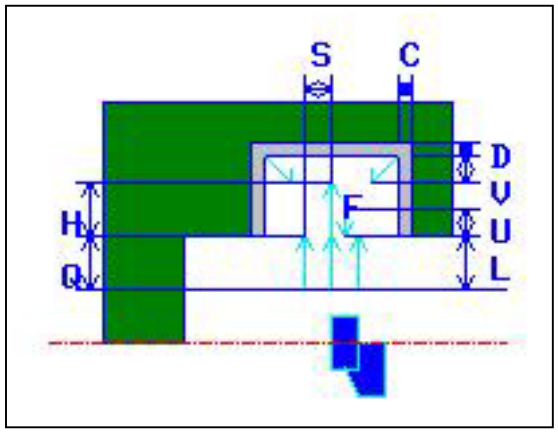
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Feedrate for cutting in the tool axis direction (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. (1 to 200, positive value) (COPY)

CUT COND.		
Data item		**Meaning**
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

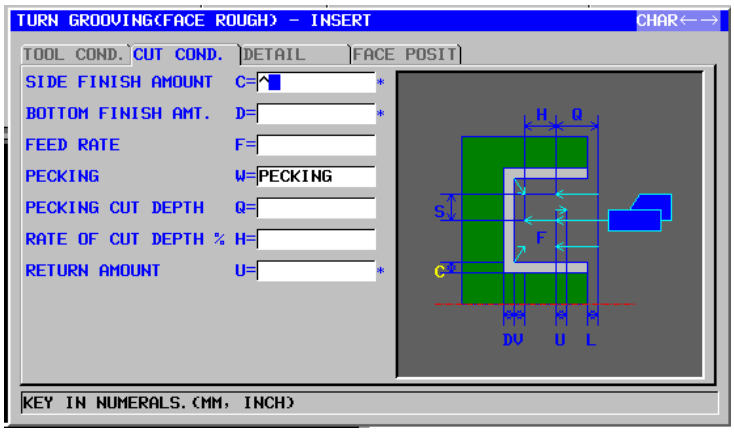
DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut. The blank is regarded as 0. (radius value, positive value) Remark) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After moving to the center of the groove (in the Z-axis direction) by rapid traverse, the tool cuts in in the X-axis direction at the feedrate (F) specified for the cutting direction. If pecking is specified, the tool cuts a specified pecking amount while retracting in the -X-axis direction between pecking cycles.
<3> When the tool reaches the groove bottom (but the finishing allowance) in the X-axis direction, the tool retracts in the -X-axis direction, shifts by a cut-in amount through the width in the -Z-axis direction, and then cuts in the X-axis direction.
<4> After step <3> is repeated until one groove wall is reached, the tool cuts another wall.
<5> After all portions are cut, the tool moves to the position "cut-in start point + clearance (M)" in the Z-axis direction by rapid traverse and is pulled out from the workpiece.

End face roughing: G1132



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J*	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

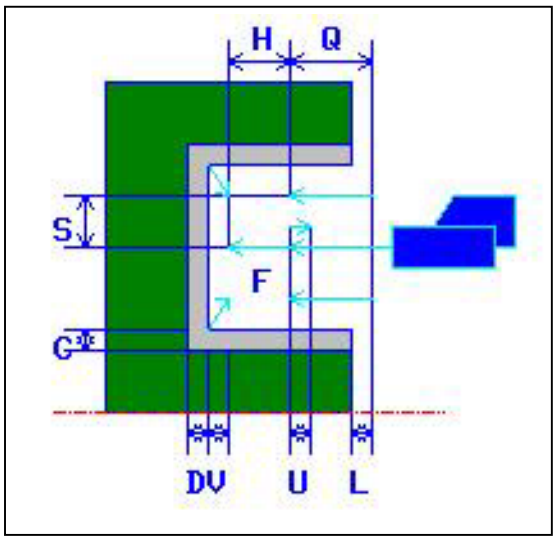
NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE	Feedrate for cutting in the tool axis direction (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. (1 to 200, positive value) (COPY)

CUT COND.		
Data item		**Meaning**
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

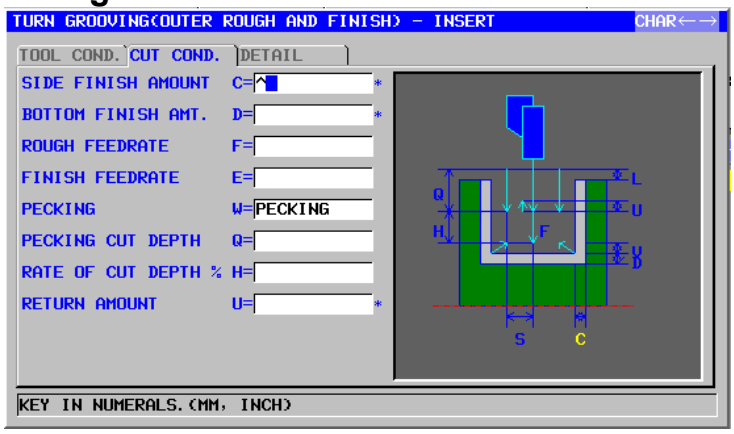
DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut. The blank is regarded as 0. (radius value, positive value) Remark) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.

- **Tool path**



<1> The tool moves to the position "cut-in start point + clearance (L, M)" by rapid traverse.
<2> After moving to the center of the groove (in the X-axis direction) by rapid traverse, the tool cuts in in the -Z-axis direction at the feedrate (F) specified for the cutting direction. If pecking is specified, the tool cuts a specified pecking amount while retracting in the +Z-axis direction between pecking cycles.
<3> When the tool reaches the groove bottom (but the finishing allowance) in the Z-axis direction, the tool retracts in the +Z-axis direction, shifts by a cut-in amount through the width in the -X-axis direction, and then cuts in the Z-axis direction.
<4> After step <3> is repeated until one groove wall is reached, the tool cuts another wall.
<5> After all portions are cut, the tool moves to the position "cut-in start position + clearance (M)" in the Z-axis direction by rapid traverse.

Outer surface roughing and finishing: G1133



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	ROUGH FEEDRATE	Feedrate for roughing in the tool axis direction (positive value)
E	FINISH FEEDRATE	Feedrate for finishing (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.

CUT COND.		
Data item		**Meaning**
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. (1 to 200, positive value) (COPY)
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

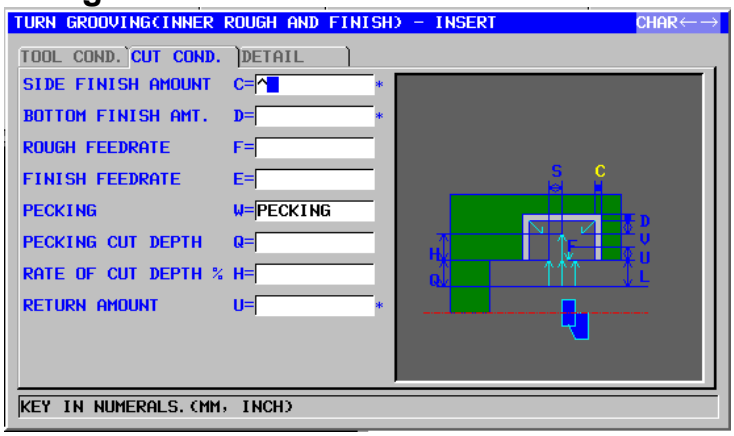
DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.

DETAIL		
Data item		Meaning
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**

Groove roughing and finishing are continued, using the same tool. See respective descriptions about the tool path for details of roughing and finishing.

Inner surface roughing and finishing: G1134



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J*	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	ROUGH FEEDRATE	Feedrate for roughing in the tool axis direction (positive value)
E	FINISH FEEDRATE	Feedrate for finishing (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.

CUT COND.		
Data item		**Meaning**
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. (1 to 200, positive value) (COPY)
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

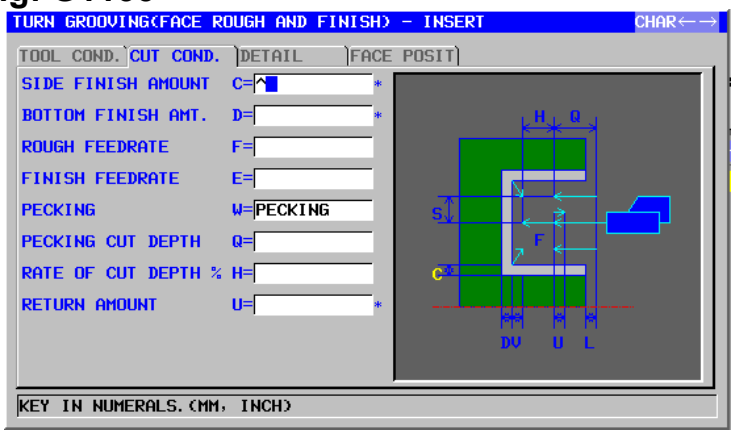
DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.

DETAIL		
Data item		Meaning
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**

Groove roughing and finishing are continued, using the same tool. See respective descriptions about the tool path for details of roughing and finishing.

End face roughing and finishing: G1135



TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab 'TOOL COND.' is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of 'TOOL DATA' in Tool Offset window. Therefore, it is not necessary to display the tab 'TOOL COND.'

CUT COND.		
Data item		**Meaning**
C*	SIDE FINISH AMOUNT	Finishing allowance for the side faces of a groove. The blank is regarded as 0. (radius value, positive value)
D*	BOTTOM FINISH AMT.	Finishing allowance for the bottom of a groove. The blank is regarded as 0. (radius value, positive value)
F	ROUGH FEEDRATE	Feedrate for roughing in the tool axis direction (positive value)
E	FINISH FEEDRATE	Feedrate for finishing (positive value)
W	PECKING	[NOTHIN] : Does not perform pecking in cutting for grooving (initial value). [PECKIN] : Performs pecking in cutting for grooving.
Q	PECKING CUT DEPTH	Depth of cut in the tool axis direction per pecking operation (radius value, positive value) Remark) This data item is indicated only when [PECKIN] is selected for PECKING.

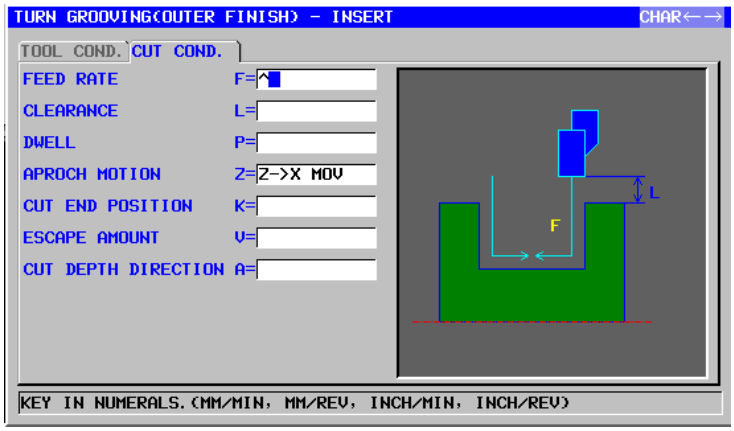
CUT COND.		
Data item		**Meaning**
H	RATE OF CUT DEPTH	Change rate for the depth of cut. Specify a change rate in steps of 1%. A second depth of cut and subsequent ones are sequentially multiplied by a specified change rate. The default is 100%, meaning that the depth of cut remains unchanged. (1 to 200, positive value) (COPY)
U*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface after each cut by pecking. (radius value, positive value) (COPY)

DETAIL		
Data item		**Meaning**
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176, the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
S	CUT DEPTH FOR WIDTH	Depth of each cut in the grooving tool width direction (radius value, positive value) (COPY)
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V*	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177, the system sets the data as an initial value.

- **Tool path**

Groove roughing and finishing are continued, using the same tool. See respective descriptions about the tool path for details of roughing and finishing.

Outer surface finishing: G1136



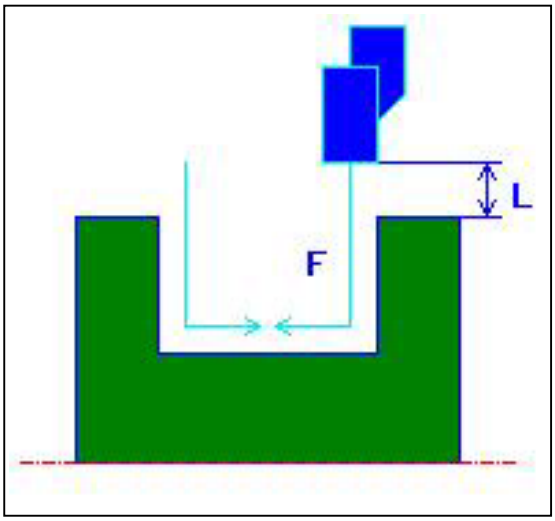
TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

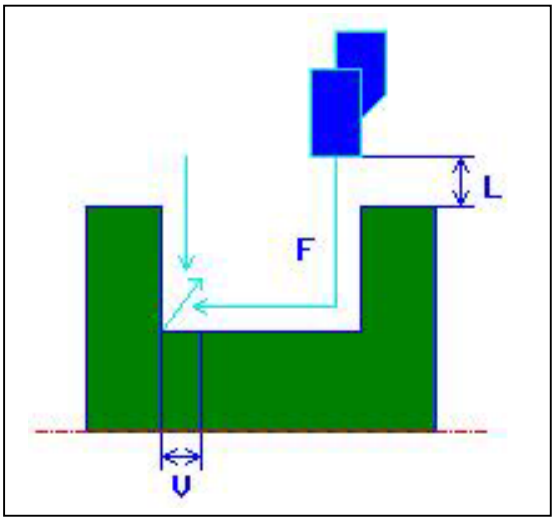
CUT COND.		
Data item		**Meaning**
F	FEED RATE	Feedrate for finishing in the tool axis direction (positive value)
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)

CUT COND.		
Data item		**Meaning**
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**



Cutting end position [CENTER]



Cutting end position [CORNER]

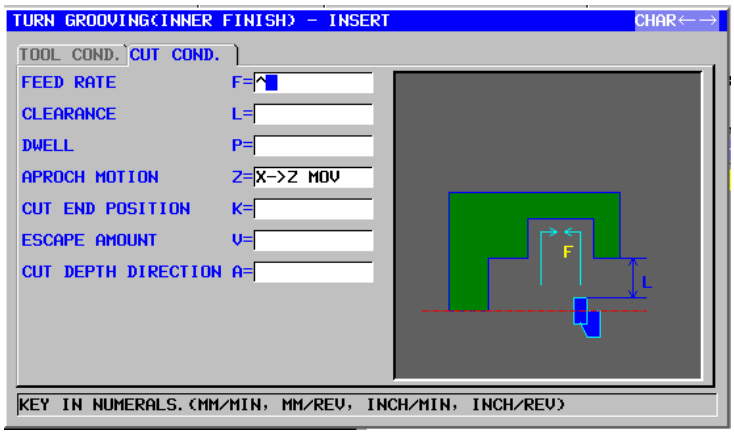
<1> The tool moves to the position "cut-in start position + clearance (L, M)" by rapid traverse.

<2> If [CENTER] is specified as the cutting end position, the tool cuts in one of the groove walls and continues cutting to the center (in the Z-axis direction) of the groove at the finishing feedrate. After retracting in the X-axis direction, the tool performs the same cutting operation for the other groove wall.

<3> If [CORNER] is specified as the cutting end point, the tool cuts in to the bottom from the wall in the -Z-axis direction, and retracts by a return clearance in the X- and Z-axis directions and then in the X-axis direction. After this, the tool starts cutting from the other groove wall and continues cutting to the edge of the bottom, and then retracts by a return clearance in the X- and Z-axis directions.

<4> The tool moves to the position "cut-in start point + clearance (L)" in the X-axis direction by rapid traverse.

Inner surface finishing: G1137



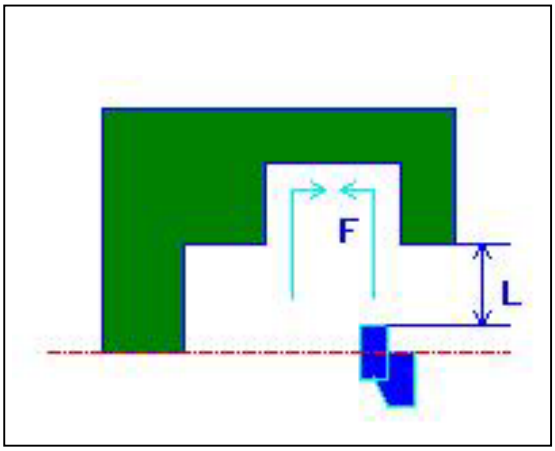
TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

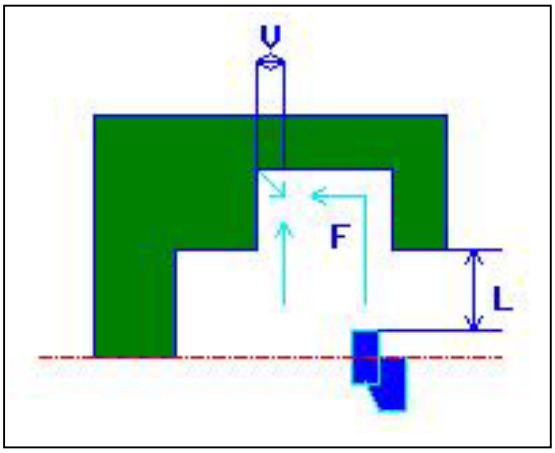
CUT COND.		
Data item		**Meaning**
F	FEED RATE	Feedrate for finishing in the tool axis direction (positive value)
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)

CUT COND.		
Data item		**Meaning**
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.
A	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**



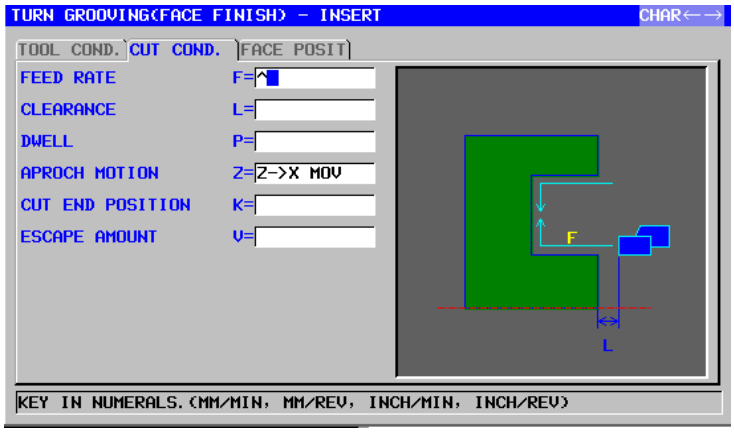
Cutting end position [CENTER]



Cutting end position [CORNER]

<1> The tool moves to the position "cut-in start position + clearance (L, M)" by rapid traverse.
<2> If [CENTER] is specified as the cutting end position, the tool cuts in one of the groove walls and continues cutting to the center (in the Z-axis direction) of the groove at the finishing feedrate. After retracting in the X-axis direction, the tool performs the same cutting operation for the other groove wall.
<3> If [CORNER] is specified as the cutting end point, the tool cuts in to the bottom from the wall in the -Z-axis direction, and retracts by a return clearance in the X- and Z-axis directions and then in the X-axis direction. After this, the tool starts cutting from the other groove wall and continues cutting to the edge of the bottom, and then retracts by a return clearance in the X- and Z-axis directions.
<4> The tool retracts to the position "cut-in start point + clearance (L)" in the X-axis direction.
<5> The tool moves to the position "cut-in start position + clearance (M)" in the Z-axis direction by rapid traverse and then is pulled out from the workpiece.

End face finishing: G1138



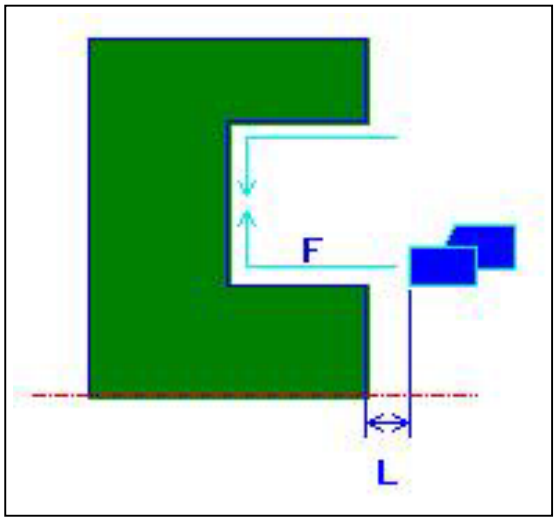
TOOL COND.		
Data item		**Meaning**
R	NOSE RADIUS	Tool nose radius of a grooving tool. (positive value)
B	TOOL WIDTH	Tool width of a grooving tool (radius value, positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a grooving tool.

NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

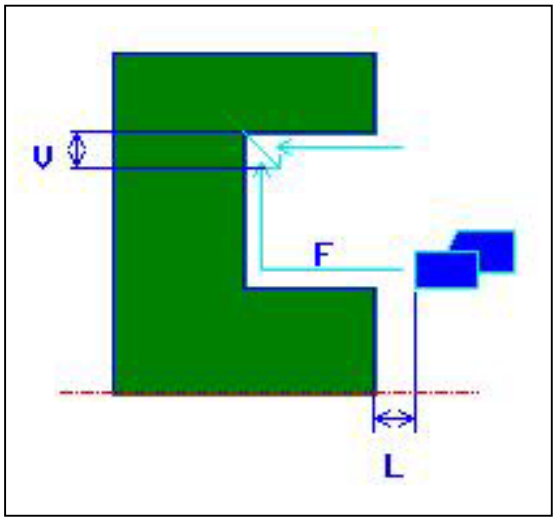
CUT COND.		
Data item		**Meaning**
F	FEED RATE	Feedrate for finishing in the tool axis direction (positive value)
L	CLEARANCE	Distance between the top surface of a groove and a machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27176 (minimum clamp value), the system sets the data as an initial value.
P	DWELL	Dwell time applicable when the tool reaches the bottom of a groove. (in seconds, positive value) (COPY)

CUT COND.		
Data item		**Meaning**
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (initial value) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
K	CUT END POSITION	[CENTER] : Cuts both of the right and left side faces of a groove evenly in finishing. [CORNER] : Cuts one side face and the entire bottom successively then cuts the other side face only in finishing. (COPY)
V	ESCAPE AMOUNT	Distance by which the tool retracts from a cutting surface when finishing is completed (radius value, positive value) Remark1) This item is displayed only when [CORNER] is selected as a cutting end position. The default value is 0. Remark2) By referring to the parameter No. 27177 (minimum clamp value), the system sets the data as an initial value.

- **Tool path**



Cutting end position [CENTER]

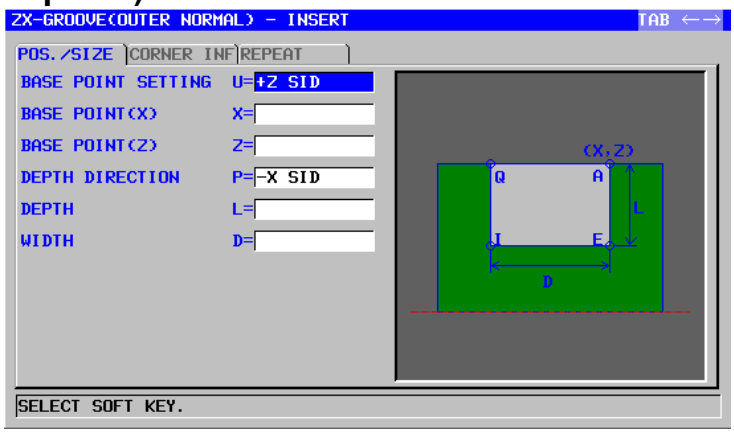


Cutting end position [CORNER]

<1> The tool moves to the position "cut-in start position + clearance (L, M)" by rapid traverse.
<2> If [CENTER] is specified as the cutting end position, the tool cuts in one of the groove walls and continues cutting to the center (in the X-axis direction) of the groove at the finishing feedrate. After retracting in the Z-axis direction, the tool performs the same cutting operation for the other groove wall.
<3> If [CORNER] is specified as the cutting end point, the tool cuts in to the bottom from the wall in the +X-axis direction, and retracts by a return clearance in the X- and Z-axis directions and then in the Z-axis direction. After this, the tool starts cutting from the other groove wall and continues cutting to the edge of the bottom, and then retracts by a return clearance in the X- and Z-axis directions.
<4> The tool moves to the position "cut-in start point + clearance (M)" in the Z-axis direction by rapid traverse.

2.3.2 Fixed Form Figure Blocks for Turning Grooving

Outer normal groove: G1470 (ZX plane)



POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+Z] : Sets the base point in the +Z direction. (initial value) [-Z] : Sets the base point in the -Z direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to X axis Depth direction, [-X] : Depth is in the –X direction. [+X] : Depth is in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

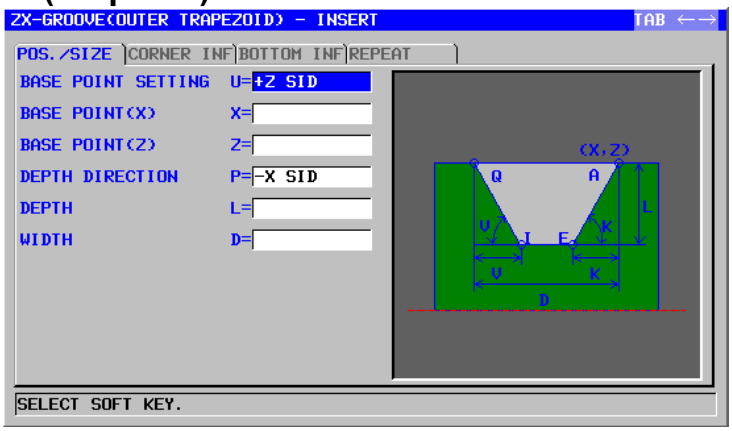
NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.

CORNER INFO		
Data item		**Meaning**
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-Z] : Places a second and subsequent grooves in the -Z direction (initial value). [+Z] : Places a second and subsequent grooves in the +Z direction.

Outer trapezoidal groove: G1471 (ZX plane)



POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+Z] : Sets the base point in the +Z direction. (initial value) [-Z] : Sets the base point in the -Z direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to X axis Depth direction, [-X] : Depth is in the –X direction. [+X] : Depth is in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

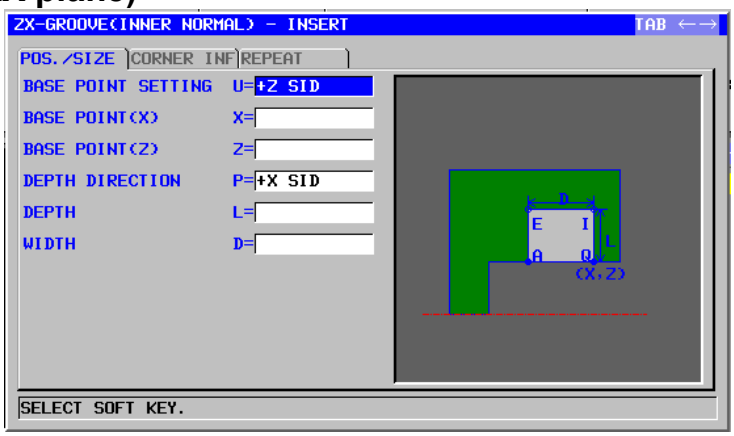
CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.

CORNER INFO		
Data item		**Meaning**
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

BOTTOM INFO		
Data item		**Meaning**
H	BOTTOM TYPE	[WIDTH] : Specifies the difference between the groove entry of each side face and the width of the groove bottom (initial value). [ANGLE] : Specifies the angle of a side face of a groove.
K*	BOTTOM SIZE/ANGLE	Difference between the groove entry of the side face on the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.
V*	BOTOM SIZE/ANGLE	Difference between the groove entry of the side face on the side opposite to the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the side opposite to the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-Z] : Places a second and subsequent grooves in the -Z direction (initial value). [+Z] : Places a second and subsequent grooves in the +Z direction.

Inner normal groove: G1472 (ZX plane)



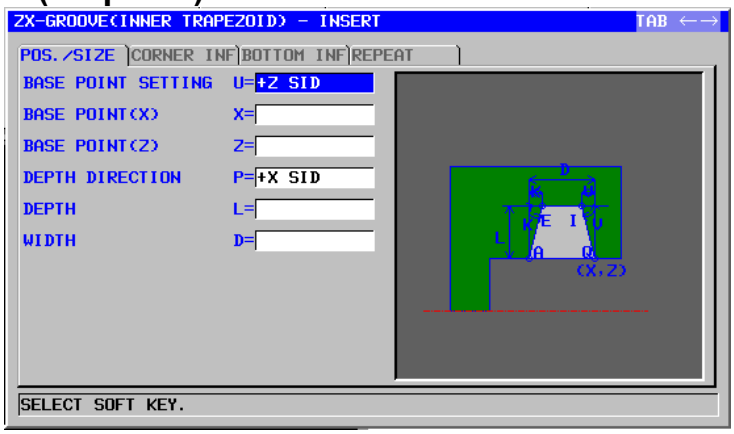
POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+Z] : Sets the base point in the +Z direction. (initial value) [-Z] : Sets the base point in the -Z direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to X axis Depth direction, [-X] : Depth is in the –X direction. [+X] : Depth is in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.

CORNER INFO		
Data item		**Meaning**
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-Z] : Places a second and subsequent grooves in the -Z direction (initial value). [+Z] : Places a second and subsequent grooves in the +Z direction.

Inner trapezoidal groove: G1473 (ZX plane)



POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+Z] : Sets the base point in the +Z direction. (initial value) [-Z] : Sets the base point in the -Z direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to X axis Depth direction, [-X] : Depth is in the –X direction. [+X] : Depth is in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

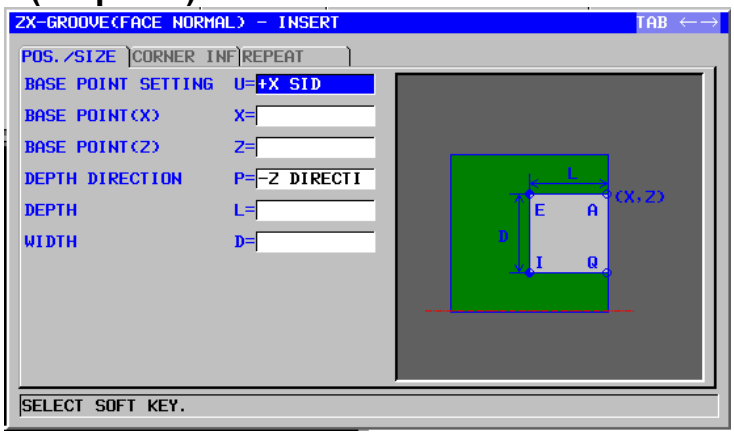
CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.

CORNER INFO		
Data item		**Meaning**
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

BOTTOM INFO		
Data item		**Meaning**
H	BOTTOM TYPE	[WIDTH] : Specifies the difference between the groove entry of each side face and the width of the groove bottom (initial value). [ANGLE] : Specifies the angle of a side face of a groove.
K*	BOTTOM SIZE/ANGLE	Difference between the groove entry of the side face on the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.
V*	BOTOM SIZE/ANGLE	Difference between the groove entry of the side face on the side opposite to the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the side opposite to the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-Z] : Places a second and subsequent grooves in the -Z direction (initial value). [+Z] : Places a second and subsequent grooves in the +Z direction.

End face normal groove: G1474 (ZX plane)



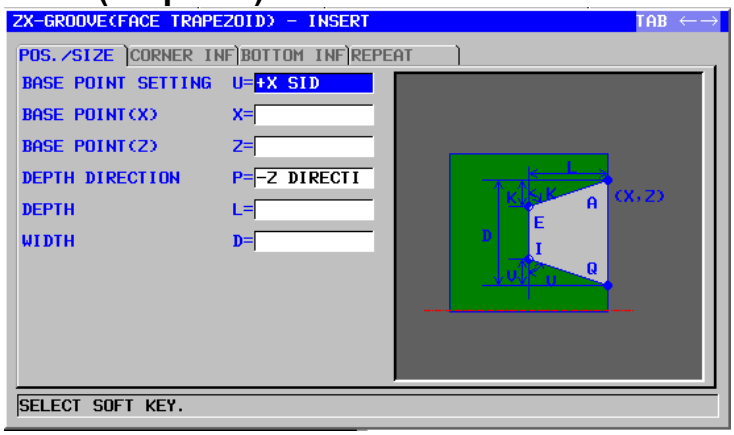
POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+X] : Sets the base point in the +X direction. (initial value) [-X] : Sets the base point in the -X direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to Z axis Depth direction, [-Z] : Depth is in the –Z direction. [+Z] : Depth is in the +Z direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.

CORNER INFO		
Data item		**Meaning**
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-X] : Places a second and subsequent grooves in the -X direction (initial value). [+X] : Places a second and subsequent grooves in the +X direction.

End face trapezoidal groove: G1475 (ZX plane)



POS./SIZE		
Data item		**Meaning**
U	BASE POINT SETTING	[+X] : Sets the base point in the +X direction. (initial value) [-X] : Sets the base point in the -X direction.
X	BASE POINT (X)	X coordinate of the reference position of a groove
Z	BASE POINT (Z)	Z coordinate of the reference position of a groove
P	DEPTH DIRECTION	As to Z axis Depth direction, [-Z] : Depth is in the –Z direction. [+Z] : Depth is in the +Z direction. Remark) This data item is enable when the parameter No.27100#0 = 1.
L	DEPTH	Groove depth (radius value, positive value)
D	WIDTH	Groove width (radius value, positive value)

CORNER INFO		
Data item		**Meaning**
A	CORNER TYPE-1	For corner (1) of the reference position [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
B	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-1.
E	CORNER TYPE-2	For corner (2) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
F	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-2.

CORNER INFO		
Data item		**Meaning**
I	CORNER TYPE-3	For corner (3) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
J	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-3.
Q	CORNER TYPE-4	For corner (4) [NOTHIN] : Specifies neither chamfering nor corner rounding (initial value). [CHAMFR] : Specifies chamfering. [ARC] : Specifies corner rounding.
R	CORNER SIZE	Chamfer amount or corner radius (radius value, positive value) Remark) This data item is indicated only when [CHAMFR] or [ARC] is selected for CORNER TYPE-4.

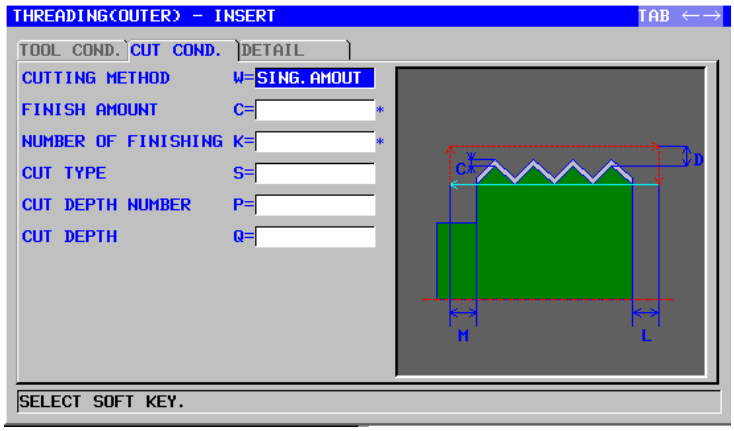
BOTTOM INFO		
Data item		**Meaning**
H	BOTTOM TYPE	[WIDTH]: Specifies the difference between the groove entry of each side face and the width of the groove bottom (initial value). [ANGLE]: Specifies the angle of a side face of a groove.
K*	BOTTOM SIZE/ANGLE	Difference between the groove entry of the side face on the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.
V*	BOTOM SIZE/ANGLE	Difference between the groove entry of the side face on the side opposite to the reference position side and the width of the groove bottom when [WIDTH] is specified (radius value, positive value). Inclination angle of the side face on the side opposite to the reference position side when [ANGLE] is specified (positive value). The blank is regarded as 0 for both.

REPEAT		
Data item		**Meaning**
M*	GROOVE NUMBER	Number of grooves of the same figure to be machined. The blank is regarded as 1. (positive value)
S	PITCH	Distance between the reference positions of two adjacent grooves (radius value, positive value)
W*	PITCH DIRECTION	[-X] : Places a second and subsequent grooves in the -X direction (initial value). [+X] : Places a second and subsequent grooves in the +X direction.

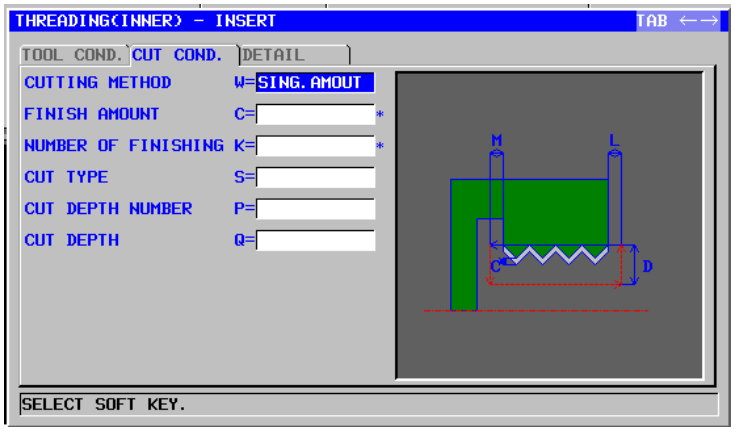
2.4 THREADING

2.4.1 Machining Type Blocks for Threading

External: G1140



Internal: G1141



TOOL COND.		
Data item		Meaning
R	NOSE RADIUS	Tool nose radius of a threading tool. (positive value)
A	NOSE ANGLE	Tool angle of a threading tool (positive value)
J	IMAGINARY TOOL NOSE	Imaginary tool nose position of a threading tool.

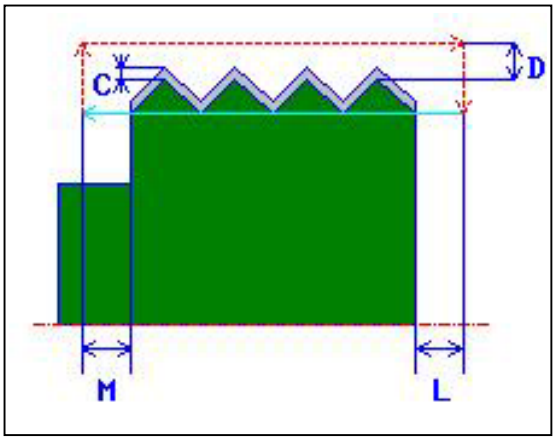
NOTE
1 Tab ‘TOOL COND.’ is enabled when bit 0 (TLG) of parameter No. 27102 = 1.
2 The operator ordinarily sets the above data on the tab of ‘TOOL DATA’ in Tool Offset window. Therefore, it is not necessary to display the tab ‘TOOL COND.’

CUT COND.		
Data item		**Meaning**
W	CUTTING METHOD	[SING.A] : Constant amount of cut, one-edge cutting [BOTH A] : Constant amount of cut, both-edge cutting [STAG.A] : Constant amount of cut, both-edge zigzag thread cutting [SING.D] : Constant depth of cut, one-edge cutting [BOTH D] : Constant depth of cut, both-edge cutting [STAG.D] : Constant depth of cut, both-edge zigzag thread cutting (COPY)
C*	FINISH AMOUNT	Finishing allowance for threading in the X-axis direction. The blank is regarded as 0. (radius value, positive value)
K*	NUMBER OF FINISHING	Number of finishing operations. (positive value) (COPY)
S	CUT TYPE	[NUMBER] : Specifies threading by the number of cuts. [DEPTH] : Specifies threading by the first amount of cut. (COPY)
P	CUT DEPTH NUMBER	Number of cuts for rough threading (999 max., positive value) Remark) The number of finishing cuts is not included. In the case of both-edge zigzag thread cutting, be sure to specify an even number of cuts. If an odd number of cuts are specified, only one additional both-edge zigzag thread cutting operation is performed. This data item is indicated only when [NUMBER] is selected.
Q	CUT DEPTH	Depth of cut per rough threading operation (radius value, positive value) Remark) The number of cuts is determined by the depth of cut and the cutting method.

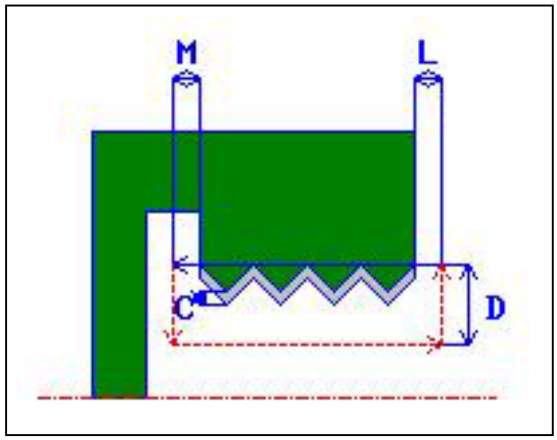
DETAIL		
Data item		**Meaning**
Z	APROCH MOTION	[Z→X] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (Initial value for external thread) [X→Z] : From the current position to the machining start point, the tool moves in the Z-axis direction and then in the X-axis direction. (Initial value for internal thread) [2 AXES] : From the current position to the machining start point, the tool moves simultaneously in the X- and Z-axis directions.
D	SURFACE CLEARANCE	Distance between a thread crest and machining start point (approach point) in the X-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27156 (minimum clamp value), the system sets the data as an initial value.

DETAIL		
Data item		**Meaning**
L	ENTRANCE CLEARANCE	Distance between a thread start point and machining start point (approach point) in the Z-axis direction (radius value, positive value) Remark) By referring to the parameter No. 27157 (minimum clamp value), the system sets the data as an initial value.
M	EXIT CLERANCE	Distance between a thread end point and threading operation end point in the Z-axis direction. (radius value, positive value) Remark) By referring to the parameter No. 27158 (minimum clamp value), the system sets the data as an initial value.
Y	CUT DEPTH DIRECTION	As to X axis Cut direction, [-X] : Cuts in the –X direction. [+X] : Cuts in the +X direction. Remark) This data item is enable when the parameter No.27100#0 = 1. (COPY)

- **Tool path**



External-thread cutting

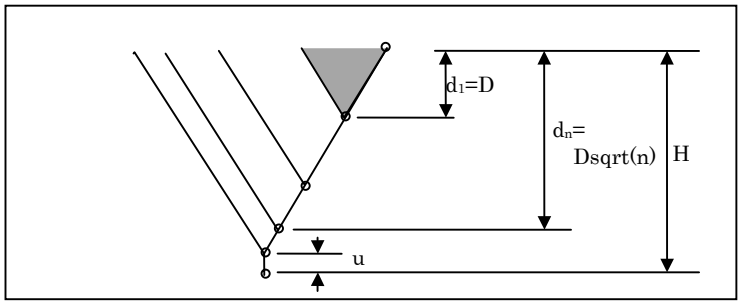


Internal-thread cutting

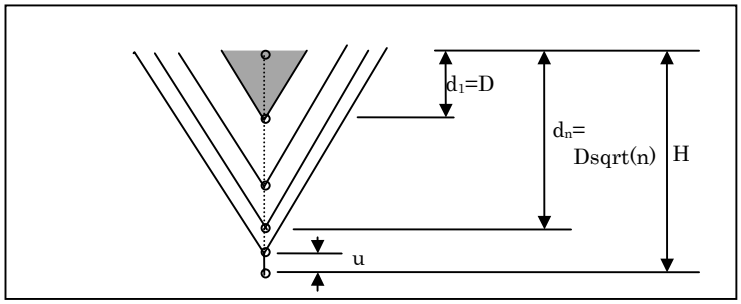
<1> The tool moves to the position "thread start point + thread surface clearance (D in the X-axis direction)" and position thread entry clearance (L in the Z-axis direction) by rapid traverse.
<2> Threading is performed using a specified cutting method. The end point of threading in the Z-axis direction is the thread exit clearance apart from the thread end point.
<3> If [NUMBER] is specified as a cut-in type, threading is performed as many times as the [NUMBER] value. If [DEPTH] is specified, this amount is used in the first cut-in and threading is performed until the specified thread figure is obtained.
<4> After all portions are cut, the tool moves to the position "cut-in start position + clearance (L)" in the X-axis direction at the rapid traverse rate.

- See the following expansions for details of the cutting methods.

[SING.A] : Constant amount of cut, one-edge cutting

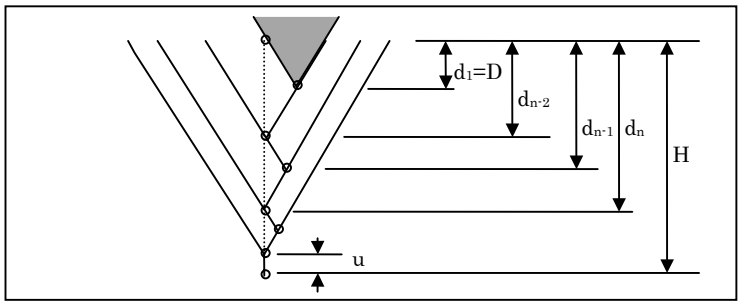


[BOTH A] : Constant amount of cut, both-edge cutting



H=Height of thread crest, D=Amount of cut, u=Finishing allowance

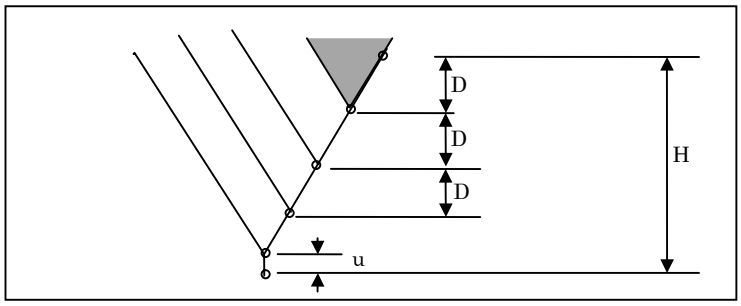
[STAG.A] : Constant amount of cut, both-edge zigzag thread cutting



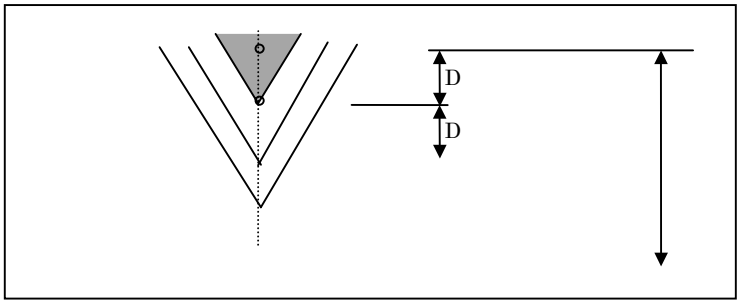
d_{n-2} = D sqrt(n-2)
d_{n-1} = (D(sqrt(n-2) + sqrt(n)))/2
d_n = D sqrt(n)

When the amount of cut is smaller than the minimum amount of cut (parameter No. 27145), the amount of cut is clamped to the minimum amount of cut.

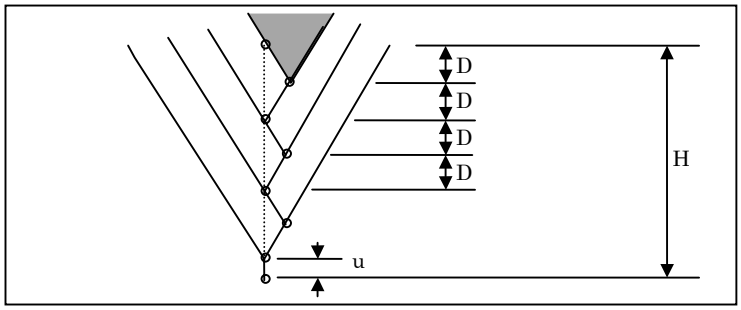
[SING.D] : Constant depth of cut, one-edge cutting



[BOTH D] : Constant depth of cut, both-edge cutting



[STAG.D] : Constant depth of cut, both-edge zigzag thread cutting



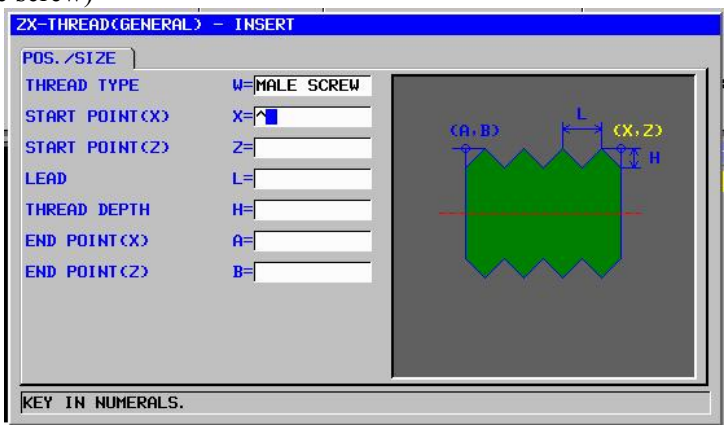
NOTE

Depending on the minimum amount of cut, the specified number of cuts may be excessive. In such a case, threading is performed by a number of cuts less than the specified number of cuts. The number of cuts actually done may differ from the specified number of cuts due to a calculation error.

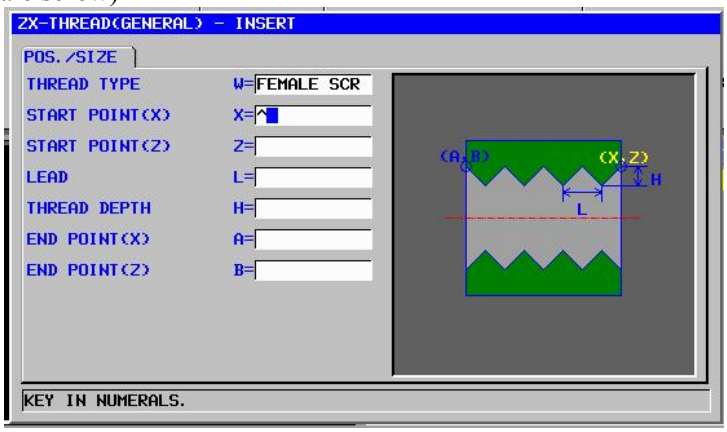
2.4.2 Fixed Form Figure Blocks for Threading

General-purpose thread: G1460 (ZX plane)

Male screw)



Female screw)



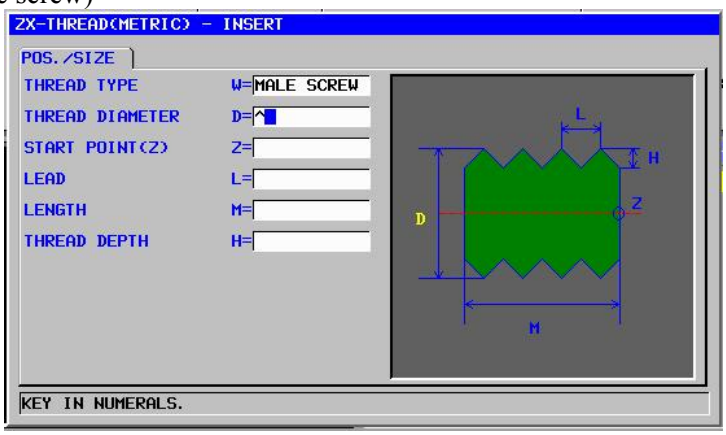
POS./SIZE		
Data item		**Meaning**
W	THREAD TYPE	[MALE] : To be selected when the external thread is specified as threading type [FEMALE] : To be selected when the internal thread is specified as threading type
X	START POINT (X)	X coordinate of a thread start point
Z	START POINT (Z)	Z coordinate of a thread start point
L	LEAD	Thread lead (radius value, positive value)
H	THREAD DEPTH	Thread depth (radius value, positive value) Remark) A thread depth is automatically calculated by pressing [CALC] after inputting a lead.
A	END POINT (X)	X coordinate of a thread end point
B	END POINT (Z)	Z coordinate of a thread end point

NOTE
The thread depth of a general-purpose thread is calculated from parameter No. 27150 (external) or parameter No. 27151 (internal) and a thread lead.

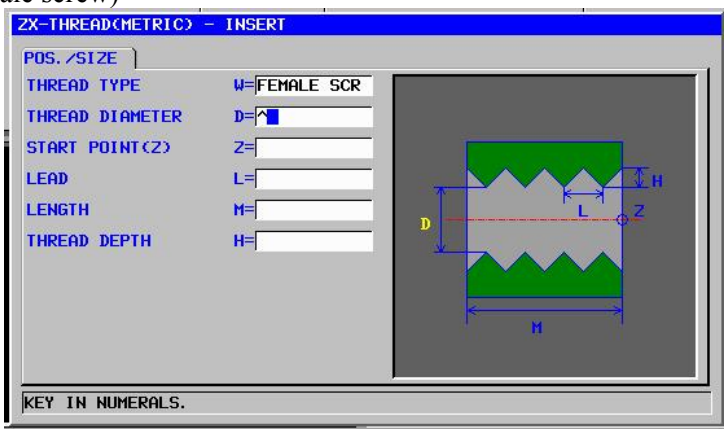
Metric thread: G1461 (ZX plane)

A metric thread is cut. Only one straight thread is machined. Be sure to set a tool angle of 60 degrees.

Male screw)



Female screw)



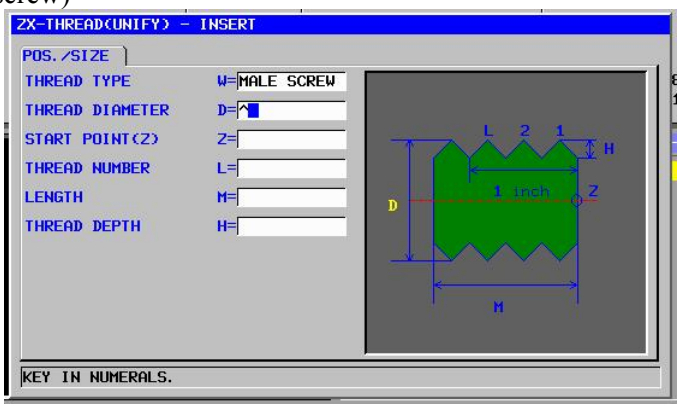
POS./SIZE		
Data item		**Meaning**
W	THREAD TYPE	[MALE] : To be selected when the external thread is specified as threading type [FEMALE] : To be selected when the internal thread is specified as threading type
D	THREAD DIAMETER	Thread diameter (positive value)
Z	START POINT (Z)	Z coordinate of a thread start point
L	LEAD	Thread lead (radius value, positive value)
M	LENGTH	Thread length (radius value, positive value)
H	THREAD DEPTH	Thread depth (radius value, positive value) Remark) A thread depth is automatically calculated by pressing [CALC] after inputting a lead.

NOTE
The thread depth of a metric thread is calculated from parameter No. 27152 (external) or parameter No. 27153 (internal) and a thread lead.

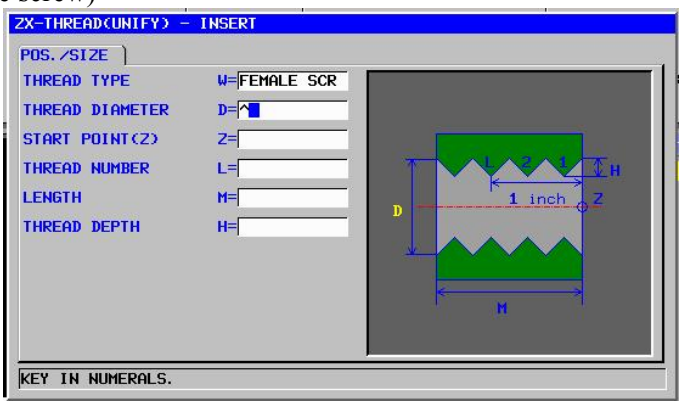
Unified thread: G1462 (ZX plane)

A unified thread is cut. Only one straight thread is machined. For a unified thread, the "number of thread crests/inch" is used instead of a thread lead. Be sure to set a tool angle of 60 degrees.

Male screw)



Female screw)



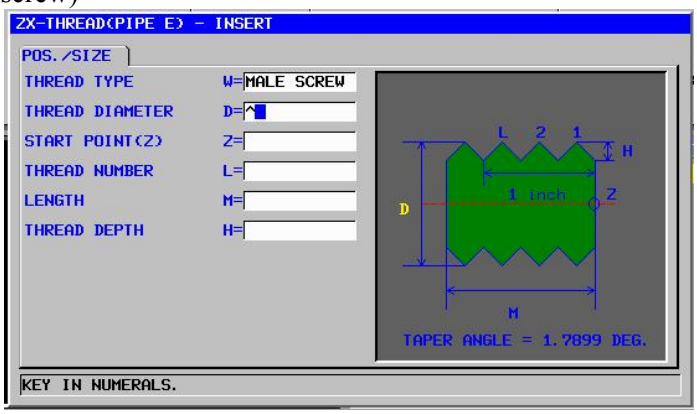
POS./SIZE		
Data item		**Meaning**
W	THREAD TYPE	[MALE] : To be selected when the external thread is specified as threading type [FEMALE] : To be selected when the internal thread is specified as threading type
D	THREAD DIAMETER	Thread diameter (positive value)
Z	START POINT (Z)	Z coordinate of a thread start point
L	THREAD NUMBER	Number of thread crests per inch
M	LENGTH	Thread length (radius value, positive value)
H	THREAD DEPTH	Thread depth (radius value, positive value) Remark) A thread depth is automatically calculated by pressing [CALC] after inputting a thread number.

NOTE
The thread depth of a unified thread is calculated from parameter No. 27152 (external) or parameter No. 27153 (internal) and a thread number.

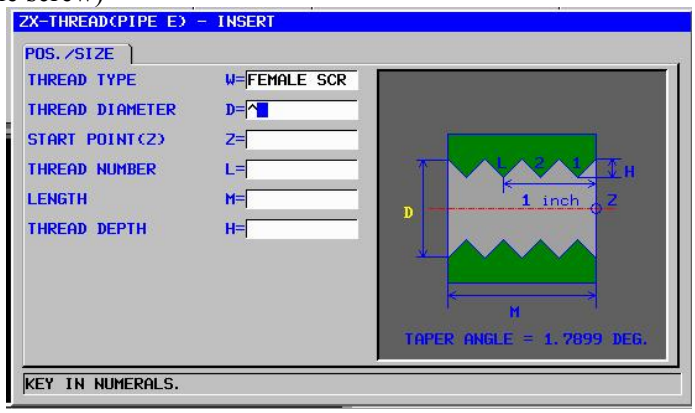
PT thread: G1463 (tapered thread for tubes, ZX plane)

A PT thread (tapered thread for tubes) is cut. Only one tapered thread (tapered by 1.7899 degrees) is machined. Be sure to set a tool angle of 55 degrees. The taper figure of an external thread (male thread) is such that the end face of a blank represents a minor diameter portion. The taper figure of an internal thread (female thread) is such that the end face of a blank represents a major diameter portion.

Male screw)



Female screw)



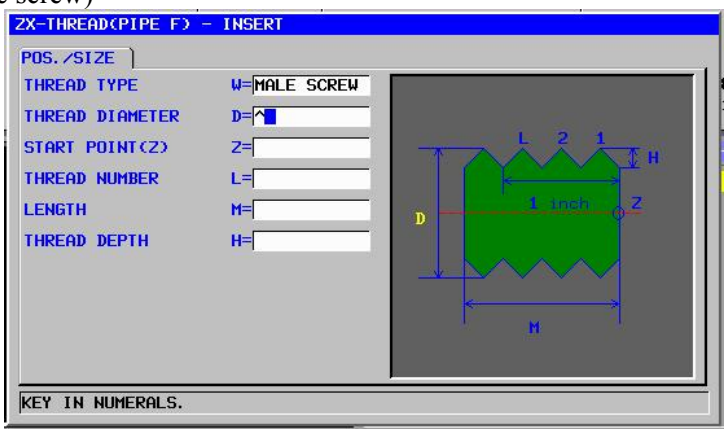
POS./SIZE		
Data item		**Meaning**
W	THREAD TYPE	[MALE] : To be selected when the external thread is specified as threading type [FEMALE] : To be selected when the internal thread is specified as threading type
D	THREAD DIAMETER	Thread diameter (positive value)
Z	START POINT (Z)	Z coordinate of a thread start point
L	THREAD NUMBER	Number of thread crests per inch
M	LENGTH	Thread length (radius value, positive value)
H	THREAD DEPTH	Thread depth (radius value, positive value) Remark) A thread depth is automatically calculated by pressing [CALC] after inputting a thread number.

NOTE
The thread depth of a PT thread is calculated from parameter No. 27154 (external) or parameter No. 27155 (internal) and a thread number.

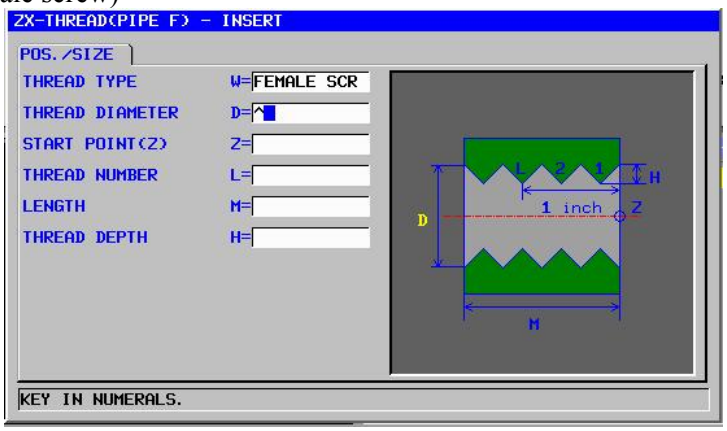
PF thread: G1464 (parallel thread for tubes, ZX plane)

A PF thread (parallel thread for tubes) is cut. Only one straight thread is machined. Be sure to set a tool angle of 55 degrees.

Male screw)



Female screw)



POS./SIZE		
Data item		**Meaning**
W	THREAD TYPE	[MALE] : To be selected when the external thread is specified as threading type [FEMALE] : To be selected when the internal thread is specified as threading type
D	THREAD DIAMETER	Thread diameter (positive value)
Z	START POINT (Z)	Z coordinate of a thread start point
L	THREAD NUMBER	Number of thread crests per inch
M	LENGTH	Thread length (radius value, positive value)
H	THREAD DEPTH	Thread depth (radius value, positive value) Remark) A thread depth is automatically calculated by pressing [CALC] after inputting a thread number.

NOTE
The thread depth of a PF thread is calculated from parameter No. 27154 (external) or parameter No. 27155 (internal) and a thread number.

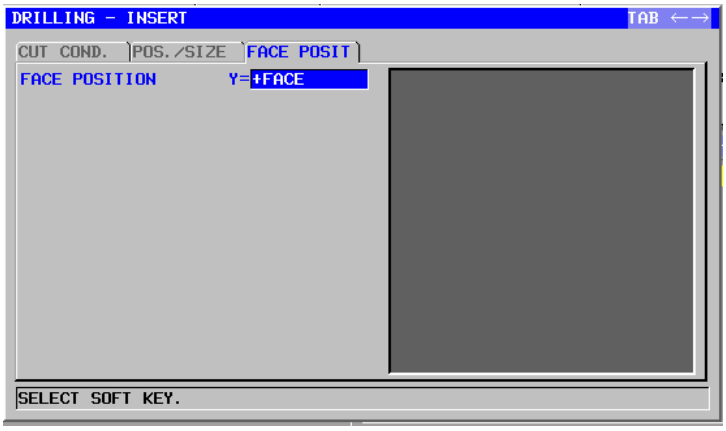
2.5 REAR END FACING BY TURNING

2.5.1 Rear End Facing

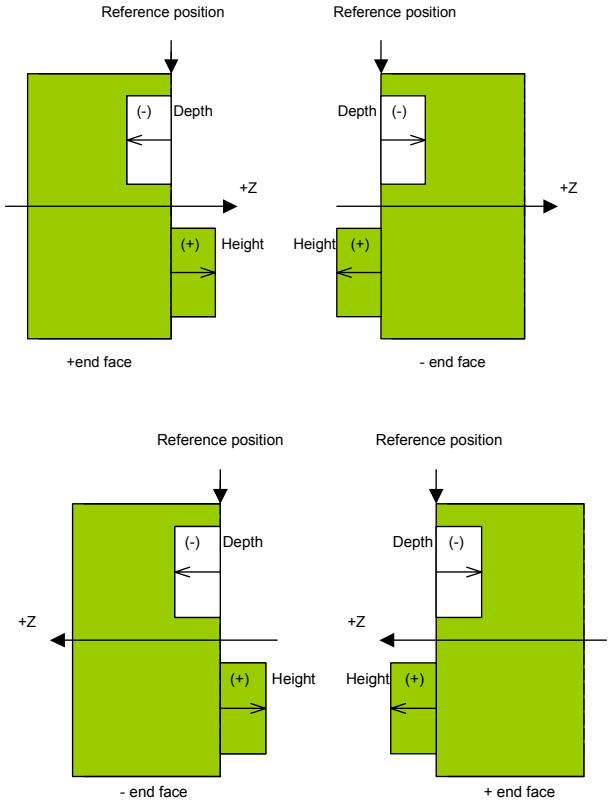
By setting bit 4 of parameter No. 27100 to 1, the input item "FACE POSITION" is displayed on the following menu. By entering this data, rear end facing is enabled.

1. Hole machining – Center drilling : G1100
2. Hole machining – Drilling : G1101
3. Hole machining – Tapping : G1102
4. Hole machining – Reaming : G1103
5. Hole machining – Boring : G1104
6. Turning – End face roughing : G1122
7. Turning – End face semifinishing : G1125
8. Turning – End face finishing : G1128
9. Turning groove – End face roughing : G1132
10. Turning groove – End face roughing and finishing : G1135
11. Turning groove – End face finishing : G1138

Example) Drilling : G1101



FACE POSIT		
Data item		**Meaning**
Y	FACE POSITION	[+FACE]: References the figure below (+ end face). (initial value) [-FACE]: References the figure below (- end face).


Reference position
(-)
Depth
+Z
(+)
Height
+end face
Reference position
Depth
(-)
+Z
Height
(+)
- end face
Reference position
(-)
Depth
+Z
(+)
Height
- end face
Reference position
Depth
(-)
+Z
Height
(+)
+ end face

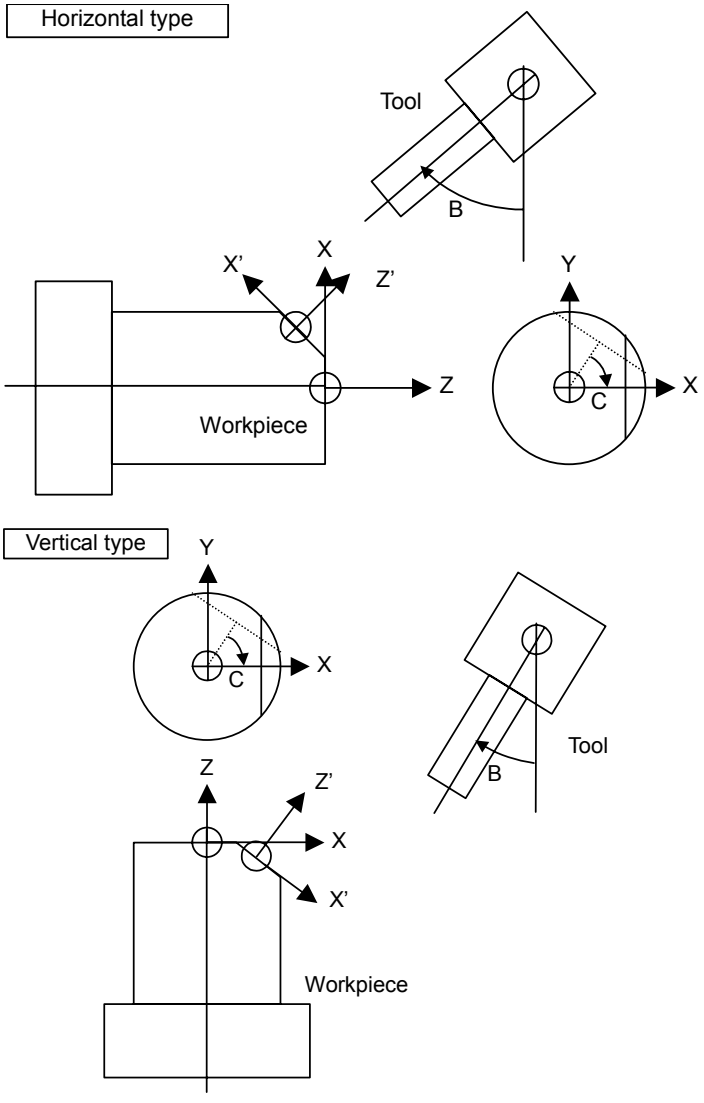
3 SLANT FACE MACHINING (COORDINATE CONVERSION)

NOTE
To use slant face machining with MANUAL GUIDE *i*, the option for the three-dimensional coordinate conversion function is required.
For details, refer to the relevant manual issued by the machine tool builder.

3.1 SUPPORTABLE MACHINE CONFIGURATION

With MANUAL GUIDE *i*, slant face machining, which is a mixture of table rotation and tool rotation, can be specified.
Those parameters that support a machine configuration used must be set beforehand.

(1) Mixture type
This type of machining machines a slant face by workpiece rotation and tool rotation.



NOTE
It is assumed that the C-axis rotates about the Z-axis, and the rotation center is on the Z-axis. Moreover, it is assumed that the B-axis rotates about the Y-axis, and the rotation center is on the Y-axis.

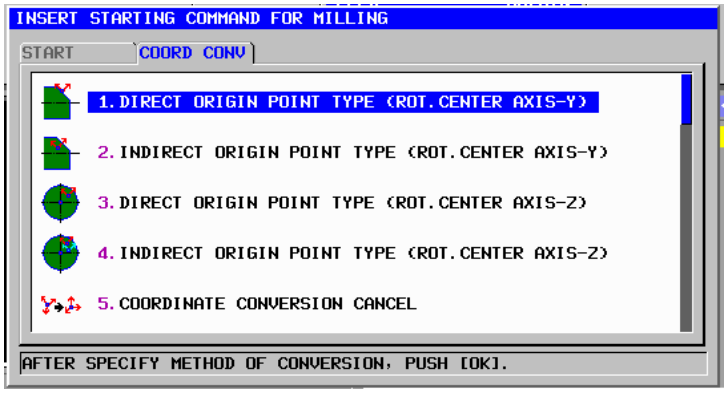
3.2 SLANT FACE MACHINING COMMAND (COORDINATE CONVERSION)

When slant face machining is performed with MANUAL GUIDE *i*, a slant face to be machined must be first specified with the coordinate conversion command, then a machining program for milling to be performed must be input.
Upon completion of slant face machining, coordinate conversion cancellation must be specified.

NOTE
Slant face machining can be performed with the following types of milling (all on the XY plane):
1. Hole machining
2. Facing
3. Contouring
4. Pocketing
5. Grooving

With MANUAL GUIDE *i*, the G code commands listed below are available to enable coordinate conversion for slant face machining.

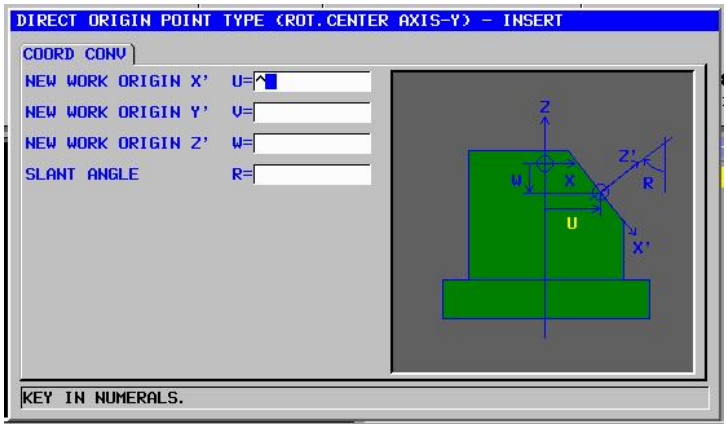
Coordinate conversion			
	Command block	G1952	Direct origin specification (with the rotation center on the Y-axis)
		G1953	Indirect origin specification (with the rotation center on the Y-axis)
		G1954	Direct origin specification (with the rotation center on the Z-axis)
		G1955	Indirect origin specification (with the rotation center on the Z-axis)
		G1959	Coordinate conversion cancel



NOTE
G code for coordinate conversion can be selected from the "COORDINATE CONVERSION" tab on the milling start command menu (displayed by pressing [START] on the milling menu).

Direct origin specification (with the rotation center on the Y-axis): G1952

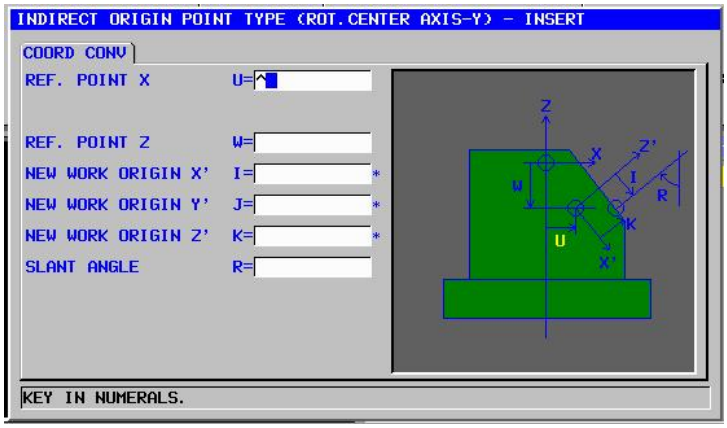
A machining surface rotates about the point (reference point) that serves as the new workpiece origin of the slant face.



COORD CONVERSION		
Data item		**Meaning**
U	NEW WORK ORIGIN X'	X coordinate of the workpiece origin of a slant face
V	NEW WORK ORIGIN Y'	Y coordinate of the workpiece origin of a slant face
W	NEW WORK ORIGIN Z'	Z coordinate of the workpiece origin of a slant face
R	SLANT ANGLE	Signed angle relative to the Z-axis. The CW direction viewed from the plus direction of the rotation axis is positive. (-90≤R≤90).

Indirect origin specification (with the rotation center on the Y-axis): G1953

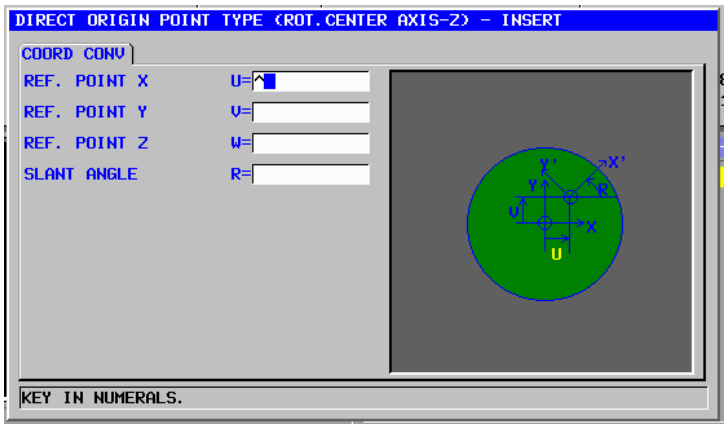
A machining surface rotates about a specified reference point, and the workpiece origin of a slant face, that is, a new machining surface, is to be specified based on the distance from the reference point (positive/negative value).



COORD CONVERSION		
Data item		**Meaning**
U	REF. POINT X	X coordinate of a reference point (workpiece coordinate system before rotation)
W	REF. POINT Z	Z coordinate of a reference point (workpiece coordinate system before rotation)
I	NEW WORK ORIGIN X'	X coordinate of the workpiece origin of a slant face
J	NEW WORK ORIGIN Y'	Y coordinate of the workpiece origin of a slant face
K	NEW WORK ORIGIN Z'	Z coordinate of the workpiece origin of a slant face
R	SLANT ANGLE	Signed angle relative to the Z-axis. The CW direction viewed from the plus direction of the rotation axis is positive. (-90≤R≤90).

Direct origin specification (with the rotation center on the Z-axis): G1954

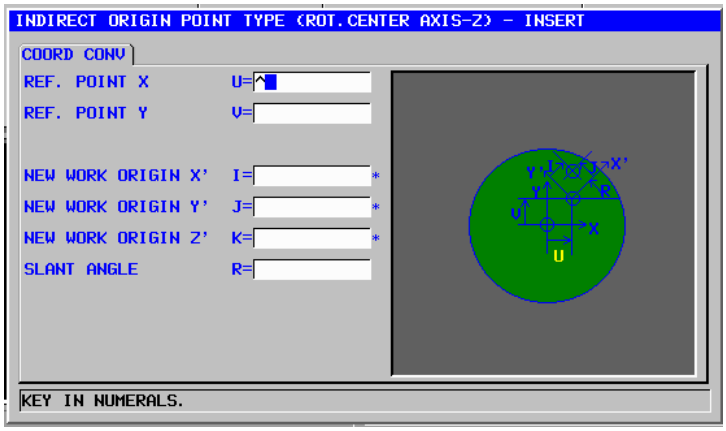
The machining plane rotates about the new point (reference point) that is to become the workpiece origin of the XY plane.



COORD CONVERSION		
Data item		**Meaning**
U	REF. POINT X	X coordinate of a new workpiece origin
V	REF. POINT Y	Y coordinate of a new workpiece origin
W	REF. POINT Z	Z coordinate of a new workpiece origin
R	SLANT ANGLE	Signed angle with respect to the X-axis. The CW direction viewed from the plus rotation axis direction is positive ($-90 \leq R \leq 90$).

Indirect origin specification (with the rotation center on the Z-axis): G1955

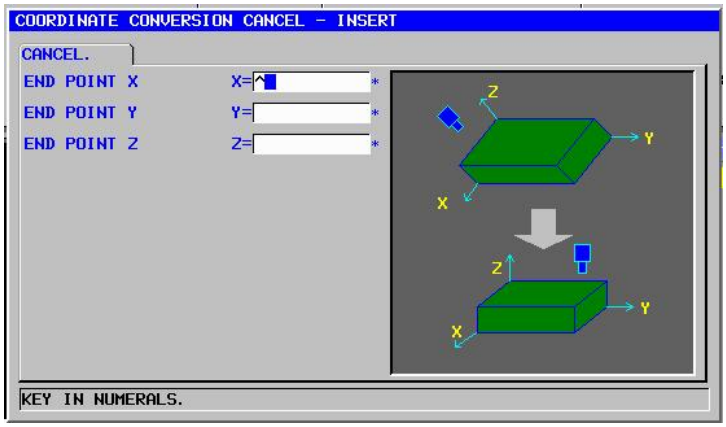
The machining plane rotates about a specified reference point. Define the workpiece origin of the XY plane, which is a new machining plane, by specifying the distance (positive/negative value) from the reference point.



COORD CONVERSION		
Data item		**Meaning**
U	REF. POINT X	X coordinate of a reference point (workpiece coordinate system before rotation)
W	REF. POINT Y	Y coordinate of a reference point (workpiece coordinate system before rotation)
I	NEW WORK ORIGIN X'	X coordinate of the workpiece origin of the slant
J	NEW WORK ORIGIN Y'	Y coordinate of the workpiece origin of the slant
K	NEW WORK ORIGIN Z'	Z coordinate of the workpiece origin of the slant
R	SLANT ANGLE	Signed angle with respect to the Z-axis. The CW direction viewed from the plus rotation axis direction is positive ($-90 \leq R \leq 90$).

Coordinate conversion cancel: G1959

After coordinate conversion is canceled, the tool moves to the entered end point by rapid traverse. If no end point is specified, the tool will not move.



CANCEL		
Data item		**Meaning**
X*	END POINT X	X coordinate to which the tool is to move by rapid traverse after coordinate conversion is canceled. If this data is omitted, the tool will not move.
Y*	END POINT Y	Y coordinate to which the tool is to move by rapid traverse after coordinate conversion is canceled. If this data is omitted, the tool will not move.
Z*	END POINT Z	Z coordinate to which the tool is to move by rapid traverse after coordinate conversion is canceled. If this data is omitted, the tool will not move.

IV. MULTI-PATH LATHE FUNCTIONS (FOR Series 16*i*/18*i*/21*i* ONLY)

1 MULTI-PATH LATHE APPLICATION

- The multi-path lathe option is required in this function.
- This function corresponds to the following CNC control units.
  2 CPU - 2-path lathe CNC , 2CPU - 3-path lathe CNC

NOTE
1 The multi-path lathe function is not supported for the Series 30*i*.
2 The SET-UP Guidance function or the Tool Management function is unsupported in this function.
3 This function cannot be used with the MANUAL GUIDE *i* simulator for the personal computer.

1.1 PREPARATION

The following preparation is needed in order to use this multi-path lathe application.

NOTE
When using the path selection soft key of Manual Guide *i*, make a setting so that the reset key on the MDI panel is enabled for all paths (bit 0 of parameter No. 8100 is 0).
(If the reset key is enabled on a path-by-path basis, (bit 0 of parameter No. 8100 is 1), the path selected in the NC is reset regardless of which path is selected in Manual Guide *i*.)

1.1.1 Machine Configuration Setting

These parameters are set according to the machine configuration.

14702#1 0 : No sub-spindle is provided.
1 : A sub-spindle is provided.
14701#1 0 : Tool post 1 cannot be used with spindle 2.
1 : Tool post 1 can be used with spindle 2.
14701#2 0 : Tool post 1 is placed above spindle 1.
1 : Tool post 1 is placed under spindle 1.
14701#3 0 : Tool post 1 is placed above spindle 2.
1 : Tool post 1 is placed under spindle 2.
27401#0 0 : Tool post 2 cannot be used with spindle 1.
1 : Tool post 2 can be used with spindle 1.
27401#1 0 : Tool post 2 cannot be used with spindle 2.
1 : Tool post 2 can be used with spindle 2.
27401#2 0 : Tool post 2 is placed above spindle 1.
1 : Tool post 2 is placed under spindle 1.
27401#3 0 : Tool post 2 is placed above spindle 2.
1 : Tool post 2 is placed under spindle 2.
27402#0 0 : Tool post 3 cannot be used with spindle 1.
1 : Tool post 3 can be used with spindle 1.
27402#1 0 : Tool post 3 cannot be used with spindle 2.
1 : Tool post 3 can be used with spindle 2.
27402#2 0 : Tool post 3 is placed above spindle 1.
1 : Tool post 3 is placed under spindle 1.
27402#3 0 : Tool post 3 is placed above spindle 2.
1 : Tool post 3 is placed under spindle 2.
14706 : Directions of the three basic axes of spindle 1
14706 : Directions of the three basic axes of spindle 2
16 : Right-handed coordinate system, right = +Z, up = +X
17 : Right-handed coordinate system, right = -Z, up = +X
18 : Right-handed coordinate system, right = -Z, up = -X
19 : Right-handed coordinate system, right = +Z, up = -X
27400#0 0 : Tool post is selection by the software key.
1: Tool post is selection by the PMC signal.

1.1.2 Set Icon for Selected Turret

The icon displayed when tool path-1 or path-2 are selected is set by the parameter.

27410 : icon number when path-1 is selected.

27411 : icon number when path-2 is selected.

27412 : icon number when path-3 is selected.

- Icon number list

Icon number	0	1	2	3	4	5	6
icon							

Icon number	10	11	12	13	14	15
icon						

Icon number	20	21	22	23	24	25
icon						

Icon number	30	31	32	33	34	35
icon						

1.2 OPERATIONS OF MULTI-PATH LATHE

1.2.1 Changing Screens for Each Path

On the MANUAL GUIDE *i* for multi-path lathe, screens and operations are done on each path respectively.
On its screen, the icon for selected path will be displayed at the upper right part.

1.2.1.1 Changing by a soft-key

On the each screen, CHPATH soft-key will be displayed.

Press [CHPATH], the displayed turret will be changed.
In case of 2-path system : turret-1 > turret-2 > turret-1 > …
In case of 3-path system : turret-1 > turret-2 > turret-3 > turret-1 > …

1.2.1.2 Changing by a switch on a machine operator's panel

Connected with the path selection signal (HEAD<G063#0>, HEAD2<G062#7>), change the display of MANUAL GUIDE *i*.

HEAD (G63.0)	HEAD2 (G62.7)	Displayed Path number
0	0	1
1	0	2
0	1	3

NOTE
Select turret, display returns to a basic screen of each mode when the system is changed occasionally.

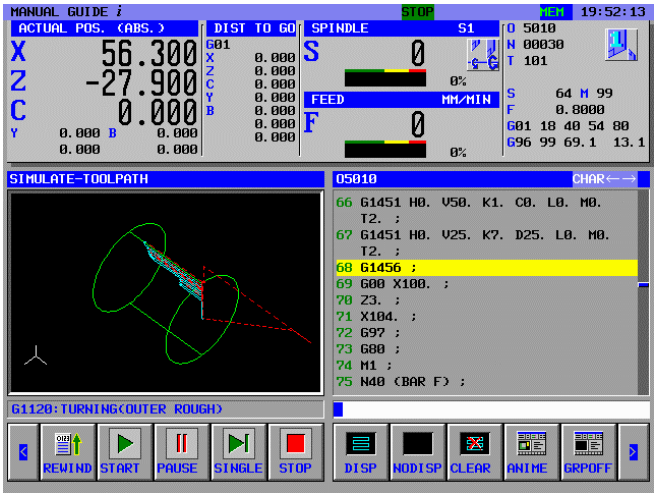
1.3 ANIMATION FOR MULTI-PATH LATHE

The tool path and animation for multi-path lathe are available.

NOTE
1 When machining simulation is started, it is necessary to set MEM mode for all path.
2 Displaying is not done for the combination where the parameter is set to the combination of the main axis and the cutlery stand as invalidity.

1.3.1 Tool Path Drawing During Machining and Tool Path Machining Simulation

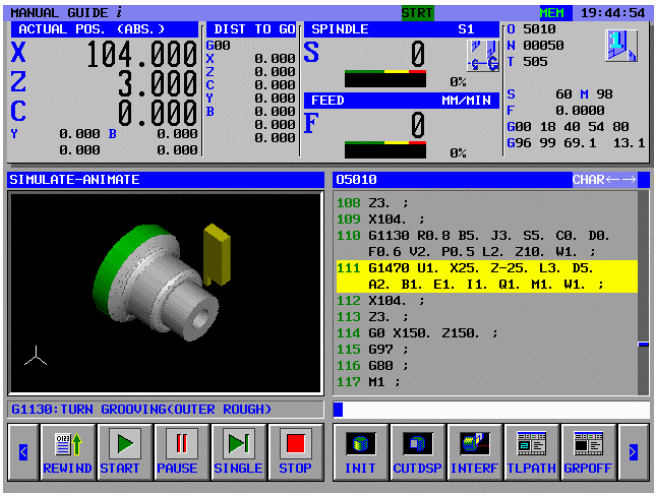
In the tool path drawing during machining (tool path), machining simulation (tool path), the turret selected now is drawn.



When the path (turret) is changed during drawing, the drawing starts from halfway status.
When the mode of changed path (turret) is other of MEM mode, the drawing will stop.

1.3.2 Machining Simulation (Animation)

In the machining simulation (animated), the Drawing for each turret is executed simultaneously, regardless of selected turret.



NOTE

Only animation for the spindle selected latest between both turrets is displayed. (The animation for the other spindle is not displayed.)

1.4 MACHINING SIMULATION FOR EACH PATH

In the multi-path system of MANUAL GUIDE *i*, the machining simulation is performed only at the selected path by the R signal.
In the multi-path system of MANUAL GUIDE *i*, the machining simulation is performed only at the selected path by a R signal that is set at the parameter No.27309 (The first figure is "bit" and the other figure is the R signal number.).
For example, with 2 paths system

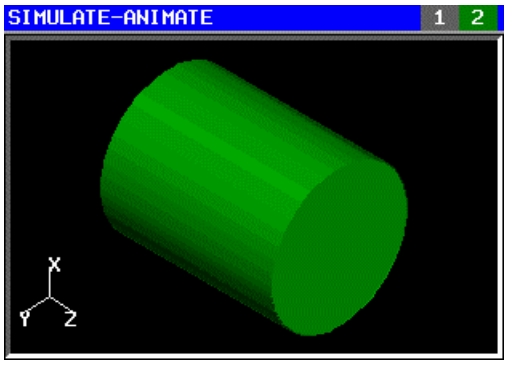
- At Path 1, the parameter No.27309=1001
- At Path 2, the parameter No.27309=2001

Then,

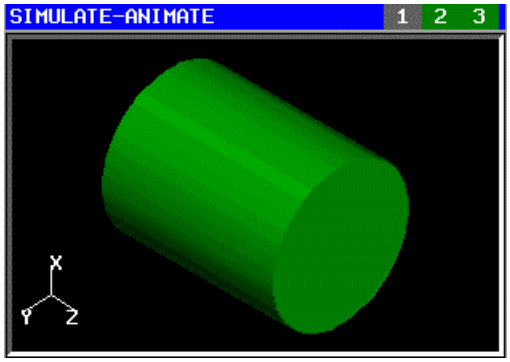
- At Path 1, if R100.1 is 1
- At Path 2, if R200.1 is 1

machining simulation is performed at each path. The status of the signal of each path is displayed at a title bar of simulation window. Green is the status for enable the simulation to be performed. Gray is the status for disable.

Ex.1 With 2 paths system, the signal of Path 1 is 0 and the one of Path 2 is 1.



Ex.2 With 3 paths system, the signal of Path 1 is 0 and the ones of Path 2/3 are 1.



NOTE

1 If M code for waiting other paths is commanded, the machining simulation is in pausing for performing of the same M code at other paths. So, if this function is made available in use of M code for waiting, it must be made disabled by the management such as the M code is disabled by using a signal of PMC for ignoring it.
2 This function needs the optional function MANUAL GUIDE *i* Animated Drawing.
3 In drawing during machining, R signal that is set at the parameter No. 27309 is disabled.
4 If the parameter No.27309 is not set at any paths, this function is disabled.
5 R signal, which is set at the parameter No. 27309, is available in converting into NC formatted program, too. The NC statement converting function is available only at the path in displaying. So, when the R signal at the path, which is the object for NC statement converting, is off, the warning message “TURN THIS PATH'S SIMULATE-SWITCH ON” is displayed.
6 In the NC statement converting function, the status of the R signal is not displayed.
7 Even if the R signal of a path is turned from “ON“ to “OFF” in the machining simulation, the machining simulation is not pausing but is performed at the path.
8 Even if the R signal of a path is turned from “ON“ to “OFF” in the machining simulation, [SINGLE] and [PAUSE] are not available at the path. In short, these soft keys are available at the path when the R signal is on.
9 If [STOP] or the RESET is pressed in the machining simulation, it stops at all paths for all status of the R signal.
10 If any paths are in machining, the machining simulation can not be performed.
11 To perform the machining simulation, the mode must be MEM at the selected path for displaying. If it is turned to another path in the other mode than MEM in the machining simulation, the machining simulation is stopped and the animation window turns off.

1.5 OTHERS

NOTE
1 Guidance window for machining cycle data entering screen are displayed following the specific coordinate system (upper direction X+: right direction Z+: parameter 14706=16).
2 The material is common with path-1 and path-2. The registration of the material is possible from path-1 and path-2. (When registered from both path, the only latest is effective.)

2 SIMULTANEOUS ALL PATH DISPLAY / EDITING FUNCTION

2.1 OUTLINE

In the multi-path lathe, simultaneous all path display and edit function. Became available.

Supported machine construction is as follows.

- 2-path 2-spindle
- 3-path 2-spindle

In order to use this feature, the following option is necessary.

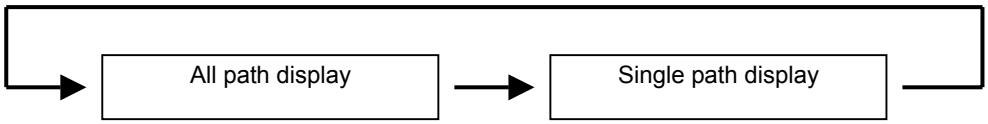
- Multi-path lathe function for MANUAL GUIDE *i*

2.2 DETAILS

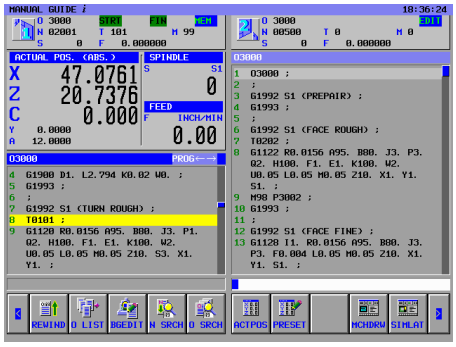
2.2.1 How to Start

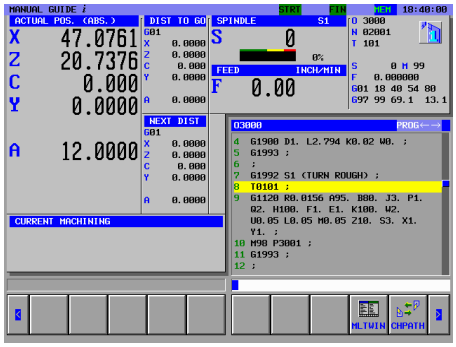
[MLTWIN] is displayed next to [CHPATH] in each basic mode.
(If the setting that [CHPATH] is not used is specified, the softkey is arranged to the same position.)

When [MLTWIN] is pressed, the display mode is changed as follows.

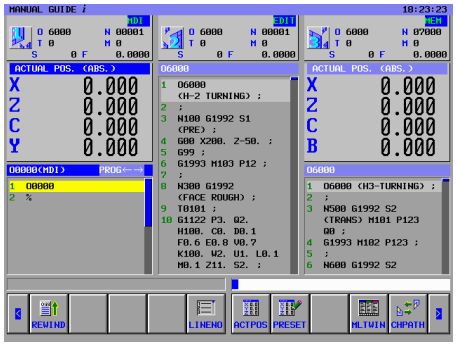


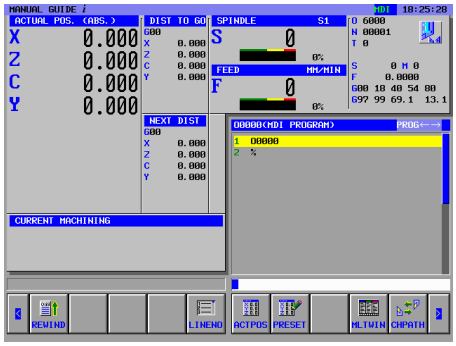
In the case of 2-path





In the case of 3-path



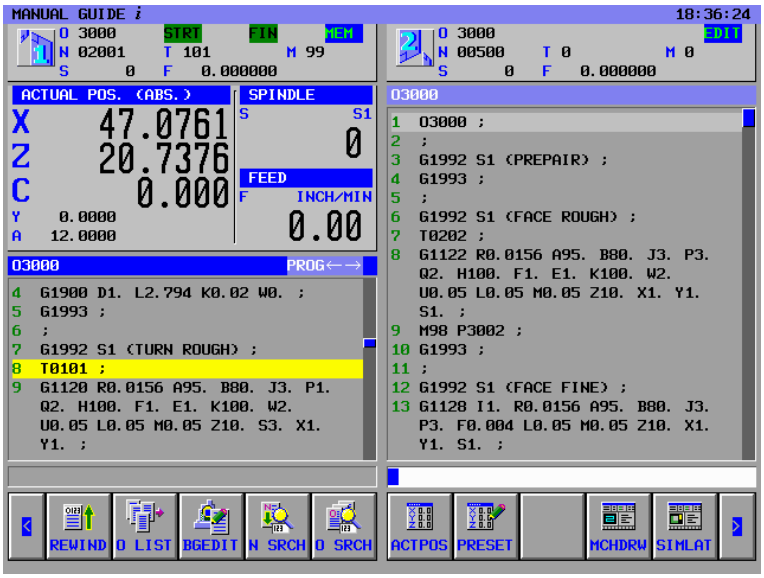


2.2.2 How to Finish

Press [MLTWIN] again, and the display mode will be changed to single display mode.

2.3 SCREEN CONFIGURATION

The screen composition of simultaneously for all path display and edit function is explained.



2.3.1 Display Position of Each Path

- 2-path
  First path : left side
  Second path : right side

- 3-path
  The display position of each path will be decided automatically according to following parameter.
  14701#1 : head 1 cannot be used in spindle 2 or can (0/1)
  27401#0 : head 2 cannot be used in spindle 1 or not (0/1)
  27401#1 : head 2 cannot be used in spindle 2 or can (0/1)
  27402#0 : head 3 cannot be used in spindle 1 or not (0/1)
  27402#1 : head 3 cannot be used in spindle 2 or can (0/1)

2.3.2 Status Display Part

The status display part display the status of each path.
This part is displayed in all operation mode.

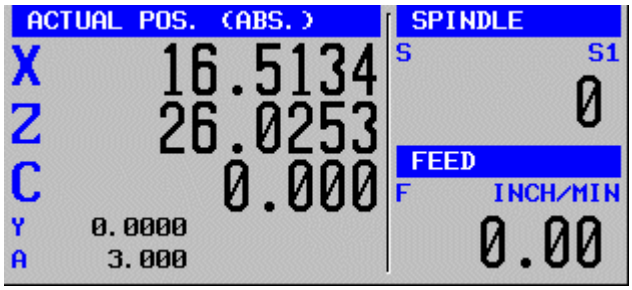
Icon of displayed path.

Operation mode
MDI, MEM, RMT, EDIT, HND, JOG, TJOG, THND, INC, REF
Alarm status
ALM
Emergency stop status
EMG
Reset status
-RESET-
Automatic operation status
STOP, HOLT, STRT
Axis motion and dwelling status
MTN, DWL
Executing auxiliary functions
FIN

O : Program number (O number)
N : Sequence number (N number)
T : Modal T code commanded data
M : Modal M code commanded data
S : Modal S code commanded data
F : Modal F code commanded data

2.3.3 Current Position Display Part

This screen is displayed out of EDIT mode.

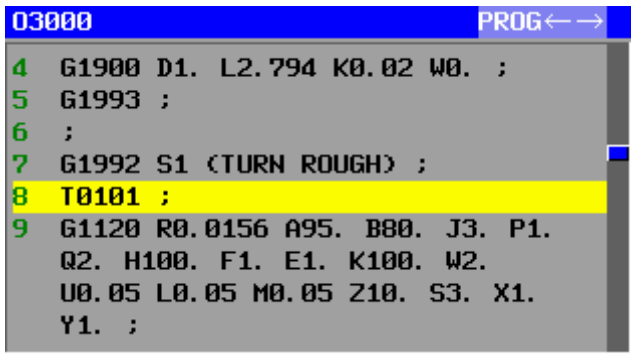


Using [ACTPOS], absolute position, relative position, machine position, and distance to go in turn.

(In case of 2-path, actual spindle speed and actual feed rate are also displayed.)

2.3.4 Program Display Part

This screen will be displayed in all operation mode.
(In case of EDIT mode, this part will be extended because current position part is lost.)



2.4 HOW TO SELECT PATH

Select the target path using [CHPATH] or path selection signal. As for the selected path, the title of position and program display part is displayed in blue. (As for the not selected path, the title will be displayed in light blue.)
For the selected path, a similar operation system to the normal screen is supported.
(In case of the screen which is not supported all path display mode, the single path display mode will be selected automatically.)

2.5 OTHERS

- The screen which need full size, like animation, convert to nc program, process list edit, chsize, and so on, will change to full screen display automatically.
- It is impossible to operate for the not selected path.
- In 3-path, actual spindle speed and actual feed rate is not supported because the display area is too small.

3 PROCESS LIST EDITING FUNCTION

Available CNC types..

- 2 CPU 2 path lathe CNC
  Lathe with 2 turrets and 2 spindles, and each turret can perform to both of spindle#1 and spindle#2 respectively.
- 2 CPU 3 path lathe CNC
  Lathe with 3 turrets and 2 spindles, turret#1 can perform to both of spindle#1 and spindle#2, turret#2 can perform to spinlde#2, and turret#3 can perform to spindle#1.
- 1 CPU 1 path lathe CNC
  Lathe with 2 spindles, 1 turret can perform to both of spindle#1 and spindle#2.

NOTE
1 Process list editing is available only at EDIT mode. Under background editing mode, it is not available.
2 When you use a process list editing function, you need a lathe machining cycle optional function. Furthermore, if you use [Add /] and [Del /] functions, you need an optional block skip function.

3.1 PREPARATION

3.1.1 Parameter

The following parameter is needed to be set.

- 14703#3 = 1 : Use the process list edit function

In case of using Add / function and Del / function,

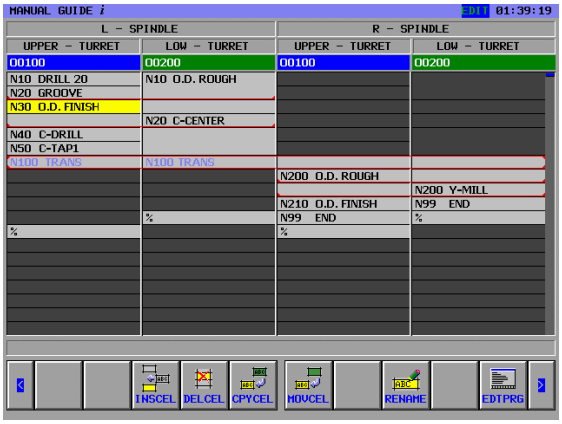
- 14701#6 = 1: Use program check function for each spindle.

3.2 START AND END OPERATIONS

3.2.1 Start

Set CNC to EDIT mode, and press [<] or [>], following soft-keys will appear.

Press this [EDTCEL], process table edit screen will appear.

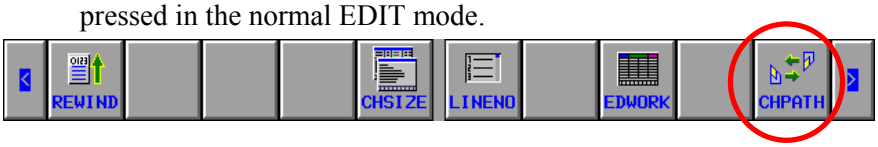


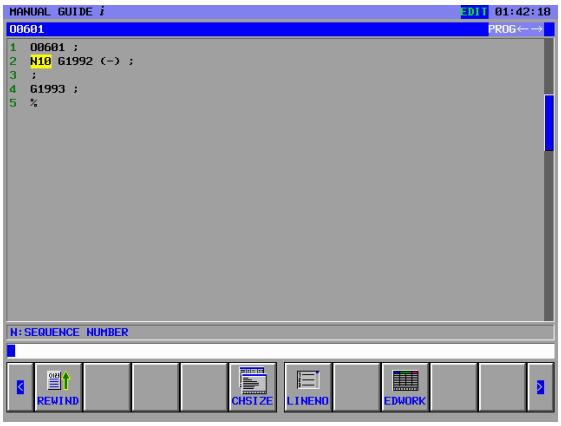
If the consistency of G1992 and G1993 is lacked when the process table edit function start, the process table edit function will not appear, and line number and message of the warning will be shown at the lower left.

Warning Message	Description
Top G1992 is short	G1993 appears without G1992.
G1993 is duplicate.(P=x/L=xxxx)	G1993 is doubled because of no G1992 in the interval.
LAST G1993 is short.	Program ends without the last G1993
G1992 is duplicate. (P=x/L=xxxx)	G1992 is doubled because of no G1993 in the interval.
M CODE is duplicate. (P=x/L=xxxx)	The same waiting M code is used again.
Waiting target is short. (P=x/L=xxxx)	Waiting M code does not exist at waiting target with P.
Illegal waiting order.(P=x/L=xxxx)	The order waiting M code appears is not correct..
No program.	The program selected now does not exist.
No waiting M code. (P=x/L=xxxx)	Waiting M code is lost at transfer.
Illegal P command. (P=x/L=xxxx)	The value of P command is different even if the value of waiting M code is the same. Own path number is lost in the P command of waiting M code.
Illegal S command. (P=x/L=xxxx)	Spindle number is not correct.
WAITING EXIST. (P=%d/L=%ld)	A waiting M code exists for a 1-path lathe.
TRANS. EXIST.(P=%d/L=%ld)	A Q command exists for a 1-path lathe.

3.2.2 End

Press [RETURN] soft-key, and the simultaneously for all path display screen appears, which also appears when [MLTWIN] soft-key is pressed in the normal EDIT mode.



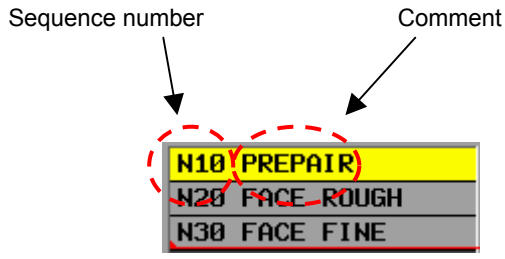


Changing CNC mode, the screen change for the other mode.

3.3 DISPLAY CONTENTS

3.3.1 Cell

Each process is corresponded to the frame in the table which is called a cell.
Only following information is displayed in this frame.



Moreover, there are following kinds of cells.

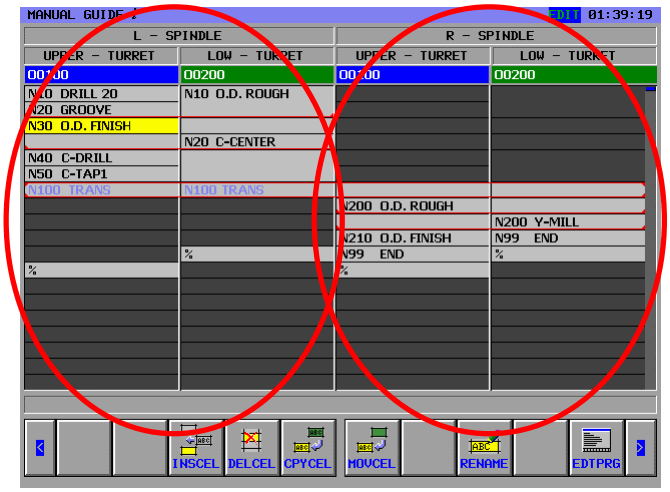
Kind	Description	Graphic
Normal Cell	Cell where the process exists.	N20 FACE ROUGH
Input Impropriety Cell	There is frame for display, but the process does not exist for it.	

3.3.2 Current Cell

The target cell for the operation is shown. It is possible to move with the cursor key. As for the selected cell, the background color is displayed in yellow.

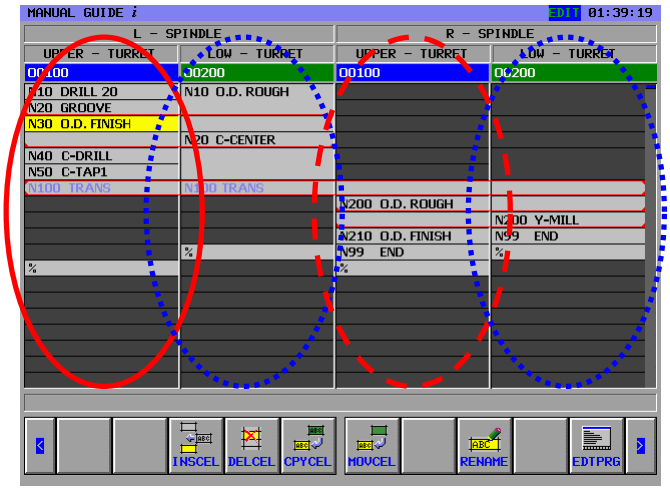
3.3.3 Spindle

First of all, each process is arranged according to the spindle.
The operator can see the process belongs to which spindle at a glance.



3.3.4 Turret

Each process is arranged further in the spindle according to the turret.

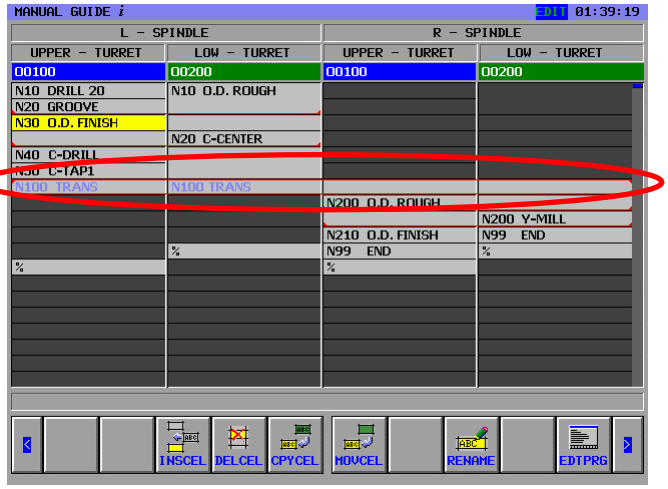


3.3.5 Waiting Line

The line that shows waiting exists is displayed.

3.3.6 Transfer

When transfer exists, that is displayed by blue character.
Transfer is arranged at the top and bottom.



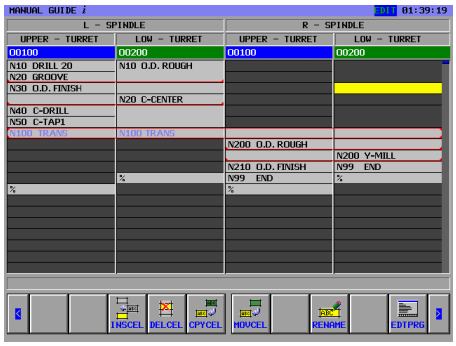
3.4 BASIC OPERATIONS

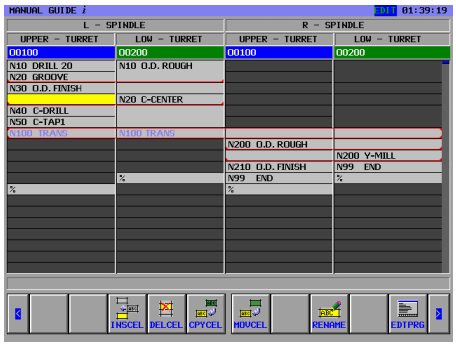
The following operations can be done in each cell.

3.4.1 Basic Operations

A current cell can be moved up, down, right, and left by operating the cursor key.

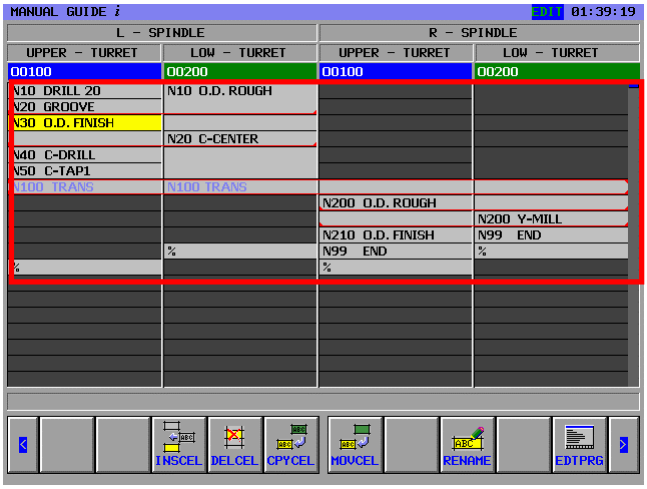
Directing left at the leftmost cell, current cell moves on the rightmost cell upper by one piece.





Directing right at the rightmost cell, current cell moves on the leftmost cell lower by one piece.

Operating page key, it is possible to scroll entire table up and down by one page.
The range of the movement of the cursor is from the head to % about each row.



3.5 EDITING OPEARTIONS

The following operations are available on each cell.

Operation	Description
INSCEL	Insert process to the upper part of the specified cell.
DELCEL	Delete the specified cell.
CPYCEL	Copy the specified cell to the specified position.
MOVCEL	Move the specified cell to the specified position.
RENAME	Modify the comment of the specified cell.
EDTPRG	Edit the program including the specified cell.
STWAIT *	Set the waiting to the specified cell.
CLWAIT *	Release the specified waiting.
STTRNS *	Set the transfer to the specified cell.
CLTRNS *	Release the specified transfer.
ADD / *	Add the optional block skip.
DEL / *	Delete the optional block skip.

NOTE
Operations marked with * cannot be used on 1-path lathe.

3.5.1 Insertion of a Cell

- Function
  - Add a process.
  - Add the process to the upper side.
  - In NC program,

    Process start block : G1992 Sx (xxxx)

    Process end block : G1993

    These codes are inserted automatically.

- Basic operation

  1. Move cursor to the cell to be inserted.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

  2. Press [INSCEL] soft-key.

TURRET 1	TURRET 2
PROC NAME MIDDLE	
%	%

The dialog for the process name input is displayed.
Input MIDDLE as an example.

  3. Press OK, and the process will be inserted.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
MIDDLE	N20 TAP
N20 FINE	%
%	

Cursor moves to the new cell.

(When cancel is pressed, return to previous state.)

- Others
  - In case of having waiting in the lower, for not breaking the waiting line, modify the height of the cell of other row that have the waiting to twice.

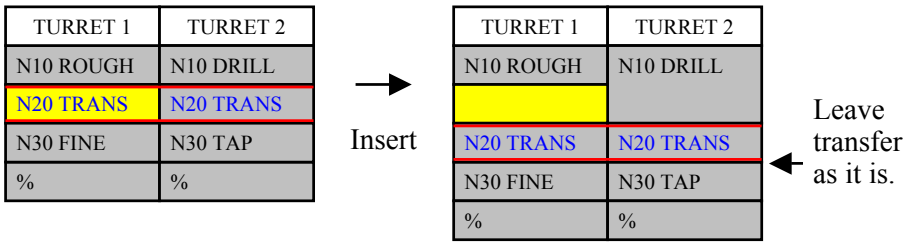
TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

Insert →

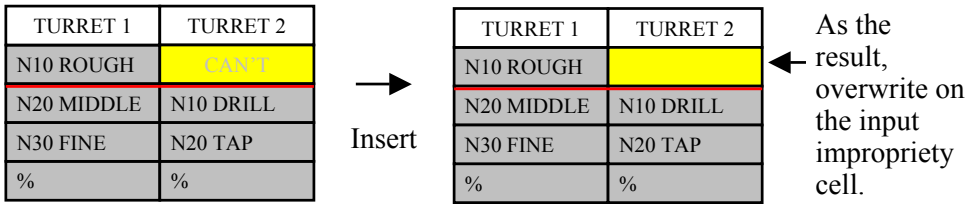
TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
	N20 TAP
N20 FINE	
%	%

Height of cell becomes twice.

- If the new cell is inserted on the cell having the waiting, the waiting do not move to the new cell. (Transfer is also similar.)



- Even if current cell is input impropriety one, insertion of the cell is possible. After insertion, the entire table is displayed again, as the result, there are some case where overwriting is executed.



3.5.2 Deletion of a Cell

- Function
  - Delete the process at the current cursor.
  - In NC program,

    Process start block : G1992 Sx (xxxx)

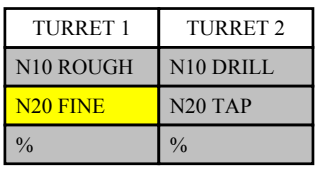
    (Normal block)

    Process end block : G1993

    These blocks are deleted automatically.

- Basic operation

  1. Move cursor to the cell to be deleted.



2. Press [DELCEL] soft-key.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

"ARE YOU SURE YOU WANT TO DELETE IT ?" is displayed in the message display part.
Press [YES] or [NO].

3. Press [YES], and the process will be deleted.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
%	N20 TAP
	%

The cursor position is left as it is.
(In case of "NO", returns to former state.)

- Others
  - In case of having waiting in the lower line, for not breaking the waiting line by adjusting after deletion, modify the height of the cell that have the waiting to twice.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

Delete →

TURRET 1	TURRET 2
N10 ROUGH	N20 TAP
N20 FINE	
%	%

↕ Height of cell becomes twice.

  - There is the cell that cannot be deleted.

TURRET 1	TURRET 2
N10 ROUGH	
N20 MIDDLE	N10 DRILL
N30 FINE	N20 TAP
%	%

← Input impropriety cell

← % cell

  - When the cell that have waiting, the waiting is released. (Transfer as well)

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

Delete →

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N30 FINE	N20 TRANS
%	N30 TAP
	%

← Release the partner of transfer, too.

(When one of the waiting member among the 3-path is deleted, the rest waiting will not be released.)

3.5.3 Copying of a Cell

- Function
  - Copy the process
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

    The blocks between above two blocks and comment in the G1992 block arc copied automatically.
- Basic Operation

1. Move the cursor to the source cell.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

2. Press [CPYCEL] soft-key.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Add "*" to the top of the source cell.

3. Move cursor to the destination cell.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Moreover background color of the source cell become green.

Press [CPYCEL] or [CANCEL].

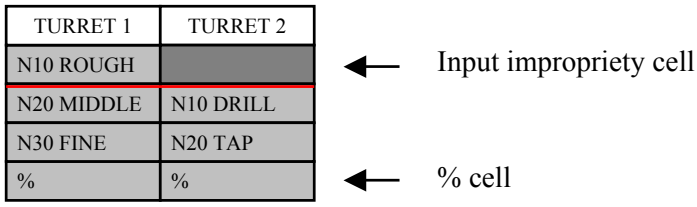
4. Press [CELCPY], overwrite copy will be done if destination process is vacant.

TURRET 1	TURRET 2
N10 ROUGH	N10 FINE
N20 FINE	N20 TAP
%	%

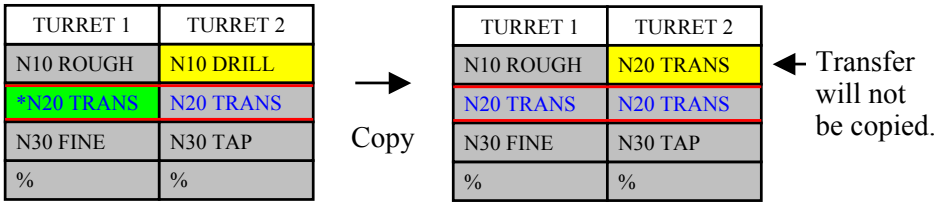
(When cancel is pressed, return to previous state.)

- Others
  - When the destination process is not vacant, it is possible to select overwrite, insert, and cancel.

  - There is the cell that cannot be specified as source and destination.



  - The waiting and transfer will not be copied.



3.5.4 Moving of a Cell

- Function
  - Move the process (The source cell is removed.)
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

    The blocks between above two blocks and comment in the G1992 block are moved automatically.
- Basic Operation

1. Move the cursor to the source cell.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

2. Press [CPYCEL].

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Add "*" to the top of the source cell.

3. Move cursor to the destination cell.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Moreover background color of the source cell become green.

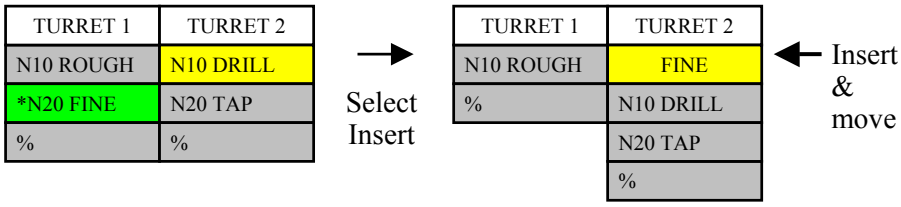
Press [CPYCEL] or [CANCEL].

4. Press [CELMOV], overwrite move will be done if destination process is vacant.

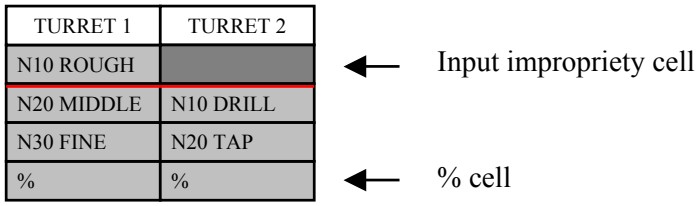
TURRET 1	TURRET 2
N10 ROUGH	N10 FINE
%	N20 TAP
	%

(When cancel is pressed, return to previous state.)

- Others
  - When the destination process is not vacant, it is possible to select overwrite, insert, and cancel.

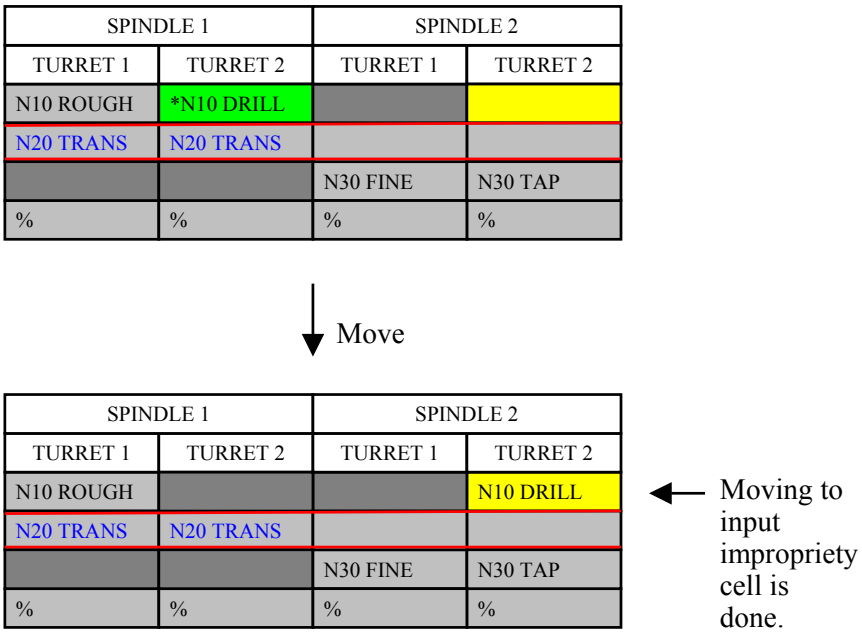


  - There is the cell that cannot be specified as source and destination.



  - The waiting and transfer will not be moved.

  - It is possible to move to input impropriety cell just in case between the same line and turret.



- It is possible to move to input impropriety cell just in case that any cell in the same line and turret is input impropriety one.

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH	CAN'T		CAN'T
N20 TRANS	N20 TRANS		
		N30 FINE	*N30 TAP
%	%	%	%

Moving

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH			N30 TAP
N20 TRANS	N20 TRANS		
		N30 FINE	%
%	%	%	

Moving to input impropriety cell is done.

3.5.5 Modification of Process Name

- Function
  - Modify the process name.
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    Modify comment in that block.

    When clear the process name, delete comment with a round bracket.
- Basic Operation

  1. Move the cursor to the cell to be modified.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

  2. Press [RENAME].

TURRET 1	TURRET 2
PROC NAME MIDDLE	
%	%

The dialog for the process name input is displayed. Input MIDDLE as an example.

  3. Press OK, and the process name will be modified.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 MIDDLE	N20 TAP
%	%

(When cancel is pressed, return to previous state.)

- Others
  - There is some cell whose name cannot be modified.

TURRET 1	TURRET 2	
N10 ROUGH		← Input impropriety cell
N20 MIDDLE	N10 DRILL	
N30 FINE	N20 TAP	
%	%	← % cell

3.5.6 Program Edit

- Function
  - Edit the process.
  - The NC Program with current cell is opened in all screen mode, and the cursor is set to the head of the process with current cell.

- Basic Operation

1. Move the cursor to the cell to be edited.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

2. Press [EDTPRG].

```
;
N10 G1992 S1 (ROUGH);
        :
        :
G1993;
;
```

The cursor of the edit screen automatically moves to the start position of the process.

3. Do the edit work.

```
;
N10 G1992 S1 (ROUGH);
        :
      G1993;
N20 G1992 S1 (FINE);
        :
G00X0.Z0.;
        :
G1993;
```

The Edit work is done with usual edit screen. (all screen display by the size substitution)

The following process can be similarly edited because of a usual edit screen.

4. Press [EDWORK].

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

The process edited on a usual edit screen becomes a current cell.

- Others
  - When select head of MANUAL GUIDE *i* according to tool post selection signal, it is necessary to set tool post selection signal to the head that the target cell belongs to in advance.
  - When edit work is started on the input impropriety cell, the next effective process in the same turret will be opened.

TURRET 1	TURRET 2
N10 ROUGH	CAN'T EDIT
N20 MIDDLE	N10 DRILL
N30 FINE	N20 TAP
%	%

EDTPRG

```
;
N10 G1992 S1 (DRILL);
        :
        :
G1993;
;
```

3.5.7 Assign of Waiting

- Function
  - Set the waiting between the process.
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

  Mxxx (Pxx) will be set to one or both of these blocks.

- Basic Operation

1. Press [STWAIT]. (Cursor position pretermission)

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

The software key array changes into set waiting mode.

2. Move cursor to the source waiting, and press [SELECT].

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Add "*" to the top of the selected cell.

3. Move cursor to the destination waiting, and press [SELECT].

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
*N20 FINE	N20 TAP
%	%

Moreover background color of the source-waiting cell become green.

Press any one of [SETTOP], [SETEND], and [STBOTH].

4. Press [SET-].

- When [SETTOP] is pressed

TURRET 1	TURRET 2
N10 ROUGH	
N20 FINE	N10 DRILL
%	N20 TAP
	%

(When finished normally, selected state will be released automatically.)

- When [SETEND] is pressed

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	
%	N20 TAP
	%

(When finished normally, selected state will be released automatically.)

- When [STBOTH] is pressed

TURRET 1	TURRET 2
N10 ROUGH	
N20 FINE	N10 DRILL
%	N20 TAP
	%

(When finished normally, selected state will be released automatically.)

5. Press [RETURN], and release set waiting mode.

- Others
  - There is some cell that cannot be specified as source and destination waiting.

TURRET 1	TURRET 2	
N10 ROUGH		← Input impropriety cell
N20 MIDDLE	N10 DRILL	
N30 FINE	N20 TAP	
%	%	← % cell

  - When the waiting has already been set, the waiting cannot be set.

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
*N20 WAIT	N20 WAIT
N30 FINE	N30 TAP
%	%

Waiting Operation

  - In case of shortage of waiting M code, the waiting cannot be set.
  - It is possible to set the waiting just between upper parts or lower parts.

- It is impossible to set waiting across other waiting.

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
N20 TRANS	N20 TRANS
*N30 FINE	N30 TAP
%	%

Waiting Operation

- It is impossible to set waiting between the process in the same path.

TURRET 1	TURRET 2
*N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
*N30 FINE	N30 TAP
%	%

Waiting Operation

3.5.8 Release of Waiting

- Function
  - Release the waiting between the process.
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

    Mxxx (Pxx) will be deleted from one or both of these blocks.
- Basic Operation

1. Press [CLWAIT]. (Cursor position pretermission)

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

The software key array changes into release waiting mode.

2. Move cursor to the cell that have waiting.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

Press any one of [CLTOP], [CLEND], and [CLBOTH].

3. Press [CL-].

- When [CLTOP] is pressed

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

(The opposite waiting will also be released.)

- When [CLEND] is pressed

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

(The opposite waiting will also be released.)

- When [CLBOTH] is pressed

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

(The opposite waiting will also be released.)

4. Press [RETURN], and release waiting mode.

- Others
  - The transfer cannot be operated by release waiting.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

Release
Waiting
Operation

3.5.9 Assign of Transfer

- Function
  - Set the transfer between the process.
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    Q0 Mxxx (Pxx) will be set to above block,

    End process block : G1993

    Mxxx (Pxx) will be set to above block.
- Basic Operation

1. Press [STTRNS]. (Cursor position pretermission)

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 FINE	N20 TAP
%	%

The software key array changes into set waiting mode.

2. Move cursor to the source transfer, and press [SELECT].

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
*N20 FINE	N20 TAP
%	%

Add "*" to the top of the selected cell.

3. Move cursor to the destination transfer, and press [SELECT].

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
*N20 FINE	N20 TAP
%	%

Add "*" to the top of the selected cell.
Moreover background color of the source waiting cell become green.
Press either [STTRNS] or [CANCEL].

4. Press [STTRNS].

TURRET 1	TURRET 2
N10 ROUGH	
N20 FINE	N10 DRILL
%	N20 TAP
	%

(When finished normally, selected state will be released automatically.)

5. Press [RETURN], and release set transfer mode.

- Others
  - There is some cell that cannot be specified as source and destination transfer.

TURRET 1	TURRET 2
N10 ROUGH	
N20 MIDDLE	N10 DRILL
N30 FINE	N20 TAP
%	%

← Input impropriety cell

← % cell

  - When the waiting or transfer has already been set, the transfer cannot be set.

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
*N20 WAIT	N20 WAIT
N30 FINE	N30 TAP
%	%

Transfer Operation

  - In case of shortage of waiting M code, the waiting cannot be set.
  - It is possible to set the transfer just between the cells that have no waiting.
  - It is impossible to set waiting across other waiting and transfer.

TURRET 1	TURRET 2
N10 ROUGH	*N10 DRILL
N20 TRANS	N20 TRANS
*N30 FINE	N30 TAP
%	%

Waiting Operation

  - It is impossible to set transfer between the process in the same path.

TURRET 1	TURRET 2
*N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
*N30 FINE	N30 TAP
%	%

Transfer Operation

3.5.10 Release Transfer

- Function
  - Release the transfer between the process.
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    Q0 Mxxx (Pxx) will be deleted from above block.

    End process block : G1993

    Mxxx (Pxx) will be deleted from above block.

- Basic Operation

1. Press [CLTRNS]. (Cursor position pretermission)

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

The software key array changes into release transfer mode.

2. Move cursor to the cell that have transfer.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

Press either [CLTRNS] or [CANCEL].

3. Press [CLTRANS].

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

(The opposite transfer will also be released.)

4. Press [RETURN], and release transfer mode.

- Others
  - The waiting cannot be operated by release transfer.

TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL
N20 TRANS	N20 TRANS
N30 FINE	N30 TAP
%	%

Release
Transfer
Operation

3.5.11 Addition of Optional Block Skip for Each Path Program Check

- Function
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

    Add any one of "/7", "/8", and "9" to the top of each block between above two blocks.

    /7 : process belong to spindle-1. (except transfer process)

    /8 : process belong to spindle-2. (except transfer process)

    /9 : transfer process

- Basic Operation

  Press [ADD /].

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL		
N20 TRANS	N20 TRANS		
		N30 FINE	N30 TAP
%	%	%	%

Adding of optional block skip

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL		
N20 TRANS	N20 TRANS		
		N30 FINE	N30 TAP
%	%	%	%

- Others
  - Additional processing is not done for the block which has G1992 and G1993.

```
G1992 S1;
T0101;
G00 X0. Z0.;
M01;
G1993;
```

→

```
G1992 S1;
/7 T0101;
/7 G00 X0. Z0.;
/7 M01;
G1993;
```

- If "/" exist, "/" will be converted to "/1" in the additional processing of optional block skip (/7, /8, /9)

```
G1992 S1;
/ T0101;
/ G00 X0. Z0.;
/ M01;
G1993;
```

→

```
G1992 S1;
/7 /1 T0101;
/7 /1 G00 X0. Z0.;
/7 /1 M01;
G1993;
```

- If any one of "/7", "/8", and "/9" has already exists at the top of the target block, exchange it instead of addting.

```
G1992 S1;
/8 T0101;
/8 G00 X0. Z0.;
/8 M01;
G1993;
```

→

```
G1992 S1;
/7 T0101;
/7 G00 X0. Z0.;
/7 M01;
G1993;
```

3.5.12 Deletion of Optional Block Skip for Each Path Program Check

- Function
  - In NC program,

    Start process block : G1992 Sx (xxxx)

    End process block : G1993

    Delete “/7”, “/8”, and “/9” at the top of each block between above two blocks.

- Basic Operation

  1. Press [DEL /].

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL		
N20 TRANS	N20 TRANS		
		N30 FINE	N30 TAP
%	%	%	%

Deleting of optional block skip

SPINDLE 1		SPINDLE 2	
TURRET 1	TURRET 2	TURRET 1	TURRET 2
N10 ROUGH	N10 DRILL		
N20 TRANS	N20 TRANS		
		N30 FINE	N30 TAP
%	%	%	%

- Others
  - Deletion processing is not done for the block which has G1992 and G1993.

```
G1992 S1;
/7 T0101;
/7 G00 X0. Z0.;
/7 M01;
G1993;
```

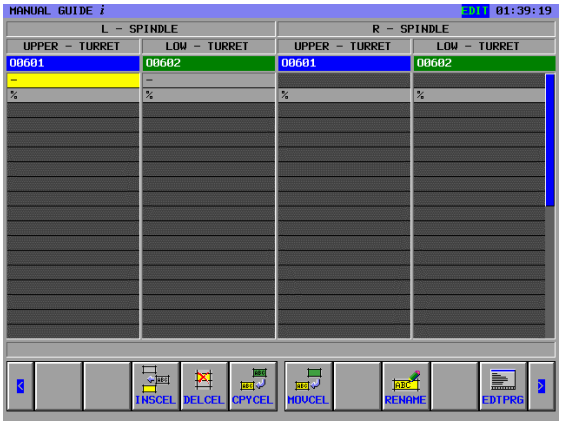
→

```
G1992 S1;
T0101;
G00 X0. Z0.;
M01;
G1993;
```

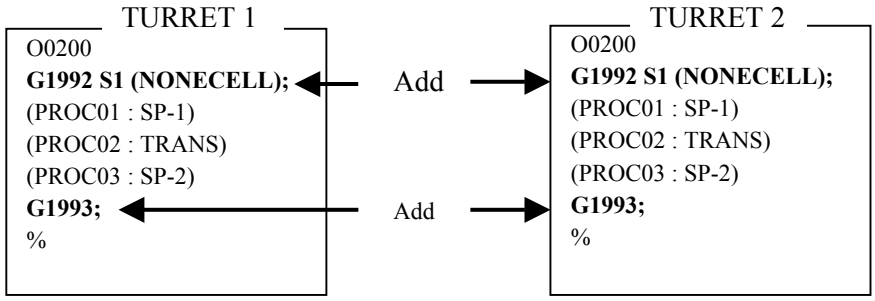
3.6 DEALING OF THE PART PROGRAM WITH UNFITTED TO PROCESS LIST FORM

1. The following screen will appear when NC program that is not fitted to process list edit function is opened.

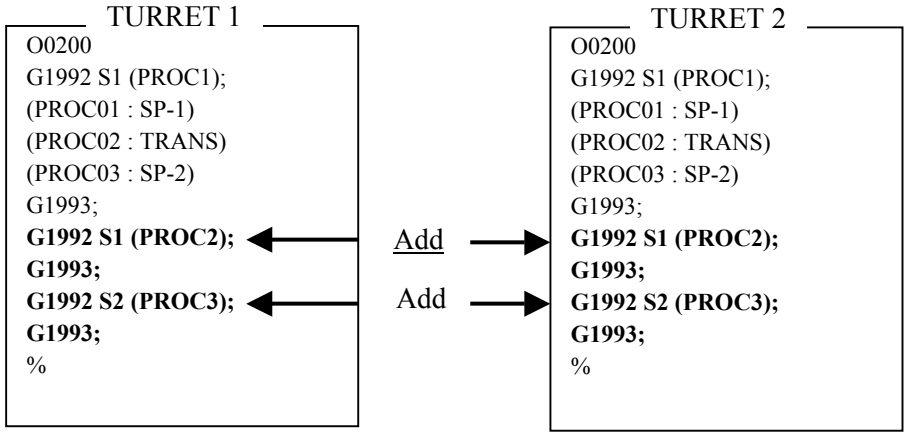
Press [YES], and process list edit function screen will appear.



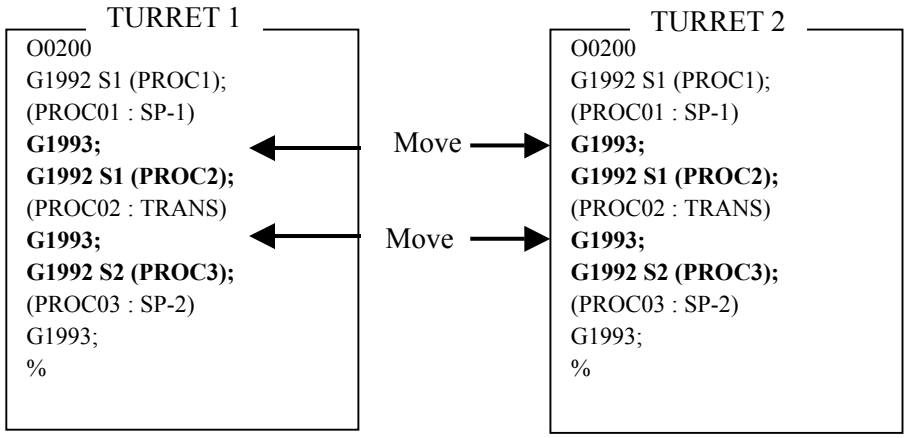
NC program will be modified as follows.



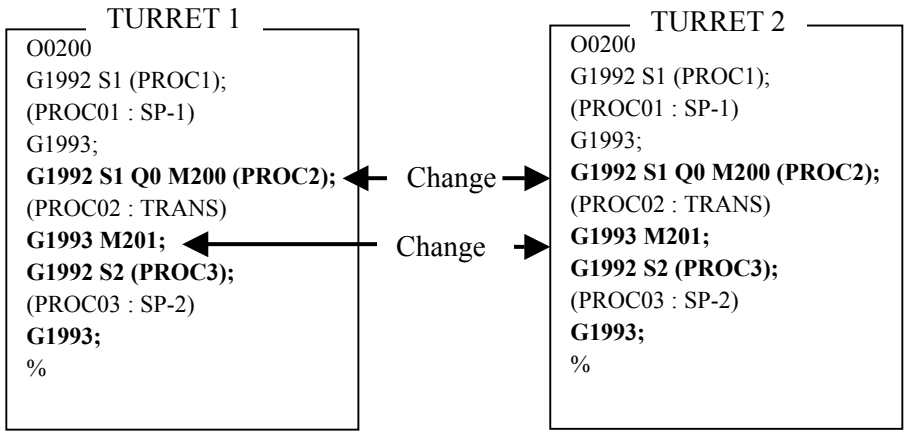
2. And then, add (sum of all process – 1) piece cell by inserting cell operation.



3. Moreover press [EDTPRG], move the G1993 and G1992 blocks between of processes using cut and paste in the normal edit screen.



4. Press [EDWORK] again, and set transfer.



5. Finish transition work.

3.7 FORMAT

Start Process : G1992
Sx : Select spindle
S1 : Spindle-1, S2 : Spindle-2
Qx : Attribute
Q0 : Transfer
Mx : Waiting M code
NC parameter from 8110 to 8111
Px : Waiting partner
Combination of existing path number

Finish process : G1993
Mx : waiting M code
NC parameter from 8110 to 8111
Px : waiting partner
Combination of existing path number

3.8 OTHERS

- Waiting M code is recognized just in the same block for the process beginning word and the process end word.
- This function is not supported in the background mode.
  (This is because a program to edit must be separately selected for each path.)
- As to moving cell and copying cell, the cell content is operated as it is.
- “/7”, “/8”, and “/9” in the combination of “/” and figure for optional block skip are reserved for “add /” function and “DEL /” function. Therefore those word should not be used in the user program freely.
- A 1-path lathe does not support the "set waiting", "cancel waiting", "set transfer", "cancel transfer", "add/", and "DEL/" functions.
- A 1-path lathe does not display the path name.
- A 1-path and 1-spindle lathe does not display the spindle name.

V. TOOL MANAGEMENT FUNCTION (FOR Series 16*i*/18*i*/21*i* ONLY)

1 ASSOCIATING TOOL NUMBERS WITH OFFSET NUMBERS

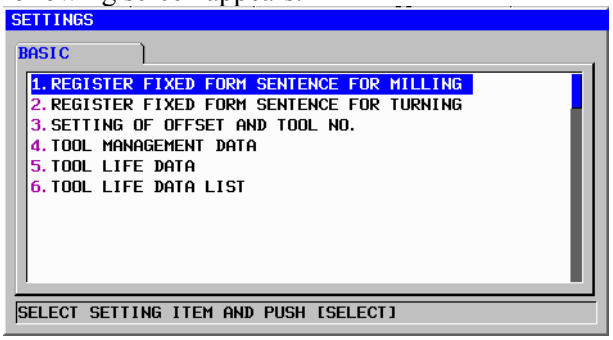
NOTE
1 To use tool management functions with MANUAL GUIDE *i*, you require tool management function options.
For details, refer to the manual issued by the machine tool builder.
2 The tool management function of MANUAL GUIDE *i* is not supported for the Series 30 *i*.
3 This function cannot be used with the MANUAL GUIDE *i* simulator for the personal computer.

On the screen for associating a tool number with a offset number, set the number of the tool that will use a offset number. When registered on this screen, the tool number is registered in the tool management data table, together with the offset number and the tool type.
This screen is effective only if bit 0 (ORT) of parameter No. 14823 is 1.

1.1 SELECTING THE SCREEN FOR ASSOCIATING A TOOL NUMBER WITH A OFFSET NUMBER

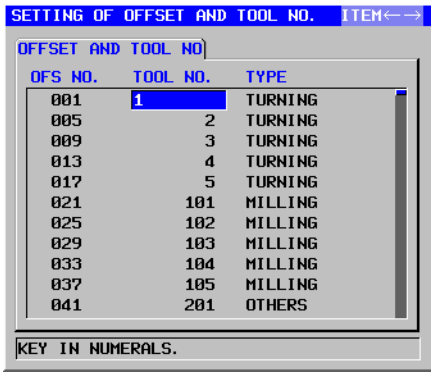
<1> Press [>] on the initial screen of each mode to display the soft keys shown below, then press [SETING]:

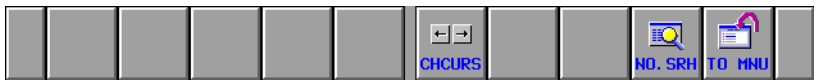
<2> The following screen appears.



<3> From this screen, select “SETTING OF OFFSET NO. AND TOOL NO.”, and the screen for associating a tool number with a tool offset number appears.

1.2 SCREEN DISPLAY ITEMS





Display items

OFS NO.:

You can only view offset numbers, and cannot set new ones.
The range of available offset numbers depends on the setting of parameter No. 14824.

TOOL NO:

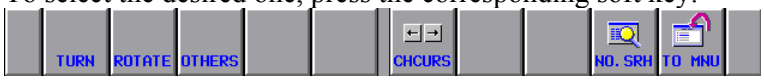
To register a new tool number in an empty tool number field, enter a new number and press the **INPUT** key.
To invalidate an existing tool number, enter 0.

NOTE

When a tool number is entered, the associated offset number is set in the “offset number” item of the data having that tool number in the tool management data table.
If the tool management data table does not contain data having that tool number, the system searches for tool management data having no tool number, and the offset number associated with the entered tool number is set in that tool management data.
If a tool number is deleted (0 is entered), all of the tool management data having that tool number is deleted.

TYPE:

To select the desired one, press the corresponding soft key.



Soft keys

[TO MNU]:
Return to the menu screen.

[CHCURS]:
Switches the system between cursor modes.

1.3 DISABLE WARNING MESSAGE

WRONG VALUE OF PARAMETER NO. 14824 :
Displayed if the value of parameter No. l4824 is outside the range of 1 to 999 and “SETTING OF OFFSET NO. AND TOOL NO.” is selected from the [SETING] menu.

INVALID INPUT :
Displayed if a tool number outside the valid range is entered.

TOOL NUMBER ALREADY EXISTS :
Displayed if the same tool number as that entered has already been set.

TOOL MANAGEMENT DATA ACCESS ERROR :
Displayed if the system fails to read or write tool and offset numbers.

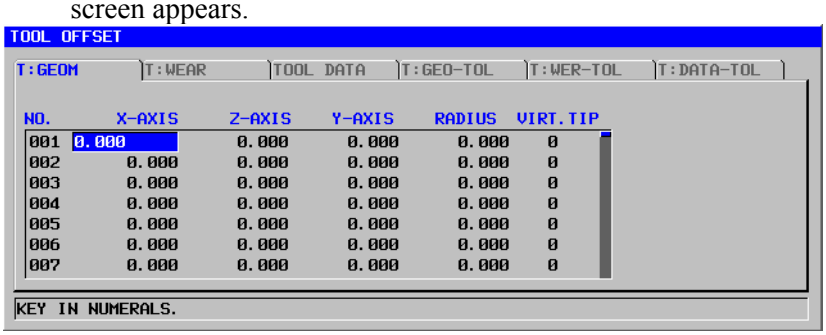
2 VIEWING AND SETTING TOOL OFFSET VALUES

In addition to the conventional tool offset setting screen, a screen is available which allows you to view and set tool offset values using tool numbers and offset types.
This screen is effective only if bit 1 (TOD) of parameter No. 14823 is 1.

2.1 SELECTING THE TOOL NUMBER-BY-TOOL NUMBER TOOL OFFSET SETTING SCREEN

<1> Press [>] on the initial screen of each mode to display the soft keys shown below:

<2> From this screen, press [T-OFS], and the tool offset setting screen appears.



<3> Moving the cursor to the “T:GEO-TOL” tab causes the “Turning geometric offset screen (on a tool number by tool number basis)” to appear.

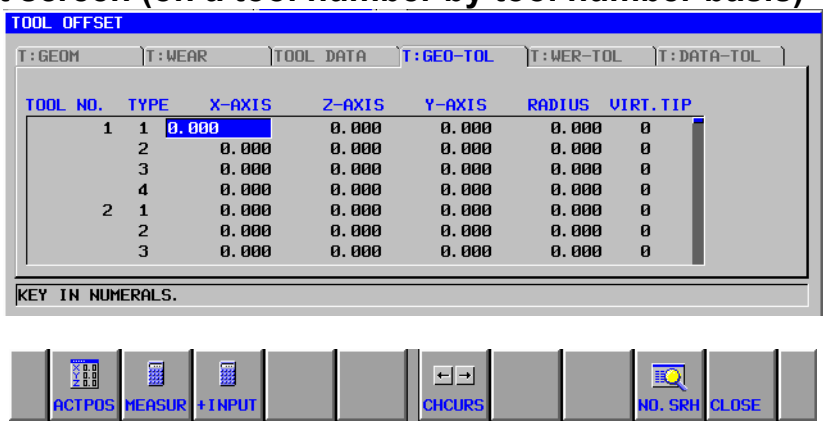
<4> Moving the cursor to the “T:WER-TOL” tab causes the “Turning wear offset screen (on a tool number by tool number basis)” to appear.

<5> Moving the cursor to “M:OFS-TOL” tab causes the “Milling offset screen (on a tool number by tool number basis)” to appear.

NOTE
If the “T:GEOM”, “T:WEAR”, and “M:OFFSET” tabs are selected, their respective conventional “T:Tool geometric offset”, “T:Tool wear offset”, and “M tool wear offset” screens appear.

2.2 SCREEN DISPLAY ITEMS

(1) Turning geometric offset screen (on a tool number by tool number basis)



- Display items

TOOL NO.:

The tool numbers in the tool management data table are displayed.
You cannot set new ones from this screen.

TYPE:

Offset types are displayed.
Values ranging from 1 to the “number of offset types” set in parameter No. 14825 are displayed sequentially.
If the setting of parameter No. 14825 is 0, nothing is displayed in the offset type column.

X-AXIS, Z-AXIS, Y-AXIS, RADIUS, VIRT. TIP:

The offset number corresponding to each combination of tool number and offset type is determined, and the offset values of the offset number are displayed.
The settings are made in the data for the offset number determined with the combination of tool number and offset type.

Valid data range:

Depends on the setting of the tool offset (described later).

Data to be referenced:

Tool offset data

- Soft keys

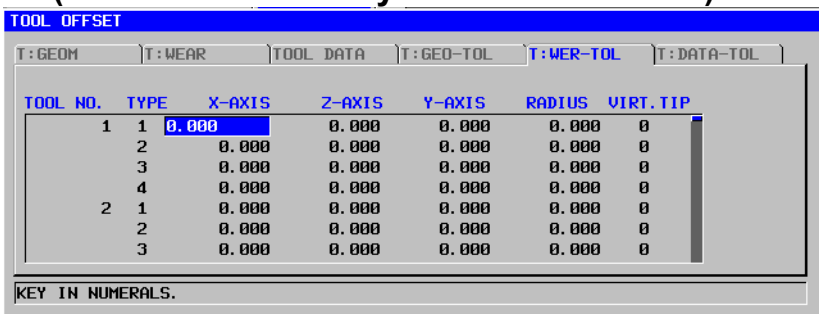
[TO MNU]:

Return to the menu screen.

[CHCURS]:

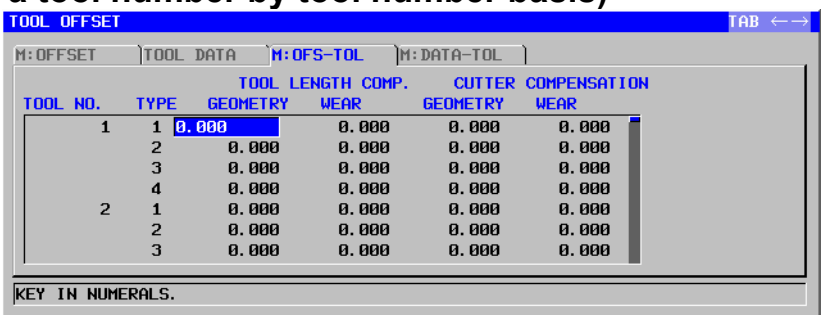
Switches the system between cursor modes.

(2) Turning wear offset screen (on a tool number by tool number basis)



The display items are the same as those on the “Turning geometric offset screen (on a tool number by tool number basis)”.

(3) Milling offset screen (on a tool number by tool number basis)



The display items are the same as those on the “Turning geometric offset screen (on a tool number by tool number basis)”.

2.3 TOOL OFFSET

A value of up to six digits (not including '-' and '.') can be set. For tool offset in T mode, if the “7-digit tool offset input” option is effective, a value of up to seven digits can be set.
The valid number of digits in the fractional part depends on the settings of the NC.

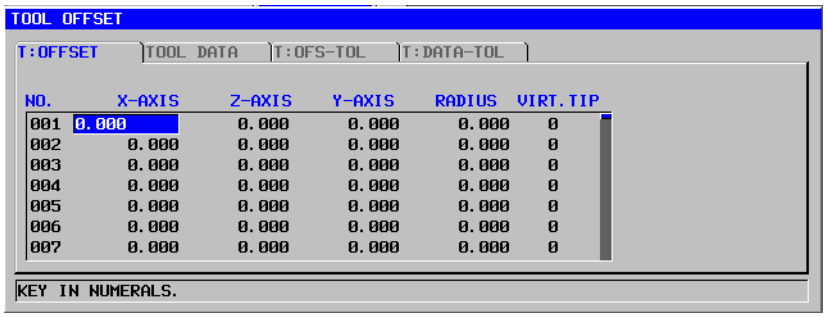
2.4 NOTES

NOTE
If bit 1 (TOF) of parameter No. 14823 is 0, the tool number-by-tool number offset value setting screen does not appear.

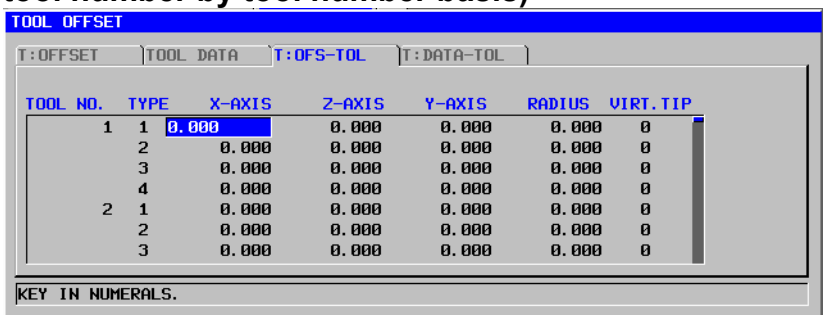
Screens that appear differently depending on whether options are provided

"Tool geometric and wear offset" (lathe systems (standard models and complex machining functions)), "tool offset memory type B", and "tool offset memory type C" (machining systems) are optional functions. If these options are not provided, screens appear as shown below.

- Screen that appears when "Set tool offset" is selected

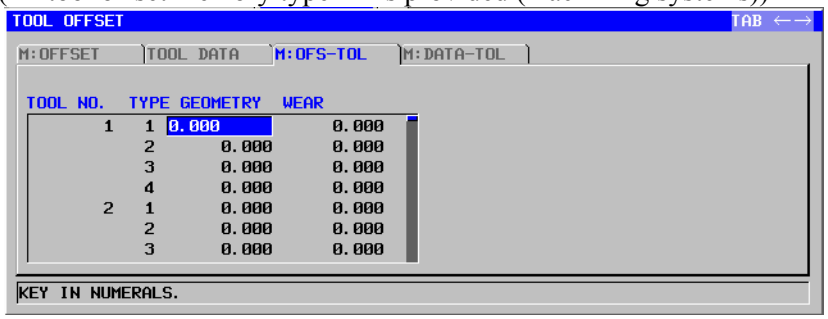


- Turning offset screen (on a tool number by tool number basis)

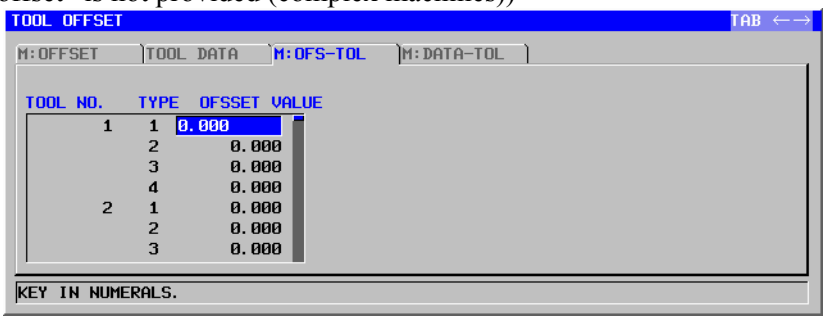


- Milling offset screen (on a tool number by tool number basis)

(If "tool offset memory type B" is provided (machining systems))

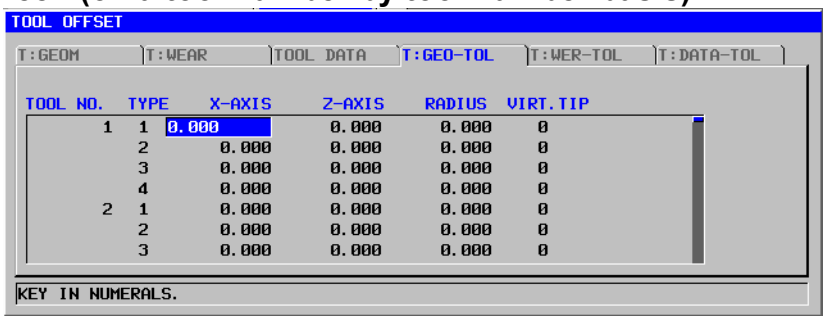


(If "tool offset memory type B" and "tool offset memory type C" are not provided (machining systems) and "tool geometric and wear offset" is not provided (complex machines))

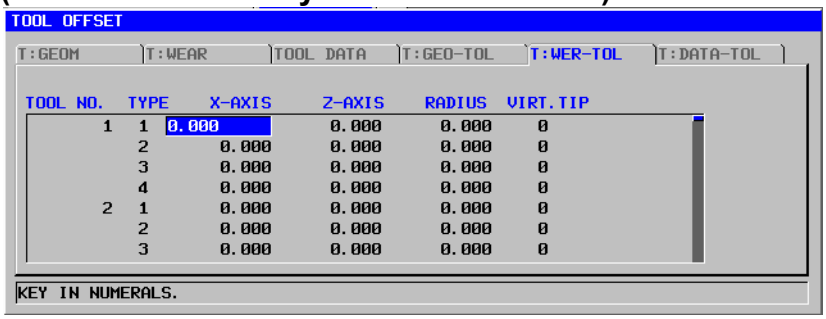


"Y-axis offset" is an optional function. If this option is not provided, screens appear as shown below.

- Turning geometric offset screen (on a tool number by tool number basis)



- Turning wear offset screen (on a tool number by tool number basis)



NOTE
1 On machining center CNCs, the turning tool offset setting screen does not appear.
2 For lathe CNCs (standard models), the milling tool offset setting screen does not appear.

2.5 DISABLE WARNING MESSAGE

WRONG VALUE OF PARAMETER No. 14823 :

Displayed if the value of parameter No. l4823 is outside the range of 1 to 999 and the tool number-by-tool number tool offset setting screen is selected. No data is displayed on the screen.

TOOL MANAGEMENT DATA ACCESS ERROR :

Displayed if the system fails to read or write tool and offset numbers.

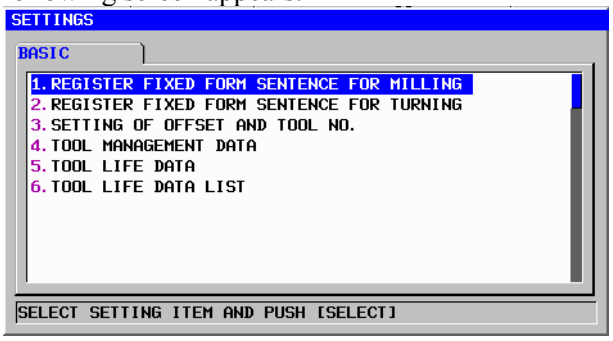
3 VIEWING AND SETTING TOOL MANAGEMENT DATA

This screen is effective only if bit 3 (TMG) of parameter No. 14823 is 1.

3.1 SELECTING THE TOOL MANAGEMENT DATA SETTING SCREEN

<1> Press [>] on the initial screen of each mode to display the soft keys shown below, then press [SETING]:

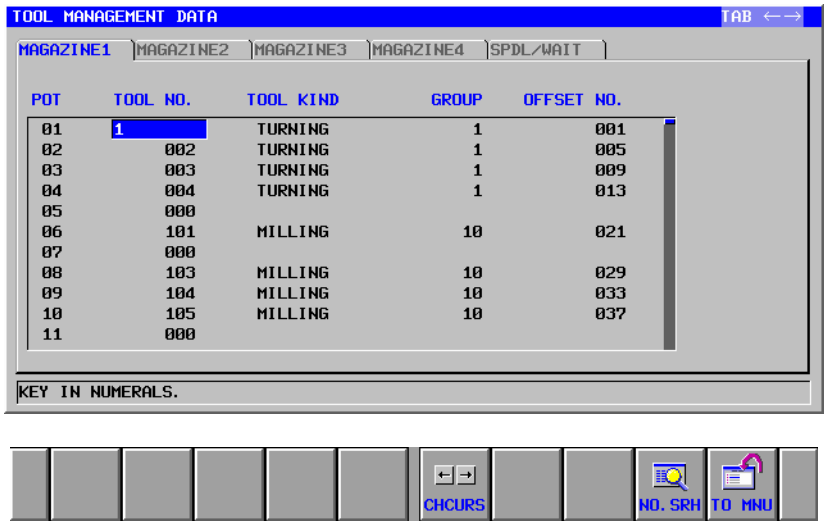
<2> The following screen appears.



<3> From this screen, select “TOOL MANAGEMENT DATA”, and the tool management data setting screen appears.

3.2 MAGAZINE DATA SCREENS (MAGAZINE 1 TO 4)

3.2.1 Screen Display Items



The tool number, type, group number, and offset number corresponding to each pot are displayed.
You can change tool numbers, types, and group numbers.

Display items

POT:

Pot numbers are displayed.
You cannot set new ones from the screen.

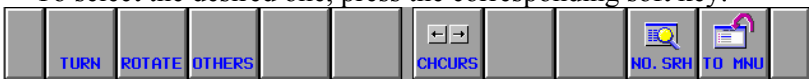
NOTE
On the individual magazine data screens, as many pots as the “number of data items” set in parameters Nos. 13222, 13227, 13232, and 13237 are displayed, beginning with the “start pot number” set in NC parameters Nos. 13223, 13228, 13233, and 13238.

TOOL NO.:

To register a new tool number in an empty tool number field, enter a new number and press the [INPUT] key.
To invalidate an existing tool number, enter 0.

TOOL KIND:

The “tool type” corresponding to each tool number, as determined from the tool management data table, is displayed.
To select the desired one, press the corresponding soft key.



GROUP:
The "group number" corresponding to each tool number, as determined from the tool management data table, is displayed.
To set a new one, enter a value.

OFFSET NO.:
The "offset number" corresponding to each tool number, as determined from the tool management data table, is displayed.
You cannot change offset numbers from this screen.

Soft keys

[TO MNU]:
Return to the menu screen.

[CHCURS]:
Switches the system between cursor modes.

3.2.2 Displayed Warning Messages

MAGAZINE MANAGEMENT DATA ACCESS ERROR :
Displayed if the system fails to normally read or write the data corresponding to the pot numbers in the magazine management data table.

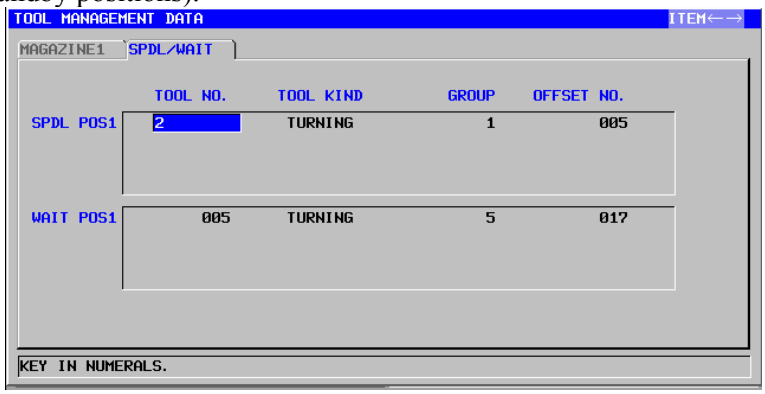
TOOL MANAGEMENT DATA ACCESS ERROR :
Displayed if the system fails to read or write tool management data such as tool numbers, types, and group numbers.

INVALID INPUT :
Displayed if the entered value is outside the valid range.

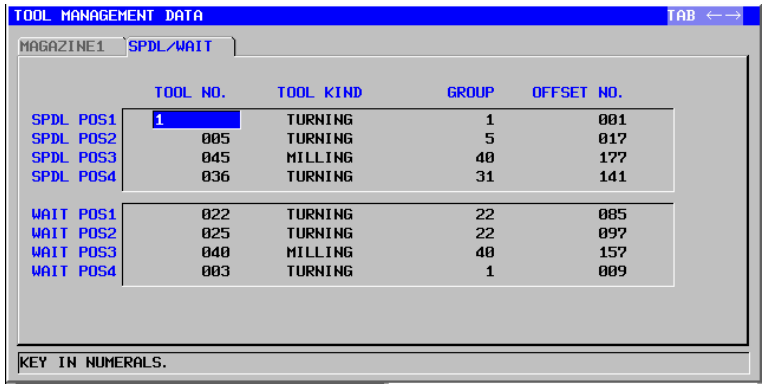
3.3 SPINDLE AND STANDBY POSITION TOOL DISPLAY SCREEN

3.3.1 Screen Display Items

This screen displays the tools at spindle positions and at subpots (standby positions).



The number of displayed spindle positions and the number of displayed standby positions vary depending on the settings of
Parameter No. 13250 (number of effective spindles)
Parameter No. 13251 (number of effective standby positions)
If the setting of parameter No. 13250 is 4 (maximum) and that of parameter No. 13251 is 4 (maximum), the screen appears as shown below.

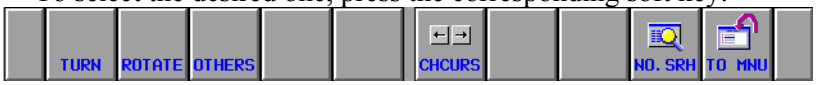


Display items

TOOL NO.:

To change the tool number at a spindle or standby position, move the cursor to that number, enter a new value, and press [INPUT]. To invalidate an existing tool number, enter 0.

TOOL KIND:
The “tool type” corresponding to each tool number, as determined from the tool management data table, is displayed.
To select the desired one, press the corresponding soft key.



GROUP:
The “group number” corresponding to each tool number, as determined from the tool management data table, is displayed.
To set a new one, enter a value.

OFFSET NO.:
The “offset number” corresponding to each tool number, as determined from the tool management data table, is displayed.
You cannot change offset numbers from this screen.

Explanation of soft keys

[TO MNU]:
Return to the menu screen.

[CHCURS]:
Switches the system between cursor modes.

3.3.2 Displayed Warning Messages

MAGAZINE MANAGEMENT DATA ACCESS ERROR :
Displayed if the system fails to normally read or write spindle or standby position data from the magazine management data table.

TOOL MANAGEMENT DATA ACCESS ERROR :
Displayed if the system fails to read or write tool management data such as tool numbers, types, and group numbers.

INVALID INPUT :
Displayed if the entered tool number is outside the valid range.

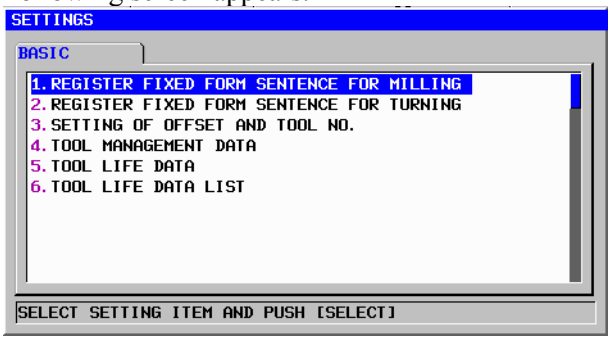
4 VIEWING AND SETTING LIFE MANAGEMENT DATA

This screen is effective only if bit 4 (TLF) of parameter No. 14823 is 1.

4.1 SELECTING THE LIFE MANAGEMENT DATA SETTING SCREEN

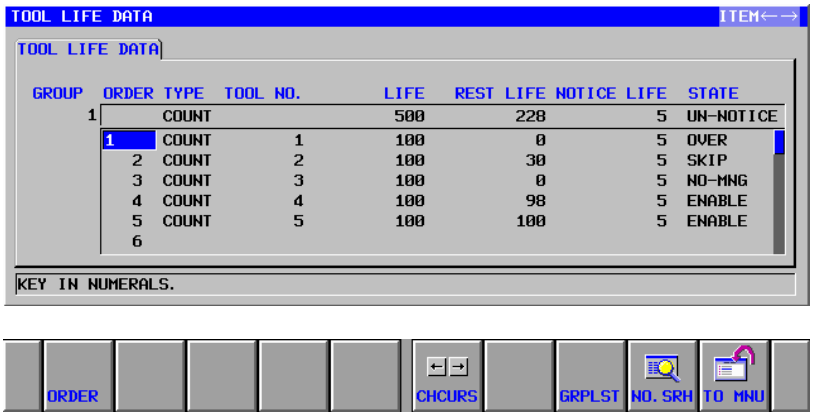
<1> Press [>] on the initial screen of each mode to display the soft keys shown below, then press [SETING]:

<2> The following screen appears.



<3> From this screen, select “TOOL LIFE DATA”, and the life management data setting screen appears.

4.2 SCREEN DISPLAY ITEMS



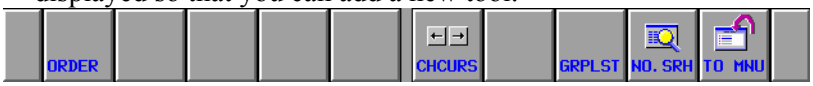
Display items

ORDER:

In the first column for each tool, the value indicating the priority of the tool is displayed.

By positioning the cursor on this item and entering a new value, you can change the priority of that tool (described in detail later).

As many values as the number of tools in the group plus one are displayed so that you can add a new tool.



GROUP:

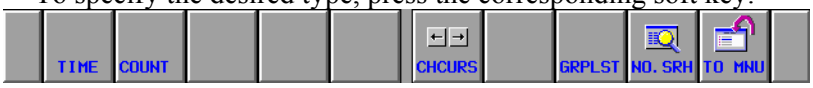
The group numbers in the tool management data table are displayed.

COUNT:

The count types (time or number of times of use) in the tool management data table are displayed.

For each group, the life count type (time or number of times of use) can be specified.

To specify the desired type, press the corresponding soft key.



TOOL NO.:

The tool numbers with the same group number are displayed.

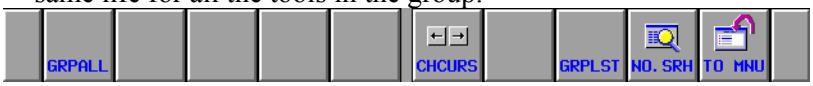
The numbers are displayed in the order in which the tools will be used.

You can register a tool number with the group.

To register a tool, move the cursor at the bottom (blank portion) of the tool number column for that group and press [REGIST] or press **INPUT**. On the contrary, if you want to delete an existing tool, move the cursor to the number of that tool and press [DELETE].

LIFE:

The life of each tool, as determined from the tool management data table, is displayed.
You can set the life of each tool.
By pressing [GRPALL] after entering a value, you can set the same life for all the tools in the group.



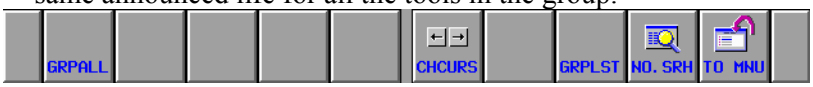
In the first row for each group, the sum of the lives of the tools registered with that group is displayed.

REST LIFE:

The rest of the life of each tool, as determined from the tool management data table, is displayed.
By reconfiguring data, you can increase the rest of the life.
In the first row for each group, the sum of the rests of the lives of the tools registered with that group is displayed.

NOTICE LIFE:

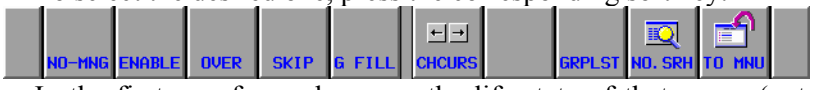
The announced life of each tool, as determined from the tool management data table, is displayed.
You can set the announced life of each tool (rest of life after which an announcement signal is issued).
By pressing [GRPALL] after entering a value, you can set the same announced life for all the tools in the group.



In the first row for each group, the rest of life after which the life state of that group changes to “announced” is displayed.
You can also set the announced life of each group.

STATE:

The state of each tool, as determined from the tool management data table (invalid, present, absent, in use, skipped (tool damaged)), is displayed.
To select the desired one, press the corresponding soft key.



In the first row for each group, the life state of that group (not announced or announced) is displayed.

4.3 CHANGING TOOL PRIORITY

You can change the priority of the tools in a group.

Procedure for changing priority

<1> Position the cursor on the priority value in the first column for the desired tool and enter a new value.

<2> Press [ORDER] or press **INPUT**, and the priority of the tool changes to the new value. Those tools that have priority values greater than the entered new value before the change are assigned their previous values plus one, respectively.

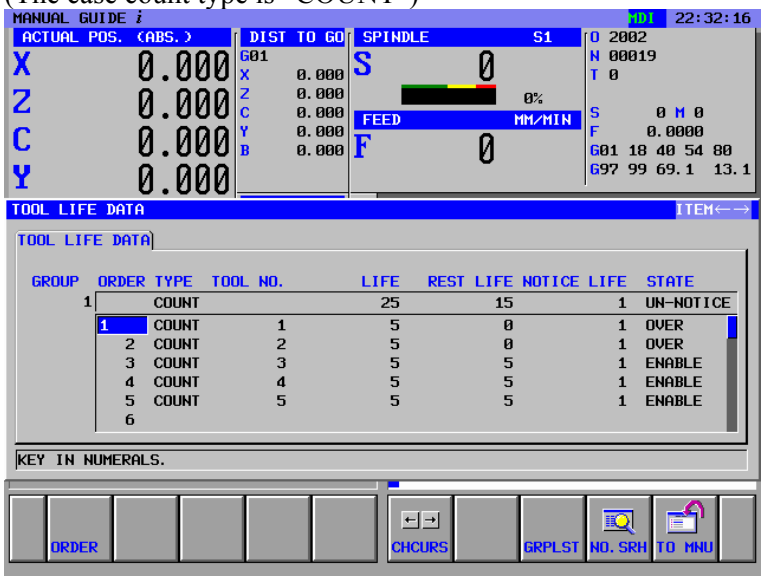
4.4 UPDATING LIFE VALUES DISPLAYED ON THE TOOL LIFE DATA SCREEN

When the tool life data is changed with operating program, the tool life data is updated on the tool life management data screen.

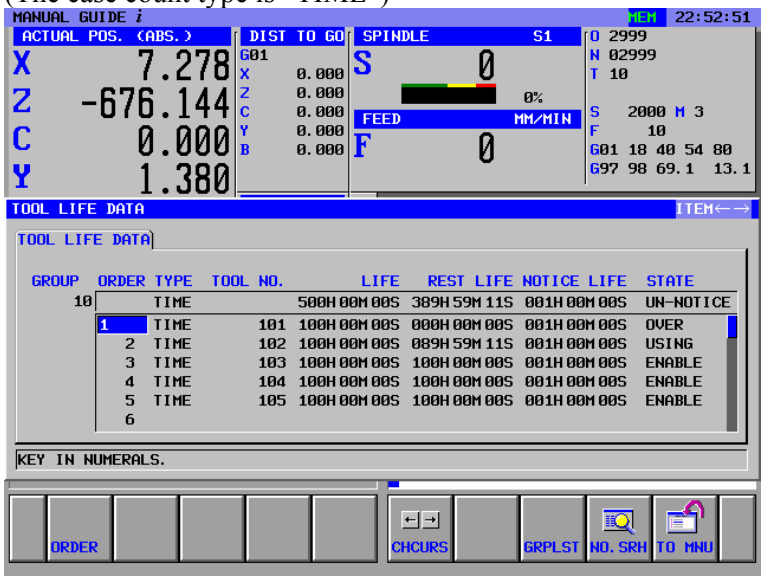
4.4.1 Operation

<1> Press [SETTING].
<2> Select “TOOL LIFE DATA” from the menu screen.
<3> The following screen appears.

(The case count type is “COUNT”)

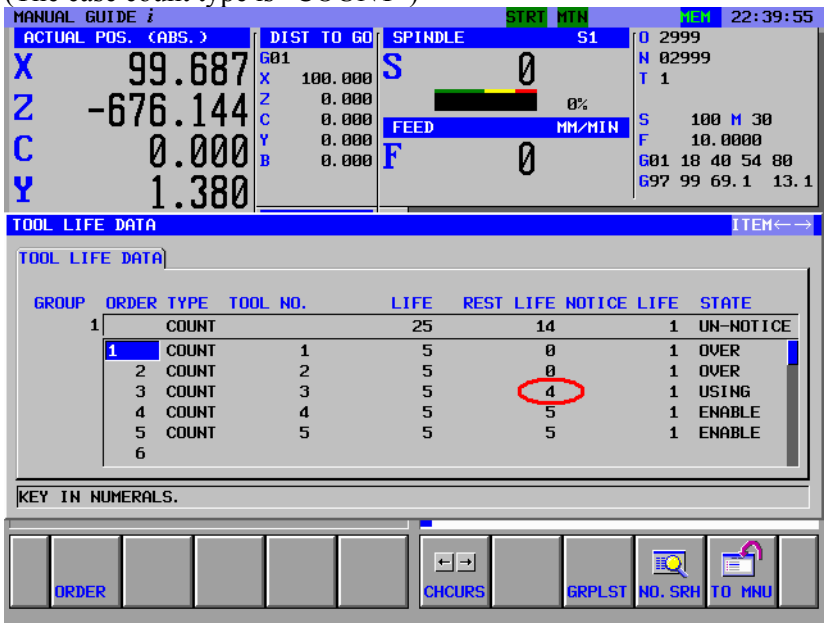


(The case count type is “TIME”)

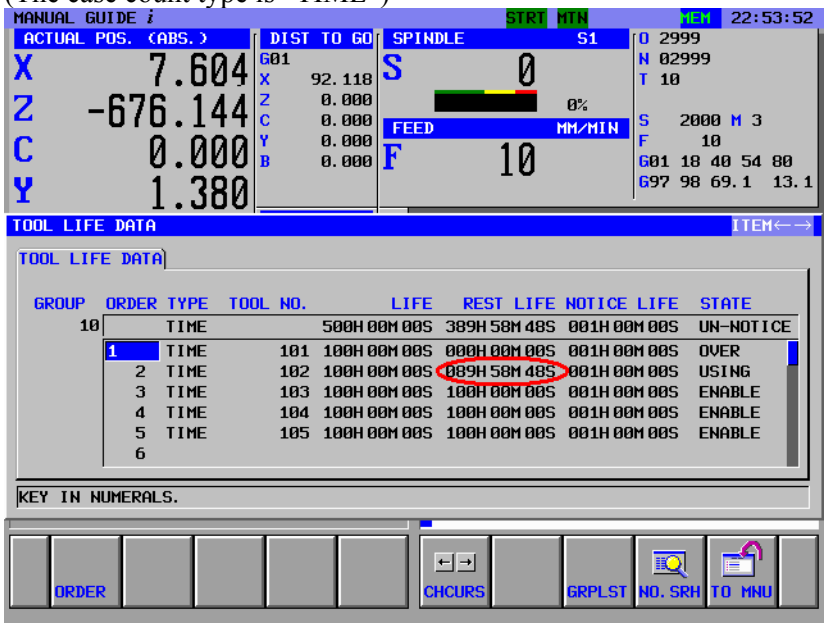


<4> If the tool life data is changed with operating program, the displayed life data is updated.

(The case count type is “COUNT”)



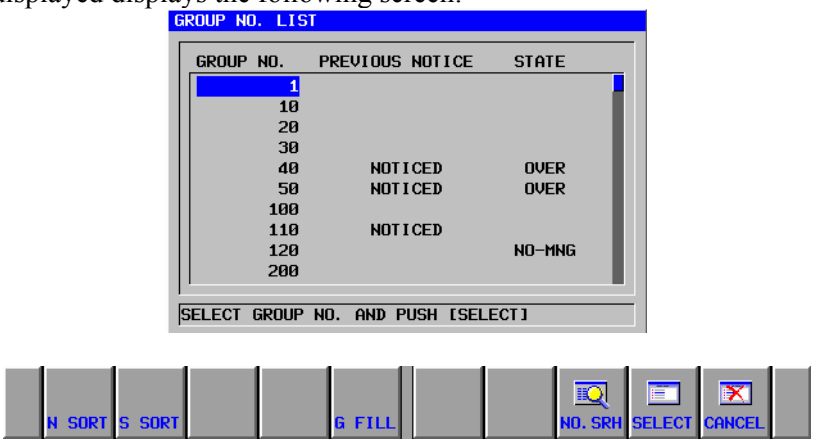
(The case count type is “TIME”)



4.5 GROUP NUMBER LIST DISPLAY

A list of the life states of groups can be displayed. Groups can be sorted in the order of number or life state.

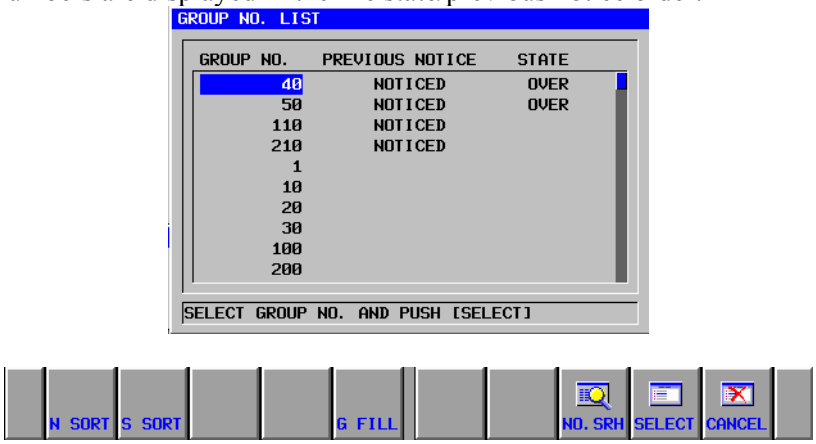
Pressing [GRPLST] when the life management data screen is displayed displays the following screen:



In the "PREVIOUS NOTICE" column, "NOTICED" is displayed only for those groups with the Previous Notice Flag set. In the "STATE" column, "OVER" is displayed for a group when the life states of all tools belonging to the group are "OVER", "SKIP", and/or "NO-NMG".

Display of groups sorted in the order of number or life state

When [S SORT] is pressed on the group number list screen, the group numbers are displayed in the life state/previous notice order.



NOTE

When group numbers are displayed in the order of life state, the groups are displayed in the following order:

<1> Groups for which "OVER" is displayed in the "STATE" column
<2> Groups for which "NOTICED" is displayed in the "PREVIOUS NOTICE" column
<3> Groups other than <1> and <2>

Pressing [N SORT] displays groups sorted by group number.

Group selection

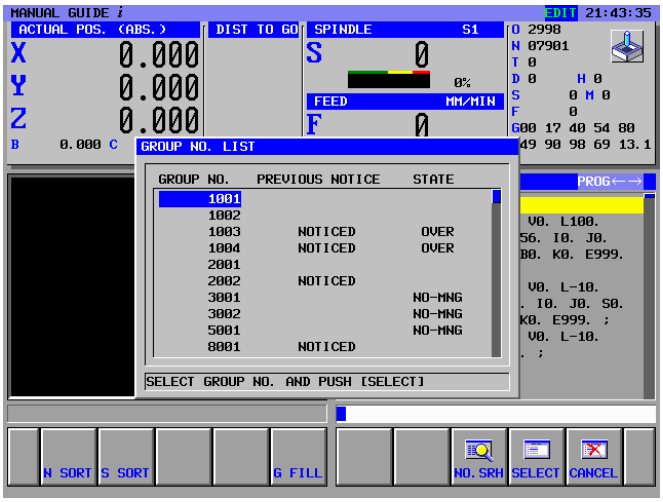
Place the cursor over a group number to be selected, then press [SELECT]. The life management data screen of the selected group number appears.

4.6 DISPLAY OF GROUP NUMBER LIST

On the group number list, the state of the group which is not managed is displayed as “NO-MNG”
The life of the group of which life state is over can be restored on the group number list.

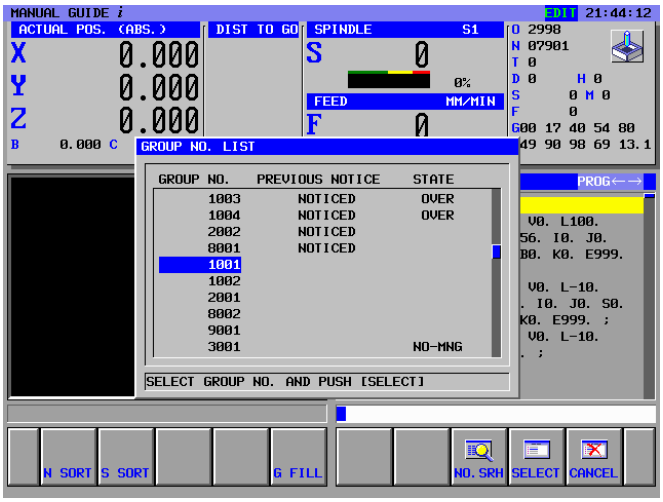
4.6.1 Display Life States of Group

On the tool life data screen, pressing [GRPLST] displays the following screen.

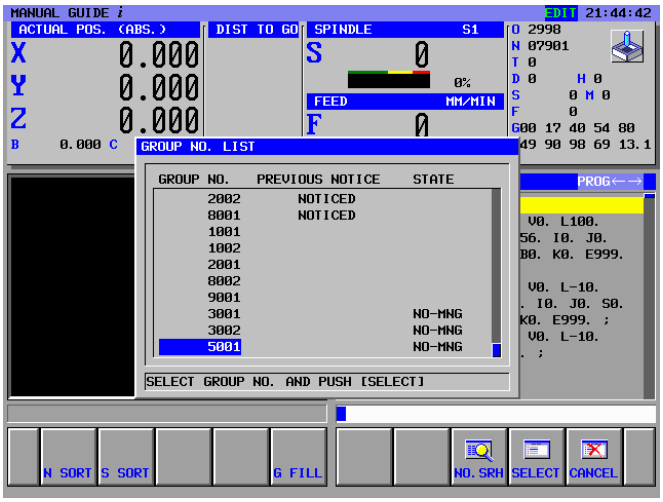


When the states of the all tools, which belong to same group, are not managed, the group life state is displayed as “NO-MNG”.
And the life state of the group, which is not “NO-MNG” and does not include “ENABLE” or “USING” tool, is displayed as “OVER”

Form this screen, pressing [S SORT] displays the following screen.

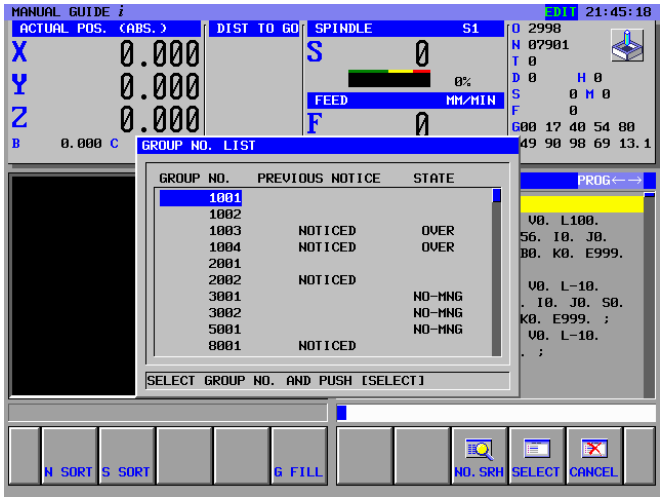


The group which state is no managed is displayed at the bottom of the list as following.

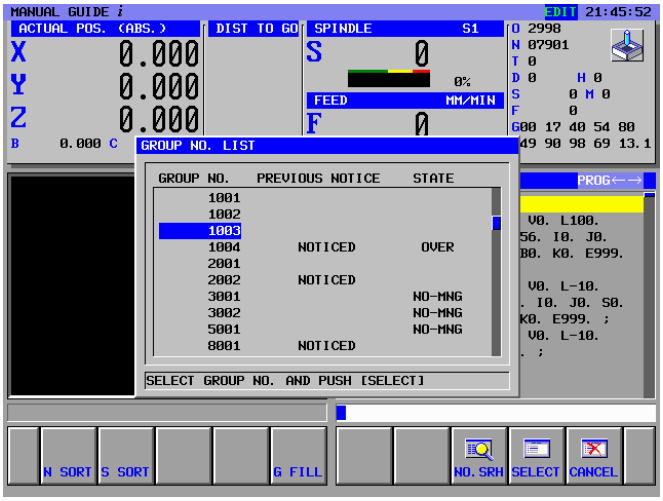


4.6.2 Restore Group Life

On the tool life data screen, pressing [G FILL] displays the following screen.



Move the cursor to the group of which state is “OVER”, and press [G FILL]. And the life states of the tools, which belong to the group, change to “ENABLE”, and the rest life value becomes same as the life value.
And the group notice life state changes “UN-NOTICE”.
The tool life state which is “NO-MNG” or “SKIP” is not updated when [G FILL] is pressed.
When the group life is restored, the group life state is no longer "OVER".



4.7 DISPLAYED WARNING MESSAGES

“TOOL MANAGEMENT DATA ACCESS ERROR”:
Displayed if the system fails to read or write tool management data such as tool numbers and group numbers.

“INVALID INPUT”:
Displayed if the entered value is outside the valid range.

“GROUP LIFE STATE IS NOT 'OVER'.”
When [G FILL] is pressed on the tool life data screen, this warning is displayed if the group state of the current cursor position is not “OVER”.

4.8 SETTING THE LIFE NOTICE FLAG

To display the life state ("NOTICED" or "UN-NOTICE") of a group on the life management data screen, the "Previous Notice Flag" of tool management data needs to be set with the PMC.

The MANUAL GUIDE *i* system displays "NOTICED" as the state of a group when the "Previous Notice Flag" is set to the state described below.

If bit 3 (ETE) of parameter No. 13200 = 0
When the "Previous Notice Flag" of one of the tools that belong to a group is set to "NOTICED"
If bit 3 (ETE) of parameter No. 13200 = 1
When the "Previous Notice Flags" of all tools that belong to a group are set to "NOTICED"

Method of modifying the PMC ladder program

Modify the ladder program so that the tool management data "Previous Notice Flag" of the tool being used is set to 1 ("NOTICED") when the NC outputs a tool life arrival notice signal.
For the "Previous Notice Flag" of tool management data, bit 7 of customization item 0 is used.

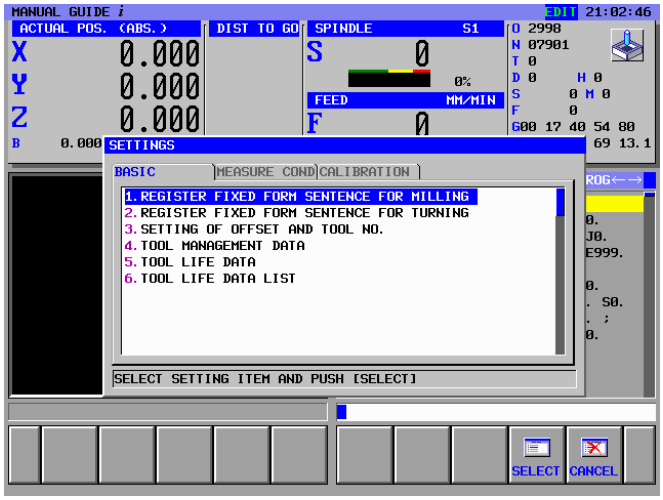
Item	Bit	Meaning	Description of data
Customization item 0	7	Previous Notice Flag	0:UN-NOTICE 1:NOTICED
	6		
	5		
	4		
	3		
	2		
	1		
	0		

5 TOOL LIFE DATA LIST SCREEN

The tool life state of all tools can be displayed on the tool life management data list screen.

5.1 SELECTING THE LIFE MANAGEMENT DATA LIST SCREEN

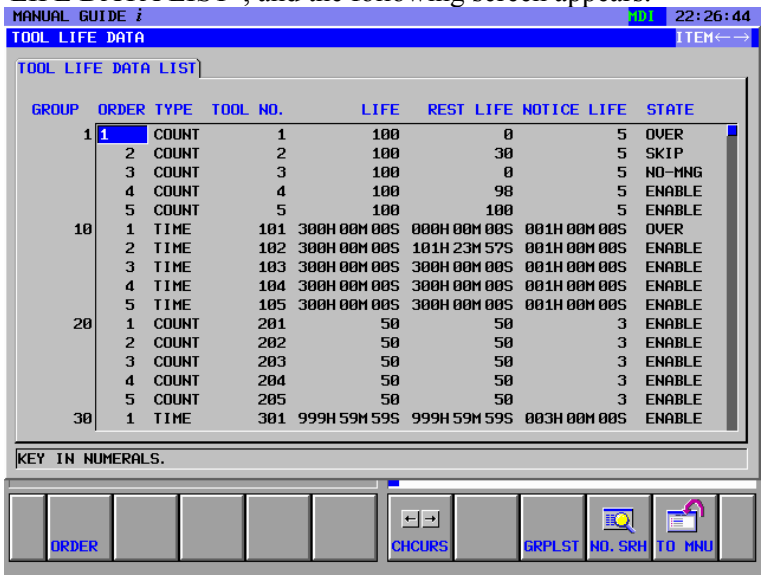
<1> Press [SETTING], and the following screen appears.



This item is displayed when the parameter No.14823#5 is ’1’.

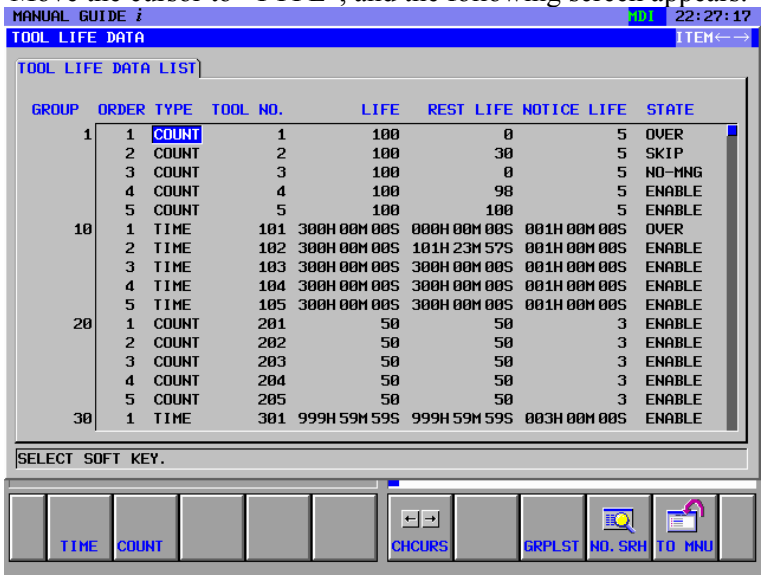
5.2 LIFE MANAGEMENT DATA LIST SCREEN

<1> From "BASIC" tab screen on SETTINGS menu, select “TOOL LIFE DATA LIST”, and the following screen appears.



- Tool life state for all tools are displayed as a list form.
- Group number is displayed at the left end.
- The life state indication of each tool is the same as that on the conventional life management data screen.
- On this screen, you can change the priority of the tools which are belong to the same group with cursor pointed tool. The action performed by pressing each soft key is the same as that on the conventional life management data screen.

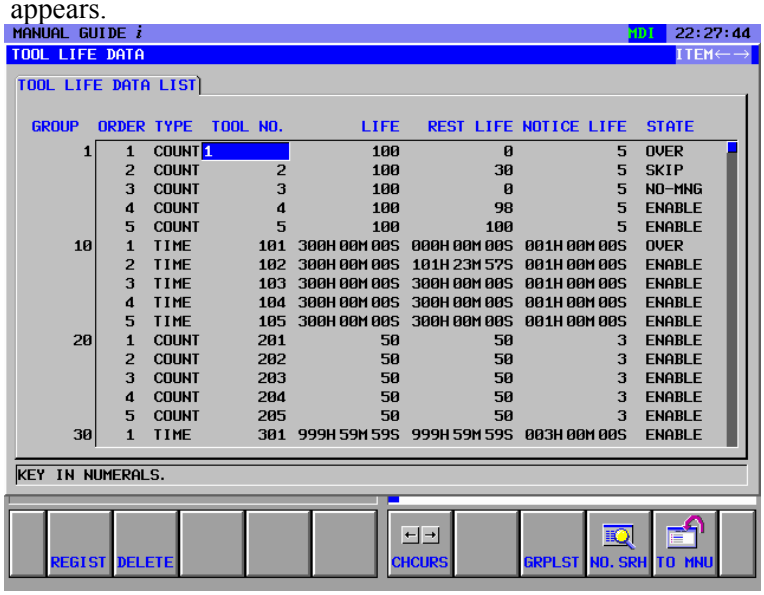
<2> Move the cursor to “TYPE”, and the following screen appears.



- On this screen, you can change the count type of the tools which are belong to the same group with cursor pointed tool.

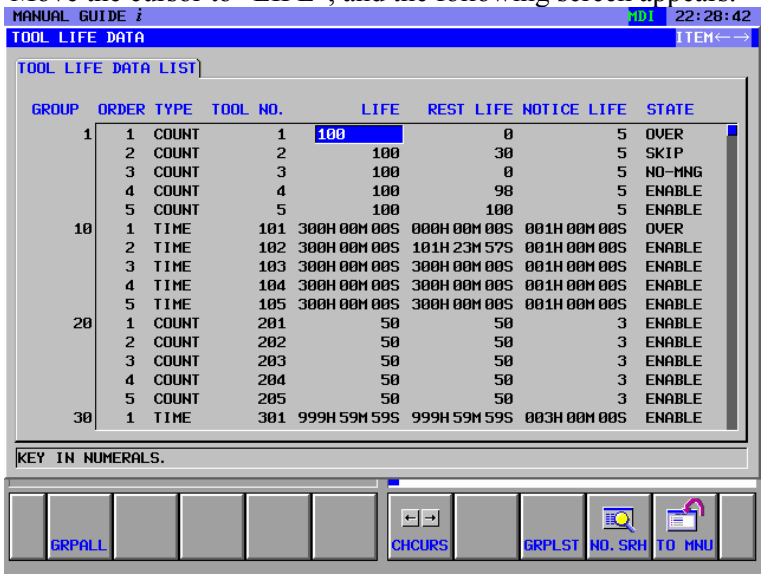
The action performed by pressing each soft key is the same as that on the conventional life management data screen.

<3> Move the cursor to “TOOL NO.”, and the following screen appears.



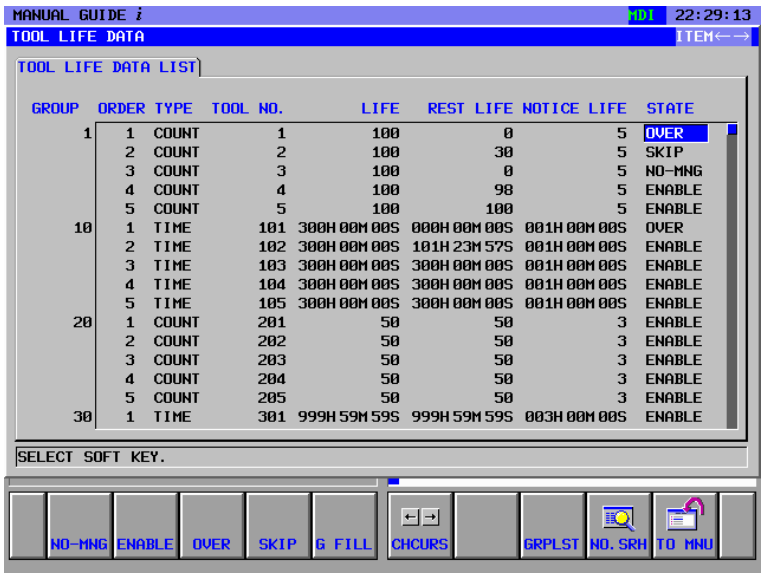
- On this screen, you can change the tool number of which is pointed by the cursor. The action performed by pressing each soft key is the same as that on the conventional life management data screen.

<4> Move the cursor to “LIFE”, and the following screen appears.



- On this screen, you can change the tool life value of which is pointed by the cursor. The action performed by pressing each soft key is the same as that on the conventional life management data screen.

<5> Move the cursor to “STATE”, and the following screen appears.



- On this screen, you can change the tool life state of which is pointed by the cursor. The action performed by pressing each soft key is the same as that on the conventional life management data screen.

<6> Pressing [GRPLST] displays the list of the life states of groups. The data displayed is the same as that on the conventional group number list screen. When a group number is selected in the group number list window, the life management data list screen appears with the cursor placed on the first tool of the selected group.

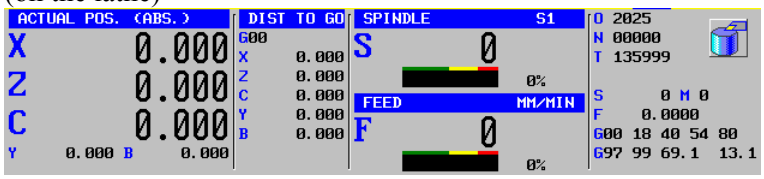
<7> When the tool life data is changed with operating program, the tool life data is updated on the tool life data list screen.

6 MODAL DISPLAY OF OFFSET TYPES

Two tool offset number specification methods are available: the conventional method in which a offset number independent of a tool number is directly specified, and the method in which a offset type associated with a tool number is specified. With the latter, when a offset type is specified, the offset type is displayed at the modal information display position as long as the offset type remains effective.

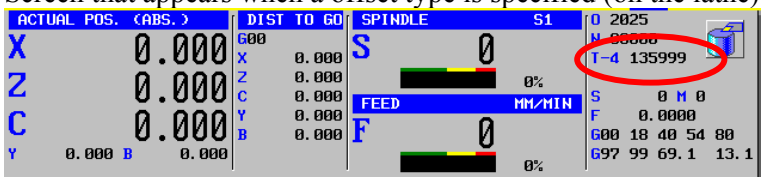
6.1 SCREEN DISPLAY ITEMS

- Screen that appears when a offset number is directly specified (on the lathe)



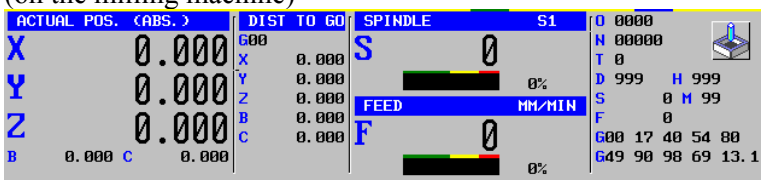
This screen is the same as the conventional one.

- Screen that appears when a offset type is specified (on the lathe)



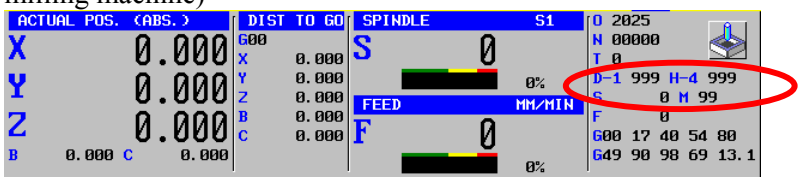
If bit 7 (STS) of parameter No. 14823 is 1 and the offset type is displayed, the offset type is displayed after 'T-' in the status display section.

- Screen that appears when a offset number is directly specified (on the milling machine)



This screen is the same as the conventional one.

- Screen that appears when a offset type is specified (on the milling machine)



If bit 7 (STS) of parameter No. 14823 is 1 and the offset type is displayed, the offset type is displayed after 'D-' and 'H-' in the status display section.

6.2 DISPLAYED OFFSET TYPES (SET BY THE MACHINE TOOL BUILDER)

In the status display section, offset types are displayed by referencing the following variables:

#90248, D code offset type on the milling machine
#90249, offset type on the lathe and H code offset type on the milling machine

When specifying a tool offset number, the machine tool builder is required to set a offset type in variables #90248 and #90249 in the called macro program, using T, D, and H codes.
If directly specifying a offset number, instead of specifying a offset type, the machine tool builder is required to set the above variables to null.

7 DISPLAY TOOL MANAGEMENT DATA OF CNC STANDARD SCREEN

By pressing the soft key displayed on the MANUAL GUIDE *i* screen, it is possible to change the screen to the tool management data table on the NC side.
In order to use this feature, it is necessary to set the TLD(No.14823#6).

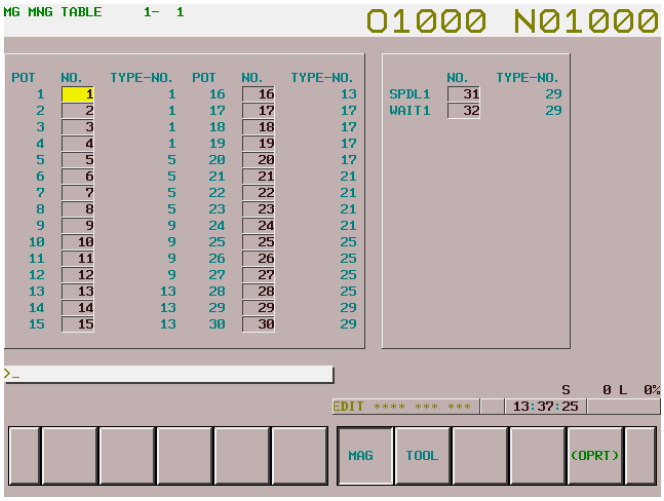
7.1 OPERATION

<1> At the case of the parameter TLD(No.14823#6) setting '1' ,the following [TL-MNG] is displayed on the base screen in the each mode.

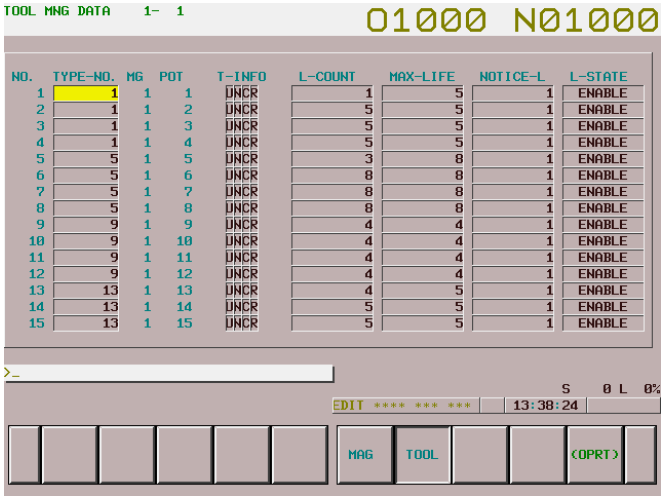
(Example) EDIT mode

<2> Pressing [TL-MNG] displays the following tool management data screen.

(Magazine management table screen)



(Tool management data table screen)



NOTE
Either “Magazine management table screen” or “Tool management data table screen” is displayed. The screen previously displayed is appeared.

<3> On this screen, if the function keys for startup MANUAL GUIDE *i* are pressed, the screen returns to the MANUAL GUIDE *i* base screen.

8 OTHERS

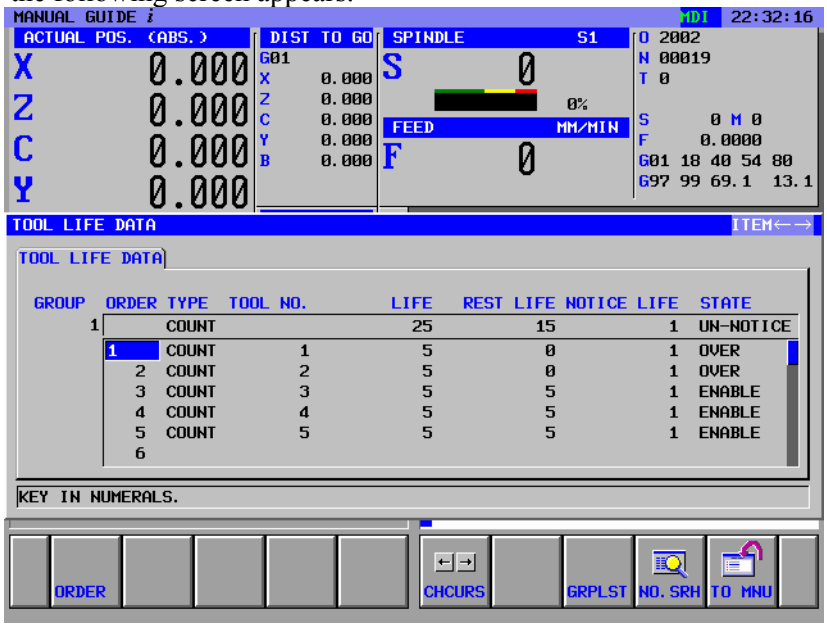
8.1 RETURN TO MENU SCREEN

It is possible to return to the menu screen from tool management screen. And it is possible to return to the base screen as before by parameter setting.

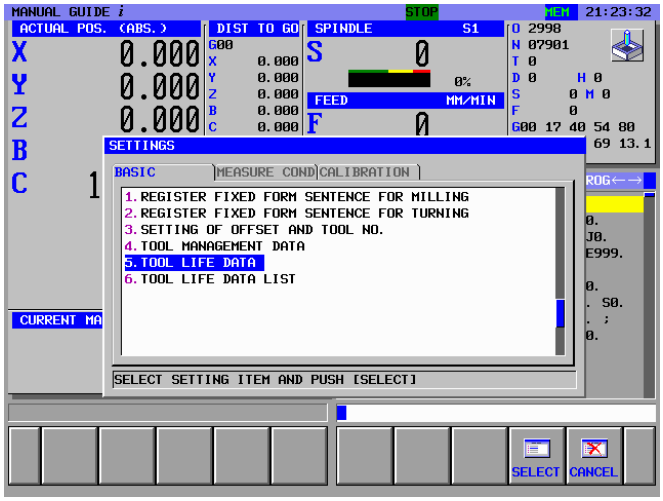
8.1.1 Return to SETTINGS Menu Screen

<1> Press [SETTING]

<2> From “BASIC” menu screen, select “TOOL LIFE DATA”,and the following screen appears.



<3> Pressing [TO MNU] displays the following menu screen. The cursor is displayed on the position of previous selected screen (in this case “TOOL LIFE DATA”)



NOTE
"SETTING OF OFFSET AND TOOL NO.","TOOL MANAGEMENT DATA", and "TOOL LIFE DATA LIST" are the same as "TOOL LIFE DATA"

When the parameter No. 14850#2 is '1', [CLOSE] is displayed instead of [TO MNU]. Pressing [CLOSE] returns to the base screen as before.

8.2 INHIBITION OF EDITING TOOL MANAGEMENT DATA AT CNC STANDARD SCREEN

On the tool management data screen of NC side, it is possible to inhabit to edit the tool management data.

8.2.1 Operations

In the case of the parameter No.14851#7 on, when [EDIT] is pressed on the tool management data screen, the following waning is displayed. And the tool management data cannot be altered at NC screen.

"WRITE PROTECTED"

8.3 USING TOOL MANAGEMENT DATA

The following customization data on the tool management data table are used by the tool management functions for MANUAL GUIDE *i*. Therefore MTB can not use these customization items when the tool management functions for MANUAL GUIDE *i* is used.

Item	Bit	Content
Customization item 0	7	Previous Notice Flag
	6	
	5	
	4	
	3	
	2	
	1	
	0	
Customization item 1		TOOL NO.
Customization item 2		OFFSET NO.
Customization item 3		TOOL KIND
Customization item 4		TOOL USING ORDER

NOTE
In order to enable "Tool Using Order", it is necessary to set the parameter No. 13203#6 to 1 and No.13260 to 4. When these parameter are 0, the shortest life tool is searched not according to this order.

VI. EXAMPLE OF PROGRAMMING OPERATION

1 EXPLANATORY NOTES

> ⚠ **WARNING**
> All data described in this Part such as parameter, offset data and part program cannot be used for actual machining. Actual data varies from one machine model to another. Refer to the applicable manual supplied by the respective machine tool builders for details.
> If the set data does not match the characteristic of a specific machine, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

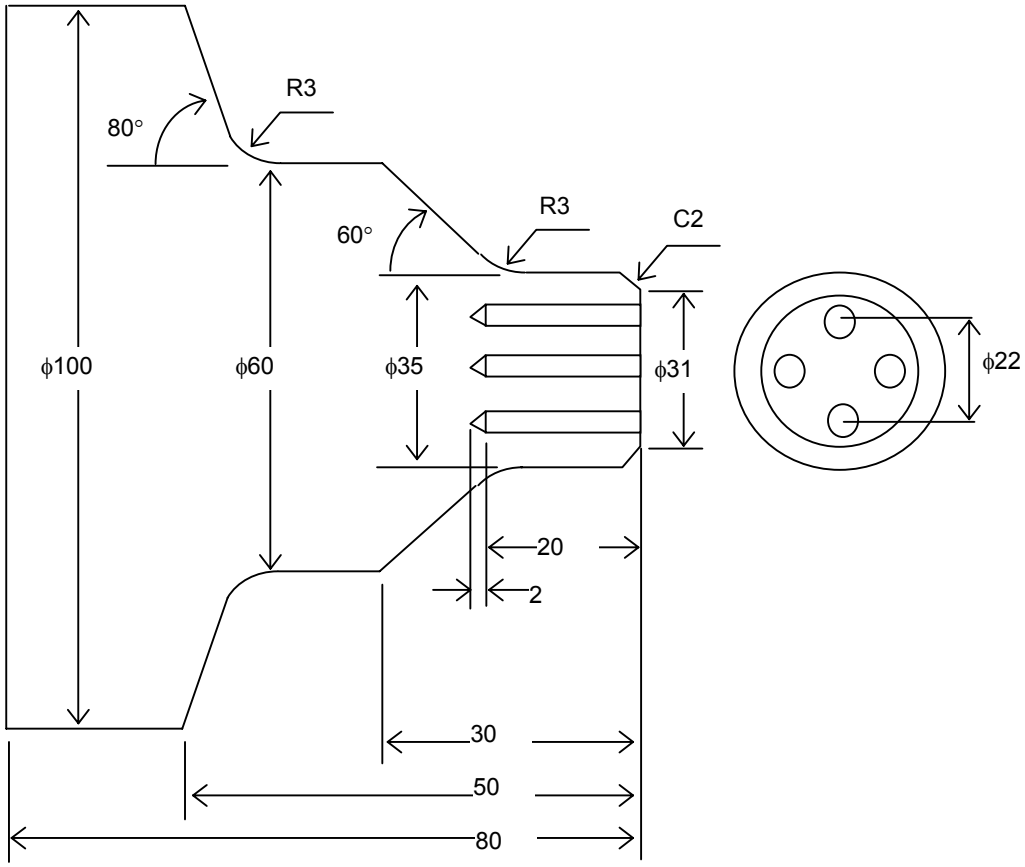
The part program made by using MANUAL GUIDE *i* has a form of ISO-code program with G-code and so on.
You must enter the program used for such as tool changing, tool offset, spindle rotation, approaching and releasing in the form of ISO-code program.
In addition to those actions, you can enter the part program for complicated machining motions, which are usually difficult to make by ISO-code form, as a "Cycle machining" by using menu programming method. This cycle machining is made in form of a block including G-4digits and necessary data items.
In the following explanations, the contents of a square frame mean the actual operations, and each operation are described as follows.

[NEWPRG]	:	Push a soft-key
12345	:	Enter numeric data
INPUT	:	Push an INPUT key
↓ ↑ → ←	:	Push a CURSOR key
⇓ ⇑	:	Push a PAGE key
(CREATE NEW PROG)	:	Name of Window or Data item
<START>	:	Name of Tab
<<1.CYLINDER>>	:	Menu item

2 LATHE

Example) Outer Roughing/Finishing, C-axis drilling

Workpiece : Round bar (ϕ100×80)
1st Process : Outer roughing by General purpose tool for roughing (T0101)
2nd Process : Outer finishing by General purpose tool for finishing (T0202)
3rd Process : C-axis end face drilling by Drill (T0303)



2.1 SETTING TOOL OFFSET DATA

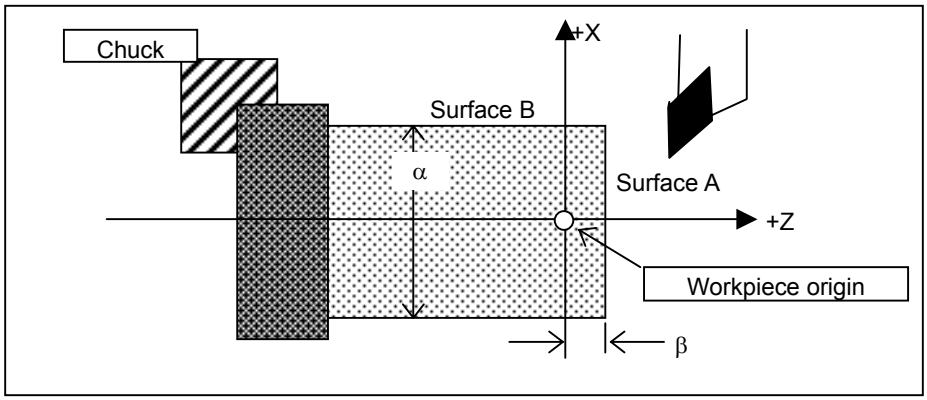
⚠ WARNING

1 Operation of tool offset setting varies from one machine model to another. So operations described in this section may differ from those ion actual machine.
As to the actual operation of tool offset setting on the actual machine, refer to the applicable manual supplied by the respective machine tool builders for details.
If the set data does not match the characteristic of a specific machine, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

2 As to the operations on a machine described in this chapter, refer to the applicable manual supplied by the respective machine tool builders for details.
If the operation does not match the characteristic of a specific machine, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

2.1.1 Setting of Z-axis Offset Data

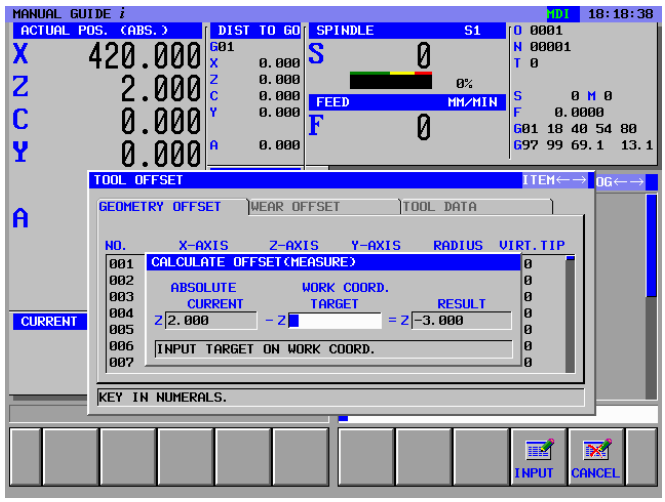
(1) Set a standard workpiece on a chuck of lathe. After then, for safety, take measure to keep fully safety such as closing the machine door.
(2) Execute the machine reference position return of X-and Z-axis.
(3) Output T-code in MDI mode, and select the tool for measuring.
(4) Make a spindle rotate by fully safety speed.
(5) Cut surface A of the following drawing in manual mode with a actual tool.



(6) Release the tool in X-axis direction only, without moving Z-axis.
(7) Stop the spindle.
(8) Measure distance β from the zero point in the workpiece coordinate system to surface A.

Operate as follows on the MANUAL GUIDE *i* screen.

[T-OFS]
(TOOL OFFSET)
<GEOMETRY OFFSET>
[CHCURS]
(ITEM <--> will be displayed on the right upper part of the window)
→
<<Z-AXIS>>
↓
<<Offset number for the used tool>>
[MEASUR]
(CALCULATE OFFSET(MEASURE))
β **INPUT** (Z WORK COORD.TARGET)
Check the result of calculation displayed in the (RESULT)
[INPUT]
(GEOMTERY OFFSET)


MANUAL GUIDE i
MDI 18:18:38
ACTUAL POS. (ABS.)
X 420.000
Z 2.000
C 0.000
Y 0.000
A
DIST TO GO
G01
X 0.000
Z 0.000
C 0.000
Y 0.000
A 0.000
SPINDLE S1
S 0
0%
FEED MM/MIN
F 0
O 0001
N 00001
T 0
S 0 M 0
F 0.0000
G01 18 40 54 80
G97 99 69.1 13.1
TOOL OFFSET
ITEM
OG
GEOMETRY OFFSET
WEAR OFFSET
TOOL DATA
NO. X-AXIS Z-AXIS Y-AXIS RADIUS VIRT.TIP
001
002
003
004
005
006
007
CALCULATE OFFSET(MEASURE)
ABSOLUTE CURRENT
WORK COORD. TARGET
RESULT
Z 2.000 -Z =Z -3.000
INPUT TARGET ON WORK COORD.
CURRENT
KEY IN NUMERALS.
INPUT
CANCEL

2.1.2 Setting of X-axis Offset Data

Continuously after setting of Z-axis offset data, set the X-axis offset data as follows.

(1) Make a spindle rotate by fully safety speed.
(2) Cut surface B of the following drawing in manual mode with a actual tool.
(3) Release the tool in Z-axis direction only, without moving X-axis.
(4) Measure the diameter α of surface B. Set this value as the measured value for X-axis in the desired offset No.

(TOOL OFFSET)
<GEOMETRY OFFSET>
(ITEM <--> will be displayed on the right upper part of the window)
←
<<X-AXIS>>
↓
<<Offset number for the used tool>>
[MEASUR]
(CALCULATE OFFSET(MEASURE))
α **INPUT** (X WORK COORD.TARGET)
Check the result of calculation displayed in the (RESULT)
[INPUT]
(GEOMTERY OFFSET)

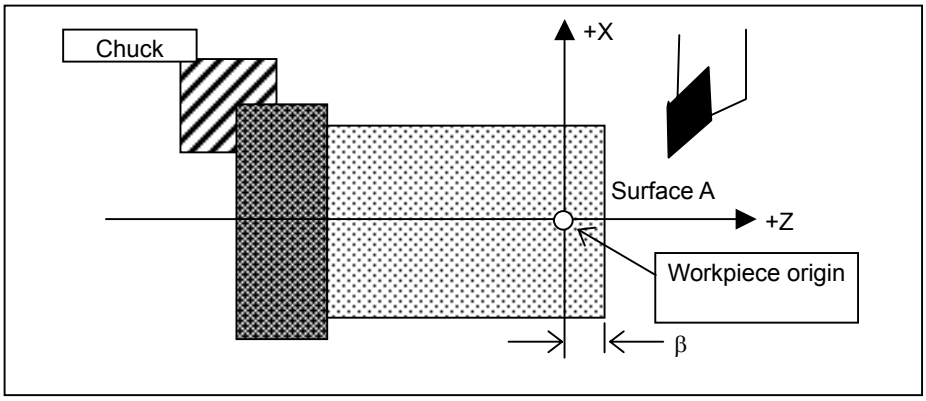
Repeat the above procedure of Z-axis and X-axis offset data measurement operations for necessary tools.

NOTE
1 Always measure the axis of diameter specification in terms of diameter value.
2 When the measured value is input as the geometry offset value by [MEASUR], the corresponding wear offset value is set 0.
3 Wear offset data is used for offsetting the error measured on the machined product or worn down amount of a tool.

2.2 SETTING OF WORKPIECE COORDINATE SYSTEM SHIFT DATA

After setting the geometry offset data for necessary tools, set the workpiece origin on the actual workpiece used for machining.
On the lathe, the rotating center line of a workpiece is usually set to workpiece origin of X-axis. So, the workpiece origin of X-axis need not be set again for new workpiece.
For Z-axis workpiece origin,, you must set again for new workpiece when you change to new one.
In this section, operations for setting the workpiece end surface to the workpiece origin are described.

(1) Set the actually machined workpiece on a chuck of lathe. After then, for safety, take measure to keep fully safety such as closing the machine door.
(2) Execute the machine reference position return of X-and Z-axis.
(3) Output T-code in MDI mode, and select the tool for measuring.
(4) Make a spindle rotate by fully safety speed.
(5) Cut surface A of the following drawing in manual mode with a actual tool.

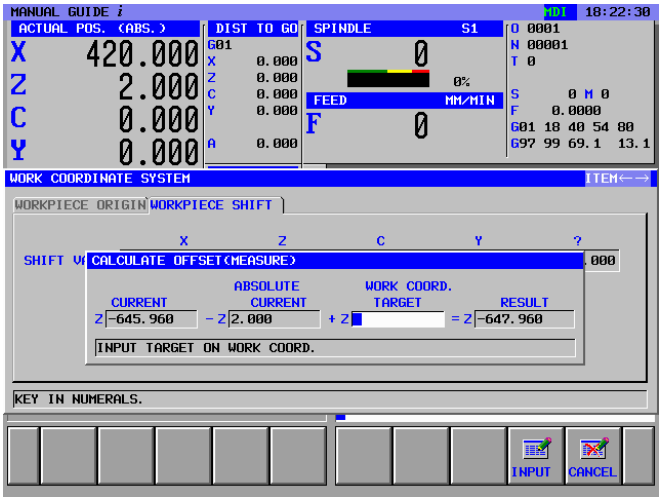


(6) Release the tool in X-axis direction only, without moving Z-axis.
(7) Stop the spindle.
(8) Define the end surface amount β.

Operate as follows on the MANUAL GUIDE *i* screen.

[WK SET]
(WORK CORRDINATE SYSTEM)
<WORKPIECE ORIGIN>
(TAB <--> will be displayed on the right upper part of the window)
→
<WORKPIECE SHIFT>
[CHCURS]
(ITEM <--> will be displayed on the right upper part of the window)
→
<<Z>>
[MEASUR]
(CALCULATE OFFSET(MEASURE))
β **INPUT** (WORK COORD.TARGET)
Check the result of calculation displayed in the (RESULT)
[INPUT]
(WORK COORDINATE SYSTEM)

As the result of the above operations, confirm that the absolute coordinate of the Z-axis is changed to the value of entered β value.



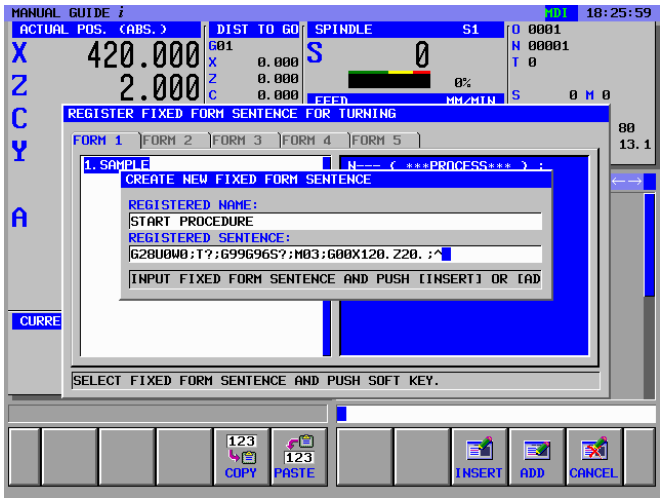
2.3 PREPARING OF THE FIXED FORM SENTENCE MENU

As to the fixed form sentence, machine tool builder usually sets the suitable menu for specified respective machine. But, you can enter his own menu on the MANUAL GUIDE *i* screen by yourself.

2.3.1 Entering the Fixed Form Sentence for Turning Machining

Enter the fixed form sentence menu which will be called by the soft-key [FIXFRM] in the soft-key group for turning machining.

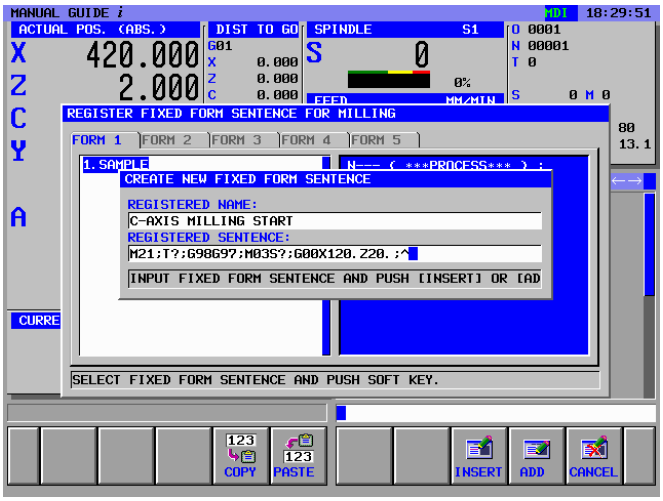
[SETTING]
(SETTING)
<BASIC>
↓
<<2.REGISTER FIXED FORM SENTENCE FOR TURNING>>
[SELECT]
(REGISTER FIXED FORM SENTENCE FOR TURNING)
<FORM1>
[NEW]
(CREATE NEW FIXED FORM SENTENCE)
<SENTENCE NAME : >
START PROCEDURE **INPUT**
<REGISTERED SENTENCE : >
G28 U0 W0 ; T? ; G99 G96 S? ; M03 ; G00 X120. Z20. ;
[INSERT]
[CLOSE]



2.3.2 Entering of the Fixed Form Sentence for Milling Machining

Enter the fixed form sentence menu which will be called by the soft-key [FIXFRM] in the soft-key group for milling machining.
Enter data for the program for milling starting procedure and program end procedure.

[SETTING]
(SETTING)
<BASIC>
↓
<<1.REGISTER FIXED FORM SENTENCE FOR MILLING>>
[SELECT]
(REGISTER FIXED FORM SENTENCE FOR MILLING)
<FORM1>
[NEW]
(CREATE NEW FIXED FORM SENTENCE)
<SENTENCE NAME : >
C-AXIS MILLING START **INPUT**
<REGISTERED SENTENCE : >
M21. ; T? ; G98 G97 ; M03 S? ; G00 X120. Z20. ;
[INSERT]
→
<FORM5>
[NEW]
(CREATE NEW FIXED FORM SENTENCE)
<SENTENCE NAME : >
PROGRAM END **INPUT**
<REGISTERED SENTENCE : >
M05. ; G00 X200. ; G28 U0 W0 ; M02 ;
[INSERT]
[CLOSE]



2.4 SETTING OF TOOL DATA

Set the necessary tool data. These tool data are used for displaying tool form of animation and calculation of cutting angle in the cycle machining.
T0101 : General purpose roughing tool
T0202 : General purpose finishing tool
T0303 : Drilling tool

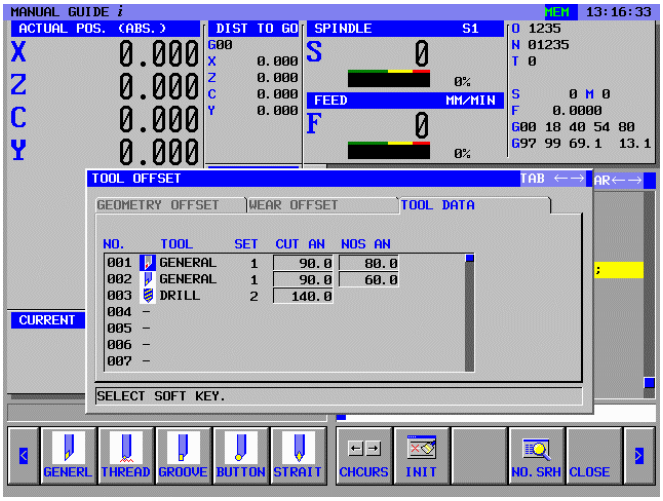
Display “TOOL OFFSET” window by [T-OFS]

GEOMETRY OFFSET				
NO.	X-AXIS	Z-AXIS	RADIUS	VIRT.TIP
001	-200.000	-300.000	0.800	3
002	-210.000	-310.000	0.400	3
003	-220.000	-330.000	3.000	0

NOTE
1 The above offset data are just an example, and they cannot be used for actual machining.
2 The above offset number 003 is used for drilling tool, and if you use it for animation, you must set the radius amount of the drill to radius offset data.

Display “TOOL DATA” tab by pushing cursor key →

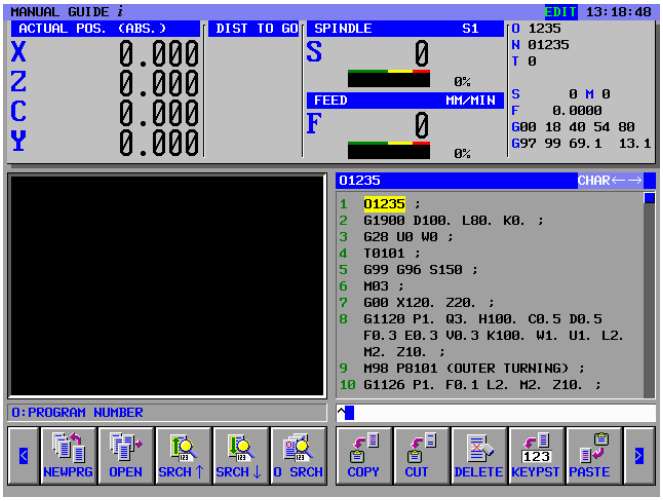
TOOL DATA				
NO.	TOOL	SET	CUT AN	NOS AN
001	GENERAL	1	90.0	80.0
002	GENERAL	1	90.0	80.0
003	DRILL	2	140.0	



2.5 CREATING OF PART PROGRAM

In the MANUAL GUIDE *i* , background editing can be used, but in this section, part program creating operations are described by using foreground editing.

2.5.1 Creating New Part Program



Create a new part program of O1234.

1. In case of creating a new part program directly

Select EDIT mode by using a mode-selecting switch on a machine-operating panel **[NEWPRG]** (CREATE NEW PROGRAM) 1234 **[CREATE]** (NEW PROGRAM NO.)

2. In case of creating a new part program on the program list window

Select EDIT mode by using a mode-selecting switch on a machine-operating panel **[O-LIST]** (OPEN PROGRAM) **[NEW]** (CREATE NEW PROGRAM) 1234 **[CREATE]** (NEW PROGRAM NO.) Select the newly entered program by ↓ **[OPEN]**

2.5.2 Operations of “START” Menu

By pushing [START] in the soft-key menu for turning machining, the window “INSERT STARTING COMMAND FOR TURNING” with the following tabs is displayed.

<START> : Fixed form sentence menu used for the top of part program or each machining process.

<BLANK> : Blank form menu, which is necessary for animation.

2.5.2.1 Entering blank form data

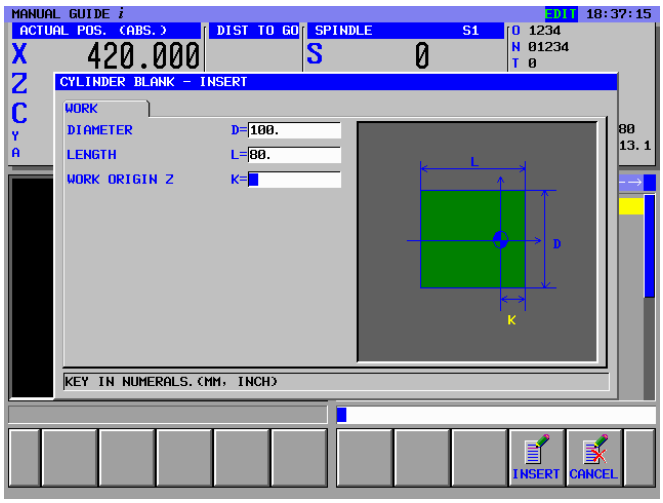
By selecting a tab <BLANK> by cursor key, blank form menu is displayed.

(INSERT STARTING COMMAND FOR TURNING)
<BLANK>
↓
<<2.CYLINDER BLANK FIGURE>>
[SELECT]

Enter blank form data as a round bar.

(CYLINDER BLANK)

100 **INPUT**	(DIAMETER)
80 **INPUT**	(LENGTH)
0 **INPUT**	(WORK ORIGIN)
[INSERT]	



2.5.3 Entering Tool Changing and Spindle Rotating Blocks for Turning Machining

2.5.3.1 Entering in ISO-code form directly

It is difficult to define the action of tool changing, spindle rotation, approaching and releasing generally because there are many difference depending on machine configuration. So, using ISO-code form program can realize more flexible and safety part program.

G28 U0 W0 ; **INSERT**	(Reference position return)
T0101 ; **INSERT**	(Tool change)
G99 G96 S150 ; **INSERT**	(Constant surface speed control, mm/rev mode)
M03 ; **INSERT**	(Spindle rotation)
G00 X120. Z20. ; **INSERT**	(Approaching)

2.5.3.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. But, in such a case, proper fixed form sentence must be prepared in advance.

(Soft-key group for Turning cycle menu)
[START]
<START>
↓
<<1.START PROCEDURE>>
[INSERT]

NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.
Place the cursor to the address with “?”, enter numeric data, then push “ALTER”.

2.5.4 Entering Outer Roughing Process

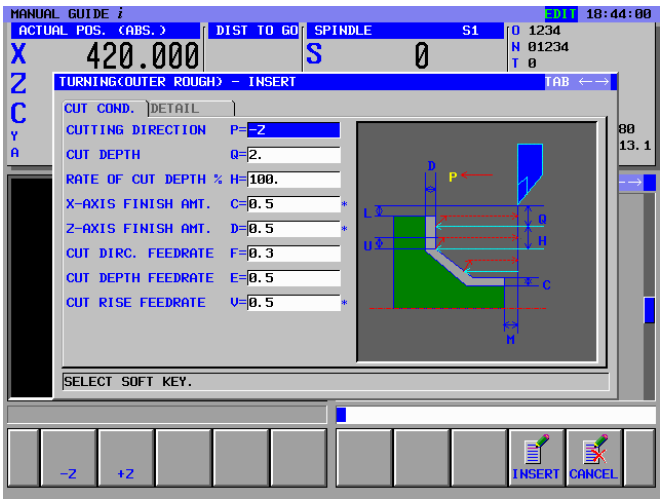
2.5.4.1 Entering outer roughing cycle block

Enter the 1st process : outer roughing by a general purpose roughing tool (T0101).
Enter machining type, cutting condition and so on.

(Soft-key group for Turning cycle menu)
[CYCLE]
→
<TURNING>
↓
<<1.TURNING(OUTER ROUGH)>>
[SELECT]
(TURNING(OUTER ROUGH) - INSERT)
<CUT COND.>

[-Z]	(CUTTING DIRECTION)
2 **INPUT**	(CUT DEPTH)
INPUT	(RATE OF CUT DEPTH %)
.5 **INPUT**	(X-AXIS FINISH AMT.)
.5 **INPUT**	(Z-AXIS FINISH AMT.)
.3 **INPUT**	(CUT DIRC. FEEDRATE)
.5 **INPUT**	(CUT DEPTH FEEDRATE)
.5 **INPUT**	(CUT RISE FEEDRATE)

[INSERT]



NOTE

1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.

2 In the cycle machining data menu window, 2 tabs, <CUT COND.> and <DETAIL>, are displayed. In the <CUT COND.>, all data must be set. In the <DETAIL>, all data are automatically set, so check those data and modify them if necessary only

2.5.4.2 Entering figure for outer roughing

By inserting the outer roughing cycle machining block, the window of free form entering is displayed, so enter the final figure of machining.

(ZX PLANE TURNING FIGURE - INSERT)	
(START POINT - INSERT)	
31 **INPUT**	(START POINT DX)
0 **INPUT**	(START POINT Z)
[OK]	
[LINE]	
(LINE - INSERT)	
[L-UP]	(LINE DIRECTION)
35 **INPUT**	(END POINT DX)
INPUT	(END POINT Z)
45 **INPUT**	(ANGLE)
[**OK]**	
[LINE]	
(LINE - INSERT)	
[LEFT]	(LINE DIRECTION)
[OK]	
[CR]	
(CORNER R - INSERT)	
3 **INPUT**	(CORNER RADIUS)
[OK]	
[LINE]	
(LINE - INSERT)	
[L-UP]	(LINE DIRECTION)
60 **INPUT**	(END POINT DX)
-30 **INPUT**	(END POINT Z)
60 **INPUT**	(ANGLE)
[OK]	
[LINE]	
(LINE - INSERT)	
[LEFT]	(LINE DIRECTION)
[OK]	
[CR]	
(CORNER R - INSERT)	
3 **INPUT**	(CORNER RADIUS)
[OK]	
[LINE]	
(LINE - INSERT)	
[L-UP]	(LINE DIRECTION)
100 **INPUT**	(END POINT DX)
-50 **INPUT**	(END POINT Z)
80 **INPUT**	(ANGLE)
[OK]	

After entering all the part figures for machining target, enter blank figure. When you use a preformed workpiece such like an cast iron, the most suitable cutting path can be made by entering the blank figure of the preformed workpiece.

In this programming example, round bar workpiece is used. So, enter the blank figure as follows.

(ZX PLANE TURNING FIGURE - INSERT)	
[LINE]	
(LINE - INSERT)	
[RIGHT]	(LINE DIRECTION)
0 **INPUT**	(END POINT Z)
→	
<ATTRIBUTE>	
[BLANK]	(ELEMENT TYPE)
[OK]	
[LINE]	
(LINE - INSERT)	
[DOWN]	(LINE DIRECTION)
31 **INPUT**	(END POINT DX)
[OK]	

Part figures are displayed in blue line, blank figures are displayed in green line, and the currently selected figure is displayed in yellow line.

Check the entered part figures and blank figures are correct by comparing with the blue print, then register them as figure blocks into CNC memory finally.

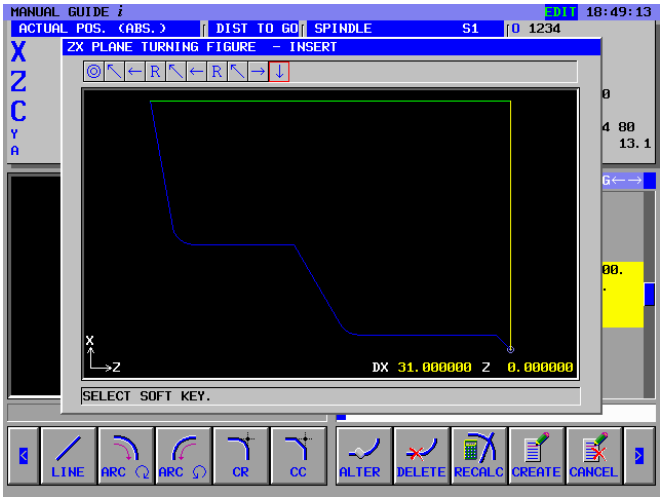
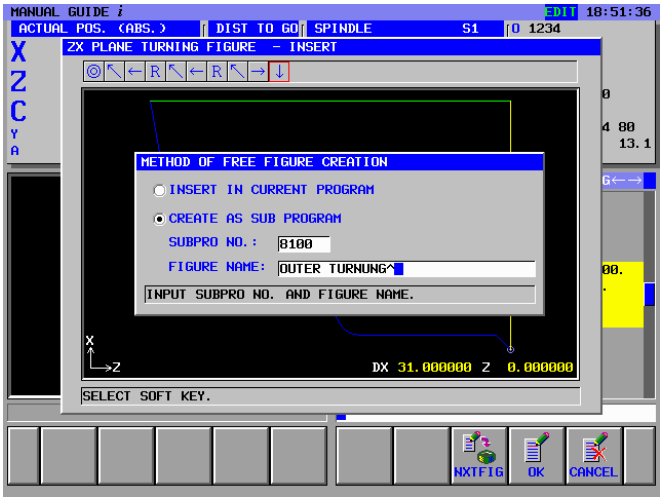


Figure blocks can be registered into the current part program directly, and also can be registered as another sub program.
Registered figure blocks can be used also for finishing, so in this example, register them as a sub program.

(ZX PLANE TURNING FIGURE - INSERT)
[CREATE]
(METHOD OF FREE FIGURE CREATION)
↓
<CREATE AS SUB PROGRAM>
8100 **INPUT** (SUBPRO NO.)
OUTER TURNING (FIGURE NAME)
[OK]



NOTE
Registered sub program can be displayed in a figure menu tab, “SUBPROGRAM”.
In this case, set the parameters No14720 to 14723 in advance.
For this example, set those parameters as follow.
No.14720=8000 (Minimum program number of sub programs displayed in the turning machining sub program menu)
No.14721=8499 (Maximum program number of sub programs displayed in the turning machining sub program menu)
No.14722=8500 (Minimum program number of sub programs displayed in the milling machining sub program menu)
No.14723=8999 (Maximum program number of sub programs displayed in the milling machining sub program menu)

2.5.5 Entering Tool Changing and Spindle Rotation Blocks for Outer Finishing in ISO-code Form

Before starting the 2nd process of outer finishing, change tool to the finishing tool (T0202), spindle rotation, and other necessary blocks in ISO-code form with G-code and so on.

```
G28 U0 W0 ; INSERT          (Reference position return)
T0202 ; INSERT              (Tool change)
G99 G96 S300 ; INSERT       (Constant surface speed control, mm/rev mode)
M03 ; INSERT                (Spindle rotation)
G00 X120. Z20. ; INSERT     (Approaching)
```

Otherwise, you can enter them from the fixed form sentence menu.

```
(Soft-key group for Turning cycle menu)
[START]
<START>
↓
<<1.START PROCEDURE>>
[INSERT]
```

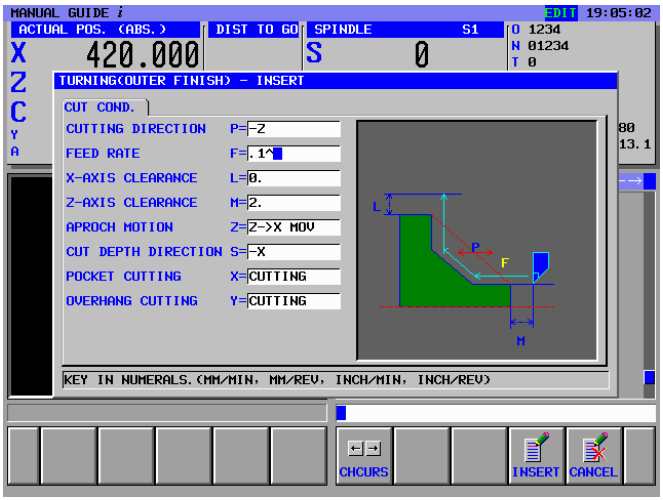
NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.

2.5.6 Entering Outer Finishing Cycle Machining Process

2.5.6.1 Entering figure for outer finishing cycle block

Enter the 2nd process : outer finishing by a general purpose finishing tool (T0202).
Enter machining type, cutting condition and so on.

(Soft-key group for Turning cycle menu)
[CYCLE]
→
<TURNING>
↓
↓
<<7.TURNING(OUTER FINISH)>>
[SELECT]
(TURNING(OUTER FINISH) - INSERT)
<CUT COND.>
[-Z] (CUTTING DIRECTION)
.1 **INPUT** (FEEDRATE)
[INSERT]

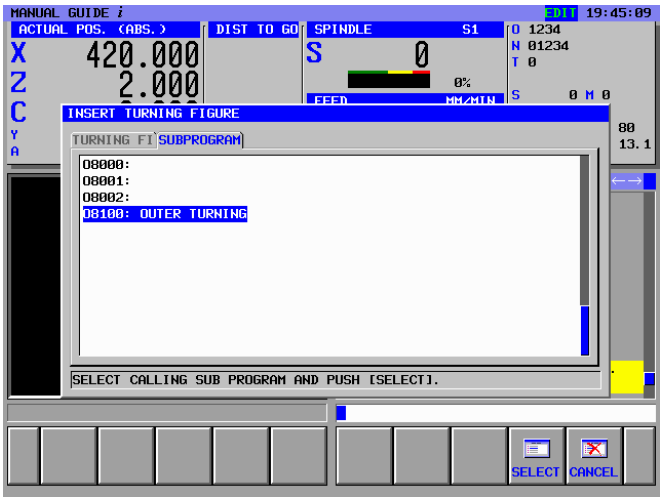


NOTE
1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 When you select the cycle machining type form cycle machining menu, instead of selecting by the cursor, you can use direct inputting of item number and **INPUT**. (In this case, enter 7 **INPUT**)

2.5.6.2 Entering figure for outer finishing

By inserting the outer finishing cycle machining block, the window for free form entering is displayed, so enter the final figure of machining. But, the former registered figure blocks for roughing can be used, so push [CANCEL] and go back to the turning figure menu window, then select from the sub program menu.

(ZX PLANE TURNING FIGURE - INSERT)
(START POINT - INSERT)
[CANCEL]
(INSERT TURNING FIGURE)
→
<SUBPROGRAM>
↓
<<08100: OUTER TURNING>>
[SELECT]



2.5.6.3 Entering releasing motion blocks in ISO-code form

After outer finishing, return the tool to safety area before the next C-axis drilling process.
Enter these motion blocks in form of ISO-code with such as G-code.

G00 X200. ; **INSERT**	(Releasing motion)
M05 ; **INSERT**	(Spindle stop)
G28 U0 W0 ; **INSERT**	(Reference position return)

2.5.7 Entering Tool Changing and Spindle Rotating Blocks for C-axis Drilling

2.5.7.1 Entering in ISO-code form directly

Enter blocks of tool changing, C-axis mode changing spindle rotation approaching and releasing for C-axis drilling. You can enter these blocks using ISO-code form.

M21. ; **INSERT**	(Change to C-axis mode)
T0303 ; **INSERT**	(Tool change)
G98 G97 ; **INSERT**	(mm/min mode)
M03 S800 ; **INSERT**	(Spindle rotation)
G00 X120. Z20. ; **INSERT**	(Approaching)

2.5.7.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. But, in such a case, proper fixed form sentence must be prepared in advance.

(Soft-key group for Milling cycle menu)
[START]
<START>
↓
<<2.C-AXIS MILLING START>>
[INSERT]

NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.

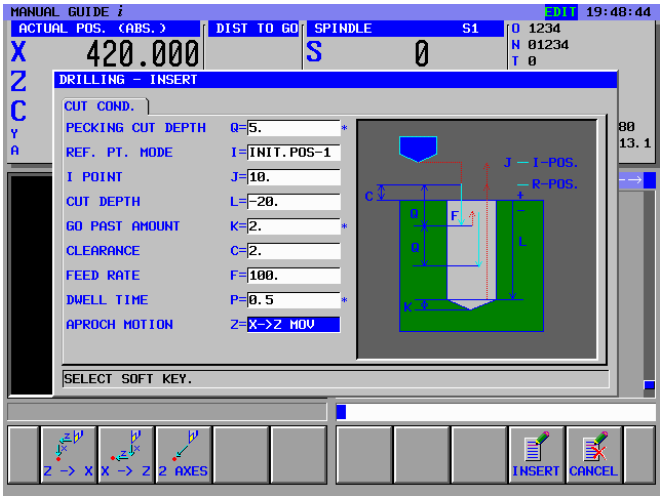
2.5.8 Entering C-axis Drilling Process

2.5.8.1 Entering C-axis drilling cycle block

Enter the 3rd process : C-axis end face drilling by the drilling tool (T0303).
Enter machining type, cutting condition and so on.

(Soft-key group for Milling cycle menu)	
[CYCLE]	
<HOLE MACH.>	
↓	
<<2.DRILLING>>	
[SELECT]	
(DRILLING - INSERT)	
<CUT COND.>	
5 **INPUT**	(PECKING CUT DEPTH)
INPUT	(REF.PT.MODE)
10 **INPUT**	(I POINT)
-20 **INPUT**	(CUT DEPTH)
2 **INPUT**	(GO PAST AMOUNT)
2 **INPUT**	(CLEARANCE)
100 **INPUT**	(FEEDRATE)
.5 **INPUT**	(DWELL TIME)
[INSERT]	

NOTE
In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.



2.5.8.2 Entering hole position block

By inserting the drilling cycle block, the window of hole position menu is displayed, so select the “Arc point” item.

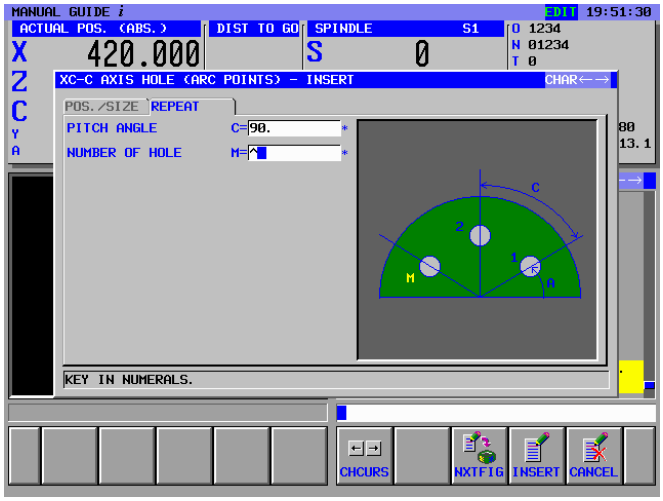
(INSERT MILLING FIGURE)
<HOLE POSI.>
↓
↓
<<17.C-AXIS HOLE ON FACE (ARC POINTS)>>
[SELECT]
(XC-C AXIS HOLE (ARC POINTS) - INSERT)
<POS./SIZE>

0 **INPUT**	(BASE POSITION)
11 **INPUT**	(X AXIS POS.(RAD.))
0 **INPUT**	(START ANGLE)

→
<REPEAT>

90 **INPUT**	(PITCH ANGLE)
4 **INPUT**	(NUMBER OF HOLE)

[INSERT]



2.5.9 Operations in the “END” Menu

All necessary machining program have been entered, so enter end procedure.

2.5.9.1 Entering in ISO-code form directly

Enter blocks for spindle stop, releasing and end M-code in ISO-code form with G-code and son on.

M05. ; **INSERT**	(Spindle stop)
G00 X200. ; **INSERT**	(Releasing motion)
G28 U0 W0 ; **INSERT**	(Reference position return)
M02 ; **INSERT**	(End M-code)

2.5.9.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. But, in such a case, proper fixed form sentence must be prepared in advance.

(Soft-key group for Milling cycle menu)
[END]
<END>
↓
<<1.PROGRAM END>>
[INSERT]

NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.

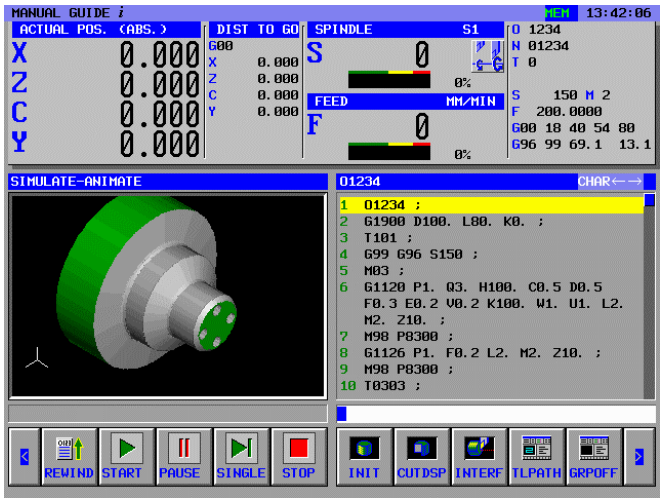
2.6 CHECKING OF THE PART PROGRAM

You can check the entered part program by animation.

2.6.1 Checking by Animation

Select MEM mode by using a mode-selecting switch on a machine-operating panel
[SIMLAT]
(SIMULATE - ANIMATE)
[REWIND]
[START]

NOTE
After checking by animation, in order to do other operation, you must close the animation window by pushing [GRPOFF] always.



3 MACHINING CENTER

Example) Outer wall contouring, Pocketing, Drilling

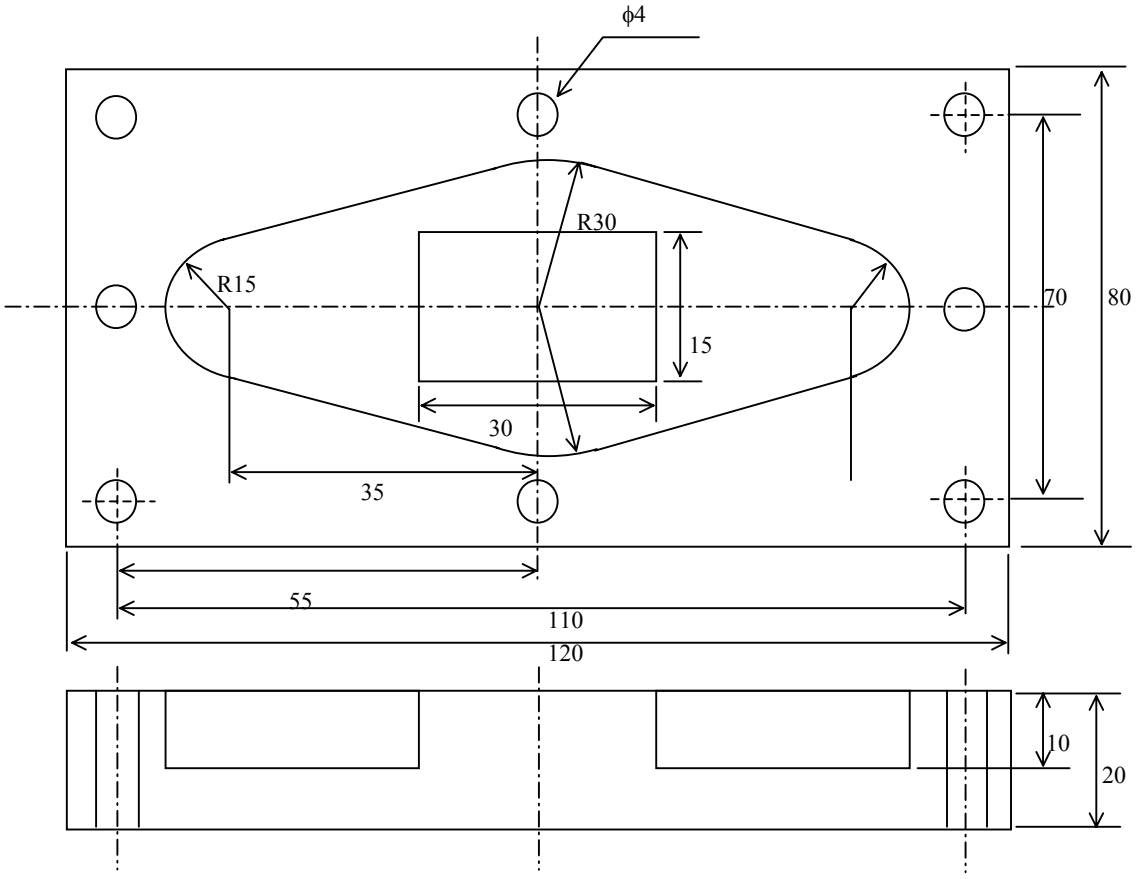
Workpiece : 90×130×30

1st Process : Outer wall contouring by Flat end mill (T01)

2nd Process: Pocket roughing by Flat end mill (T01)

3rd Process : Pocket finishing by Flat end mill (T02) (side, bottom)

4th Process : Drilling by Drill tool (T03)



3.1 SETTING OF TOOL LENGTH OFFSET DATA

⚠ WARNING

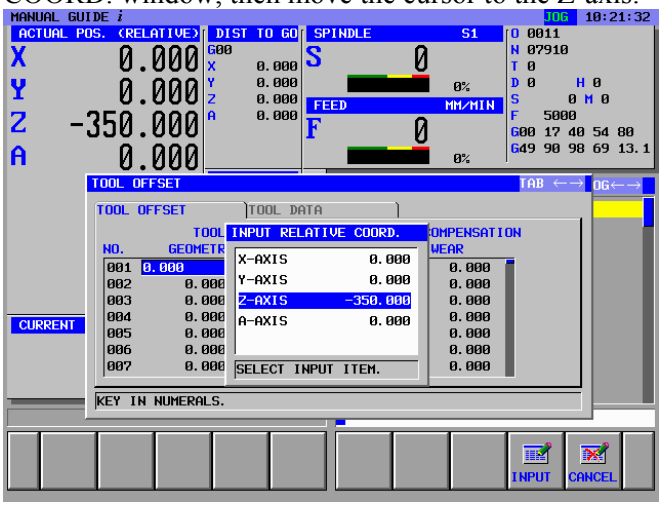
1 Operation of tool offset setting varies from one machine model to another. So operations described in this section may differ from those ion actual machine. As to the actual operation of tool offset setting on the actual machine, refer to the applicable manual supplied by the respective machine tool builders for details.
If the set data does not match the characteristic of a specific machine, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

2 As to the operations on a machine described in this chapter, refer to the applicable manual supplied by the respective machine tool builders for details.
If the operation does not match the characteristic of a specific machine, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

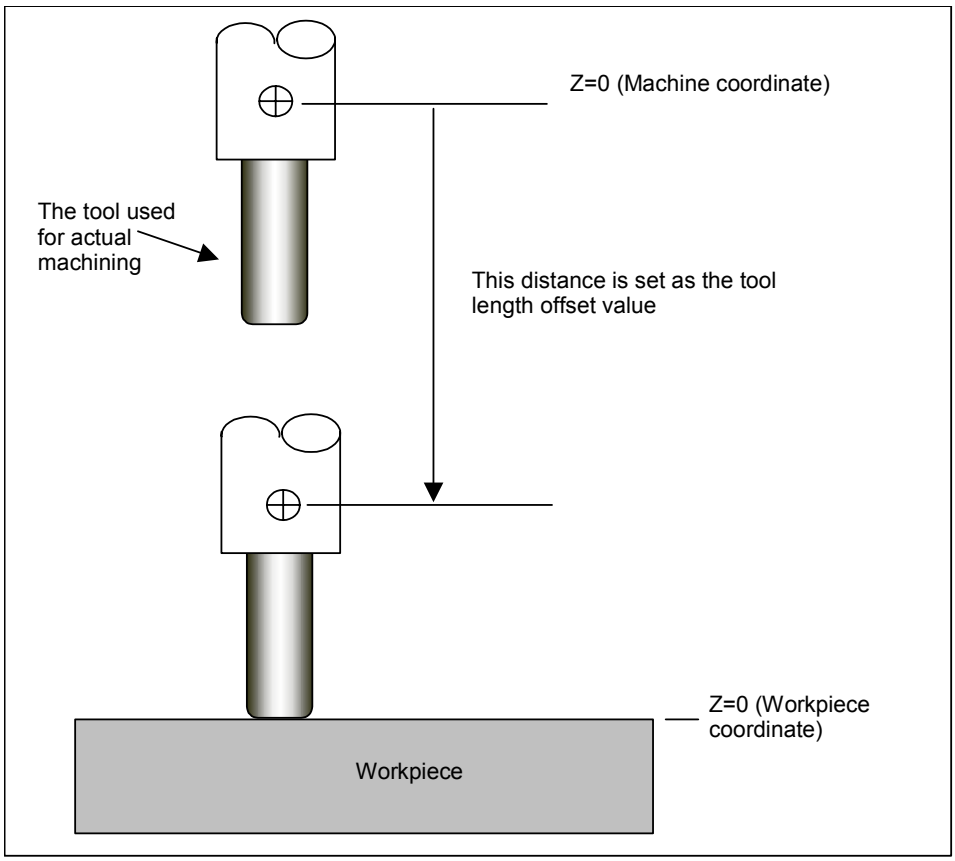
Assume that the Z=0 position of the workpiece coordinate is the surface of the workpiece, while the Z=0 position of the machine coordinate is the machine origin.
By setting the distance between these 2 points to the tool length offset data, workpiece coordinate of Z-axis can be defined.
Tool length is different between each tool for the actual machining, so set the offset data respectively.
The tool length offset value is the relative coordinate value of Z-axis at the point where the tool tip is touched to the surface of the workpiece.

(1) Select the tool used for actual machining by switch on the machine operator's panel.
(2) Execute the machine reference position return of Z-axis.
(3) Press [ACTPOS] and make the relative coordinate data display.
(4) Press [PRESET], then push [ALL 0], and press [ALTER], then the relative coordinate value of all axes are reset to 0. But, the Z-axis coordinate only is used for measuring.
(5) Press [T-OFS] and make the tool offset widow open.
(6) Make the tool tip touch at the surface of the workpiece by JOG or Manual handwheel. Then, the distance from the machine origin is displayed as the Z-axis relative coordinate value.
(7) Select the tool length compensation column of the tool used for measuring by moving the cursor by cursor key.

(8) Pressing [INP.C.] displays the window of INPUT RELATIVE COORD. window, then move the cursor to the Z-axis.



(9) Pressing [INPUT] makes the Z-axis relative coordinate value enter as the tool offset length data.



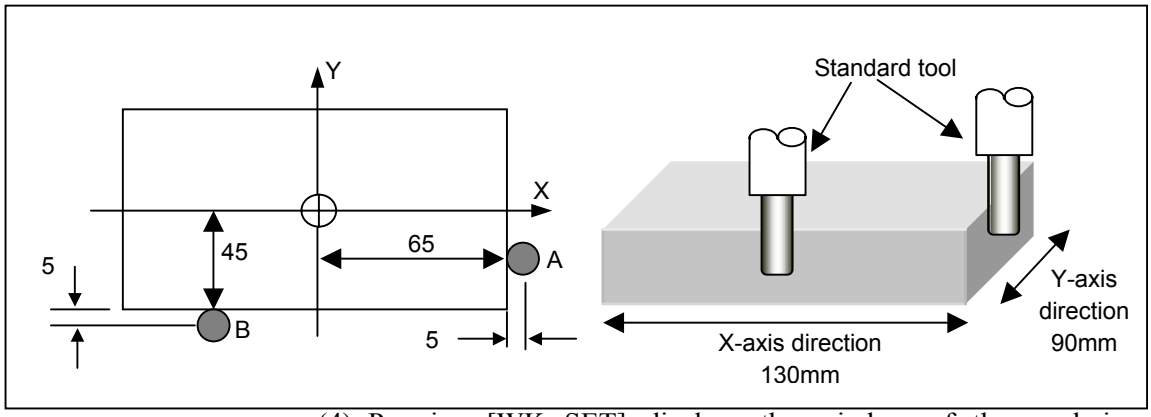
NOTE

There is a case such as using a sensor called base master instead of touching the tool to the workpiece surface, so into details and actual operations, refer to the manual made by machine tool builder.

3.2 SETTING OF WORKPIECE ORIGIN OFFSET VALUE

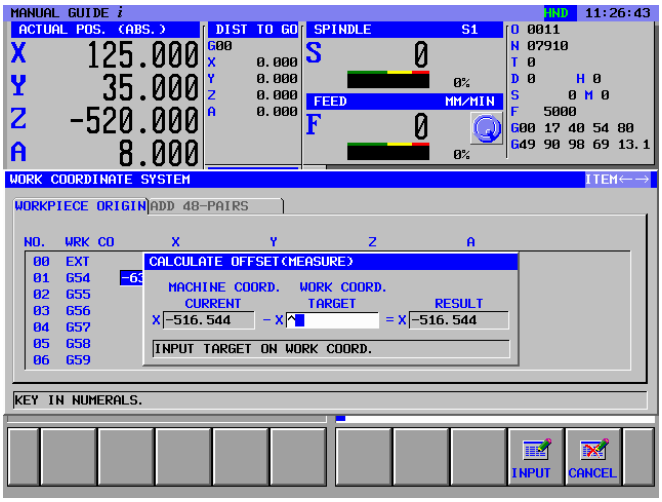
After setting the geometry offset data for necessary tools, set the workpiece origin on the actual workpiece used for machining.
In order to carry out the actual machining by using the part program made on the workpiece coordinate, set the distance between machine coordinate and workpiece coordinate to the workpiece coordinate system as the workpiece origin offset value.
In this section, operations, of which setting the center of the workpiece (90x130) as the workpiece origin of X/Y/Z-axis by using standard too with the radius 5mm, are described.
And, G54 is used as the workpiece coordinate system.

(1) Select the standard tool by using the switch on the machine operator's panel. There is a case of using special tool such as the centering lever as the standard tool for easy operations and high precision accuracy, so refer to the manual made by machine tool builder for actual operations.
(2) After then, for safety, take measure to keep fully safety such as closing the machine door.
(3) Select the manual handwheel mode on the machine operator's panel, and make the standard tool touch to the right side of the workpiece (A-position in the drawing) by referencing the following drawing. At this time, make a spindle rotate by fully safety speed if necessary, then you can avoid over-cutting by stopping the tool moving at the position the tool begins cutting. As to actual operations, refer to the manual made by the machine tool builder.



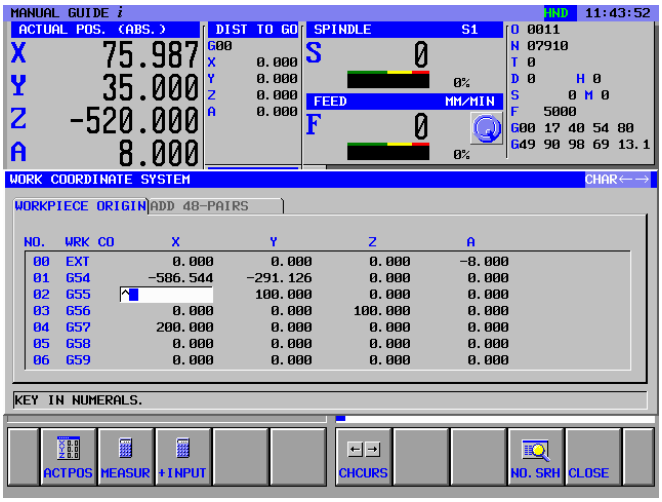
(4) Pressing [WK SET] displays the window of the workpiece coordinate system.
(5) Select the G54 X-axis data by moving the cursor. When other tab is displayed in the window, the cursor motion type might be the tab transition type, so press [CHCURS] for changing to item transition type of cursor.

(6) Pressing [MEASUR] displays the window of offset calculating.



(7) When the tool touches to the right side of the workpiece, the X-axis position should be X=70.0mm, 65mm of the right side position + 5mm of the tool radius, so enter 70.0 to the target workpiece coordinate value. Then, the calculation result value is displayed at the RESULT column, so confirm the result is correct.

(8) Pressing [INPUT] sets the above calculated value to the workpiece origin offset data.



(9) Release the tool from the workpiece once, and set the Y-axis workpiece origin offset data by similar operations of (3) to (8). In this case, touch the point B in the drawing and the entering target value is Y=50.0mm.

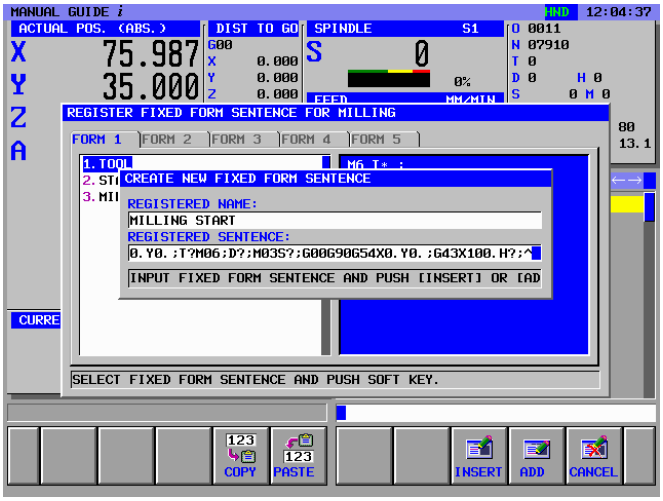
3.3 PREPARING OF THE FIXED FORM SENTENCE MENU

As to the fixed form sentence, machine tool builder usually sets the suitable menu for specified respective machine. But, you can enter his own menu on the MANUAL GUIDE *i* screen by yourself.

3.3.1 Entering the Fixed Form Sentence for Milling

Enter the fixed form sentence menu which will be called by the soft-key [FIXFRM] in the soft-key group for milling.
Enter programs for START and END.

[SETTING]
(SETTING)
<BASIC>
↓
<<1. REGISTER FIXED FORM SENTENCE FOR MILLING>>
[SELECT]
(REGISTER FIXED FORM SENTENCE FOR MILLING)
<FORM 1>
[NEW]
(CRATE NEW FIXED FORM SENTENCE)
<REGISTERED NAME : >
MILLING START **INPUT**
<REGISTERED SENTENCE : >
G28 G91 Z0. ; G28 X0. Y0. ; T? M06 ; D? ; M03 S? ; G00 G90 G54 X0. Y0.; G43 Z100. H? ;
[ADD]
→
<FORM5>
[NEW]
(CREATE NEW FIXED FORM SENTENCE)
<REGISTERED NAME : >
PROGRAM END **INPUT**
<REGISTERED SENTENCE : >
M05. ; G00 G90 Z100. ; G28 G91 Z0, ; M06 T0 ;M30 ;
[ADD]
[TO MNU]



3.4 SETTING OF THE TOOL DATA

Set the necessary tool data. These tool data are used for displaying tool form of animation and calculation of cutting angle in the cycle machining.
The tool length offset data were already set in section 3.1.
T01 : Flat end mill for roughing
T02 : Flat end mill for finishing
T03 : Drill

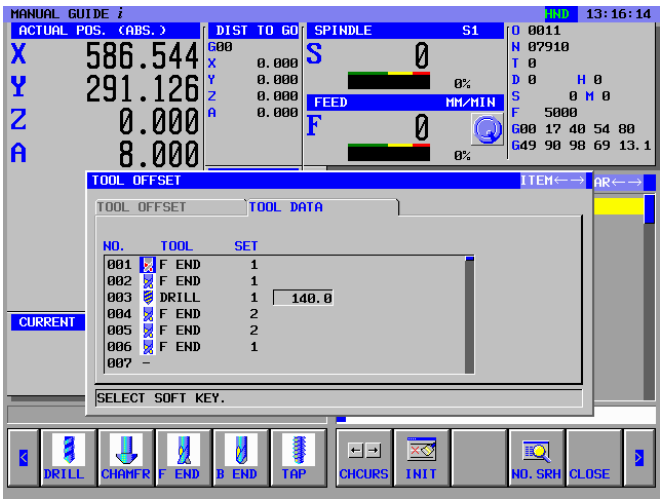
Display “TOOL OFFSET” window by [T-OFS]

TOOL OFFSET				
NO.	TOOL LENGTH. COMP.		CUTTER COMPENSATOIN	
	GEOMETRY	WEAR	GEOMETRY	WEAR
001	Measured value	0.000	4.000	0.000
002	Measured value	0.000	3.000	0.000
003	Measured value	0.000	2.000	0.000

NOTE
1 The above offset data are just an example, and they cannot be used for actual machining.
2 The above offset number 003 is used for drilling tool, and if you use it for animation, you must set the radius amount of the drill to radius offset data.
3 There is a case that the above offset screen might not be displayed depending on the attached optional function. Into details, refer to the manual made by machine tool builder.

Display “TOOL DATA” tab by cursor key →

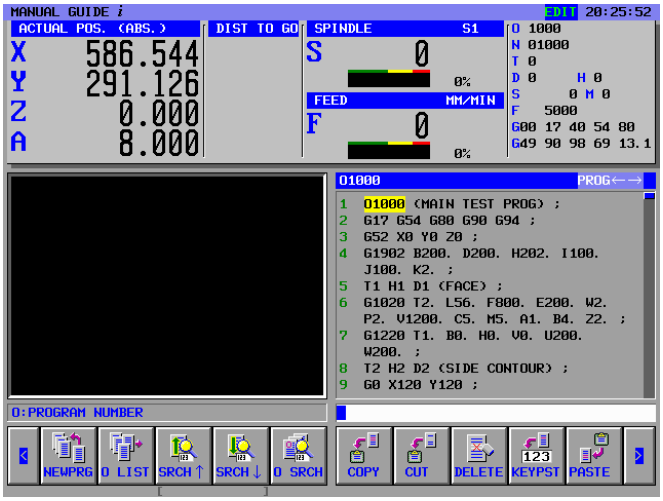
TOOL DATA				
NO.	TOOL	SET		
001	F END	1		
002	F END	1		
003	DRILL	1	140.0	



3.5 CREATING OF PART PROGRAM

In the MANUAL GUIDE *i* , background editing can be used, but in this section, part program creating operations are described by using foreground editing on the EDIT mode.

3.5.1 Creating New Part Program



Create a new part program of O1234.

1. In case of creating a new part program directly

Select EDIT mode by using a mode-selecting switch on a machine-operating panel **[NEWPRG]** (CREATE NEW PROGRAM) 1234 **[CREATE]** (NEW PROGRAM NO.)

2. In case of creating a new part program on the program list window

Select EDIT mode by using a mode-selecting switch on a machine-operating panel **[O-LIST]** (OPEN PROGRAM) **[NEW]** (CREATE NEW PROGRAM) 1234 **[CREATE]** (NEW PROGRAM NO.) Select the newly entered program by ↓ **[OPEN]**

3.5.2 Operations of “START” Menu

By pushing [START] in the soft-key menu for milling, the window “INSERT STARTING COMMAND FOR MILLING” with the following tabs is displayed.

<START> : Fixed form sentence menu used for the top of part program or each machining process.

<COORD CONV> : Coordinate conversion menu, which is necessary for the machine on the inclined surface, this is not used in this section.

<BLANK> : Blank form menu, which is necessary for animation.

3.5.2.1 Entering blank form data

By selecting a tab <BLANK> by cursor key, blank form menu is displayed.

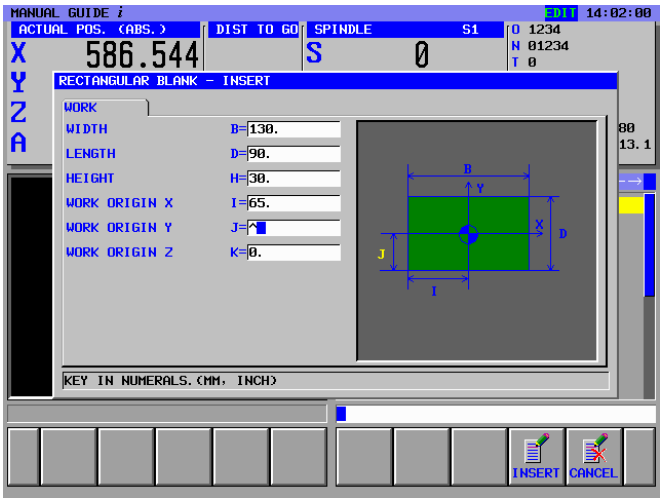
(INSERT STARTING COMMAND FOR MILLING)
<BLANK>
↓
<<1. RECTANGULAR BLANK FIGURE>>
[SELECT]

Enter blank form data as a round bar.

(RECTANGULAR BLANK FIGURE - INSERT)

130 **INPUT**	(WIDTH)
90 **INPUT**	(LENGTH)
30 **INPUT**	(HEIGHT)
65 **INPUT**	(WORK ORIGIN X)
45 **INPUT**	(WORK ORIGIN Y)
0 **INPUT**	(WORK PROGIN Z)

[INSERT]



3.5.3 Entering Tool Changing and Spindle Rotating Blocks for Roughing Flat End Mill

3.5.3.1 Entering in ISO-code form directly

It is difficult to define the action of tool changing, spindle rotation, approaching and releasing generally because there are many difference depending on machine configuration. So, using ISO-code form program can realize more flexible and safety part program.

```
G28 G91 Z0. ; INSERT          (Z-axis reference position return)
G28 G91 X0. Y0. ; INSERT      (X/Y-axis reference position return)
T01 M06 ; INSERT              (Tool change)
D1 ; INSERT                   (Selecting tool data)
M03 S1000 ; INSERT            (Spindle rotation)
G00 G90 G54 X0. Y0. ; INSERT  (X/Y-axis approaching)
G43 Z100. H1 ; INSERT         (Z-axis approaching)
```

3.5.3.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. But, in such a case, proper fixed form sentence must be prepared in advance.

```
(Soft-key group for milling cycle menu)
[START]
(INSERT STARTING COMMAND FOR MILLING)
<START>
↓
<<3.MILLING START>>
[INSERT]
```

NOTE

There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.
Place the cursor to the address with “?”, enter numeric data, then push “ALTER”.
After altering the data, return the cursor to the EOB at the end of the program.

3.5.4 Entering Outer Wall Contouring Process

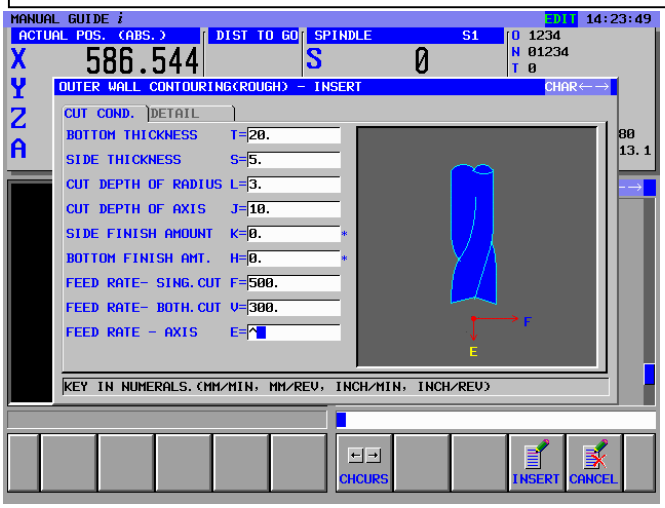
3.5.4.1 Entering outer wall contouring (rough) cycle block

Enter the 1st process : outer wall contouring process by the roughing flat endmill (T01).
Enter machining type, cutting condition and so on.

(Soft-key menu for milling cycle menu)
[CYCLE]
(INSERT MILLING CYCLE)
→
<CONTOURING>
↓
<<1.OUTER WALL CONTOURING(ROUGH)>>
[SELECT]
(OUTER WALL CONTOURING(ROUGH) - INSERT)
<CUT COND.>

20 **INPUT**	(BOTTOM THICKNESS)
5 **INPUT**	(SIDE THICKNESS)
3 **INPUT**	(CUT DEPTH OF RADIUS)
10 **INPUT**	(CUT DEPTH OF AXIS)
0 **INPUT**	(SIDE FINISH AMOUNT)
0 **INPUT**	(BOTTOM FINISH AMOUNT)
500 **INPUT**	(FEED RATE - SING.CUT)
300 **INPUT**	(FEED RATE - BOTH CUT)
200 **INPUT**	(FEED RATE - AXIS)

[INSERT]



NOTE
1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 In the cycle machining data menu window, 2 tabs, <CUT COND.> and <DETAIL>, are displayed. In the <CUT COND.>, all data must be set. In the <DETAIL>, all data are automatically set, so check those data and modify them if necessary only

3.5.4.2 Entering figure for outer wall contouring (rough)

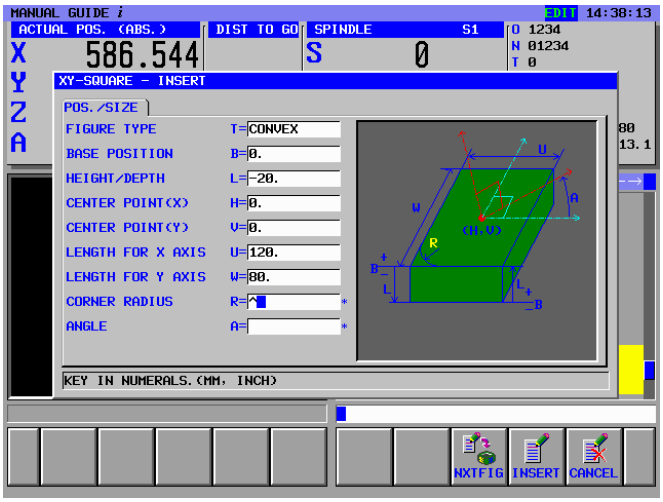
By inserting the cycle machining block, the window of contouring figure menu is displayed, so select the XY-SQUARE CONVEX

(INSERT MILLING FIGURE)
<CONT. FIG.>

<<1.XY-SQUARE CONVEX>>
[SELECT]
(XY-SQUARE - INSERT)
<POS./SIZE>

[CONVEX]	(FIGURE TYPE)
0 **INPUT**	(BASE POSITION)
-20 **INPUT**	(HEIGHT/DEPTH)
0 **INPUT**	(CENTER POINT (X))
0 **INPUT**	(CENTER POINT (Y))
120 **INPUT**	(LENGTH FOR X AXIS)
80 **INPUT**	(LENGTH FOR Y AXIS)

[INSERT]



3.5.5 Entering Pocket Roughing Process

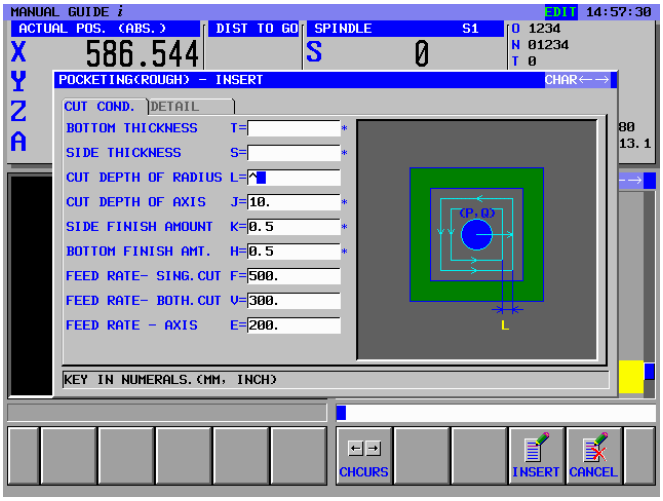
3.5.5.1 Entering pocket roughing cycle block

Enter the 2nd process : pocket roughing by a roughing flat endmill (T01).
Since same tool with the 1st process is used, tool changing blocks are not necessary.
Enter machining type, cutting condition and so on.

(Soft-key group for Milling cycle menu)
[CYCLE]
(INSERT MILLING CYCLE)
→
<POCKETING>
↓
<<1.POCKETING(ROUGH)>>
[SELECT]
(POCKETING(ROUGH) - INSERT)
<CUT COND.>

INPUT	(BOTTOM THICKNESS)
INPUT	(SIDE THICKNESS)
3 **INPUT**	(CUT DEPTH OF RADIUS)
10 **INPUT**	(CUT DEPTH OF AXIS)
0.5 **INPUT**	(SIDE FINISH AMOUNT)
0.5 **INPUT**	(BOTTOM FINISH AMOUNT)
500 **INPUT**	(FEED RATE - SING.CUT)
300 **INPUT**	(FED RATE - BOTH CUT)
200 **INPUT**	(FEED RATE - AXIS)

[INSERT]



NOTE
1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 In the cycle machining data menu window, 2 tabs, <CUT COND.> and <DETAIL>, are displayed. In the <CUT COND.>, all data must be set. In the <DETAIL>, all data are automatically set, so check those data and modify them if necessary only

3.5.5.2 Entering figure for pocket roughing

By inserting the cycle machining block, the window of pocketing figure menu, so select the XY-FREE CONCAVE FIGURE.

(INSERT MILLING FIGURE)	
<POCKET FIG>	
↓	
<<4.XY-FREE CONCAVE FIGURE>>	
[SELECT]	
(XY PLANE FREE FIGURE - INSERT)	
(START POINT - INSERT)	
INPUT	(FIGURE TYPE)
-50 **INPUT**	(START POINT X)
0 **INPUT**	(START POINT Y)
0 **INPUT**	(BASE POSITION)
-10 **INPUT**	(HEIGHT/DEPTH)
[OK]	
[ARC ↷]	
(ARC (CW) - INSERT)	
INPUT	(END POINT X)
INPUT	(END POINT Y)
INPUT	(RADIUS)
-35 **INPUT**	(CENTER POINT CX)
0 **INPUT**	(CENTER POINT CY)
INPUT	(LAST CONNECTION)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[LINE]	
(LINE - INSERT)	
[R-UP]	(LINE DIRECTION)
INPUT	(END POINT X)
INPUT	(END POINT Y)
INPUT	(ANGLE)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[ARC ↷]	
(ARC (CW) - INSERT)	
INPUT	(END POINT X)
INPUT	(END POINT Y)
30 **INPUT**	(RADIUS)
0 **INPUT**	(CENTER POINT CX)
0 **INPUT**	(CENTER POINT CY)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[LINE]	
(LINE - INSERT)	
[R-DOWN]	(KINE DIRECTION)
INPUT	(END POINT X)
INPUT	(END POINT Y)
INPUT	(ANGLE)
[TANGNT]	(NEXT CONNECTION)
[OK]	

[ARC ↻]	
(ARC (CW) - INSERT)	
INPUT	(END POINT X)
INPUT	(END POINT Y)
15 **INPUT**	(RADIUS)
35 **INPUT**	(CENTER POINT CX)
0 **INPUT**	(CENTER POINT CY)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[LINE]	
(LINE - INSERT)	
[L-DOWN]	(LINE DIRECTION)
INPUT	(END POINT X)
INPUT	(END POINT Y)
INPUT	(ANGLE)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[ARC ↻]	
(ARC (CW) - INSERT)	
INPUT	(END POINT X)
INPUT	(END POINT Y)
30 **INPUT**	(RADIUS)
0 **INPUT**	(CENTER POINT CX)
0 **INPUT**	(CENTER POINT CY)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[LINBE]	
(LINE - INSERT)	
[L-UP]	(LINE DIRECTION)
INPUT	(END POINT X)
INPUT	(END POINT Y)
INPUT	(ANGLE)
[TANGNT]	(NEXT CONNECTION)
[OK]	
[ARC ↻]	
(ARC (CW) - INSERT)	
-50 **INPUT**	(END POINT X)
0 **INPUT**	(END POINT Y)
15 **INPUT**	(RADIUS)
-35 **INPUT**	(CENTER POINT CX)
0 **INPUT**	(CENTER POINT CY)
INPUT	(NEXT CONNECTION)
[OK]	

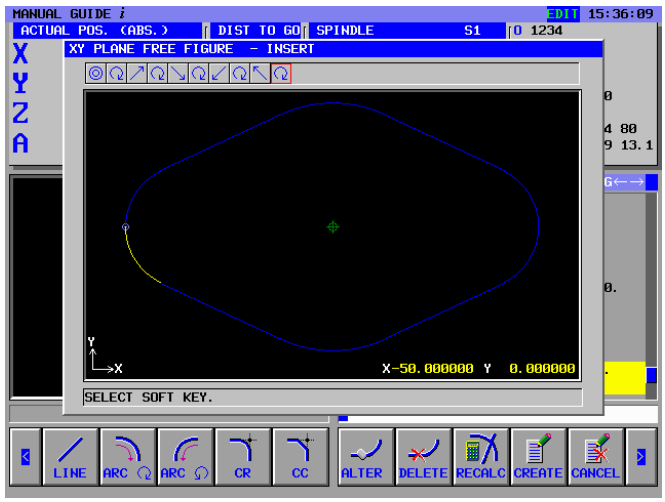
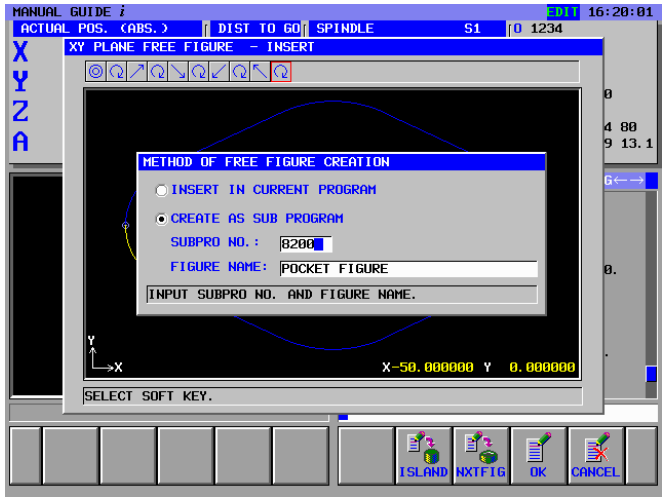


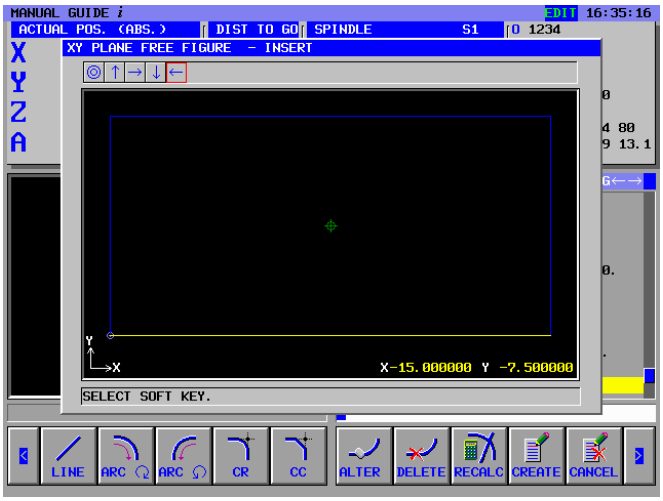
Figure blocks can be registered into the current part program directly, and also can be registered as another sub program.
Registered figure blocks can be used also for finishing, so in this example, register them as a sub program.

(XY PLANE FREE FIGURE - INSERT)
[CREATE]
(METHOD OF FREE FIGURE CREATION)
↓
<CREATE AS SUB PROGRAM>
8200 **INPUT** (SUBPRO NO.)
POCKET FIGURE (FIGURE NAME)
[ISLAND]



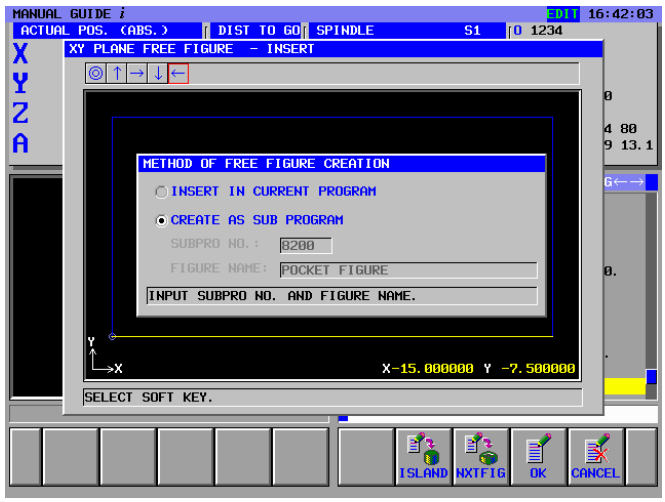
There is an island in a pocket, so enter island figure continuously.

(START POINT - INSERT)	
INPUT	(FIGURE TYPE)
-15 **INPUT**	(START POINT X)
-7.5 **INPUT**	(START POINT Y)
0 **INPUT**	(BASE POSITION)
-10 **INPUT**	(HEIGHT / DEPTH)
[OK]	
[LINE]	
(LINE - INSERT)	
[UP]	(LINE DIRETION)
7.5 **INPUT**	(END POINT Y)
[OK]	
[LINE]	
(LINE - INSERT)	
[RIGHT]	(LINE DIRECTION)
15 **INPUT**	(END POINT X)
[OK]	
[LINE]	
(LINE - INSERT)	
[DOWN]	(LINE DIRECTION)
-7.5 **INPUT**	(END POINT Y)
[OK]	
[LINE]	
(LINE - INSERT)	
[LEFT]	(LINE DIRECTION)
-15 **INPUT**	(END POINT X)
[OK]	



Entered figure blocks were already defined to make as a subprogram, so check the contents of them and create it as a subprogram by pressing [OK]

(XY PLANE FREE FIGURE - INSERT)
[CREATE]
(METHOD OF FREE FIGURE CREATION)
[OK]



NOTE

Registered sub program can be displayed in a figure menu tab, “SUBPROGRAM”.
In this case, set the parameters No14720 to 14723 in advance.
For this example, set those parameters as follow.

No.14720=8000 (Minimum program number of sub programs displayed in the turning machining sub program menu)
No.14721=8499 (Maximum program number of sub programs displayed in the turning machining sub program menu)
No.14722=8500 (Minimum program number of sub programs displayed in the milling machining sub program menu)
No.14723=8999 (Maximum program number of sub programs displayed in the milling machining sub program menu)

3.5.6 Entering Tool Changing and Spindle Rotating Blocks for Finishing Flat End Mill

3.5.6.1 Entering in ISO-code form directly

For pocket finishing, enter commands for operations including changing tools, specifying the spindle, and approaching the machining start point.

G28 G91 Z0. ; **INSERT**	(Z-axis reference position return)
G28 G91 X0. Y0. ; **INSERT**	(X/Y-axis reference position return)
T02 M06 ; **INSERT**	(Tool change)
D2 ; **INSERT**	(Selecting tool data)
M03 S1500 ; **INSERT**	(Spindle rotation)
G00 G90 G54 X0. Y0. ; **INSERT**	(X/Y-axis approaching)
G43 Z100. H2 ; **INSERT**	(Z-axis approaching)

3.5.6.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. The already prepared fixed form sentence as 3.MILLING START was made with entering undefined data as “?”, so it can be used also for finishing.

(Soft-key group for milling cycle menu)
[START]
(INSERT STARTING COMMAND FOR MILLING)
<START>
↓
<<3.MILLING START>>
[INSERT]

NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.
Place the cursor to the address with “?”, enter numeric data, then push “ALTER”.
After altering the data, return the cursor to the EOB at the end of the program.

3.5.7 Entering Pocket Bottom and Side Finishing Process

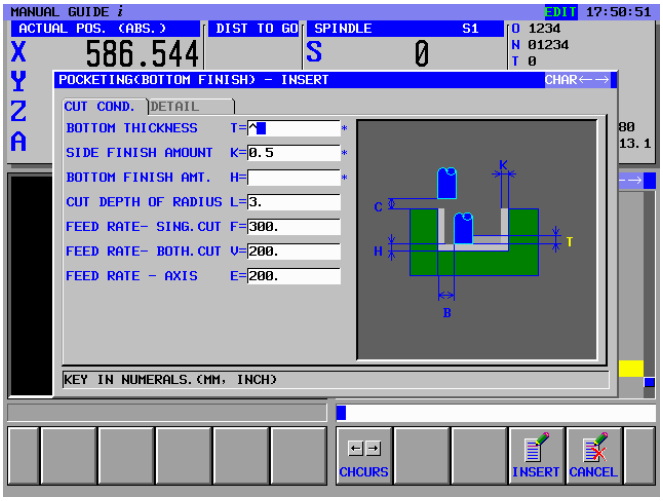
3.5.7.1 Entering pocket bottom finishing cycle block

Enter 3rd process : pocket bottom finishing process by the roughing flat end mill (T01).
Enter machining type, cutting condition and so on.

(Soft-key menu for milling cycle menu)
[CYCLE]
(INSERT MILLING CYCLE)
→
<POCKETING>
↓
<<2.POCKETING (BOTTOM FINISH)>>
[SELECT]
(POCKETING (BOTTOM FINISH) - INSERT)
<CUT COND.>

INPUT	(BOTTOM THICKNESS)
0.5 **INPUT**	(SIDE FINISH AMOUNT)
INPUT	(BOTTOM FINISH AMT.)
3 **INPUT**	(CUT DEPTH OF RADIUS)
300 **INPUT**	(FEED RATE - SING.CUT)
200 **INPUT**	(FEED RATE - BOTH.CUT)
200 **INPUT**	(FEED RATE - AXIS)

[INSERT]

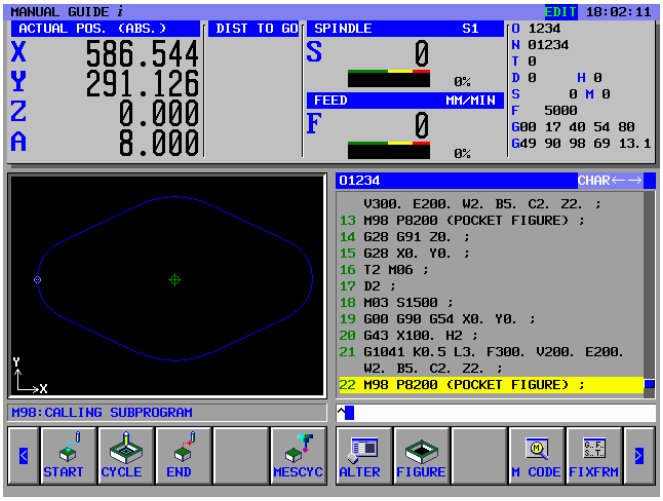


NOTE
1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 In the cycle machining data menu window, 2 tabs, <CUT COND.> and <DETAIL>, are displayed.
In the <CUT COND.>, all data must be set. In the <DETAIL>, all data are automatically set, so check those data and modify them if necessary only

3.5.7.2 Entering figure for pocket bottom finishing

By inserting the cycle block, the window of pocketing figure is displayed, so enter the figure for finishing. But, the former registered figure blocks for roughing can be used, so select from the subprogram menu.

(INSERT MILLING FIGURE)
→
<SUBPROGRAM>
↓
<<08200: POCKET FIGURE>>
[SELECT]



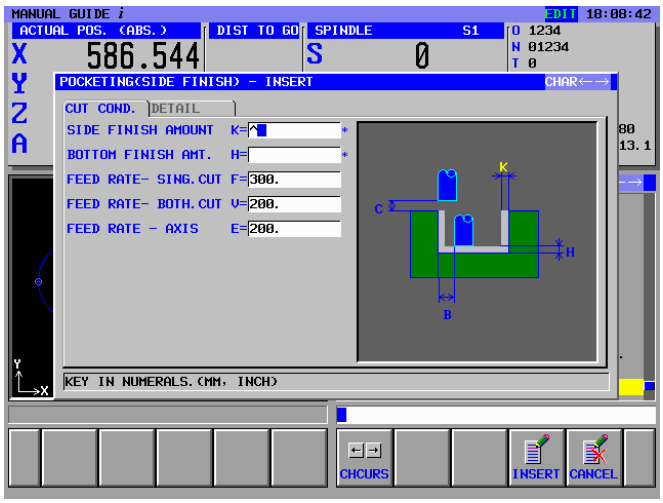
3.5.7.3 Entering pocket side finishing cycle block

Enter 3rd process : pocket side and bottom finishing process by the roughing flat end mill (T01).
Enter machining type, cutting condition and so on.

(Soft-key menu for milling cycle menu)
[CYCLE]
(INSERT MILLING CYCLE)
→
<POCKETING>
↓
<<3.POCKETING (SIDE FINISH)>>
[SELECT]
(POCKETING (SIDE FINISH) - INSERT)
<CUT COND.>

INPUT	(SIDE FINISH AMOUNT)
INPUT	(BOTTOM FINISH AMT.)
300 **INPUT**	(FEED RATE - SING.CUT)
200 **INPUT**	(FEED RATE - BOTH.CUT)
200 **INPUT**	(FEED RATE - AXIS)

[INSERT]

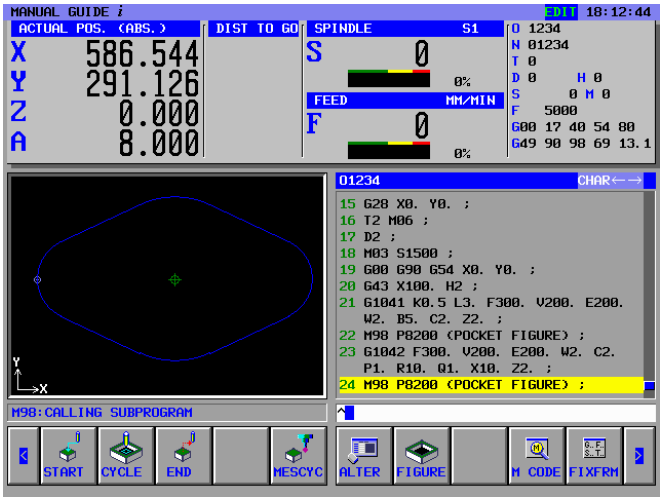


NOTE
1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 When you select the cycle machining type form cycle machining menu, instead of selecting by the cursor, you can use direct inputting of item number and INPUT. (In this case, enter 3 INPUT)

3.5.7.4 Entering figure for pocket side finishing

By inserting the cycle block, the window of pocketing figure is displayed, so enter the figure for finishing. But, the former registered figure blocks for roughing can be used, so select from the subprogram menu.

(INSERT MILLING FIGURE)
→
<SUBPROGRAM>
↓
<<08200: POCKET FIGURE>>
[SELECT]



3.5.8 Entering Tool Changing and Spindle Rotating Blocks for Drill

3.5.8.1 Entering in ISO-code form directly

For drilling, enter commands for operations including changing tools, specifying the spindle, and approaching the machining start point.

G28 G91 Z0. ; **INSERT**	(Z-axis reference position return)
G28 G91 X0. Y0. ; **INSERT**	(X/Y-axis reference position return)
T03 M06 ; **INSERT**	(Tool change)
D3 ; **INSERT**	(Selecting tool data)
M03 S800 ; **INSERT**	(Spindle rotation)
G00 G90 G54 X0. Y0. ; **INSERT**	(X/Y-axis approaching)
G43 Z100. H3 ; **INSERT**	(Z-axis approaching)

3.5.8.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. The already prepared fixed form sentence as 3.MILLING START was made with entering undefined data as “?”, so it can be used also for finishing.

(Soft-key group for milling cycle menu)
[START]
(INSERT STARTING COMMAND FOR MILLING)
<START>
↓
<<3.MILLING START>>
[INSERT]

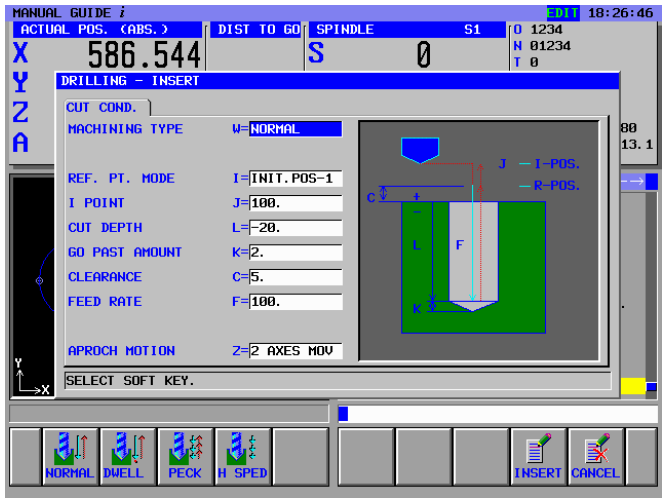
NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.
Place the cursor to the address with “?”, enter numeric data, then push “ALTER”.
After altering the data, return the cursor to the EOB at the end of the program.

3.5.9 Entering Drilling Process

3.5.9.1 Entering drilling cycle block

Enter 4th process : Drilling by drill (T0303).
Enter machining type, cutting condition and so on.

(Soft-key menu for milling cycle menu)	
[CYCLE]	
(INSERT MILLING CYCLE)	
<HOLE MACH.>	
↓	
<<2.DRILLING>>	
[SELECT]	
(DRILLING - INSERT)	
<CUT COND.>	
INPUT	(MACHINING TYPE)
INPUT	(REF.PT.MODE)
100 **INPUT**	(I POINT)
-20 **INPUT**	(CUT DEPTH)
2 **INPUT**	(GO PAST AMOUNT)
5 **INPUT**	(CLEARANCE)
100 **INPUT**	(FEED RATE)
[INSERT]	



NOTE

1 In the cycle machining data menu window, all data excepting cutting condition data are set automatically. However, the data entered at previously entered cycle of same kind are copied, so you must enter the data if you have not entered the same kind of cycle.
2 When you select the cycle machining type form cycle machining menu, instead of selecting by the cursor, you can use direct inputting of item number and INPUT. (In this case, enter 2 INPUT)

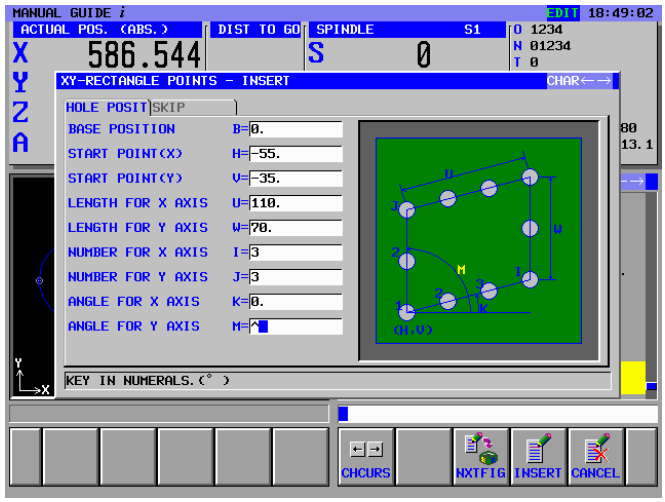
3.5.9.2 Entering hole position block

By inserting the drilling cycle block, the window of hole position menu is displayed, so select the “YY-RECTANGLE POINTS” item.

(INSERT MILLING FIGURE)
<HOLE POSI.>
↓
↓
<<5.XY-RECTANGLE POINTS>>
[SELECT]
(XY-RECTANGLE POINTS - INSERT)
<HOLE POSIT>

0 **INPUT**	(BASE POSITION)
-55 **INPUT**	(START POINT (X))
-35 **INPUT**	(START POINT (Y))
110 **INPUT**	(LENGTH FOR X AXIS)
70 **INPUT**	(LENGTH FOR Y AXIS)
3 **INPUT**	(NUMBER FOR X AXIS)
3 **INPUT**	(NUMBER FOR Y AXIS)
0 **INPUT**	(ANGLE FOR A AXIS)
90 **INPUT**	(ANGLE FOR Y AXIS)

[INSERT]



3.5.10 Operations in the “END” Menu

All necessary machining program have been entered, so enter end procedure.

3.5.10.1 Entering in ISO-code form directly

Enter blocks for spindle stop, releasing and end M-code in ISO-code form with G-code and son on.

M05. ; **INSERT**	(Spindle stop)
G00 G90 Z100.. ; **INSERT**	(Releasing motion)
G28 G91 Z0. ; **INSERT**	(Reference position return)
M06 T0 ; **INSERT**	(Tool changing)
M30 ; **INSERT**	(End M-code)

3.5.10.2 Entering by fixed form sentence menu

ISO-code form part program can be entered from the fixed form sentence menu. But, in such a case, proper fixed form sentence must be prepared in advance.

(Soft-key group for Milling cycle menu)
[END]
<END>
↓
<<1.END OF PROGRAM>>
[INSERT]

NOTE
There may be a case that undefined value is entered by “?” in the fixed form sentence menu, so in this case, you need to replace the “?” by proper value can be used in actual machining.

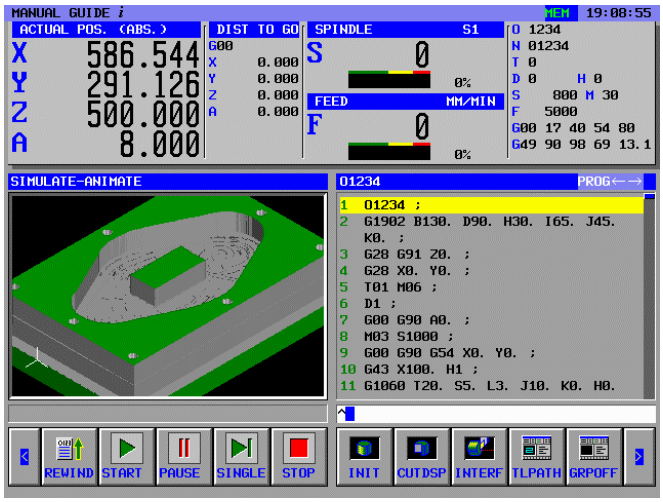
3.6 CHECKING OF THE PART PROGRAM

You can check the entered part program by animation.

3.6.1 Checking by Animation

Select MEM mode by using a mode-selecting switch on a machine-operating panel
[SIMLAT]
(SIMULATE - ANIMATE)
[REWIND]
[START]

NOTE
After checking by animation, in order to do other operation, you must close the animation window by pushing [GRPOFF] always.



APPENDIX

A PARAMETERS

> ⚠ **WARNING**
> Be sure to use the parameters set by the machine tool builder.
> If you change the setting of a parameter, the machining program may not work correctly.
> If the machining program does not work correctly, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

A.1 REQUIRED PARAMETERS

A.1.1 Parameters Required for Basic Options

To use MANUAL GUIDE *i*, be sure to set the following parameters:

(1) No.8701#4 = 1
Read of "vacant" P code macro variables is enabled.
(In Series 30*i*, this parameter is not necessary.)

(2) No.3201#6 = 1
Program registration does not end with M end codes (M02, M30, and M99).

(3) No.8650#0 = 1
When the reset key is pressed, C-EXE passes a key code to an application program.

(4) No.8701#6 = 1
When the TV check is set to ON, edit operations are not regarded as errors.
(In Series 30*i*, this parameter is not necessary.)

(5) No.3112#0 (SGD) = 0
This parameter enables or disables servo waveform display. When this parameter is set to 1, other graphic functions become unavailable.
(In Series 30*i*, this parameter is not necessary.)

(6) No.3103#3 = 1
In the FS160i and NC systems connected to a personal computer (with an Ethernet board or HSSB board), NC software reads the [HELP] key and changes screens even when a C-EXE screen is being displayed.
To suppress this, set bit 3 of parameter No. 3103 to 1. The NC software will then ignore any press of the [HELP] key while a C-EXE screen is being displayed.
(In Series 30*i*, this parameter is not necessary.)

(7) No.9000#0 = 0
This parameter is for debugging machining macro program.
And if this parameter is set to 1, Manual Guide I does not work correctly.

(8) No.3106#6 = 1
The function of C language executor refers to this parameter. If this parameter is set to 1, Slant face machining does not work correctly.
(The system checks this parameter in the case that three-dimensional coordinate conversion is available.)

(9) No.8650#1 = 1
During display of the screen of C language executor, Alarm screen is not changed when the alarm occurs.

(10) No.9000#7 = 0
If this bit is set, MANUAL GUIDE *i* does not update the screen display at the time of operation, and cycle execution is not performed normally.

(11) No.8650#2 = 1
When the standard MDI key for Series 30i, please sure to set to ON.
(In Series 16*i*/18*i*/21*i*, this parameter is not necessary.)

(12) No.14853#7 = 1
The program window screen in machining based on the new specifications is used. (Scroll bar is displayed)
(In Series 16*i*/18*i*/21*i*, this parameter is not necessary.)

A.1.2 Parameters Required for Machining Simulation

To perform machining simulation, be sure to set the following parameters:

(1) No.14706≠0
Directions of the three basic axes of spindle 1
Conditions)
- One-path T series, M series, and CNCs for complex machining
- Two-path T series
  1) For path 1, always set this parameter.
  2) For path 2
     When bit 0 (SME) of parameter No. 27401 is set to 1
- Three-path T series
  1) For path 1, always set this parameter.
  2) For path 2
     When bit 0 (SME) of parameter No. 27401 is set to 1
  3) For path 3
     When bit 0 (TME) of parameter No. 27402 is set to 1

(2) No.14707≠0
Directions of the three basic axes of spindle 2
Conditions)
When bit 1 (SUB) of parameter No. 14702 is set to 1 (A sub-spindle is provided.) :
- For the one-path T series, always set this parameter.
- For the two-path T series:
  1) For path 1
     When bit 1 (FSE) of parameter No. 14701 is set to 1
  2) For path 2
     When bit 1 (SSE) of parameter No. 27401 is set to 1
- For the three-path T series:
  1) For path 1
     When bit 1 (FSE) of parameter No. 14701 is set to 1
  2) For path 2
     When bit 1 (SSE) of parameter No. 27401 is set to 1
  3) For path 3
     When bit 1 (TSE) of parameter No. 27402 is set to 1.

A.1.3 Parameters Required for Other Options except Basic Option (For Series 30*i* only)

To use other optional function except Basic option in Series 30*i*, be sure to set the following parameters:

(1) No.9071 ≠ 0
This parameter is set to P-CODE Macro number of MANUAL GUIDE *i* as follows.

Machining Center := 90
Lathe := 91

(2) No.9072 ≠ 0
This parameter is set to the block number for dealing with Macro sentence continuously in Execute Macro program. MANUAL GUIDE *i* recommend it to 1.

A.2 BASIC PARAMETERS

A.2.1 Settings for the Color Palette for Screen Display (No.2)

These parameters set the colors used to display screen components.
Color setting data for a screen display color number* (1 to 16)

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14480	DSPCOL17

DSPCOL17 : Color of the special character.
If the value is 0, the color of the special character is red (630000).

A.2.2 Parameters for Operations in General

	#7	#6	#5	#4	#3	#2	#1	#0
14700	MGI				CS2	CS1		PWD

PWD 0 : When the power is turned on, the system is not switched to the Manual Guide screen.
1 : When the power is turned on, the system is switched to the Manual Guide screen.

CS1= 0, CS2 = 0 : When the power is turned on, the custom screen is not displayed.
CS1= 1, CS2 = 0 : When the power is turned on, Custom Screen 1 (AUX) is displayed.
CS1= 0, CS2 = 1 : When the power is turned on, Custom Screen 3 (MCR) is displayed.
CS1= 1, CS2 = 1 : When the power is turned on, Custom Screen 2 (MENU) is displayed.

MGI 0 : MANUAL GUIDE *i* is enabled.
1 : MANUAL GUIDE *i* is disabled.

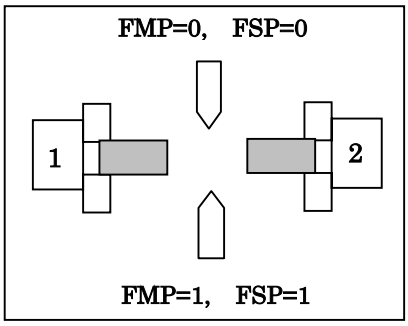
	#7	#6	#5	#4	#3	#2	#1	#0
14701		PCK	CLP2	CLP1	FSP	FMP	FSE	

FSE 0 : Tool post 1 cannot be used with spindle 2.
1 : Tool post 1 can be used with spindle 2.

NOTE
FSE is available in the case of No.14702 SUB(#1) = 1 only.

FMP 0 : Tool post 1 is placed above spindle 1.
1 : Tool post 1 is placed under spindle 1.

FSP 0 : Tool post 1 is placed above spindle 2.
1 : Tool post 1 is placed under spindle 2.



NOTE
FSP is available in the case of No.14702 SUB(#1) = 1 and No.14701 FSE(#1) = 1 only.

CLP1= 0, CLP2 = 0 : The size of the clipboard is set to 1024 bytes.
CLP1= 1, CLP2 = 0 : The size of the clipboard is set to 2048 bytes.
CLP1= 0, CLP2 = 1 : The size of the clipboard is set to 4096 bytes.
CLP1= 1, CLP 2 = 1 : The size of the clipboard is set to 8192 bytes.

PCK 0 : Process List Editing Function : The soft-key [ADD /] and [DEL /] for program check function for each spindle are not displayed.
1 : Process List Editing Function : The soft-key [ADD /] and [DEL /] for program check function for each spindle are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
14702	SFA	SFB	SFC	SFD			SUB	MT1

MT1 0 : Vertical.
1 : Horizontal (chuck located on the left side).

SUB 0 : No sub-spindle is provided.
1 : A sub-spindle is provided.

SFD 0 : Normal rotation is assumed if G266#5 (SFRD)=0 and G266#4 (SRVD)=1
Reverse rotation is assumed if G266#5 (SFRD)=1 and G266#4 (SRVD)=0
1 : Normal rotation is assumed if G266#5 (SFRD)=1 and G266#4 (SRVD)=0
Reverse rotation is assumed if G266#5 (SFRD)=0 and G266#4 (SRVD)=1

SFC 0 : Normal rotation is assumed if G204#5 (SFRC)=0 and G204#4 (SRVC)=1
Reverse rotation is assumed if G204#5 (SFRC)=1 and G204#4 (SRVC)=0
1 : Normal rotation is assumed if G204#5 (SFRC)=1 and G204#4 (SRVC)=0
Reverse rotation is assumed if G204#5 (SFRC)=0 and G204#4 (SRVC)=1

SFB 0 : Normal rotation is assumed if G074#5 (SFRB)=0 and G074#4 (SRVB)=1
Reverse rotation is assumed if G074#5 (SFRB)=1 and G074#4 (SRVB)=0
1 : Normal rotation is assumed if G074#5 (SFRB)=1 and G074#4 (SRVB)=0
Reverse rotation is assumed if G074#5 (SFRB)=0 and G074#4 (SRVB)=1

SFA 0 : Normal rotation is assumed if G070#5 (SFRA)=0 and G070#4 (SRVA)=1
Reverse rotation is assumed if G070#5 (SFRA)=1 and G070#4 (SRVA)=0
1 : Normal rotation is assumed if G070#5 (SFRA)=1 and G070#4 (SRVA)=0
Reverse rotation is assumed if G070#5 (SFRA)=0 and G070#4 (SRVA)=1

	#7	#6	#5	#4	#3	#2	#1	#0
14703		G62	NCC	TAB	LST	GDM	SFN	FDS

FDS 0 : During feed per revolution, actual feedrate is displayed as that of feed per minute on the base screen.
1 : During feed per revolution, actual feedrate is displayed as that of feed per revolution on the base screen. (Refer to “3. All in one screen”.)

SFN 0 : In lower part of the soft keys, shortcut numbers are not displayed.
1 : In lower part of the soft keys, shortcut numbers are displayed.

GDM 0 : When the CNC is a CNC for complex machining, just one guidance message is displayed according to the T or M mode.
1 : When the CNC is a CNC for complex machining, guidance messages for the T and M modes are both displayed regardless of the mode set. (Refer to “3. All in one screen”.)

LST 0 : The soft-key [EDTCEL] is not displayed.
1 : The soft-key [EDTCEL] is displayed.

TAB 0 : On the multiple-tab screen, the cursor does not automatically move to the next tab when [INPUT] is pressed in the lowermost edit box.
1 : On the multiple-tab screen, the cursor automatically moves to the next tab when [INPUT] is pressed in the lowermost edit box.

NCC 0 : In NC statement conversion, 4-digit G code cycle machining commands before expansion are output as comments.
1 : In NC statement conversion, 4-digit G code cycle machining commands before expansion are not output as comments.

G62 0 : The capability of allowing the G62.4 signal to make the screen display visible or invisible is disabled.
1 : The capability of allowing the G62.4 signal to make the screen display visible or invisible is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
14704	DXC	DZX	DXY	IJK	LDM	SLM	PWO	PTO

PTO 0 : During operation, data entry is prohibited on the tool offset setting screen.
1 : During operation, data entry is not prohibited on the tool offset setting screen.

PWO 0 : During operation, data entry is prohibited on the workpiece origin offset setting screen.
1 : During operation, data entry is not prohibited on the workpiece origin offset setting screen.

SLM 0 : The spindle load meter is displayed.
1 : The spindle load meter is not displayed.

LDM 0 : The servo load meter is displayed.
1 : The servo load meter is not displayed.

IJK 0 : The arc command in a contour program is output in IJK format.
1 : The arc command in a contour program is output in R format.
Remark) contour program : Refer to “Entering Contour Programs”.

DXY 0 : The X coordinate in the XY plane contour program is output as a radius value.
1 : The X coordinate in the XY plane contour program is output as a diameter value.

DZX 0 : The X coordinate in the ZX plane contour program is output as a radius value.
1 : The X coordinate in the ZX plane contour program is output as a diameter value.

DXC 0 : The X coordinate in the XC plane contour program is output as a radius value.
1 : The X coordinate in the XC plane contour program is output as a diameter value.

NOTE
When Diameter programming is used in the lathe, please be sure to set DXY, DZX and DXC to 1. (The above case is the parameter No.1006 #3(DIA) = 1.)

A.2.3 Parameters for the Axial Configuration of the Machine

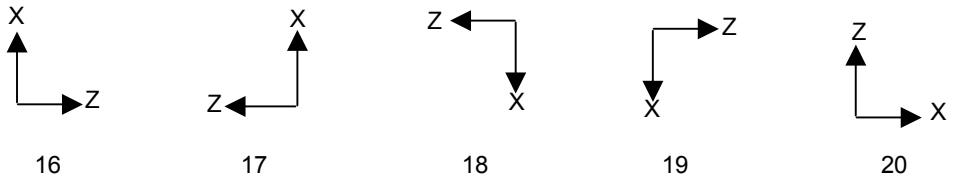
These parameters set the axial configuration of the machine. (Used in machining simulation.)

14706	DRCTS1

(FANUC standard settings = 20 or 16)

DRCTS 1 : Number of Workpiece coordinate for main spindle

16 : Right-handed coordinate system, right = +Z, up = +X
17 : Right-handed coordinate system, right = -Z, up = +X
18 : Right-handed coordinate system, right = -Z, up = -X
19 : Right-handed coordinate system, right = +Z, up = -X
20 : Right-handed coordinate system, right = +X, up = +Z



FANUC standard settings = 20 :
Vertical machining center/vertical lathe
FANUC standard settings = 16 :
Horizontal machining center/horizontal lathe

14707	DRCTS2

(FANUC standard settings = 0 : Not use)

DRCTS 2 : Number of Workpiece coordinate for sub spindle if existing

16 : Right-handed coordinate system, right = +Z, up = +X
17 : Right-handed coordinate system, right = -Z, up = +X
18 : Right-handed coordinate system, right = -Z, up = -X
19 : Right-handed coordinate system, right = +Z, up = -X
20 : Right-handed coordinate system, right = +X, up = +Z

A.2.4 Settings for Spindle Status Display

These parameters set spindle status display on the base screen.

14710	AST

(FANUC standard settings = 0)

AST 0 : When a CNC unit for complex machining is used, actual spindle speed/spindle load ratio/spindle status display on the base screen is not automatically switched in turning mode.

≠0 : Spindle number: When a CNC unit for complex machining is used, actual spindle speed/spindle load ratio/spindle status display on the base screen is automatically switched to the display for a specified spindle number in turning mode.

14711	ASM

(FANUC standard settings = 0)

ASM 0 : When a CNC unit for complex machining is used, actual spindle speed/spindle load ratio/spindle status display on the base screen is not automatically switched in milling mode.

≠0 : Spindle number: When a CNC unit for complex machining is used, actual spindle speed/spindle load ratio/spindle status display on the base screen is automatically switched to the display for a specified spindle number in milling mode.

A.2.5 Setting for the Display Language

This parameter sets the display language on the Manual Guide *i* screen.

14712	MSGLANG

(FANUC standard settings = 0)

MSGLANG 0 : The setting of language which NC displays is assumed. But if it is not any other than English, Japanese, German, French, Italian, Spanish, Swedish, Czech, Portuguese or Polish, MANUAL GUIDE *i* is displayed in English.

1 : English
2 : Japanese
3 : German
4 : French
5 : Italian
6 : Spanish
7 : Swedish
8 : Czech
9 : Portuguese
10 : Polish
except above : English

NOTE
Language file 1 is necessary to display Japanese, Germany, French or Italian.
And language file 2 is necessary to display Spanish, Czech, Portuguese or Polish.

A.2.6 Settings for Graphic Display

These parameters set graphic display.

14713	GRPSCALE

(FANUC standard settings = 0)

GRPSCALE : Scaling unit for graphic enlargement and reduction.
Scaling unit = 64/GRPSCALE
(Valid data range: 0 to 255. If 0 is set, 64 is assumed.)

14714	GRPMOVEH

(FANUC standard settings = 0)

GRPMOVEH : Graphic horizontal movement unit (dots)
(Valid data range: 0 to 255. If 0 is set, 64 dots is assumed.)

14715	GRPMOVEV

(FANUC standard settings = 0)

GRPMOVEV : Graphic vertical movement unit (dots)
(Valid data range: 0 to 255. If 0 is set, 35 dots is assumed.)

14716	GRPROTA

(FANUC standard settings = 0)

GRPROTA : Graphic rotation direction movement unit (angle)
(Valid data range: 0 to 255. If 0 is set, 10 degrees is assumed.)

A.2.7 Settings for Machining Simulation Axes

These parameters set machining simulation.

14717	SMLCNO

(FANUC standard settings = 0)

SMLCNO : Rotate (Cs) axis number
Valid data range: from 0 to the number of controlled axes

NOTE
1 In case of one Cs axis of main spindle, please be sure to set No.14717 only. If no Cs axis or Animation option, it is unnecessary to set.
2 In case of two Cs axis with main and sub spindle, please be sure to set No.27301, No.27302 and No.27312 #0. And No.14717 is used as working number for Cs axis, as follows.
3 When 27312#0 = 1, No. 14717 is rewritten to the value of No.27301 or No.27302 by a spindle selection command (G1998). Therefore, in the case of No.27312 #0=1, please be sure to specify Spindle selection command (G1998) before Milling cycles.
4 As to Apindle selection command (G1998), please refer to “9.5 Setting of data for Animation”.

14718	SMLRTNO

(FANUC standard settings = 0)

SMLRTNO : Rotate (Cs) axis number, which tilts tool head or table
Valid data range: from 0 to the number of controlled axes

NOTE
No.14718 is used in Simulation or Turning cycles. Therfore, if the machine has a rotate axis which tilts tool head or table, please sure to set this parameter.
If no such axis or animation option, it is not necessary to set.

A.2.8 Settings for Subprogram Selection Screens

These parameters set the registration start/end numbers of subprogram selection screens.

14720	TFIGSNO

(FANUC standard settings = 0)

TFIGSNO : Registration start number of the turning subprogram selection screen.

14721	TFIGENO

(FANUC standard settings = 0)

TFIGENO : Registration end number of the turning subprogram selection screen.

14722	MFIGSNO

(FANUC standard settings = 0)

MFIGSNO : Registration start number of the milling subprogram selection screen.

14723	MFIGENO

(FANUC standard settings = 0)

MFIGENO : Registration end number of the milling subprogram selection screen.

A.2.9 Settings for the Color Palette for Screen Display

These parameters set the colors used to display screen components.
Color setting data for a screen display color number* (1 to 16)

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14724	DSPCOL1

DSPCOL1 : Color of the cursor in the program display section on the base screen.

14725	DSPCOL2

DSPCOL2 : Used to display alarms in the system title display section.

14726	DSPCOL3

DSPCOL3 : Used to display the mode and line number in the system title display section.

14727	DSPCOL4

DSPCOL4 : Used to display the background of the alarm indication and the load meter.

14728	DSPCOL5

DSPCOL5 : Used to display the background of the title in the status display section, soft key characters, item name characters, and the scroll box of the scroll bar.

14729	DSPCOL6

DSPCOL6 : Used to display the serial numbers of selection screens.

14730	DSPCOL7

DSPCOL7 : Used to display the background of the system title display section.

14731	DSPCOL8

DSPCOL8 : Used to display characters in the title section.

14732	DSPCOL9

DSPCOL9 : Used to display the mode on the base screen and the material elements of arbitrary figures.

14733	DSPCOL10

DSPCOL10 : Used to display frames.

14734	DSPCOL11

DSPCOL11 : Used to display cells that cannot be edited by the process list edit function.

14735	DSPCOL12

DSPCOL12 : Used to display the background of the status display section.

14736	DSPCOL13

DSPCOL13 : Used to display the bar of the load meter.

14737	DSPCOL14

DSPCOL14 : Used to display the background of the cursor mode.

14738	DSPCOL15

DSPCOL15 : Used to display window shadows.

14739	DSPCOL16

DSPCOL16 : Used to display the background of the base screen.

If these parameters are set to 0, the following values are used as their respective initial values.

No.14724 = 636300 Yellow
No.14725 = 630000 Red
No.14726 = 003200 Green
No.14727 = 636300 Yellow
No.14728 = 000063 Blue
No.14729 = 420042 Purple (pinkish)
No.14730 = 323260 Light ultramarine
No.14731 = 636363 White
No.14732 = 163616 Bright green
No.14733 = 000000 Black
No.14734 = 121212 Very dark gray
No.14735 = 484848 Bright gray
No.14736 = 006363 Bright, light blue
No.14737 = 20203C Light ultramarine
No.14738 = 242424 Dark gray
No.14739 = 404040 Rather bright gray

A.2.10 Settings for the Color Palette for Icon Display

These parameters set the color palette colors used to display icons.
Color setting data for an ICOCOL* screen display color number* (1 to 16)

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14740	ICOCOL1
14741	ICOCOL2
14742	ICOCOL3
14743	ICOCOL4
14744	ICOCOL5
14745	ICOCOL6
14746	ICOCOL7
14747	ICOCOL8
14748	ICOCOL9
14749	ICOCOL10
14750	ICOCOL11
14751	ICOCOL12
14752	ICOCOL13
14753	ICOCOL14
14754	ICOCOL15
14755	ICOCOL16

If these parameters are set to 0, the following values are used as their respective initial values.

No.14740 = 630000	Red
No.14741 = 003200	Green
No.14742 = 636300	Yellow
No.14743 = 000063	Blue
No.14744 = 420042	Purple
No.14745 = 480040	Dark pink
No.14746 = 636363	White
No.14747 = 163616	Bright green
No.14748 = 000000	Black
No.14749 = 006060	Bright, light blue
No.14750 = 484848	Bright gray
No.14751 = 006363	Bright blue
No.14752 = 320000	Dark red
No.14753 = 242424	Dark gray
No.14754 = 404040	Rather bright gray
No.14755 = 000000	Black

A.2.11 Settings for the Color Palette for Guide Display

These parameters set the colors used to display guides.
Color setting data for a GIDCOL* screen display color* (1 to 16)

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14756	GIDCOL1

GIDCOL 1 : Material color.

14757	GIDCOL2

GIDCOL2 : Material frame color.

14758	GIDCOL3

GIDCOL3 : Tool color.

14759	GIDCOL4

GIDCOL4 : Tool frame color.

14760	GIDCOL5

GIDCOL5 : Tool path (cutting feed).

14761	GIDCOL6

GIDCOL6 : Tool path (rapid traverse).

14762	GIDCOL7

GIDCOL7 : Dimensional line.

14763	GIDCOL8

GIDCOL8 : Character color (unselected character).

14764	GIDCOL9

GIDCOL9 : Character color (selected character).

14765	GIDCOL10

GIDCOL10 : Portion to be cut.

14766	GIDCOL11

GIDCOL11 : Finishing allowance.

14767	GIDCOL12

GIDCOL12 : Reserved.

14768	GIDCOL13

GIDCOL13 : Reserved.

14769	GIDCOL14

GIDCOL14 : Explanation of measurement (static sentence)

14770	GIDCOL15

GIDCOL15 : Explanation of measurement (dynamic sentence)

14771	GIDCOL16

GIDCOL16 : Background color.

If these parameters are set to 0, the following values are used as their respective initial values.

No.14756 = 003200	Green
No.14757 = 000063	Blue
No.14758 = 000063	Blue
No.14759 = 006060	Light blue
No.14760 = 006060	Light blue
No.14761 = 600000	Red
No.14762 = 000063	Blue
No.14763 = 000063	Blue
No.14764 = 636300	Yellow
No.14756 = 484848	Bright gray
No.14766 = 404040	Rather bright gray
No.14767 = 000000	Black (reserved)
No.14768 = 000000	Black (reserved)
No.14769 = 636363	White
No.14770 = 006060	Light blue
No.14771 = 242424	Dark gray

A.2.12 Settings for Tool Path Drawing Colors

These parameters set the tool path drawing colors.

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14773	DATA

DATA : Color of blank figure during tool path drawing.
If the value is set to 0, Green is used as the respective initial values (003200).

A.2.13 Settings for Machining Simulation Animation Colors

These parameters set machining simulation animation colors.

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14777	ANMCOL1

ANMCOL1 : Color of part figures during animation.

14778	ANMCOL2

ANMCOL2 : Color of material cut-off portions during animation.

14779	ANMCOL3

ANMCOL3 : Color of the tool during animation.

14780	ANMCOL4

ANMCOL4 : Color of the coordinate axes during animation.

14781	ANMCOL5

ANMCOL5 : Color of the background during animation.

If these parameters are set to 0, their standard colors are assumed.

A.2.14 Settings for Path Colors During Tool Path Plotting

These parameters set the path colors used during tool path plotting.

- Specify color setting data with a 6-digit number in the format of "xxyyzz".
  (xx:Value for red, yy:Value for green, zz:Value for blue)
- The valid data range of each color value is 0 to 63. A value greater than 63 is interpreted as 63.
- For a number with less than six digits, all the unspecified digits are interpreted as high-order ones and are all assumed 0.

14785	PATHCOL1

PATHCOL1 : Color of the path of a rapid traverse tool.

14786	PATHCOL2

PATHCOL2 : Color of the path of a cutting feed tool.

14787	PATHCOL3

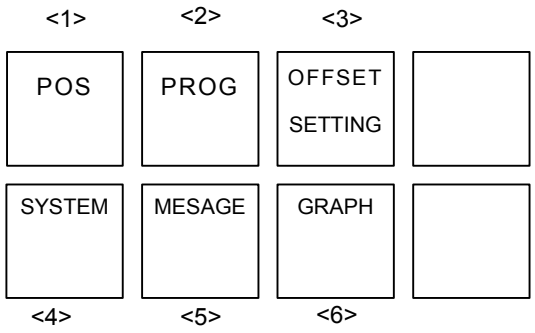
PATHCOL3 : Color of the path of a threading tool.

If these parameters are set to 0, their standard colors are assumed.

A.2.15 Settings for the Allocation of Startup Function Keys

	#7	#6	#5	#4	#3	#2	#1	#0
14794			GRP	MES	SYS	OFS	PRG	POS

POS 0 : The Manual Guide does not start when function key <1> is pressed.
1 : The Manual Guide starts when function key <1> is pressed.
PRG 0 : The Manual Guide does not start when function key <2> is pressed.
1 : The Manual Guide starts when function key <2> is pressed.
OFS 0 : The Manual Guide does not start when function key <3> is pressed.
1 : The Manual Guide starts when function key <3> is pressed.
SYS 0 : The Manual Guide does not start when function key <4> is pressed.
1 : The Manual Guide starts when function key <4> is pressed.
MES 0 : The Manual Guide does not start when function key <5> is pressed.
1 : The Manual Guide starts when function key <5> is pressed.
GRP 0 : The Manual Guide does not start when function key <6> is pressed.
1 : The Manual Guide starts when function key <6> is pressed.



	#7	#6	#5	#4	#3	#2	#1	#0
14795		PS3	PS2	PS1	FPT	CS3	CS2	CS1

CS1 0 : The Manual Guide does not start on Custom Screen 1 (AUX) when function key <1> is pressed.
1 : The Manual Guide starts on Custom Screen 1 (AUX) when function key <1> is pressed.

NOTE
1 If the conversational macro screen is not provided, bit 5 of parameter No. 8652 (CMEC1) must be set to 1.
2 This parameter is not supported in Series 30*i*.

CS2 0 : The Manual Guide does not start on Custom Screen 3 (MCR) when function key <1> is pressed.
1 : The Manual Guide starts on Custom Screen 3 (MCR) when function key <1> is pressed.

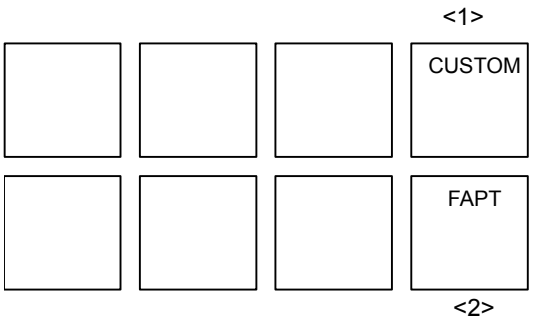
NOTE
1 If the conversational macro screen is not provided, bit 6 of parameter No. 8652 (CMEC2) must be set to 1.
2 This parameter is not supported in Series 30*i*.

CS3 0 : The Manual Guide does not start on Custom Screen 2 (MENU) when function key <1> is pressed.
1 : The Manual Guide starts on Custom Screen 2 (MENU) when function key <1> is pressed.

NOTE
1 If the conversational macro screen is not provided, bit 7 of parameter No. 8652 (CMEC3) must be set to 1.
2 This parameter is not supported in Series 30*i*.

FPT 0 : The Manual Guide does not start when function key <2> is pressed.
1 : The Manual Guide starts when function key <2> is pressed.

NOTE
1 Bit 4 of parameter No. 8652 (CMECF) must be set to 1.
2 This parameter is not supported in Series 30*i*.



PS3=0, PS2=0, PS1=0 : The maximum memory size is set to 250K byte.
PS3=0, PS2=0, PS1=1 : The maximum memory size is set to 500K byte.
PS3=0, PS2=1, PS1=0 : The maximum memory size is set to 1M byte.
PS3=0, PS2=1, PS1=1 : The maximum memory size is set to 2M byte.
PS3=1, PS2=0, PS1=0 : The maximum memory size is set to 4M byte.
PS3=1, PS2=0, PS1=1 : The maximum memory size is set to 5M byte.
PS3=1, PS2=1, PS1=0 : The maximum memory size is set to 6M byte.
PS3=1, PS2=1, PS1=1 : The maximum memory size is set to 7M byte.

NOTE

PS3, PS2 and PS1 are set in the 1 path parameter only.

To specify a maximum allowable memory size greater than 250K bytes in parameter No. 14795, set an appropriate value in parameter No. 8781 (DRAM size that can be used by a C language application).

To increase the DRAM size, the custom capacity option is separately required.

* If the DRAM size is increased by 1M bytes by using parameter No. 8781, about 500,000 characters (about 20,000 blocks) can be increased as a guideline when one block consists of 25 characters on average.

A.2.16 Settings for Current Position Display

14799	DS1AXS

DS1AXS 0 : The first controlled axis is displayed in display area 1.
≠0 : Number of the controlled axis to be displayed in display area 1.

14800	DS2AXS

DS2AXS 0 : The second controlled axis is displayed in display area 2.
≠0 : Number of the controlled axis to be displayed in display area 2.

14801	DS3AXS

DS3AXS 0 : The third controlled axis is displayed in display area 3.
≠0 : Number of the controlled axis to be displayed in display area 3.

14802	DS4AXS

DS4AXS 0 : The fourth controlled axis is displayed in display area 4.
≠0 : Number of the controlled axis to be displayed in display area 4.

14803	DS5AXS

DS5AXS 0 : The fifth controlled axis is displayed in display area 5.
≠0 : Number of the controlled axis to be displayed in display area 5.

14804	DS6AXS

DS6AXS 0 : The sixth controlled axis is displayed in display area 6.
≠0 : Number of the controlled axis to be displayed in display area 6.

14805	DS7AXS

DS7AXS 0 : The seventh controlled axis is displayed in display area 7.
≠0 : Number of the controlled axis to be displayed in display area 7.

14806	DS8AXS

DS8AXS 0 : The eighth controlled axis is displayed in display area 8.
≠0 : Number of the controlled axis to be displayed in display area 8.

A.2.17 Settings for F Load Meter Compensation

Parameters Nos. 14815 to 14822 are independent ones for respective paths.
These parameters are used to compensate a CNC controlled axis to which load is applied constantly, such as a vertical axis, for that load, using the load meter.

14815	ELOFS1

ELOFS1 : Value of the load current of the first CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14816	ELOFS2

ELOFS2 : Value of the load current of the second CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14817	ELOFS3

ELOFS3 : Value of the load current of the third CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14818	ELOFS4

ELOFS4 : Value of the load current of the fourth CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14819	ELOFS5

ELOFS5 : Value of the load current of the fifth CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14820	ELOFS6

ELOFS6 : Value of the load current of the sixth CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14821	ELOFS7

ELOFS7 : Value of the load current of the seventh CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

14822	ELOFS8

ELOFS8 : Value of the load current of the eighth CNC controlled axis in the steady status, as converted into a digital value (-6554 to +6554).

A.2.18 Settings for Tool Management Functions

These parameters are for the settings for tool management functions.

	#7	#6	#5	#4	#3	#2	#1	#0
14823	STS	TLD	LIA	LIF	TMG	MSR	TOF	ORT

ORT 0 : The screen for associating a tool number with a compensation number is not displayed.
1 : The screen for associating a tool number with a compensation number is displayed.

TOF 0 : The per-tool-number tool offset setting screen is not displayed.
1 : The per-tool-number tool offset setting screen is displayed.

MSR 0 : The compensation type, tool number, and group number input fields are not displayed on the measurement screen.
1 : The compensation type, tool number, and group number input fields are displayed on the measurement screen.

TMG 0 : The tool management data setting screen is not displayed.
1 : The tool management data setting screen is displayed.

LIF 0 : The tool life management data setting screen is not displayed.
1 : The tool life management data setting screen is displayed.

LIA 0 : The screen of Tool Life Data List is not displayed.
1 : The screen of Tool Life Data List is displayed.

TLD 0 : The soft-key for changing screen from Manual Guide *i* to Tool Management Data Table is not displayed.
1 : The soft-key for changing screen from Manual Guide *i* to Tool Management Data Table is displayed.

STS 0 : When a compensation type is specified, the compensation type is not displayed in the status display area.
1 : When a compensation type is specified, the compensation type is displayed in the status display area.

14824	OFSRELTL

OFSRELTL : Start compensation number (0 to 999) to be associated with a tool number.

14825	OFSTYPNO

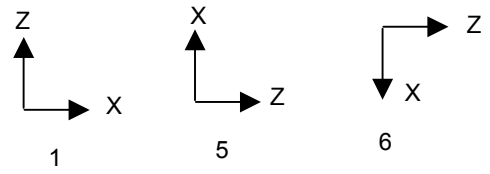
OFSTYPO : Number of compensation types (0 to 9)

A.2.19 Settings for Arbitrary Figures

These parameters are for the settings for arbitrary shapes.

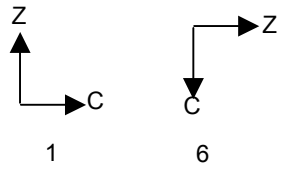
14840	DSPCRDZX

DSPCRDZX : Drawing coordinates when an arbitrary ZX figure is programmed.
= 0 Same effect as that of setting 5.
= 1 Plan view, horizontal axis = +X, vertical axis = +Z
= 5 Plan view, horizontal axis = +Z, vertical axis = +X
= 6 Plan view, horizontal axis = +Z, vertical axis = -X



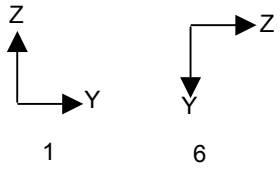
14841	DSPCRDZC

DSPCRDZC : Drawing coordinates when an arbitrary ZC figure is programmed.
= 0 Same effect as that of setting 6.
= 1 Plan view, horizontal axis = +C, vertical axis = +Z
= 6 Plan view, horizontal axis = +Z, vertical axis = -C



14842	DSPCRDYZ

DSPCRDYZ : Drawing coordinates when an arbitrary YZ figure is programmed.
= 0 Same effect as that of setting 6.
= 1 Plan view, horizontal axis = + Y, vertical axis = +Z
= 6 Plan view, horizontal axis = +Z, vertical axis = -Y



A.2.20 Other Parameters

14843

Number of blocks which is used to judge if a subprogram calling "M98 P****" is arbitrary figure data when a cursor is on the block of the subprogram calling in the program-editing screen.

= A positive number Number of blocks
= 0 All of the subprogram
= -1 None of the subprogram (The arbitrary figures of the subprogram are not drawn.)

14844

A cycle time when the date and the time are recorded in an operating history.
The date and the time are recorded in an operating history every the set time. However, the date and the time are not recorded if there is no operating data in the time.
Valid data range : 0 to 1439
Unit of data : 1 [min]

A.2.21 Settings for Operations in General (All Common Path)

These parameters are for the settings for operations in general.

	#7	#6	#5	#4	#3	#2	#1	#0
14850								

#0 0 : In the tool offset window, the tab of [TOOL DATA] is displayed.
1 : In the tool offset window, the tab of [TOOL DATA] is not displayed.

NOTE
#0 is necessary to order the optional function of Milling Cycles or Turning Cycles.

#1 0 : In the workpiece coordinate system window and tool offset window, [CHCURS] can not be changed.
1 : In the workpiece coordinate system window and tool offset window, [CHCURS] can be changed into [TAB→].

#2 0 : The soft-key [RETURN] is displayed on Tool Management Data screen and Manual Measurement screen.
1 : The soft-key [CLOSE] is displayed on Tool Management Data screen and Manual Measurement screen.

#3 0 : Fixed form sentence editing is not disabled with the Memory Protection key
1 : Fixed form sentence editing is disabled with the Memory Protection key

#4 0 : The soft-key [M CODE] is displayed on the basic screen.
1 : The soft-key [M CODE] is not displayed on the basic screen

#5 0 : When a program is opened, a check for an invalid character is not made.
1 : When a program is opened, a check for an invalid character is made. (When bit 5 is set to 1, a longer time is required to open a program.)

#6 0 : When MANUAL GUIDE *i* is used with the function key [PROG], switching from the MGi screen to the NC screen to the MGi screen is performed in mode switching.
1 : When MANUAL GUIDE *i* is used with the function key [PROG], switching from the MGi screen to the NC screen to the MGi screen is not performed in mode switching.

#7 0 : If no modification is made to the program size and time stamp (in minutes) at the time of screen switching from the NC screen to the MGi screen, the program is not read again.
1 : The program is always read again at the time of screen switching from the NC screen to the MGi screen.

	#7	#6	#5	#4	#3	#2	#1	#0
14851		GCC	PKW	W12			SBP	

#0 0 : Corner element between a blank element and a part element is created in the normal direction at creating free figure.
1 : Corner element between a blank element and a part element is created in the opposite direction at creating free figure.

SBP 0 : When the [INSERT] soft key is pressed on the fixed form figure input screen, the subprogram output selection screen is not displayed.
1 : When the [INSERT] soft key is pressed on the fixed form figure input screen, the subprogram output selection screen is displayed.

W12 0 : In the case of editing on the process list screen, the address W1 and W2 are not output with the Process Start Block(G1992).
1 : In the case of editing on the process list screen, the address W1 and W2 are output with the Process Start Block(G1992).

PWK 0 : The “PROGRAM COORDINATE” input item does not appear on the entry window for the arbitrary figures.
1 : The “PROGRAM COORDINATE” input item appears on the entry window for the arbitrary figures.

GCC 0 : On tool offset screen and workpiece shift offset screen, the display data for each program coordinate system changes by soft-key.
1 : On tool offset screen and workpiece shift offset screen, the display data for each program coordinate system changes by DO-signal(F0347#GCO).

#7 0 : Not inhibit to edit the tool management data at NC screen.
1 : Inhibit to edit the tool management data at NC screen

	#7	#6	#5	#4	#3	#2	#1	#0
14852				CFF		G4E	NTC	

NTC In the drawing during machining,
0 : Conpensates the tool offset.
1 : Not conpensates the tool offset.

G4E 0 : When a cursor is placed on the word in the cycle machining block and an operator is going to insert a new word, the warning will be displayed.
1 : When a cursor is placed on the word in the cycle machining block and an operator is going to insert a new word, the warning will not be displayed.

CFF 0 : The number of fixed form sentences per tab is 10 and the characters per fixed form sentence is 128.
1 : The number of fixed form sentences per tab decreases to 5 and the characters per fixed form sentence increase to 256.

#6 0 : The tool data access function is not used.
1 : The tool data access function is used.

#7 0 : Each time an operation is performed, check is made to see if BG editing is in progress.
1 : Each time an operation is performed, no check is made to see if BG editing is in progress.

A.2.22 Settings for Operations in General (For Series 30*i*)

These parameters are for the settings for operations in general in Series 30*i*.

	#7	#6	#5	#4	#3	#2	#1	#0
14853								

#0 0 : The program list screen based on the new specifications is used.
1 : The program list screen based on the old specifications is used.

#1 0 : When the function key [PROG] is pressed, the screen display toggles between the NC program screen and the program list screen.
1 : When the function key [PROG] is pressed, the NC program screen is displayed at all times.

#2 0 : When the [O SRCH] key is pushed with empty of key-in-buffer , the program number is searched
1 : When the [O SRCH] key is pushed with empty of key-in-buffer , the program number is not searched.

#7 0 : The program window screen in machining based on the old specifications is used. (Scroll bar is not displayed)
1 : The program window screen in machining based on the new specifications is used. (Scroll bar is displayed)

A.2.23 Settings for Operations in General (For Multi C Executor)

These parameters are for the settings for operations in general in multi C executor.

	#7	#6	#5	#4	#3	#2	#1	#0
14854								

#7 0 : When multiple applications are used, MANUAL GUIDE *i* exercise screen display control immediately after the power is turned on.
1 : When multiple applications are used, MANUAL GUIDE *i* does not exercise screen display control immediately after the power is turned on. (An application of the machine tool builder exercises screen display control.)

A.2.24 Settings for Operations in General (Each Path)

These parameters are for the settings for operations in general.

	#7	#6	#5	#4	#3	#2	#1	#0
14855								

#0 0 : In the tool offset window, Y-axis offset data is displayed.
1 : In the tool offset window, Y-axis offset is not displayed.

PRC 0 : When the Program Coordinate System Changing Function is enabled, the coordinate system on the first spindle side is the program coordinate system-1 and the coordinate system on the second spindle side is the program coordinate system-2.
1 : When the Program Coordinate System Changing Function is enabled, the coordinate system on the first spindle side is the program coordinate system-2 and the coordinate system on the second spindle side is the program coordinate system-1.

NOTE
This parameter #0 is necessary to order the optional function of Y-axis offset data display

A.2.25 Settings for Operations in General (All Common Path)

These parameters are for the settings for operations in general.

14860	DATA

DATA : Special character for search function in the program display window on the base screen.
Please set the decimal number of ASCII.
If the value is 0, the special character is assumed to be "?".

Special characters are used with the following functions:
<1> In program display, the color of special characters is changed for display.
<2> If a fixed form sentence to be inserted includes special characters, a warning message is indicated.

14861	UNDOBUF

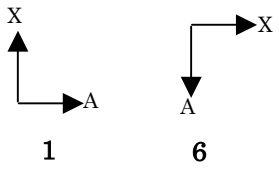
UNDOBUF : Specify the total size of the buffer for use with the redo and undo functions.
If 0 is input, 5 KB of buffer space is allocated. If a negative value is input, the redo and undo functions cannot be used.
Valid data range : -127 to 127
Unit of data : 1 [KB]

A.2.26 Settings for Arbitrary Figures(XA Plane)

These parameters are for the settings for arbitrary figures.

14862	DSPCRDXA

DSPCRDXA : Drawing coordinates when an arbitrary ZC figure is programmed.
=0 Same effect as that of setting 6.
=1 Plan view, horizontal axis = +A, vertical axis = +X
=6 Plan view, horizontal axis = +X, vertical axis = -A



1 6

A.3 PARAMETERS FOR MILLING CYCLE MACHINING

A.3.1 Parameters for Milling Cycles in General

These parameters are for the settings for milling cycles in general.

	#7	#6	#5	#4	#3	#2	#1	#0
27000	MC7	MC6	MC5	MC4	MC3	MC2	MC1	MC0

MC0 0 : In ZC plane cycle output, G02/G03 are inverted.
1 : In ZC plane cycle output, G02/G03 are not inverted.

MC1 0 : The hole machining menu has hole machining items for M.
Remark) Refer to “1.1 Hole Machining by Milling”.
1 : The hole machining menu has hole machining items for T.
Remark) Refer to “1.2 Hole Machining by Turning”.

MC2 0 : In XC plane cycle output, G12.1/G13.1 are not output.
1 : In XC plane cycle output, G12.1/G13.1 are output.

Remark) Refer to the paragraph of XC plane figure of Facing, Contouring, Pocketing or Grooving.

MC3 0 : In ZC plane cycle output, G07.1 is not output.
1 : In ZC plane cycle output, G07.1 is output.

Remark) Refer to the paragraph of ZC plane figure of Facing, Contouring, Pocketing or Grooving.

MC4 0 : In cycle input screens, the tab of [FACE POSIT] for end face position is not displayed.
1 : In cycle input screens, the tab of [FACE POSIT] for end face position is displayed.

Remark) Refer to “1.8 Rear End Facing by Milling”.

MC5 0 : In the soft-key of [START],[CYCLE],[END] and [FIGURE], the icons for Milling are displayed.
1 : In the soft-key of [START],[CYCLE],[END] and [FIGURE], the icons for Turning are displayed.

MC6 0 : In [START] menu screens, the tab of [COORD CONV] is displayed.
1 : In [START] menu screens, the tab of [COORD CONV] is not displayed.

Remark) Refer to “III-3. Slant Face Machining (coordinate conversion)”.

MC7 0 : The tab of [EMBOSSING] is not displayed on the cycle machining menu.
1 : The tab of [EMBOSSING] is displayed on the cycle machining menu.

	#7	#6	#5	#4	#3	#2	#1	#0
27001					P3	P2	P1	P0

P0 0 : The tab of [ROT. AXIS] for rotation axis names are not displayed.
1 : The tab of [ROT. AXIS] for rotation axis names are displayed.

P1 0 : Invalid
1 : Rotation axis name selection soft keys [C] and [A] are used.
(It is necessary to set P0 to 1.)

P2 0 : Invalid
1 : Rotation axis name selection soft keys [C] and [B] are used.
(It is necessary to set P0 to 1.)

P3 0 : Invalid
1 : Rotation axis name selection soft keys [C] and [E] are used.
(It is necessary to set P0 to 1.)

Remark) Refer to “III-1.9 Address Setting of Rotation Axis”.

	#7	#6	#5	#4	#3	#2	#1	#0
27002	ESC	MDL					TYP	TLG

TLG 0 : In Milling cycle menu, the tab of [TOOL COND.] is not displayed.
1 : In Milling cycle menu, the tab of [TOOL COND.] is displayed.

TYP 0 : When the data input window for a milling figure block is opened for modification, the item "FIGURE TYPE" displays all figure types.
1 : When the data input window for a milling figure block is opened for modification, the item "FIGURE TYPE" displays only the figure type that is applicable to the immediately preceding machining process.

MDL 0 : In Milling cycles, after completing the whole cycle motions, the modal will be returned to the state of beginning cycle.
1 : In Milling cycles, after completing the whole cycle motions, the modal will not be returned to the state of beginning cycle.

ESC 0 : In Milling cycles, after completing the whole cycle motions, the tool will go back to the point where machining cycle started.
1 : In Milling cycles, after completing the whole cycle motions, the tool will not go back to the point where machining cycle started.

	#7	#6	#5	#4	#3	#2	#1	#0
27003						ML2	ML1	ML0

By setting this parameter, the optimum cycle menus can be displayed on the screen. Please set 1 bit only according to the machine configuration.

ML0 1 : Machining center X/Y/Z-axis
ML1 1 : Machining center X/Y/Z/C/B-axis (B-axis : Tool tilting axis)
ML2 1 : Machining center X/Y/Z/A (A-axis : work rotation axis)/B-axis (B-axis : Tool tilting axis)

Remarks) If both of ML0 and ML1 are set, ML1 takes priority.

NOTE

1 When the parameter No.27003 is set, please sure to push [F] key on NOW LOADING screen after Power ON. The necessary parameters are set automatically. (When the necessary parameters are set, the message of "NOW SETTING PARAMETERS" is displayed on the left side of NOW LOADING screen.)

2 By setting the parameter No.27003, the following parameters related to display are set autoimatically.

	No.14702#1 Spindl select	No.27000#1 Hole select	No.27000#4 Face pos-M	No.27100#4 Face pos-T	No.27000#6 Coord conv	No.27001#0,#1 Rot. axis nam	No.27004#0 Menu 1	No.27004#1 Menu 2
No.27003 #0=1	0	0	0	0	1	0	1	0
No.27003 #1=1	0	0	0	0	0	0	0	0
No.27003 #2=1	0	0	0	0	0	0	0	0

	#7	#6	#5	#4	#3	#2	#1	#0
27004						**MM2**	**MM1**	**MM0**

If the machine configuration does not meet the parameter No.27003 or No.27103, this parameter is used to display the optimum menu on the screen.

MM0 1 : The following menus are displayed.
- Hole Machining (G1000 to G1006) or (G1110 to G1114)
- Facing (G1020 to G1021)
- Contouring (G1030 to G1033)
- Pocketing (G1040 to G1043)
- Grooving (G1050 to G1053)
- XY-plane : Free figure (G1200 to G1206)
- XY-plane : Hole position (G1210 to G1217)
- XY-plane : Fixed form figure (G1220 to G1223)

MM1 1 : The following menus are displayed.
- Hole Machining (G1000 to G1006) or (G1110 to G1114)
- Facing (G1020 to G1021)
- Contouring (G1030 to G1033)
- Pocketing (G1040 to G1043)
- Grooving (G1050 to G1053)
- C-axis Grooving (G1056)
- XC-plane : Free figure (G1500 to G1506)
- XC-plane : Fixed form figure (G1520 to G1523)
- XC-plane : C-axis figure (G1570 to G1573)
- ZC-plane : Free figure (G1600 to G1606)
- ZC-plane : C-axis figure (G1670 to G1673)

MM2 1 : The following menus are displayed.(It is effective only at MM0 = 1.)

- Hole Machining (G1000 to G1006) or (G1110 to G1114)
- Facing (G1020 to G1021)
- Contouring (G1030 to G1033)
- Pocketing (G1040 to G1043)
- Grooving (G1050 to G1053)
- XA-plane : Free figure (G1700 –> G1706)
- XA-plane : A-axis figure (G1770 –> G1773)

NOTE
1 When these parameter are all 0, the all milling cycle menu except the menu of XA plame are displayed on the screen
2 When the parameter No.27003 or No.27103 is set, this parameter are initialize at the power ON and pushing “F” key.

27005	CLMPM

CLMPM 0 : M code for Main spindle C-axis clamping is not output.
≠0 : M code for Main spindle C-axis clamping.
(Path-specific parameter)

Remark) refer to “III-1.10 C-axis Clamping M code Output”.

27006	UCLMPM

UCLMPM 0 : M code for Main spindle C-axis unclamping is not output.
≠0 : M code for Main spindle C-axis unclamping.
(Path-specific parameter)

Remark) refer to “III-1.10 C-axis Clamping M code Output”.

27007	CFCODM

CFCODM : Feedrate to replace all rapid traverse feedrate during C-axis machining for **feed per minute.**
If 0 is set, the feedrate is assumed 2000 (mm/min) or 78.7 (inch/min) .

Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

NOTE
In XC plane, Polar coordinate interpolation is used. That is, it is impossible to use G0 command. Therefore, No.27007 is used instead of rapid traverse.

27008	CFCODR

CFCODR : Feedrate to replace all rapid traverse feedrate during C-axis machining for **feed per revolution.**
If 0 is set, the feedrate is assumed 2 (mm/min) or 0.0787 (inch/min).

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

NOTE
In XC plane, Polar coordinate interpolation is used. That is, it is impossible to use G0 command. Therefore, No.27008 is used instead of rapid traverse.

27009	CLERCLMP

CLERCLMP : Minimum Clump value of the clearance for Milling cycles.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27010	APESCLMP

APESCLMP : Minimum Clump value of Radius or Distance of Approach or Escape for Milling cycles.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27011	CLMPMS

CLMPMS 0 : M code for Sub spindle C-axis clamping is not output.
≠0 : M code for Sub spindle C-axis clamping.
(Path-specific parameter)

Remark) refer to “1.10 C-axis Clamping M code Output”.

27012	UCLMPMS

UCLMPMS 0 : M code for Sub spindle C-axis unclamping is not output.
≠0 : M code for Sub spindle C-axis unclamping.
(Path-specific parameter)

Remark) refer to “1.10 C-axis Clamping M code Output”.

A.3.2 Parameters for Facing Cycles

These parameters are for the settings for facing cycles.

	#7	#6	#5	#4	#3	#2	#1	#0
27030							FC1	FC0

FC0 0 : The input data item of [PATH MOVE METHOD] and [PATH MOVE FEED RATE] are displayed on Facing cycle menu.
1 : The input data item of [PATH MOVE METHOD] and [PATH MOVE FEED RATE] are not displayed on Facing cycle menu.

FC1 0 : all menu is displayed in “CUT SHIFT DIRECTION”.
1 : only the perpendicular direction to “CUTTING DIRECTION” is displayed in “CUT SHIFT DIRECTION”.

A.3.3 Parameters for Contouring Cycles

These parameters are for the settings for contouring cycles.

	#7	#6	#5	#4	#3	#2	#1	#0
27040		CN6		CN4	CN3	CN2	CN1	CN0

CN0 0 : During in-feed in Roughing, the tool moves by retracting to the height of the top workpiece surface plus the clearance.
1 : During in-feed in Roughing, the tool moves by retracting to the height of the machining surface plus the clearance.

CN1 0 : In Roughing, the tool moves at a hollow by retracting to the height of the top workpiece surface plus the clearance.
1 : In Roughing, the tool moves at a hollow by retracting to the height of the machining surface plus the clearance.

CN2 0 : In Roughing, the tool moves at an opening by retracting to the height of the top workpiece surface plus the clearance..
1 : In Roughing, the tool moves at an opening by retracting to the height of the machining surface plus the clearance.

CN3 0 : In side finishing, NC does not perform cutter compensation.
1 : In side finishing, NC performs cutter compensation.

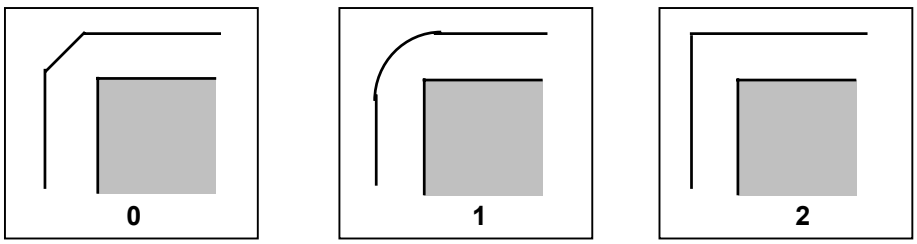
Remark) Tool path is not calculated with Cutter compensation in inside, but G41 or G42 is outputted directly.

CN4 0 : The system performs an interference check.
1 : The system does not perform an interference check.

CN6 0 : In Roughing, when the cutting start point is equal to the end point, the tool cuts directly without escape to the radius direction.
1 : In Roughing, when the cutting start point is equal to the end point, the tool cuts with escape to the radius direction.

27045	COFSW

COFSW : Offset method for side finishing and chamfering in contouring.
=0 : Corner cut interpolation.
=1 : Circular interpolation.
=2 : Extended straight line.

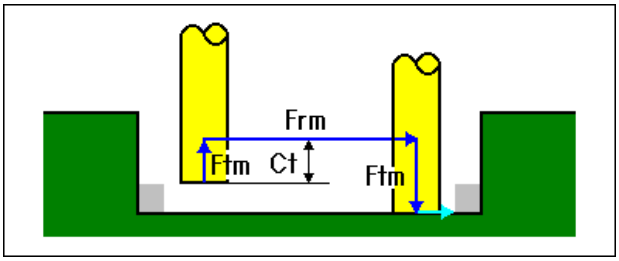


27046	CMVFR

CMVFR : Feedrate during movement in the cutter radius direction in contouring.
For **feed per minute.**
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the cutter radius direction is 'Frm' as the following illustration.



27047	CMVFT

CMVFT : Feedrate during movement in the tool axis direction in contouring for **feed per minute**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

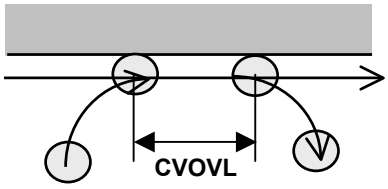
Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the tool axis direction is 'Ftm' as the above illustration.

27048	CVOVL

CVOVL : Amount of overlapping for an approach/escape during contouring.

Unit of data :
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)



27049	CMVFR

CMVFR : Feedrate during movement in the cutter radius direction in contouring for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27046.

27050	CMVFT

CMVFT : Feedrate during movement in the tool axis direction in contouring for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27047.

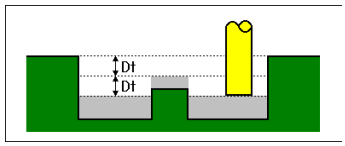
A.3.4 Parameters for Pocketing Cycles

These parameters are for the settings for pocketing cycles.

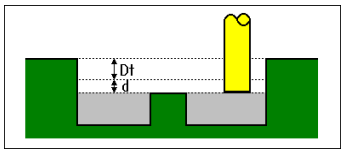
	#7	#6	#5	#4	#3	#2	#1	#0
27060	PR7	PR6	PR5	PR4	PR3	PR2	PR2	PR0

PR0 0 : Machining starts on the inside during roughing and bottom finishing.
1 : Machining starts on the outside during roughing and bottom finishing.

PR1 0 : The top of an island is not machined during roughing and bottom finishing.
1 : Machining is performed by controlling the depth of cut during roughing and bottom finishing.

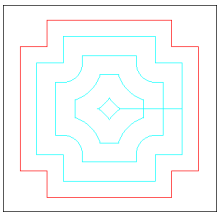


PR1 = 0

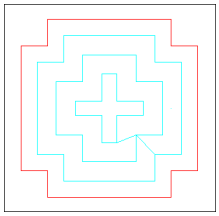


PR1 = 1

PR2 0 : Method of interpolation during roughing and bottom finishing (interpolation of elements with arcs).
1 : Method of interpolation during roughing and bottom finishing (interpolation of elements by extending them).

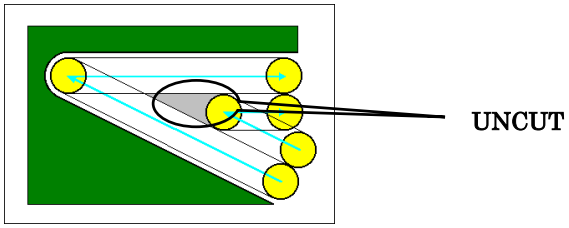


PR2 = 0



PR2 = 1

PR3 0 : Method of machining portions left uncut during roughing and bottom finishing (no machining of portions left uncut).
1 : Method of machining portions left uncut during roughing and bottom finishing (machining of portions left uncut).

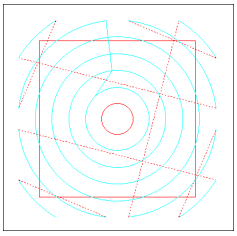


PR4 0 : The tool moves by retracting to the height of the top workpiece surface plus the clearance at an opening during roughing and bottom finishing.

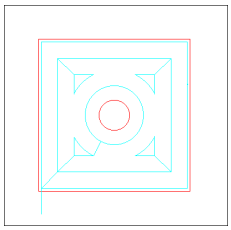
1 : The tool moves by retracting to the height of the machining surface plus the clearance at an opening during roughing and bottom finishing.

PR5 0 : Paths for all material elements are created with an island reference during roughing and bottom finishing.

1 : Paths for all material elements are created with a pocket reference during roughing and bottom finishing.



PR5 = 0



PR5 = 1

PR6 0 : The tool moves by retracting to the height of the top workpiece surface plus the clearance during movement in the tool axis direction during roughing and bottom finishing.

1 : The tool moves by retracting to the height of the machining surface plus the clearance during movement in the tool axis direction during roughing and bottom finishing.

PR7 0 : The input data item of [START PT.(1st AXIS)] and [START PT.(2nd AXIS)] are not displayed on the Pocketing cycle menu..

1 : The input data item of [START PT.(1st AXIS)] and [START PT.(2nd AXIS)] are displayed on the Pocketing cycle menu..

	#7	#6	#5	#4	#3	#2	#1	#0
27061				PF4	PF3	PF2	PF1	PF0

PF0 0 : During in-feed in side finishing and chamfering, the tool moves by retracting to the height of the top workpiece surface plus the clearance.

1 : During in-feed in side finishing and chamfering, the tool moves by retracting to the height of the machining surface plus the clearance.

PF1 0 : In side finishing and chamfering, the tool moves at a hollow by retracting to the height of the top workpiece surface plus the clearance.

1 : In side finishing and chamfering, the tool moves at a hollow by retracting to the height of the machining surface plus the clearance.

PF2 0 : In side finishing and chamfering, the tool moves at an opening by retracting to the height of the top workpiece surface plus the clearance.
1 : In side finishing and chamfering, the tool moves at an opening by retracting to the height of the machining surface plus the clearance.

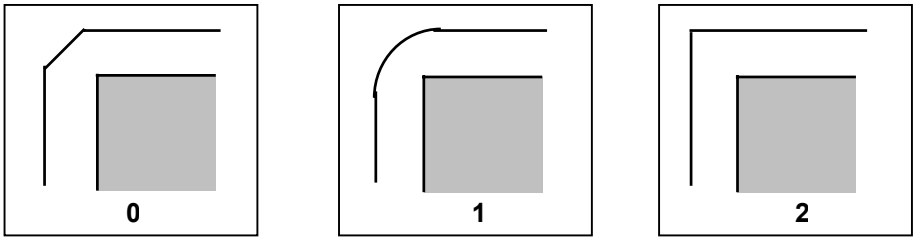
PF3 0 : In side finishing, NC does not perform cutter compensation.
1 : In side finishing, NC performs cutter compensation.

Remark) Tool path is not calculated with Cutter compensation in inside, but G41 or G42 is outputted directly.

PF4 0 : The system performs an interference check.
1 : The system does not perform an interference check.

27065	POFSW

POFSW : Offset method for side finishing and chamfering in pocketing.
= 0 : Corner cut interpolation.
= 1 : Circular interpolation.
= 2 : Extended straight line.

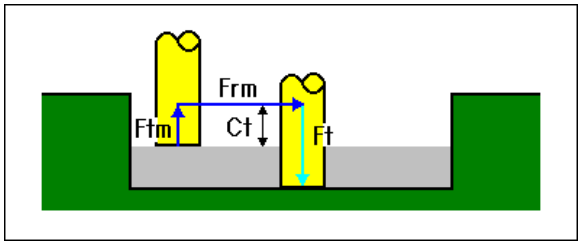


27066	PKTFR

PKTFR : Feedrate during movement in the cutter radius direction in in-feed for **feed per minute**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the cutter radius direction is 'Frm' as the following illustration.



27067	PKTFT

PKTFT : Feedrate during movement in the tool axis direction in in-feed for **feed per minute**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

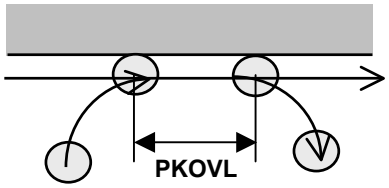
Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the tool axis direction is 'Ftm' as the above illustration.

27068	PKOVL

PKOVL : Amount of overlapping for an approach/escape during side finishing and chamfering.

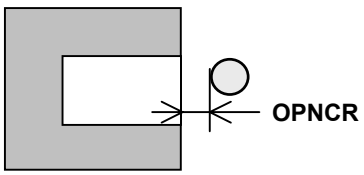
Unit of data :
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)



27069	OPNCR

OPNCR : Clearance for an open portion in pocketing
(When OPNCR = 0)
For metric input (0000#2=0), the clearance for the open portion is assumed to be 3 mm.
For inch input (0000#2=1), the clearance for the open portion is assumed to be 0.3 inch.

Unit of data :
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)



27070	PKTFR

PKTFR : Feedrate during movement in the cutter radius direction in in-feed for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27066.

27071	PKTFT

PKTFT : Feedrate during movement in the tool axis direction in in-feed for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27067.

A.3.5 Parameters for Grooving Cycles

These parameters are for the settings for grooving cycles.

	#7	#6	#5	#4	#3	#2	#1	#0
27080						GR2	GR1	GR0

GR0 0 : During roughing and bottom finishing, in-feed in the cutter radius direction is performed with a uniform depth of cut.
(A uniform depth is the calculated depth automatically.)
1 : During roughing and bottom finishing, in-feed in the cutter radius direction is performed with [CUT DEPTH OF RADIUS].

GR1 0 : During roughing and bottom finishing, in-feed in the tool axis direction is performed with a uniform depth of cut.
(A uniform depth is the calculated depth automatically.)
1 : During roughing and bottom finishing, in-feed in the tool axis direction is performed with [CUT DEPTH OF AXIS].

GR2 0 : During roughing and bottom finishing, tool retracts to the top workpiece surface plus the clearance..
1 : During roughing and bottom finishing, the tool retracts to the position of the machining surface plus the clearance.

	#7	#6	#5	#4	#3	#2	#1	#0
27081				GF4	GF3	GF2	GF1	GF0

GF0 0 : During in-feed in side finishing and chamfering, the tool moves by retracting to the height of the top workpiece surface plus the clearance.
1 : During in-feed in side finishing and chamfering, the tool moves by retracting to the height of the machining surface plus the clearance.

GF1 0 : In side finishing and chamfering, the tool moves at a hollow by retracting to the height of the top workpiece surface plus the clearance.
1 : In side finishing and chamfering, the tool moves at a hollow by retracting to the height of the machining surface plus the clearance.

GF2 0 : In side finishing and chamfering, the tool moves at an opening by retracting to the height of the top workpiece surface plus the clearance.
1 : In side finishing and chamfering, the tool moves at an opening by retracting to the height of the machining surface plus the clearance.

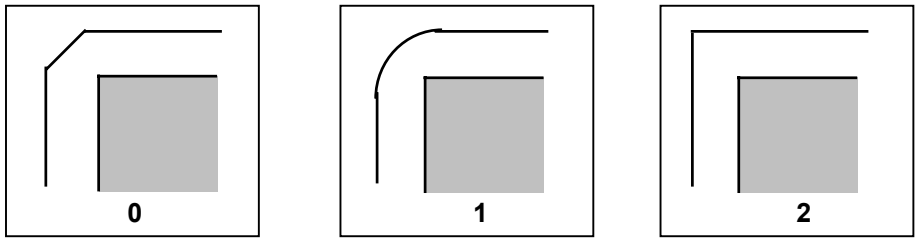
GF3 0 : In side finishing, NC does not perform cutter compensation.
1 : In side finishing, NC performs cutter compensation.

Remark) Tool path is not calculated with Cutter compensation in inside, but G41 or G42 is outputted directly.

GF4 0 : The system performs an interference check.
1 : The system does not perform an interference check.

27085	GOFSW

GOFSW : Offset method for groove finishing paths.
= 0 : Corner cut interpolation.
= 1 : Circular interpolation.
= 2 : Extended straight line.

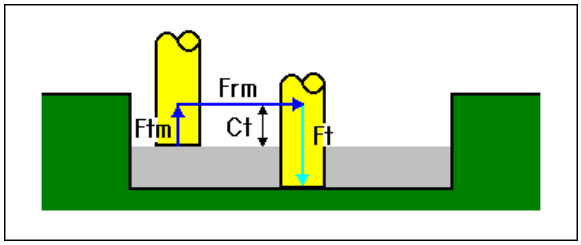


27086	GMVFR

GMVFR : Feedrate during movement in the cutter radius direction in grooving for **feed per minute**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the cutter radius direction is 'Frm' as the following illustration.



27087	GMVFT

GMVFT : Feedrate during movement in the tool axis direction in contouring for **feed per minute**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

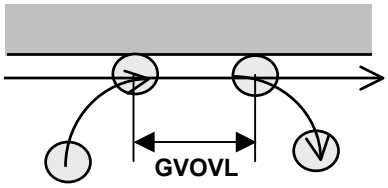
Unit of data:
For metric input (0000#2=0) : 1(mm/min)
For inch input (0000#2=1) : 0.01(inch/min)

Remark) Feedrate during movement in the tool axis direction is 'Ftm' as the above illustration.

27088	GVOVL

GVOVL : Amount of overlapping for an approach/escape during side finishing and chamfering.

Unit of data :
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)



27089	GMVFR

GMVFR : Feedrate during movement in the cutter radius direction in grooving for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27086.

27090	GMVFT

GMVFT : Feedrate during movement in the tool axis direction in contouring for **feed per revolution**.
If 0 is set, the feedrate is assumed Rapid traverse feedrate.

Unit of data:
For metric input (0000#2=0) : 0.0001(mm/rev)
For inch input (0000#2=1) : 0.000001(inch/rev)

Remark) Refer to No.27087.

A.4 PARAMETERS FOR TURNING CYCLE OPTIONS

A.4.1 Parameters Common to Turning Cycles

These parameters are used for settings common to Turning cycles.

	#7	#6	#5	#4	#3	#2	#1	#0
27100				TC4			TC1	TC0

TC0 0 : The input item of [CUT DEPTH DIRECTION] is not displayed.
1 : The input item of [CUT DEPTH DIRECTION] is displayed.

TC1 0 : The input item of [POCKET CUTTING] and [OVERHANG CUTTING] are not displayed.
1 : The input item of [POCKET CUTTING] and [OVERHANG CUTTING] are displayed.

TC4 0 : The tab of [FACE POSIT] for end face position is not displayed.
1 : The tab of [FACE POSIT] for end face position is displayed.

Remark) Refer to “III-2.5 Rear End Facing by Turning”.

	#7	#6	#5	#4	#3	#2	#1	#0
27102	ESC	MDL				LOC	TYP	TLG

TLG 0 : In Turning cycle menu, the tab of [TOOL COND.] is not displayed.
1 : In Turning cycle menu, the tab of [TOOL COND.] is displayed.

TYP 0 : When the data input window for a turning figure block is opened for modification, the item "FIGURE TYPE" displays all figure types.
1 : When the data input window for a turning figure block is opened for modification, the item "FIGURE TYPE" displays only the figure type that is applicable to the immediately preceding machining process.

LOC 0 : The turning cycle menu lists "HOLE MACHINING", "TURNING", "TURNING GROOVING", and "THREADING" in this order
1 : The turning cycle menu lists "TURNING", "TURNING GROOVING", "THREADING", and "HOLE MACHINING" in this order.

MDL 0 : In Turning cycles, after completing the whole cycle motions, the modal will be returned to the state of beginning cycle.
1 : In Turning cycles, after completing the whole cycle motions, the modal will not be returned to the state of beginning cycle.

ESC 0 : In Turning cycles, after completing the whole cycle motions, the tool will go back to the point where machining cycle started.
1 : In Turning cycles, after completing the whole cycle motions, the tool will not go back to the point where machining cycle started.

	#7	#6	#5	#4	#3	#2	#1	#0
27103	LT7				LT3	LT2	LT1	LT0

By setting this parameter, the optimum cycle menus can be displayed on the screen. Please set 1 bit only according to the machine configuration.

LT0 1 : Lathe - X/Z-axis
LT1 1 : Lathe - X/Z/C-axis
LT2 1 : Lathe - X/Z/C/Y-axis
LT3 1 : Lathe -X/Z/C/Y/B-axis (B-axis : Tool tilting axis)

Remarks) If over two of LT0 to LT3 are set, larger number bit takes priority.

LT7 1 : Lathe - Lathe with sub spindle

Remarks) This LT7 is used together with LT0 to LT3.

NOTE
1 When the parameter No.27103 is set, please sure to push [F] key on NOW LOADING screen after Power ON. The necessary parameters are set automatically. (When the necessary parameters are set, the message of “NOW SETTING PARAMETERS” is displayed on the left side of NOW LOADING screen.)
2 By setting the parameter No.27103, the following parameters related to display are set autoimatically.

	No.14702#1 Spindl select	No.27000#1 Hole select	No.27000#4 Face pos-M	No.27100#4 Face pos-T	No.27000#6 Coord conv	No.27001#0,#1 Rot. axis nam	No.27004#0 Menu 1	No.27004#1 Menu 2
No.27103 #0=1 No.27103 #7=0	0	0	0	0	1	0	0	0
No.27103 #1=1 No.27103 #7=0	0	1	0	0	1	0	0	1
No.27103 #2=1 No.27103 #7=0	0	1	0	0	1	0	0	0
No.27103 #3=1 No.27103 #7=0	0	1	0	0	0	0	0	0
No.27103 #0=1 No.27103 #7=1	1	0	0	1	1	0	0	0
No.27103 #1=1 No.27103 #7=1	1	1	1	1	1	1	0	1
No.27103 #2=1 No.27103 #7=1	1	1	1	1	1	1	0	0
No.27103 #3=1 No.27103 #7=1	1	1	1	1	0	1	0	0

A.4.2 Parameters for Turning Cycle Machining

These parameters are for the settings for turning cycles.

	#7	#6	#5	#4	#3	#2	#1	#0
27120								BLN

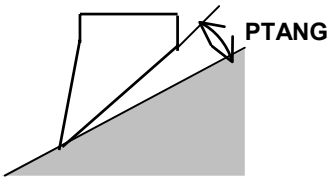
BLN 0 : When the tool advances in the cutting direction, the excessive amount of travel of the tool is nose radius R if the attribute of the figure across which the tool moves is the blank element.

1 : When the tool advances in the cutting direction, the excessive amount of travel of the tool is [clearance + nose radius R (×2) if the attribute of the figure across which the tool moves is the blank element (conventional specifications).

27125	PTANG

PTANG : Cutting edge protection angle.

Unit of data : 1 deg



27126	DCLMP

DCLMP : clamping value of 'CUT DEPTH' in Turning, Turning grooving and Threading

Unit of data :
For metric input (0000#2=0) : 0.001 mm
For inch input (0000#2=1) : 0.0001 inch

When DCLMP = 0, the depth-of-cut clamping value is assumed to be one-tenth of a specified depth of cut.

27128	ESCPCLMP

ESCPCLMP : Minimum clump value of ESCAPE AMOUNT for Turning Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27129	XAXSCLMP

XAXSCLMP : Minimum clump value of X-AXIS CLEARANCE for Turning Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27130	ZAXSCLMP

ZAXSCLMP : Minimum clump value of Z-AXIS CLEARANCE for Turning Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

A.4.3 Parameters for Threading Cycles

These parameters are for the settings for threading cycles.

27145	TDMIN

TDMIN : Minimum depth of cut during threading.

Unit of data :
For metric input (0000#2=0) : 0.001 mm
For inch input (0000#2=1) : 0.0001 inch

27150	TGNOUT

TGNOUT : Thread height factor for general-purpose threads (for outside diameters). The value 0 is regards as 0.6495.

Unit of data : 0.0001

NOTE
No.27150 is used to calculate [THREAD DEPTH] in general-purpose threads (for outside diameters). The formula is as follows.
(Thread Depth for Outside) = (No.27150) × (Lead)

27151	TGNIN

TGNIN : Thread height factor for general-purpose threads (for inside diameters). The value 0 is regards as 0.6495.

Unit of data : 0.0001

NOTE
No.27151 is used to calculate [THREAD DEPTH] in general-purpose threads (for inside diameters). The formula is as follows.
(Thread Depth for Inside) = (No.27151) × (Lead)

27152	TMTOUT

TMTOUT : Thread height factor for metric and unified threads (for outside diameters). The value 0 is regards as 0.6495.

Unit of data : 0.0001

NOTE
1 No.27152 is used to calculate [THREAD DEPTH] in metric threads (for outside diameters).
The formula is as follows.
(Thread Depth for Outside) = (No.27152) × (Lead)
2 No.27152 is used to calculate [THREAD DEPTH] in unified threads (for outside diameters).
The formula is as follows.
(Thread Depth for Outside) (mm)
= ((No.27152) ÷ (Thread Number)) × 25.4
(Thread Depth for Outside) (inch)
= (No.27152) ÷ (Thread Number)

27153	TMTIN

TMTIN : Thread height factor for metric and unified threads (for inside diameters). The value 0 is regards as 0.6495.

Unit of data : 0.0001

NOTE
1 No.27153 is used to calculate [THREAD DEPTH] in metric threads (for Inside diameters).
The formula is as follows.
(Thread Depth for Inside) = (No.27153) × (Lead)
2 No.27153 is used to calculate [THREAD DEPTH] in unified threads (for Inside diameters).
The formula is as follows.
(Thread Depth for Inside) (mm)
= ((No.27153) ÷ (Thread Number)) × 25.4
(Thread Depth for Inside) (inch)
= (No.27153) ÷ (Thread Number)

27154	TPTOUT

TPTOUT : Thread height factor for PT and PF threads (for outside diameters). The value 0 is regards as 0.6403.

Unit of data : 0.0001

NOTE
No.27154 is used to calculate [THREAD DEPTH] in PT and PF threads (for outside diameters). The formula is as follows.
(Thread Depth for Outside) (mm)
= ((No.27154) ÷ (Thread Number)) × 25.4
(Thread Depth for Outside) (inch)
= (No.27154) ÷ (Thread Number)

27155	TPTIN

TPTIN : Thread height factor for PT and PF threads (for inside diameters). The value 0 is regards as 0.6403.

Unit of data : 0.0001

NOTE
No.27155 is used to calculate [THREAD DEPTH] in PT and PF threads (for Inside diameters). The formula is as follows.
(Thread Depth for Inside) (mm)
= ((No.27155) ÷ (Thread Number)) × 25.4
(Thread Depth for Inside) (inch)
= (No.27155) ÷ (Thread Number)

27156	SURFSCLMP

SURFCLMP : Minimum clump value of SURFACE CLEARANCE for Threading Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27157	ENTRCLMP

ENTRCLMP : Minimum clump value of ENTRANCE CLEARANCE for Threading Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27158	EXITSCLMP

EXITCLMP : Minimum clump value of EXIT CLEARANCE for Threading Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

A.4.4 Parameter for Turning and Grooving Cycles

This parameter is for the setting for turning and grooving cycles.

27175	GDMIN

GDMIN : Minimum depth of cut in turning and grooving (rough cutting).
Unit of data :
For metric input (0000#2=0) : 0.001 mm
For inch input (0000#2=1) : 0.0001 inch

When GDMIN = 0, the minimum depth of cut is assumed to be one-tenth of a specified depth of cut.

27176	CLRECLMP

CLRECLMP : Minimum clump value of CLEARANCE for Turning grooving Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

27177	ESCPCLMP

ESCPCLMP : Minimum clump value of ESCAPE CLEARANCE for Turning grooving Cycle.

Unit of data:
For metric input (0000#2=0) : 0.001(mm)
For inch input (0000#2=1) : 0.0001(inch)

A.4.5 Parameters for Program Coordinate System Changing Function and Tool Offset Memory Changing Function

These parameters are for the settings for program coordinate system changing function and tool offset memory changing Function.

27180	G1992W1M

G1992W1M : The M-code output to change to the program coordinate system-1 when G1992 S** W1 block is executed.
valid data range : 0-99999999

27181	G1992W2M

G1992W2M : The M-code output to change to the program coordinate system-2 when G1992 S** W2 block is executed.
valid data range : 0-99999999

27184	G1992W1O

G1992W1O : The P-code macro program number called to change to the program coordinate system-1 when G1992 S** W1 block is executed.
valid data range : 0-99999999

27185	G1992W2O

G1992W2O : The P-code macro program number called to change to the program coordinate system-2 when G1992 S** W2 block is executed.
valid data range : 0-99999999

27188	PGC1IC

PGC1IC : Icon number for program coordinate system-1. (Each Path)

27189	PGC2IC

PGC1IC : Icon number for program coordinate system-2. (Each Path)

The values set to No.27188 and No.27189 must be selected from the following table.

Icon Number	11	12	13	14
icon				
Icon Number	21	22	23	24
icon				
Icon Number	31	32	33	34
icon				

If the icon number which does not exist the above table is set to the parameter, the icon for selected coordinate system is not displayed.

A.4.6 Parameters for Machining Simulation (Animated)

These parameters are for the settings for machining simulation (animated).

27300	SCALE OF THE BLANK

(Byte type, FANUC standard settings = 0)

Scale magnification for automatic scaling in the machining simulation function

Unit of data : 10 %

Valid data range : 0 to 20 (When 0 is set, 100% is assumed.)

NOTE

1 In case of one Cs axis of main spindle, please be sure to set No.14717 only. If no Cs axis or Animation option, it is unnecessary to set.

2 In case of two Cs axis with main and sub spindle, please be sure to set No.27301, No.27302 and No.27312 #0. And No.14717 is used as working number for Cs axis, as follows.

3 When 27312#0 = 1, No. 14717 is rewritten to the value of No.27301 or No.27302 by a spindle selection command (G1998). Therefore, in the case of No.27312 #0=1, please be sure to specify Spindle selection command (G1998) before Milling cycles.

4 As to Apindle selection command (G1998), please refer to “9.5 Setting of data for Animation”.

27301	ROTATE AXIS NO. (SP1)

(Byte type, FANUC standard settings = 0)

Rotation axis (Cs axis) number of Main spindle

<Related parameters : No.14717, No.27312#0, #1, #2, No.27302>

NOTE

Please be sure to specify Spindle selection command (G1998 S1) before each Milling cycles.

27302	ROTATE AXIS NO. (SP2)

(Byte type, FANUC standard settings = 0)

Rotation axis (Cs axis) number of Sub spindle

<Related parameters : No.14717, No.27312#0, #1, #2, No.27301>

NOTE

Please be sure to specify Spindle selection command (G1998 S2) before each Milling cycles.

27303	MTYPE

(Byte type, FANUC standard settings = 0)

MTYPE : Type of machine mechanism

Type	Controlled rotary axis	Parameter setting for the axis
0	Without a rotary axis Or With a tool rotary axis	Parameter No.14178 is a tool rotary axis.
1	With a workpiece table rotary axis	Parameter No.14178 is a workpiece table rotary axis.

<Related parameters : No.27301#1, No.27305, No.27307>

NOTE
1 No. of workpiece rotated axis (usually C-axis) is set to the parameter No.14717. In the case of machine mechanism type=1, this parameter must not be set to workpiece table rotary axis No.
2 Drawing of machining simulation is not available for a machine which takes a tool rotary axis with a workpiece table rotary axis.
3 Drawing of machining simulation is not available for a machine which has the roatry axis slant for the basic three axis.
4 Machine mechanism type=2(With a workpiece table rotary axis) is not available with multi-path system or with a subspindle.
5 The axis to set to the parameter No.14718 is only a rotary axis about Y axis.

27305	TBLDISTX

(2-word type, FANUC standard settings = 0)

TBLDISTX : In the case that type of machine mechanism is type 1(With a workpiece table rotary axis), distance (X-axis) from the rotary center point to the rotary standard point of drawing blank figure. The direction of + X axis is "+" for this data.

<Related parameters : No.27301#1, No.27303, No.27307>

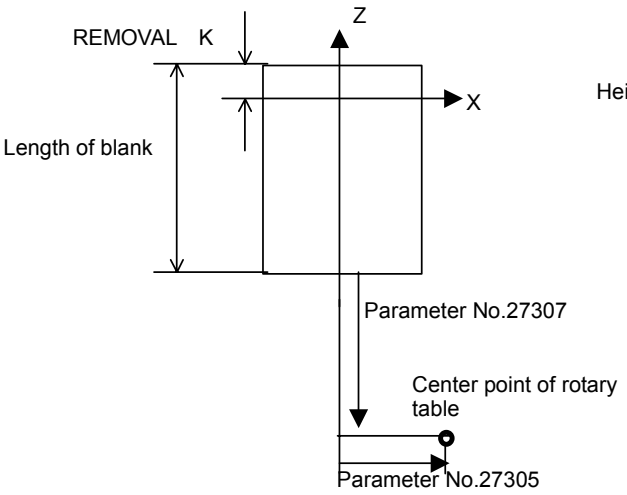
27307	TBLDISTZ

(2-word type, FANUC standard settings = 0)

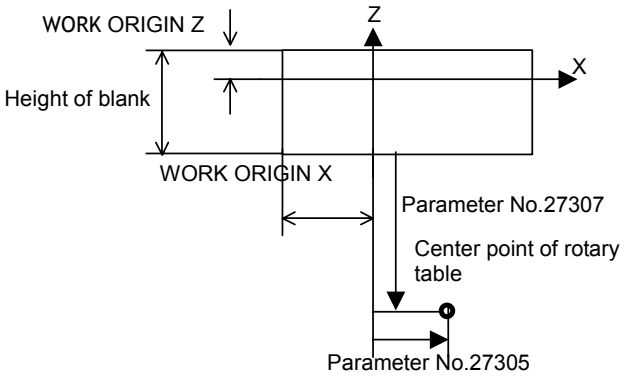
TBLDISTZ : In the case that type of machine mechanism is type 1(With a workpiece table rotary axis), distance (Z-axis) from the rotary center point to the rotary standard point of drawing blank figure. The direction of - Z-axis is "+" for this data.

<Related parameters : No.27301#1, No.27303, No.27307>

Column blank form



Rectangular solid blank form



27309	DATA

(2-word type, FANUC standard settings = 0)

DATA : R signal for selected path simulation in Multi path lathe function (The first figure is "bit" and other figure is the R signal number.)

In the multi-path lathe, the machining simulation is performed only at the selected path by R signal that is set at this parameter.
For example, with 2 paths system

- At Path 1, the parameter No.27309=1237 (R123.7 is 1)
- At Path 2, the parameter No.27309=11237 (R1123.7 is 1)

	#7	#6	#5	#4	#3	#2	#1	#0
27310		TLD		WOK				PRC

(FANUC standard settings = 00000000)

PRC 0 : The soft-key of [STOP] is displayed on the animation soft-key row.
1 : The soft-key of [PROC] is displayed instead of the soft-key [STOP].

NOTE
[STOP] is for the operation to stop with the end block.
[PROC] is for the operation to stop with 'M01'.

WOK 0 : A blank figure is displayed on the tool path drawing screen or the machining drawing screen when the drawing screen is opened.
1 : A blank figure is displayed on the tool path drawing screen or the machining drawing screen when a G code for blank figure definition is executed.

TLD 0 : The tag of [TOOL] is not displayed on START menu window.
1 : The tag of [TOOL] is displayed on START menu window

	#7	#6	#5	#4	#3	#2	#1	#0
27311							ACD	ITF

(FANUC standard settings = 00000000)

ITF 0 : Animated simulation continues even when the interference check function issues an interference alarm.
1 : Animated simulation is stopped temporarily even when the interference check function issues an interference alarm.

NOTE
Parameters No. 27310 and No. 27311 can not be used in some editions.

ACD 0 : Even If the workpiece coordinate is changed in performing machining simulation, the drawing is performed on the same workpiece coordinate as one on the top of simulation.(The workpiece coordinate by drawing definition of blank form block is same as the continuous-state workpiece coordinate on the top of simulation.)
1 : If the workpiece coordinate is changed in performing machining simulation, the drawing is performed on it.

NOTE
1 This parameter is availabel only in drawing of machining sumulation. Drawing during machining is always performed on the same workpiece coordinate as the top of machining.
2 In multi-paths system, this parameter is a common parameter at all paths.
3 In the case of ACD = 1, spindle at which animated simulation is performed is the last commanded by G1998 command at one of paths. In the case of ACD = 0, spindle at which animated simulation is performed is the last commanded by G1992 or G1998 command at one of paths.

	#7	#6	#5	#4	#3	#2	#1	#0
27312						INS	INA	SPA

(FANUC standard settings = 00000000)

SPA 0 : The rotation axis number for simulation based on spindle 1 or spindle 2 with a subspindle attached is not switched by a spindle selection command.

1 : The rotation axis number for simulation based on spindle 1 or spindle 2 with a subspindle attached is switched by a spindle selection command.

<Related parameters : No.14717, No.27312#1, #2, No.27301, No.27302>

NOTE
When SPA = 1, No. 14717 is rewritten to the value of No.27301 (in the case of G1998 S1) or No.27302 (in the case of G1998 S2) by a spindle selection command (G1998). So, please be sure to specify a spindle selection command when SPA = 1.

INA 0 : INS (#2) is not available.

1 : INS (#2) is available.

NOTE
It is necessary to set SPA (#0) to 1.

INS 0 : After animated simulation, No.14717 (the rotation axis number for simulation) is rewritten to the value of No.27301 (axis number of Main spindle).

1 : After animated simulation, No.14717 (the rotation axis number for simulation) is rewritten to the value of No.27302 (axis number of Sub spindle).

NOTE
It is necessary to set SPA (#0) and INA (#1) to 1.

27323	MINMOV

(2-word type, FANUC standard settings=0)

MINMOV : Least command increment for the tool in machining simulation
Unit of data : 1 [dot]

	#7	#6	#5	#4	#3	#2	#1	#0
27350								GTP

(FANUC standard settings = 00000000)

GTP 0 : When animate general tool, Tip position is in front

1 : When animate general tool, Tip position is in rear

27351	GENR TIP LENGTH

(2-word type, FANUC standard settings=0)

GENR TIP LENGTH : Cutter length when animate general tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 12mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.4724inch.

27352	GENR HOLD LENGTH

(2-word type, FANUC standard settings=0)

GENR HOLD LENGTH : Holder length when animate general tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27353	GENR HOLD WIDTH

(2-word type, FANUC standard settings=0)

GENR HOLD WIDTH : Holder width when animate general tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)
Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 14mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.5512inch.

27354	GENR HOLD LENGTH2

(2-word type, FANUC standard settings=0)

GENR HOLD LENGTH2 : Holder length2 when animate general tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

27355	GENR HOLD WIDTH2

(2-word type, FANUC standard settings=0)

GENR HOLD WIDTH2 : Holder width2 when animate general tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

	#7	#6	#5	#4	#3	#2	#1	#0
27356								TTP

(FANUC standard settings = 00000000)

TTP 0 : When animate threading tool, tip position is in front
1 : When animate threading tool, tip position is in rear

27357	THREAD TIP WIDTH

(2-word type, FANUC standard settings=0)

THREAD TIP WIDTH : Tip width when animate threading tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 3mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.1181inch.

27358	THREAD HOLD LENGTH

(2-word type, FANUC standard settings=0)

THREAD HOLD LENGTH : Holder length when animate threading tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27359	THREAD HOLD WIDTH

(2-word type, FANUC standard settings=0)

THREAD HOLD WIDTH : Holder width when animate threading tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 14mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.5512inch.

	#7	#6	#5	#4	#3	#2	#1	#0
27360								GVP

(FANUC standard settings = 00000000)

GVP 0 : When animate grooving tool tip position is in front
1 : When animate grooving tool tip position is in rear

27361	GROOVE HOLD LENGTH

(2-word type, FANUC standard settings=0)

GROOVE HOLD LENGTH : Holder length when animate grooving tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27362	GROOVE HOLD WIDTH

(2-word type, FANUC standard settings=0)

GROOVE HOLD WIDTH : Holder width when animate grooving tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 14mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.5512inch.

	#7	#6	#5	#4	#3	#2	#1	#0
27363								BTP

(FANUC standard settings = 00000000)

BTP 0 : When animate button turning tool tip position is in front
1 : When animate button turning tool tip position is in rear

27364	BUTTON HOLD LENGTH

(2-word type, FANUC standard settings=0)

BUTTON HOLD LENGTH : Holder width when animate button turning tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27365	BUTTON HOLD WIDTH

(2-word type, FANUC standard settings=0)

BUTTON HOLD WIDTH : Holder width when animate button turning tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 14mm.

If 0 is set in case of inch input(0000#2=1), default data will be 0.5512inch.

	#7	#6	#5	#4	#3	#2	#1	#0
27366								STP

(FANUC standard settings = 00000000)

STP 0 : When animate straight tool, tip position is in front

1 : When animate straight tool, tip position is in rear

27367	STRAI TIP LENGTH

(2-word type, FANUC standard settings=0)

STRAI TIP LENGTH : Tip length when animate straight tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 12mm.

If 0 is set in case of inch input(0000#2=1), default data will be 0.4724inch.

27368	STRAI HOLD LENGTH

(2-word type, FANUC standard settings=0)

STRAI HOLD LENGTH : Holder length when animate straight tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 50mm.

If 0 is set in case of inch input(0000#2=1), default data will be 1.9865inch.

27369	STRAI HOLD WIDTH

(2-word type, FANUC standard settings=0)

STRAI HOLD WIDTH : Holder width when animate straight tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 14mm.
If 0 is set in case of inch input(0000#2=1), default data will be 0.5512inch.

27370	STRAI HOLD LENGTH2

(2-word type, FANUC standard settings=0)

STRAI HOLD LENGTH2 : Holder length 2 when animate straight tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

27371	STRAI HOLD WIDTH2

(2-word type, FANUC standard settings=0)

STRAI HOLD WIDTH2 : Holder width 2 when animate straight tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

27372	DRILL TIP LENGTH

(2-word type, FANUC standard settings=0)

DRILL TIP LENGTH : Holder width when animate straight tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9865inch.

27373	F E MIL TIP LENGTH

(2-word type, FANUC standard settings=0)

F E MIL TIP LENGTH : Tip length when animate flat end mill tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9865inch.

27374	TAP TIP LENGTH

(2-word type, FANUC standard settings=0)

TAP TIP LENGTH : Tip length when animate tapping tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 50mm.

If 0 is set in case of inch input(0000#2=1), default data will be 1.9865inch.

27376	C SINK TIP LENGTH

(2-word type, FANUC standard settings=0)

C SINK TIP LENGTH : Tip length when animate counter sink tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 26mm.

If 0 is set in case of inch input(0000#2=1), default data will be 1.0236inch.

27377	C SINK CUT LENGTH

(2-word type, FANUC standard settings=0)

C SINK CUT LENGTH : Cutter length when animate counter sink tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 50mm.

If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27378	C SINK SHANK LENGTH

(2-word type, FANUC standard settings=0)

C SINK SHANK LENGTH : Shank length when animate counter sink tool

Input unit :

mm input (0000#2=0) : 0.001(mm)

inch input (0000#2=1) : 0.0001(inch)

Remarks)

If 0 is set in case of metric input (0000#2=0), default data will be 130mm.

If 0 is set in case of inch input(0000#2=1), default data will be 5.1181inch.

27379	C SINK SHANK DIA

(2-word type, FANUC standard settings=0)

C SINK SHANK DIA : Shank diameter when animate counter sink tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 32mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.2598inch.

27380	B E MIL TIP LENGTH

(2-word type, FANUC standard settings=0)

B E MIL TIP LENGTH : Shank diameter when animate counter sink tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27381	REAM TIP LENGTH

(2-word type, FANUC standard settings=0)

REAM TIP LENGTH : Tip length when animate reamer tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27382	BORING TIP LENGTH

(2-word type, FANUC standard settings=0)

BORING TIP LENGTH : Tip length when animate boring tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 50mm.
If 0 is set in case of inch input(0000#2=1), default data will be 1.9685inch.

27383	F MIL TIP LENGTH

(2-word type, FANUC standard settings=0)

F MIL TIP LENGTH : Tip length when animate face mill tool
Input unit :
mm input (0000#2=0) : 0.001(mm)
inch input (0000#2=1) : 0.0001(inch)

Remarks)
If 0 is set in case of metric input (0000#2=0), default data will be 63mm.
If 0 is set in case of inch input(0000#2=1), default data will be 2.4803inch.

A.4.7 Parameter for Machining Simulation (Animated)

27390	M-CODE AVERAGE TIME

(2-word type, FANUC standard settings=0)

M-CODE AVERAGE TIME
: Execution time of averaged M code
This parameter is used for calculating machining time.
data unit : 10(msec)
valid data range : 0-99999999

27391	S-CODE AVERAGE TIME

(2-word type, FANUC standard settings=0)

S-CODE AVERAGE TIME
: Execution time of averaged S code
This parameter is used for calculating machining time.
data unit : 10(msec)
valid data range : 0-99999999

27390	T-CODE AVERAGE TIME

(2-word type, FANUC standard settings=0)

T-CODE AVERAGE TIME
: Execution time of averaged T code
This parameter is used for calculating machining time.
data unit : 10(msec)
valid data range : 0-99999999

A.4.8 Parameters for Multi-path Lathe Function

These parameters are for the Multi-path lathe function.

	#7	#6	#5	#4	#3	#2	#1	#0
27400								SPT

(FANUC standard settings = 00000000)

SPT 0 : Tool post is selection by the software key
1 : Tool post is selection by the HEAD switch signal
(common parameter among paths)

	#7	#6	#5	#4	#3	#2	#1	#0
27401	MR2		SR2		SSP	SMP	SSE	SME

(FANUC standard settings = 00000000)

SME 0 : Tool post 2 cannot be used with spindle 1.
1 : Tool post 2 can be used with spindle 1.

SSE 0 : Tool post 2 cannot be used with spindle 2.
1 : Tool post 2 can be used with spindle 2.

NOTE
SSE is available in the case of Parameter No.14702 SUB(#1) = 1 only.

SMP 0 : Tool post 2 is placed above spindle 1.
1 : Tool post 2 is placed under spindle 1.

NOTE
SMP is available in the case of Parameter No.27401 SME(#0) = 1 only.

SSP 0 : Tool post 2 is placed above spindle 2.
1 : Tool post 2 is placed under spindle 2.
(common parameter among paths)

NOTE
SSP is available in the case of Parameters No.14702 SUB(#1) = 1 and No.27401 SSE(#1) = 1 only.

	#7	#6	#5	#4	#3	#2	#1	#0
27402					TSP	TMP	TSE	TME

(FANUC standard settings = 00000000)

TME 0 : Tool post 3 cannot be used with spindle 1.
1 : Tool post 3 can be used with spindle 1.

TSE 0 : Tool post 3 cannot be used with spindle 2.
1 : Tool post 3 can be used with spindle 2.

NOTE
TSE is available in the case of Parameter No.14702 SUB(#1) = 1 only.

TMP 0 : Tool post 3 is placed above spindle 1.
1 : Tool post 3 is placed under spindle 1.

NOTE
TMP is available in the case of Parameter No.27402 TME(#0) = 1 only.

TSP 0 : Tool post 3 is placed above spindle 2.
1 : Tool post 3 is placed under spindle 2.
(common parameter among paths)

NOTE
TSP is available in the case of Parameters No.14702 SUB(#1) = 1 and No.27402 TSE(#1) = 1 only.

A.4.9 Parameters for Icon of Path Number Display

These parameters are for Icon of path number display.

27410	P1ICON

(Byte type, FANUC standard settings=0)

P1ICON : ICON number when path-1 is selected
(common parameter among paths)

27411	P2ICON

(Byte type, FANUC standard settings=0)

P2ICON : ICON number when path-2 is selected
(common parameter among paths)

27412	P3ICON

(Byte type, FANUC standard settings=0)

P3ICON : ICON number when path-3 is selected
(common parameter among paths)
= 0 : nothing
= 1 : path-1 with lower direction and opposite to another (2 path system)
= 2 : path-1 with upper direction and opposite to another (2 path system)
= 3 : path-2 with upper direction and opposite to another (2 path system)
= 4 : path-2 with lower direction and opposite to another (2 path system)
= 5 : path-2 with upper direction and parallel to another (2 path system)
= 6 : path-2 with lower direction and parallel to another (2 path system)
= 10 : single path-1 with lower direction (3 path system)
= 11 : right side path-1 with upper direction (3 path system)
= 12 : left side path-1 with upper direction (3 path system)
= 13 : single path-1 with upper direction (3 path system)
= 14 : right side path-1 with lower direction (3 path system)
= 15 : left side path-1 with lower direction (3 path system)
= 20 : right side path-2 with upper direction (3 path system)
= 21 : left side path-2 with upper direction (3 path system)
= 22 : single side path-2 with lower direction (3 path system)
= 23 : right side path-2 with lower direction (3 path system)
= 24 : left side path-2 with lower direction (3 path system)
= 25 : single path-2 with upper direction (3 path system)
= 30 : left side path-3 with upper direction (3 path system)
= 31 : right side path-3 with upper direction (3 path system)
= 32 : single side path-3 with lower direction (3 path system)
= 33 : left side path-3 with lower direction (3 path system)
= 34 : right side path-3 with lower direction (3 path system)
= 35 : single path-3 with upper direction (3 path system)

A.4.10 Other Parameters

This parameter is set for Macro executor and available only in Series 16*i*/18*i*/21*i* systems.

	#7	#6	#5	#4	#3	#2	#1	#0
27500								FSV

(FANUC standard settings = 00000000)

FSV In the case of using “Controlling conversational macro function screens” (#8510) in a Macro executor which is made by MTB, if #8510 is set in displaying a window screen of MANUAL GUIDE *i,*

0 : #8510 is stored and when the window display is closed, #8510 setting is restored.

1 : #8510 is disregarded.

B ALARMS

If the input program or one or more parameter settings are not correct, the following P/S alarms are raised.
When an alarm other than the following P/S alarms is raised, refer to the relevant NC operator's manual.

NOTE
In Series 30*i*, the alarm is not P/S, but MC.

Alarm		Description	
16*i*	**30*i***		
3003	3503	Cause	There is no area that can be machined. The tool is too large for the specified machining area.
		Action	Modify the machining program to use a smaller tool.
3004	3504	Cause	The number of cuts has exceeded the limit.
		Action	Modify the machining program to reduce the number of cuts by, for example, dividing the machining profile.
3005	3505	Cause	The machining start point is invalid.
		Action	Either change the machining start point to a normal one or modify the machining program to automatically decide the point.
3006	3506	Cause	The machining area is invalid.
		Action	Modify the machining program to specify a correct machining area, such as a part figure and a blank figure in a machining profile.
3007	3507	Cause	The cutting conditions are invalid.
		Action	Modify the machining program to specify normal cutting conditions such as the feedrate.
3008	3508	Cause	Finishing is not possible under the specified conditions.
		Action	Review the finishing program.
3009	3509	Cause	An interference occurred. For example, the tool path may interfere with the opposite machining area.
		Action	For example, the cutter radius may be too large for the machining area. Modify the machining program.
3010	3510	Cause	No machining cycle is found. No machining type block is found; only a figure block is specified.
		Action	Modify the machining program by, for example, adding the necessary machining type block.
3011	3511	Cause	No machining cycle is found. An unavailable machining cycle is specified.
		Action	Necessary options may not be added. Contact the machine tool builder.
3012	3512	Cause	G codes that cannot be combined together. A figure block not available to the machining type block is specified.
		Action	Modify the machining program to specify a correct combination of machining type block and figure block.
3013	3513	Cause	The arbitrary figure is invalid.
		Action	Modify the machining program to specify a correct arbitrary figure.
3014	3514	Cause	The figure data is invalid.
		Action	Modify the machining program to specify correct figure data.

Alarm		Description	
16*i*	30*i*		
3015	3515	Cause	The tool offset cannot be read correctly.
		Action	Necessary options, such as the number of offset sets, may not be set. Modify the machining program by, for example, changing the offset number to an available one.
3016	3516	Cause	With a cycle machining command or other 4-digit G commands, necessary arguments are not entered.
		Action	Modify the machining program by, for example, adding necessary arguments.
3025	3525	Cause	The parameter setting is invalid.
		Action	Check the parameter setting.
3030	3530	Cause	The machining type specification is invalid.
		Action	Modify the machining program to specify an appropriate machining type.
3031	3531	Cause	The return mode is invalid.
		Action	Modify the machining program to specify an appropriate return mode.
3032	3532	Cause	The I point coordinate specification is invalid.
		Action	Modify the machining program to specify an appropriate I point coordinates.
3033	3533	Cause	The dwell time specification is invalid.
		Action	For example, a negative value may be entered as the dwell time. Modify the machining program to specify an appropriate dwell time.
3034	3534	Cause	The overrun specification is invalid.
		Action	For example, a negative value may be entered as the overrun. Modify the machining program to specify an appropriate overrun.
3035	3535	Cause	The thread lead specification is invalid.
		Action	For example, a negative value may be entered as the thread lead. Modify the machining program to specify an appropriate thread lead.
3036	3536	Cause	The shift specification is invalid.
		Action	For example, a negative value may be entered as the shift. Modify the machining program to specify an appropriate shift.
3037	3537	Cause	The cutting method specification is invalid.
		Action	A value not specifiable as threading or other cutting methods is entered. Modify the machining program to specify an appropriate cutting method.
3038	3538	Cause	The cutting direction specification is invalid.
		Action	A value not specifiable as turning or other cutting directions is entered. Modify the machining program to specify an appropriate cutting direction.
3039	3539	Cause	The cut direction specification is invalid.
		Action	A value not specifiable as turning or other cut directions is entered. Modify the machining program to specify an appropriate cut direction.
3040	3540	Cause	The movement method specification is invalid.
		Action	A value not specifiable as facing or other movement methods is entered. Modify the machining program to specify an appropriate movement method.
3041	3541	Cause	The chamfer amount specification is invalid.
		Action	A value not specifiable as chamfering or other chamfer amounts is entered, such as a negative value. Modify the machining program to specify an appropriate chamfer amount.
3042	3542	Cause	The ejection amount specification is invalid.
		Action	A value not specifiable as chamfering or other face ejection amounts is entered, such as a negative value. Modify the machining program to specify an appropriate ejection amount.
3043	3543	Cause	The surplus thickness specification is invalid.
		Action	A value not specifiable as pocketing or other surplus thicknesses is entered, such as a negative value. Modify the machining program to specify an appropriate surplus thickness.
3044	3544	Cause	The cutter radius is invalid.
		Action	A value not specifiable as a cutter radius is entered, such as a negative value. Modify the machining program to specify an appropriate cutter radius.

Alarm		Description	
16*i*	30*i*		
3045	3545	Cause	The depth of cut is invalid.
		Action	A value not specifiable as a depth of cut is entered, such as a negative value. Modify the machining program to specify an appropriate depth of cut.
3046	3546	Cause	The cutting angle is invalid.
		Action	A value not specifiable as a cutting angle is entered. Modify the machining program to specify an appropriate cutting angle.
3047	3547	Cause	The cut angle is invalid.
		Action	A value not specifiable as pocketing or other cut angles is entered. Modify the machining program to specify an appropriate cut angle.
3048	3548	Cause	The clearance is invalid.
		Action	A value not specifiable as a clearance is entered, such as a negative value. Modify the machining program to specify an appropriate clearance.
3049	3549	Cause	The finishing allowance is invalid.
		Action	A value not specifiable as a finishing allowance is entered, such as a negative value. Modify the machining program to specify an appropriate finishing allowance.
3050	3550	Cause	The feedrate is invalid.
		Action	A value not specifiable as a feedrate is entered, such as zero. Modify the machining program to specify an appropriate feedrate.
3051	3551	Cause	The number of finishing operations is invalid.
		Action	A value not specifiable as the number of threading or other finishing operations is entered, such as 0. Modify the machining program to specify an appropriate number of finishing operations.
3052	3552	Cause	The approach specification is invalid.
		Action	A value not specifiable as contouring or other approaches is entered. Modify the machining program to specify an appropriate approach.
3053	3553	Cause	The escape specification is invalid.
		Action	A value not specifiable as contouring or other escapes is entered. Modify the machining program to specify an appropriate escape.
3054	3554	Cause	The compensation number is invalid.
		Action	A value not specifiable as contour finishing or other compensation numbers is entered. Modify the machining program to specify an appropriate compensation number.
3055	3555	Cause	The radius of the tool nose is invalid.
		Action	A value not specifiable as turning or other tool nose radii is entered. Modify the machining program to specify an appropriate radius of the tool nose.
3056	3556	Cause	The cutting edge angle/tool angle is invalid.
		Action	A value not specifiable as turning or other cutting edge angles or tool angles is entered. Modify the machining program to specify an appropriate value.
3057	3557	Cause	The virtual tool tip position is invalid.
		Action	A value not specifiable as turning or other virtual tool tip positions is entered. Modify the machining program to specify an appropriate virtual tool tip position.
3058	3558	Cause	The depth-of-cut change ratio is invalid.
		Action	A value not specifiable as turning or other depth-of-cut change ratios is entered. Modify the machining program to specify an appropriate value.
3059	3559	Cause	The return clearance is invalid.
		Action	A value not specifiable as turning or other return clearances is entered. Modify the machining program to specify an appropriate value.
3060	3560	Cause	The tool width is invalid.
		Action	A value not specifiable as turning and grooving or other tool widths is entered. Modify the machining program to specify an appropriate value.
3061	3561	Cause	The pecking is invalid.
		Action	A value not specifiable as turning and grooving or other peckings is entered. Modify the machining program to specify an appropriate value.

Alarm		Description	
16*i*	30*i*		
3062	3562	Cause	The first feed override is invalid.
		Action	A value not specifiable as turning or other first feed overrides is entered. Modify the machining program to specify an appropriate value.
3063	3563	Cause	The spindle speed is invalid.
		Action	A value not specifiable as the spindle speed, for example 0, is entered. Modify the machining program to specify an appropriate spindle speed.
3064	3564	Cause	Tool data aren't enough to execute cycle.
		Action	Set tool data in offset window or tool condition in machining blocks.
3065	3565	Cause	Tool type doesn't fit the cycle type.
		Action	Modify tool data in offset window or the cycle type in machining program.
3066	3566	Cause	Program coordinate system changing function is invalid.
		Action	Necessary options or parameters concerning with program coordinate system changing function may not be set. Confirm these setting.
3070	3570	Cause	Modal can't be sheltered and restored.
		Action	Command G00, G01, G02 or G03 before the cycle is executed.
3071	3571	Cause	Modal can't be sheltered and restored.
		Action	Command G40 before the cycle is executed.
3072	3572	Cause	Modal can't be sheltered and restored.
		Action	Command G80 before the cycle is executed.
3075	3575	Cause	The figure type is invalid.
		Action	A type not specifiable as the figure type of a figure block is selected. Modify the machining program to specify an appropriate type.
3076	3576	Cause	The reference position is invalid.
		Action	A value not specifiable as the reference position of a figure block is entered. Modify the machining program to specify an appropriate value.
3077	3577	Cause	The height/depth is invalid.
		Action	A value not specifiable as the height/depth of a figure block is entered. Modify the machining program to specify an appropriate value.
3078	3578	Cause	The figure length is invalid.
		Action	A value not specifiable as the length of a figure block is entered. Modify the machining program to specify an appropriate value.
3079	3579	Cause	The corner rounding specification is invalid.
		Action	A value not specifiable as the corner rounding of a figure block is entered. Modify the machining program to specify an appropriate value.
3080	3580	Cause	The figure angle specification is invalid.
		Action	A value not specifiable as the angle of a figure block is entered. Modify the machining program to specify an appropriate value.
3081	3581	Cause	The groove width specification is invalid.
		Action	A value not specifiable as the groove width of a figure block is entered. Modify the machining program to specify an appropriate value.
3082	3582	Cause	The figure radius specification is invalid.
		Action	A value not specifiable as the arc radius of a figure block is entered. Modify the machining program to specify an appropriate value.
3083	3583	Cause	The center-to-center distance specification is invalid.
		Action	A value not specifiable as the center-to-center distance of track or other figures is entered. Modify the machining program to specify an appropriate value.
3084	3584	Cause	The pitch specification is invalid.
		Action	A value not specifiable as the pitch angle of a figure block is entered. Modify the machining program to specify an appropriate value.
3085	3585	Cause	The number of holes/grooves specification is invalid.
		Action	A value not specifiable as the number of holes or grooves in a figure block is entered. Modify the machining program to specify an appropriate value.

Alarm		Description	
16*i*	30*i*		
3086	3586	Cause	The coordinate specification is invalid.
		Action	A value not specifiable as a coordinate of a figure block is entered. Modify the machining program to specify an appropriate value.
3087	3587	Cause	The groove depth specification is invalid.
		Action	A value not specifiable as the groove depth of a figure block is entered. Modify the machining program to specify an appropriate value.
3088	3588	Cause	The groove corner specification is invalid.
		Action	A value not specifiable as the corner rounding of a figure block for grooving is entered. Modify the machining program to specify an appropriate value.
3089	3589	Cause	The groove bottom specification is invalid.
		Action	A value not specifiable as the groove bottom of a figure block for grooving is entered. Modify the machining program to specify an appropriate value.
3090	3590	Cause	The thread/hole diameter specification is invalid.
		Action	A value not specifiable as a thread or hole diameter in a figure block is entered. Modify the machining program to specify an appropriate value.
3091	3591	Cause	The number-of-threads specification is invalid.
		Action	A value not specifiable as the number of threads in a figure block to be threaded is entered. Modify the machining program to specify an appropriate value.
3092	3592	Cause	An arbitrary figure is not closed.
		Action	An arbitrary figure entered for facing, pocketing, or turning is not closed. Modify the machining program to specify a closed figure that uses the same point both as the start and end points.
3093	3593	Cause	All figure elements of an arbitrary figure are specified as parts.
		Action	All figure elements of an arbitrary figure entered for turning are specified as "parts." Modify the machining program to specify the figure elements corresponding to actual blanks as "blanks."
3094	3594	Cause	The end face specification is invalid.
		Action	A value not specifiable as the end face of the cycle machining is entered. Modify the machining program to specify an appropriate value.
3095	3595	Cause	The arbitrary figure group specification is not correct.
		Action	In the group specification used in, for example, pocketing with islands, either the start or end block is not entered. Enter an appropriate block.
3098	3598	Cause	The arbitrary figure group specification is not correct.
		Action	In the group specification used in, for example, pocketing with islands, either the start or end block is not entered. Enter an appropriate block.

NOTE
Alarm numbers 3503 through 3598 for the Series 30*i* are changed to 0503 through 0598 when bit 1 of parameter No. 6008 is 1.

C MANUAL GUIDE *i* SETUP METHOD

C.1 GENERAL

In this chapter, the fundamental method of starting up MANUAL GUIDE *i* is described. If it is already installed and running correctly, you need not the following operations.

C.2 HARDWARE

The configuration of hardware for running MANUAL GUIDE *i* is determined according to the combination of other CNC functions ordered. In this manual, it is assumed that the necessary hardware is completed.

C.3 SOFTWARE

Software described below is necessary for MANUAL GUIDE *i*.

C.3.1 Lathe (Series 16*i*/18*i*/21*i*)

(1) In case of using only MANUAL GUIDE *i* Basic function (S781), following software is necessary. As to BY43 and BY44, please select one according to the machine configuration.

File name	Note
BY45_1.MEM	Control software
BY43_2.MEM / BY43_3.MEM	Horizontal Lathe defination software
BY44_2.MEM / BY44_3.MEM	Vertical Lathe defination software

(2) In case of using MANUAL GUIDE *i* Optional Functions like Milling Cycle (S782), Turning Cycle (S783), Animation (S784), Set Up Guidance (S785), or Multi Path Lathe function (S786), following software is necessary instead of that mentioned in (1). As to BY43 and BY44, please select one according to the machine configuration.

File name	Note
BY40_1.MEM	Control software
BY43_2.MEM / BY43_3.MEM	Horizontal Lathe defination software
BY44_2.MEM / BY44_3.MEM	Vertical Lathe defination software
BH00.MEM	Lathe optional software

(3) For 2-path Lathe, following software is necessary addition to that mentioned in (2).

File name	note
BY41.MEM	Control software
BH01.MEM	Lathe optional software

(4) For 3-path Lathe, following software is necessary addition to that mentioned in (2).

File name	note
BY42.MEM	Control software
BH01.MEM	Lathe optional software
BH02.MEM	Lathe optional software

NOTE
BY41*.MEM is not required.

C.3.2 Machining Center (Series 16*i*/18*i*/21*i*)

(1) In case of using only MANUAL GUIDE *i* Basic function (S781), following software is necessary. As to BY46 and BY47, please select one according to the machine configuration.

File name	Note
BY45_1.MEM	Control software
BY46_2.MEM / BY46_3.MEM	Horizontal Milling defination software
BY47_2.MEM / BY47_3.MEM	Vertical Milling defination software

(2) In case of using MANUAL GUIDE *i* Optional Functions like Milling Cycle (S782), Turning Cycle (S783), Animation (S784), or Set Up Guidance (S785), following software is necessary instead of that mentioned in (1). As to BY46 and BY47, please select one according to the machine configuration.

File name	Note
BY40_1.MEM	Control software
BY46_2.MEM / BY46_3.MEM	Horizontal Milling defination software
BY47_2.MEM / BY47_3.MEM	Vertical Milling defination software
BJ00.MEM	Milling optional software

C.3.3 Lathe or Machining Center (Series 30*i*)

(1) In case of using only MANUAL GUIDE *i* Basic function (S781), following software is necessary.
As to BY80 - BY83, please select one according to the machine configuration.

File name	note
BY75.MEM	Control software
BY80.MEM	Horizontal Lathe defination software
BY81.MEM	Vertical Lathe definition software
BY82.MEM	Horizontal Milling defination software
BY83.MEM	Vertical Milling definition software
BY95.MEM	Language software

NOTE
Graphic software is 60VD.

(2) In case of using MANUAL GUIDE *i* Optional Functions like Milling Cycle (S782), Turning Cycle (S783), Animation (S784), or Set Up Guidance (S785), following software is necessary instead of that mentioned in (1).
As to BY80 - BY83 and BJ11- BH11, please select each one according to the machine configuration.

File name	note
BY70.MEM	Control software
BY80.MEM	Horizontal Lathe defination software
BY81.MEM	Vertical Lathe definition software
BY82.MEM	Horizontal Milling defination software
BY83.MEM	Vertical Milling definition software
BY95.MEM	Language software
BJ11.MEM	Lathe optional software
BH11.MEM	Milling optional software

NOTE
Graphic software is 60VA.

C.3.4 Lathe with Compound Machining Function (Series 16*i*/18*i*/21*i*)

(1) In case of using only MANUAL GUIDE *i* Basic function (S781), following software is necessary. As to BY43 and BY44, please select one according to the machine configuration.

File name	Note
BY45_1.MEM	Control software
BY43_2.MEM / BY43_3.MEM	Horizontal Lathe defination software
BY44_2.MEM / BY44_3.MEM	Vertical Lathe defination software

(2) In case of using MANUAL GUIDE *i* Optional Functions like Milling Cycle (S782), Turning Cycle (S783), Animation (S784), Set Up Guidance (S785), following software is necessary instead of that mentioned in (1). As to BY43 and BY44, please select one according to the machine configuration.

File name	Note
BY40_1.MEM	Control software
BY43_2.MEM / BY43_3.MEM	Horizontal Lathe defination software
BY44_2.MEM / BY44_3.MEM	Vertical Lathe defination software
BH10.MEM	Lathe optional software
BJ10.MEM	Milling optional software

C.3.5 Other Machines (Series 16*i*/18*i*/21*i*)

MANUAL GUIDE *i* Basic function (S781) is common for all machine configurations. Following software is necessary. As to BY43 - BY47, please select one according to the machine configuration.

File name	Note
BY45_1.MEM	Control software
BY43_2.MEM / BY43_3.MEM	Horizontal Lathe defination software
BY44_2.MEM / BY44_3.MEM	Vertical Lathe defination software
BY46_2.MEM / BY46_3.MEM	Horizontal Milling defination software
BY47_2.MEM / BY47_3.MEM	Vertical Milling defination software

C.3.6 Other Machines (Series 30*i*)

MANUAL GUIDE *i* Basic function (S781) is common for all machine configurations. Following software is necessary.
As to BY80 - BY83, please select one according to the machine configuration.

File name	Note
BY75.MEM	Control software
BY80.MEM	Horizontal Lathe defination software
BY81.MEM	Vertical Lathe definition software
BY82.MEM	Horizontal Milling defination software
BY83.MEM	Vertical Milling definition software
BY95.MEM	Language software

NOTE
Graphic software is 60VD.

C.3.7 Note

Some functions of MANUAL GUIDE *i* may not work if CNC and Graphic (*) software are old, because MANUAL GUIDE *i* uses the CNC and Graphic functions. So it is recommended to install the latest version of those software.

* In case of 160*i*s/180*i*s/210*i*s, 160*i*/180*i*/210*i* or 30*i*s/300*i*, CNC screen display function corresponds to graphic software.

C.4 PARAMETER SETTING

C.4.1 Lathe

Set the parameters of the cells ☐ in the first path and set those of cells ▒ in each path in case of multi path lathe and lathe with compound machining function.

(1) Set following parameters for MANUAL GUIDE *i* Basic function.

No.	Value	note
3103#3	1	CNC ignores [HELP] key during displaying C executor screen in open CNC. (It is necessary in Series 30*i*.)
3106#2	1	[NEXT DISTANCE] display is available in single block mode It sumultaneously enables to look ahead 1 block. So if you stop the execution by SINGLE BLOCK signal, then edit the next block and re-start, the block before editting is executed.
3206#6	1	No.3106#6, position display change for 3D conversion mode, is effective in MANUAL GUIDE *i* screen
3112#0	0	Servo wave display is not available (It is necessary in Series 30*i*.)
3201#6	1	M02,M30,M99 are not regarded as the end of resister
8661	40	SRAM size for application's backup variables (40KB) (It is necessary in Series 30*i*.)
8662	23	SRAM size for application's backup (23KB) (It is necessary in Series 30*i*.)
8650#1	1	not switched to alarm screen automatically during displaying C executor screen
8650#0	1	C executor transfer [RESET] key code to application
8701#6	1	TV check is skipped during edit operation (It is necessary in Series 30*i*.)
8701#4	1	Vacant value of P code variables can be read. (It is necessary in Series 30*i*.)
8781	0	DRAM size for user program of C executor must be zero (It is necessary in Series 30*i*.)
9000#0	0	Debug parameter for macro program must be zero
14700#7=0	0	Enable to run MANUAL GUIDE *i*
14794-14795	*	Assign function key for start MANUAL GUIDE *i*
9071	90	P code number for MANUAL GUIDE *i* (in Series 30*i* with optional function)
9072	1	Block number for dealing with Macro sentence continuously in Execute Macro program for MANUAL GUIDE *i*. (in Series 30*i* with optional function)

*

14794#0=1: [POS] key is assigned for start
#1=1: [PRG] key is assigned for start
#2=1: [OFS] key is assigned for start
#3=1: [SYS] key is assigned for start
#4=1: [MES] key is assigned for start
#5=1: [GRP] key is assigned for start

14795#0=1: [CUSTOM](AUX screen) key is assigned for start
(No.8652#5 must be set to 1 in case macro screen does not exist)
#1=1: [CUSTOM](MCR screen) key is assigned for start
(No.8652#6 must be set to 1 in case macro screen does not exist)
#2=1: [CUSTOM](MENU screen) key is assigned for start
(No.8652#7 must be set to 1 in case macro screen does not exist)
#3=1: Blank key is assigned for start
(No.8652#4 must be set to 1)

(2) Set following parameters for displaying load meter. Following parameters depend on machining configuration. Refer to Parameter Manual (B-63530EN) for detail.

No.	Value	note
1023	≠0	Servo axis number of control axis
2086	≠0	regular current parameter
2165	≠0	maximum value of amplifier current
3151-3158	≠0	Axis number to display load meter (Note)
4127	≠0	Displayed value in maximum load

NOTE
In Series 30*i*, there are not the parameters from No.3154 to No.3158.

(3) Set following parameters for optimizing cycle menu and icons of Turning Cycle and Milling Cycle functions according to machine configuration. In case of no above cycle options, it is unnecessary to set.

No.	Value	note
14702#0	→	0:Vertical, 1:Horizontal for icons
27003#1-#0	Xxxxxx00	xxxxxx00:Lathe *
27103#7,#3-#0	→	0xxx0001: 2 axes (XZ) lathe 0xxx0010: 3 axes (XZC) lathe 0xxx0100: 4 axes (XZCY) lathe 0xxx1000: 5 axes (XZCYB) lathe 1xxx0001: 2 axes (XZ) lathe with sub spindle 1xxx0010: 3 axes (XZC) lathe with sub spindle 1xxx0100: 4 axes (XZCY) lathe with sub spindle 1xxx1000: 5 axes (XZCYB) lathe with sub spindle

* In case 27003= xxxxxx00 & 27103=0xxx0000, all of cycle menu is displayed
* This parameter is available on BY40/04 and later.

NOTE
When the parameter No.27003 or No.27103 is set, please sure to push [F] key on NOW LOADING screen after Power ON. The necessary parameters are set automatically. (When the necessary parameters are set, the message of "NOW SETTING PARAMETERS" is displayed on the left side of NOW LOADING screen.)

(4) Set following parameters for output of polar coordinate interpolation command (G12.1) and cylindrical interpolation command (G7.1) in cycle motion. In case of no Cs axis or Milling Cycle option, it is unnecessary to set.

No.	Value	note
27000#2	→	0: G12.1/G13.1 are not output in cycle motion on XC plane 1: G12.1/G13.1 are automatically output in cycle motion on XC plane
27000#3	→	0: G7.1Cxx/G7.1C0 are not output in cycle motion on ZC plane 1: G7.1Cxx/G7.1C0 are automatically output in cycle motion on ZC plane

(5) Set following parameters for optimizing C axis figure data input screen, if two rotate (Cs) axes in main and sub spindle exist and have different axis name. In case of no Milling Cycle option, it is unnecessary to set.

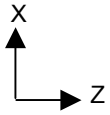
No.	Value	note
27001#3-#0	→	xxxx0011: Tab for C or A axis selection is displayed in figure data input screen xxxx0101: Tab for C or B axis selection is displayed in figure data input screen xxxx1001: Tab for C or E axis selection is displayed in figure data input screen

(6) Set following parameters for adapting tool motion to workpiece coordinate in MANUAL GUIDE *i* Animation function. In case of no Animation option, it is unnecessary to set.

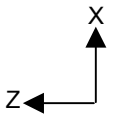
No.	Value	note
14706	*	Workpiece coordinate for main spindle
14707	*	Workpiece coordinate for sub spindle if existing

* Setting value for 14706 and14707

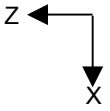
16 : Right hand coordinate, right =+Z and upper =+X



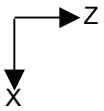
17 : Right hand coordinate, right =-Z and upper =+X



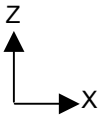
18 : Right hand coordinate, right =-Z and upper =-X



19 : Right hand coordinate, right =+Z and upper =-X



20 : Right hand coordinate, right =+X and upper =+Z



(7) Set following parameters for adapting rotate (Cs) axis motion to workpiece coordinate. If no Cs axis or Animation option, it is unnecessary to set.

In case of one Cs axis of main spindle

No.	Value	note
14717	→	Cs axis number

In case of two Cs axis with main and sub spindle

No.	Value	note
27301	→	Cs axis number of main spindle
27302	→	Cs axis number of sub spindle
27312	xxxxxxx1	switch Cs axis number by spindle selection command for animation, “G1998 S__;” .

NOTE
14717 is used as working number for Cs axis.

(8) Set following parameters for adapting rotate axis motion, which tilts tool head or table, to workpiece coordinate. If no such axis or Animation option, it is unnecessary to set.

No.	Value	note
14718	→	rotate axis number, which tilts tool head or table

(9) Set following parameters for Set up Guidance function. If no Set up Guidance option, it is unnecessary to set.

No.	Value	note
12381	10000*	Top number of variables for measurement condition
12382	10180*	Top number of variables for calibration
12383	10300*	Top number of variables for measurement execution
12384	10500*	Top number of variables for storing measurement result
12385	699*	Amount number of variables for storing measurement result
12386	951*	Address of PMC R signal for measurement
12387	0*	Macro program number for tool select
12388	3600*	Macro program number for measurement execution

* Settings described above is a sample. Set proper value according to macro program made by MTB.

(10) Set following parameters for Multi Path Lathe function. If no Multi Path Lathe option, it is unnecessary to set.

No.	Value	note
14703#3	1	Process list editing is available
27400#0	→	0:Softkey switches the display for each path 1:Head select signal switches the display for each path
14701#1	→	0:Head#1 can not machine with spindle#2 1:Head#1 can machine with spindle#2
14701#2	→	0:Head#1 machines upper part of spindle#1 1:Head#1 machines lower part of spindle#1
14701#3	→	0:Head#1 machines upper part of spindle#2 1:Head#1 machines lower part of spindle#2
27401#0	→	0:Head#2 can not machine with spindle#1 1:Head#2 can machine with spindle#1
27401#1	→	0:Head#2 can not machine with spindle#2 1:Head#2 can machine with spindle#2
27401#2	→	0:Head#2 machines upper part of spindle#1 1:Head#2 machines lower part of spindle#1
27401#3	→	0:Head#2 machines upper part of spindle#2 1:Head#2 machines lower part of spindle#2
27402#0	→	0:Head#3 can not machine with spindle#1 1:Head#3 can machine with spindle#1
27402#1	→	0:Head#3 can not machine with spindle#2 1:Head#3 can machine with spindle#2
27402#2	→	0:Head#3 machines upper part of spindle#1 1:Head#3 machines lower part of spindle#1
27402#3	→	0:Head#3 machines upper part of spindle#2 1:Head#3 machines lower part of spindle#2
27410	*	Icon number for path 1 display
27411	*	Icon number for path 2 display
27412	*	Icon number for path 3 display

* Icon number

Number	0	1	2	3	4	5	6
Icon							

Number	10	11	12	13	14	15
Icon						

Number	30	31	32	33	34	35
Icon						

C.4.2 Machining Center

(1) Set following parameters for MANUAL GUIDE *i* Basic function.

No.	Value	note
3103#3	1	CNC ignores [HELP] key during displaying C executor screen in open CNC. (It is necessary in Series 30*i*.)
3106#2	1	[NEXT DISTANCE] display is available in single block mode It sumultaneously enables to look ahead 1 block. So if you stop the execution by SINGLE BLOCK signal, then edit the next block and re-start, the block before editting is executed.
3206#6	1	No.3106#6, position display change for 3D conversion mode, is effective in MANUAL GUIDE *i* screen
3112#0	0	Servo wave display is not available (It is necessary in Series 30*i*.)
3201#6	1	M02,M30,M99 are not regarded as the end of resister
8661	40	SRAM size for application's backup variables (40KB) (It is necessary in Series 30*i*.)
8662	23	SRAM size for application's backup (23KB) (It is necessary in Series 30*i*.)
8650#1	1	not switched to alarm screen automatically during displaying C executor screen
8650#0	1	C executor transfer [RESET] key code to application
8701#6	1	TV check is skipped during edit operation (It is necessary in Series 30*i*.)
8701#4	1	Vacant value of P code variables can be read. (It is necessary in Series 30*i*.)
8781	0	DRAM size for user program of C executor must be zero (It is necessary in Series 30*i*.)
9000#0	0	Debug parameter for macro program must be zero
14700#7=0	0	Enable to run MANUAL GUIDE *i*
14794-14795	*	Assign function key for start MANUAL GUIDE *i*
9071	90	P code number for MANUAL GUIDE *i* (in Series 30*i* with optional function)
9072	1	Block number for dealing with Macro sentence continuously in Execute Macro program for MANUAL GUIDE *i*. (in Series 30*i* with optional function)

*

14794#0=1: [POS] key is assigned for start
#1=1: [PRG] key is assigned for start
#2=1: [OFS] key is assigned for start
#3=1: [SYS] key is assigned for start
#4=1: [MES] key is assigned for start
#5=1: [GRP] key is assigned for start

14795#0=1: [CUSTOM](AUX screen) key is assigned for start
(No.8652#5 must be set to 1 in case macro screen does not exist)
#1=1: [CUSTOM](MCR screen) key is assigned for start
(No.8652#6 must be set to 1 in case macro screen does not exist)
#2=1: [CUSTOM](MENU screen) key is assigned for start
(No.8652#7 must be set to 1 in case macro screen does not exist)
#3=1: Blank key is assigned for start
(No.8652#4 must be set to 1)

(2) Set following parameters for displaying load meter. Following parameters depend on machining configuration. Refer to Parameter Manual (B-63530EN) for detail.

No.	Value	note
1023	≠0	Servo axis number of control axis
2086	≠0	regular current parameter
2165	≠0	maximum value of amplifier current
3151-3158	≠0	Axis number to display load meter (Note)
4127	≠0	Displayed value in maximum load

NOTE
In Series 30*i*, there are not the parameters from No.3154 to No.3158.

(3) Set following parameters for optimizing cycle menu and icons of Turning Cycle and Milling Cycle functions according to machine configuration. In case of no above cycle options, it is unnecessary to set.

No.	Value	note
14702#0	→	0:Vertical, 1:Horizontal for icons
27003#1-#0	→	Xxxxxx01: 3 axes (XYZ) Machining Center xxxxxx10: 5 axes (XYZCB) Machining Center
27103#7,#3-#0	0xxx0000	0xxx0000: Machining Center

* In case 27003= xxxxxx00 & 27103=0xxx0000, all of cycle menu is displayed
* This parameter is available on BY40/04 and later.

NOTE
When the parameter No.27003 or No.27103 is set, please sure to push [F] key on NOW LOADING screen after Power ON. The necessary parameters are set automatically. (When the necessary parameters are set, the message of “NOW SETTING PARAMETERS” is displayed on the left below of NOW LOADING screen.)

(4) Set following parameters for output of polar coordinate interpolation command (G12.1) and cylindrical interpolation command (G7.1) in cycle motion. In case of no Cs axis or Milling Cycle option, it is unnecessary to set.

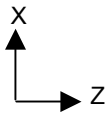
No.	Value	note
2700#2	→	0: G12.1/G13.1 are not output in cycle motion on XC plane 1: G12.1/G13.1 are automatically output in cycle motion on XC plane
2700#3	→	0: G7.1Cxx/G7.1C0 are not output in cycle motion on ZC plane 1: G7.1Cxx/G7.1C0 are automatically output in cycle motion on ZC plane

(5) Set following parameters for adapting tool motion to workpiece coordinate in MANUAL GUIDE *i* Animation function. In case of no Animation option, it is unnecessary to set.

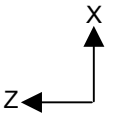
No.	Value	note
14706	*	Workpiece coordinate

* Setting value for 14706

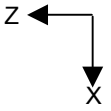
16 : Right hand coordinate, right =+Z and upper =+X



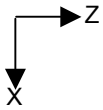
17 : Right hand coordinate, right =-Z and upper =+X



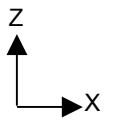
18 : Right hand coordinate, right =-Z and upper =-X



19 : Right hand coordinate, right =+Z and upper =-X



20 : Right hand coordinate, right =+X and upper =+Z



(6) Set following parameters for adapting rotate (Cs) axis motion to workpiece coordinate. If no Cs axis or Animation option, it is unnecessary to set.

No.	Value	note
14717	→	Cs axis number

(7) Set following parameters for adapting rotate axis motion, which tilts tool head or table, to workpiece coordinate. If no such axis or Animation option, it is unnecessary to set.

No.	Value	note
14718	→	Rotate axis number, which tilts tool head or table

(8) Set following parameters for Set up Guidance function. If no Set up Guidance option, it is unnecessary to set.

No.	Value	note
12381	10000*	Top number of variables for measurement condition
12382	10180*	Top number of variables for calibration
12383	10300*	Top number of variables for measurement execution
12384	10500*	Top number of variables for storing measurement result
12385	699*	Amount number of variables for storing measurement result
12386	951*	Address of PMC R signal for measurement
12387	0*	Macro program number for tool select
12388	3600*	Macro program number for measurement execution

* Settings described above is a sample. Set proper value according to macro program made by MTB.

C.4.3 Other Machines

Refer to C.4.1 Lathe or C.4.2 Machining Center, which is closer to your machine.

C.5 M CODE OUTPUTED DURING CYCLE EXECUTING

Following M code is output in milling cycle

- M code for rigid tapping
- M code for clamping and unclamping rotate axis as “C”

C.5.1 M code for Rigid Tapping

(1) M code is output in following cycles in case that rigid tap is specified and parameter No.5200#0 is zero.

G1002 (Tapping Cycle for Machining Center by live tool)
G1112 (Tapping Cycle for Lathe by live tool)
G1102 (Tapping Cycle for Lathe by rotating work)

(2) M code is output in following timing

G90
G00 X(x1) Y(y1) Z(I point)
G99
M29 S(spindle)*
G84 Z(depth) R(R point) P(dwell) F(feed)
X(x2) Y(y2)
X(x3) Y(y3)
G80
G00 Z(I point)

* M code value depends on parameter No.5210 and No.5212.

C.5.2 M code for Clamping and Unclamping Rotate Axis as “C”

(1) M code is automatically output in C axis, which position C axis in the cycle motion.

In Hole Machining Cycles combined with following figures, C axis clamping and unclamping M codes are automatically output.

a) C-axis hole on the polar coordinate plane (circle points): G1572
b) C-axis hole on the polar coordinate plane (random points): G1573
c) C-axis hole on the cylindrical surface (circle points): G1672
d) C-axis hole on the cylindrical surface (random points): G1673

In Grooving Cycles combined with following figures, C axis clamping and unclamping M codes are automatically output.

e) X-axis groove on the polar coordinate plane: G1571
f) Z-axis groove on the cylindrical surface: G1671

(2) M code is output in following timing

In case of Hole Machining Cycles

```
G90
G17
Mb*
G00 X(x1) C(c1) Z(I point)
G99
G81 Z(depth) R(R point) F(feed) Ma*
Mb
X(x2) C(c2) Ma*
Mb
X(x3) C(c3) Ma*
Mb
G80
G00 Z(I point)
```

* Ma means C axis clamping M code, Mb means C axis unclamping one.
M code value depends on parameter No.27005, No.27006, No.27011 and No.27012

In case of Grooving Cycles

```
Mb *
G17
G0 X 80. C0.
G0 Z2.
Ma *
G1Z-10.F100.
G1 X40. F100.
G1Z2.F100.
Mb *
G17
G0 X80. C120.
G0 Z2.
Ma *
G1Z-10.F100.
G1 X40. F100.
G1Z2.F100.
Mb *
```

* Ma means C axis clamping M code, Mb means C axis unclamping one.
  M code value depends on parameter No.27005, No.27006, No.27011 and No.27012

C.6 OPTIONAL FUNCTIONS AVAILABLE

Following optional functions are installed with MANUAL GUIDE *i*, when a CNC is shipped.

C.6.1 Lathe (Series 16*i*/18*i*/21*i*)

Function		NO.
MANUAL GUIDE *i* Basic	(S781)	J734(*) J872 J972 J973 J738#256K
MANUAL GUIDE *i* Turning Cycle	(S783)	J878 J930 J855

* User application on C executor is not available.

C.6.2 Machining Center (Series 16*i*/18*i*/21*i*)

Function		NO.
MANUAL GUIDE *i* Basic	(S781)	J734(*) J872 J972 J973 J956 J738#256K
MANUAL GUIDE *i* Milling Cycle	(S782)	J890 J931

* User application on C executor is not available.

C.6.3 Lathe, Machining Center (Series 30*i*)

Function		NO.
MANUAL GUIDE *i* Basic	(S781)	J888 J873 J972 J973 J956 J760 J738#256K
MANUAL GUIDE *i* Turning Cycle	(S783)	J890 J855 J930
MANUAL GUIDE *i* Milling Cycle	(S782)	J890 J930

INDEX

<D>

<E>

<F>

<G>

<N>

<O>

<P>

<*Q*>

<*R*>

<*S*>

<T>

<U>

<V>

<W>

Revision Record

FANUC MANUAL GUIDE *i* OPERATOR'S MANUAL (B-63874EN)

Edition	Date	Contents	Edition	Date	Contents
04	Dec., 2003	Addition of descriptions for Series 30*i* Addition of following items Notes on creating programs, Undo and redo, Arbitrary figure copy functions, Editing a fixed form figure subprogram, Program restart function, Accessing tool data base function, Memory card input/output of fixed form sentences, Spindle movement animation for automatic lathes, Support for folder management Correction of errors			
03	Jul., 2003	It is changed to the contents that the whole of the manual is often understood drastically. Addition of following items Changing work coordinate during machining simulation, Tool data base function, Editing of free figure of subprogram form, Multi-path lathe functions, Modal display of offset types, Example of programming, and Manual guide *i* setup method Correction of errors			
02	Jan., 2003	Addition of following items Displaying the drawing-during-machining window, BG editing, NC statement conversion, Next-block display function, Operations in background editing, Operations with the NC statement conversion function, Shortcut key operations, Help screen, Memory card input/output function, Handling large programs, Fixed-point format calculation function, Automatic setting of initial value data, Rear end facing by milling, Subspindle-based c-axis machining, Rear end facing by turning hole machining (workpiece rotation), Required parameters, Parameters common to lathing cycles, Parameters for machining simulation (animated) Correction of errors			
01	Aug., 2002	———	05	Jul., 2004	Addition of following items Machining time display, Prism blank figure, XA plane, MANUAL GUIDE i Simulator for PC Correction of errors

Series 16*i* /18*i* /21*i* - MB/TB MANUAL GUIDE *i*
Addition of supplemental instructions into an Operator's Manual

1 . Type of applied technical documents

Name	FANUC MANUAL GUIDE *i* Operator's Manual
Spec. No.／Ed.	B-63874EN/05

2 . Summary of Change

Group	Name／Outline	New, Add, Correct, Delete	Applicable Date
Basic Function			
Optional Function	VII. Supplemental Instructions	Add	Immediately
Unit			
Maintenance Parts			
Notice			
Correction			
Another			

VII. SUPPLEMENTAL INSTRUCTIONS

1 Outline

Various functions described in this Part VII are available from the version 06 of MANUAL GUIDE *i* for Series16*i*/18*i*/21*i*.
As to the version 05 and before it, refer to the other Parts. In case of that there is overlapping part between this Part and other Parts, the content described in this Part becomes available if you are using the version 06.

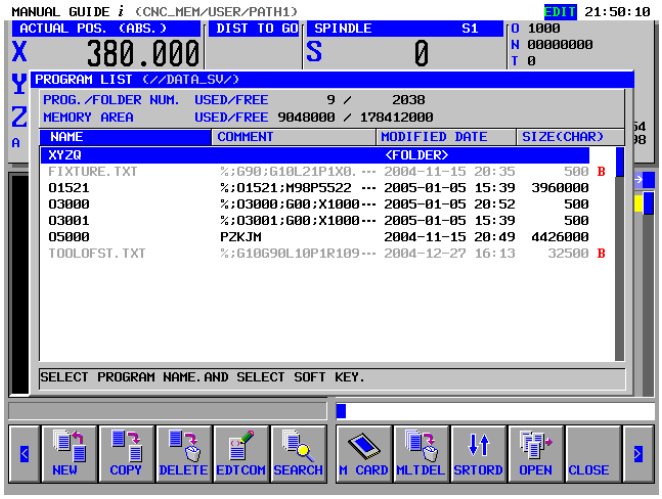
The version number on MANUAL GUIDE *i* for Series30*i* and Series 0*i* is as follows.

For Series30*i*, the version is 08.
For Series 0*i*, the version is 02.

2 Display Attribute of Binary File (Only FS30i)

2.1. Details

On program list window, if there is a file other than program (hereinafter call Binary File) on data server, the file is displayed in gray character, and shows a letter “B” in red at very right on a screen as a figure below.



3 Alter the Cursor Position After the Copy

3.1. Details

NEW PROG	O LIST	↑ SRCH	↓ SRCH	OSRCH	COPY	CUT	DELETE	KEYPST	PASTE

By pressing [COPY] to display a message of copy range selection, press [COPY] after selecting the range by cursor key. (Displays in yellow) Copied contents are saved to clipboard.

After operated copy, whether return a cursor to the top block of a selected range, or keep a cursor to the end block of a selected range, are depending on a setting of below parameter.

Parameter No.14705#1=0 : Return a cursor to the top block of a selected range.

Parameter No.14705#1=1 : Keep a cursor to the end block of a selected range.

4 Maximum Characters in a Block

4.1. Details

On MANUAL GUIDE *i*, program containing over 199 characters cannot be operated.
When selecting a program containing over 200 characters (including spaces), message ”Block exceed maximum size” is displayed, and all operations cannot be performed.
When the message is displayed, please select another program on NC screen.

5 Specify Side Finish Feedrate of Free Figure at Milling

5.1. Outline

On free figure at milling, side finish feed rate per element can be set. This function is available with setting parameter No27002<#4> = 1.
When side finish feed rate is available, side finish federate at [ATTRIBUTE] tab of figure element indicate as following plane

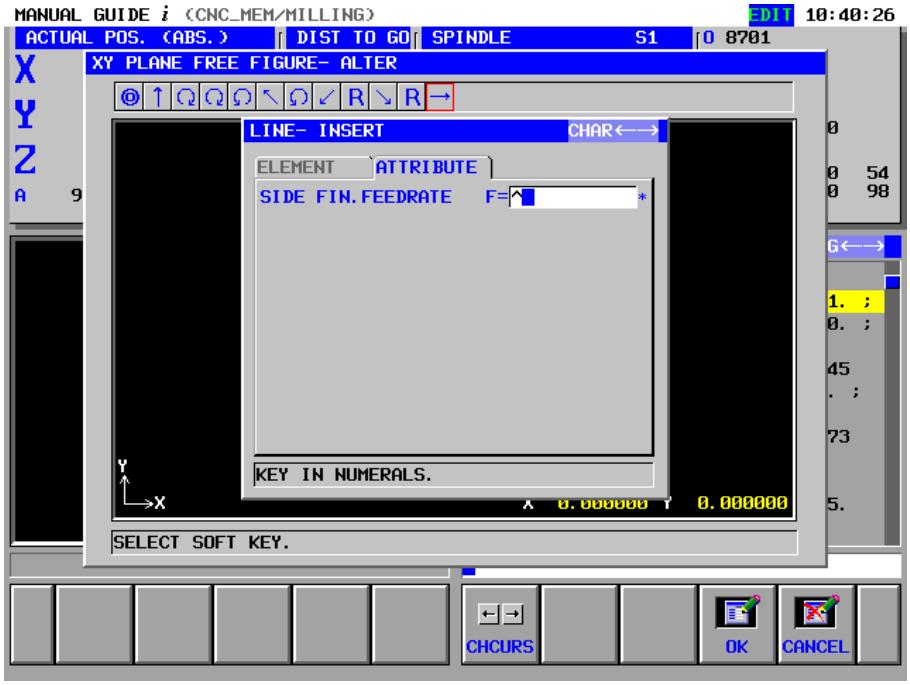
- XY FREE FIGURE BLOCK
  - ・Line: G1201
  - ・Arc(CW): G1202
  - ・Arc (CCW): G1203
  - ・Chamfer: G1204
  - ・Corner R: G1205

- YZ FREE FIGURE BLOCK
  - ・Line: G1301
  - ・Arc(CW): G1302
  - ・Arc (CCW): G1303
  - ・Chamfer: G1304
  - ・Corner R: G1305

- XC FREE FIGURE BLOCK
  - ・Line: G1501
  - ・Arc(CW): G1502
  - ・Arc (CCW): G1503
  - ・Chamfer: G1504
  - ・Corner R: G1505

- ZC FREE FIGURE BLOCK
  - ・Line: G1601
  - ・Arc(CW): G1602
  - ・Arc (CCW): G1603
  - ・Chamfer: G1604
  - ・Corner R: G1605

- XA FREE FIGURE BLOCK
  - Line: G1701
  - Arc(CW): G1702
  - Arc (CCW): G1703
  - Chamfer: G1704
  - Corner R: G1705



5.2. Machining Type Side Finish Feed Rate is Available

When a value [Side Finish Feed Rate] is set, it is cut by [Side Finish Feed Rate] instead of [Single Cut Feed Rate] set by machining type block on cut feed rate used [Single Cut Feed Rate].
At this event, the available cycles to set a side finish federate are as followings.

- **CONTOURING**
  - Outer Wall Side Finishing (G1062)
  - Inner Wall Side Finishing (G1066)
  - Partial Side Finishing (G1070)

- **EMBOSSING**
  - Side Finishing (G1082)

However, [Side Finish Feed Rate] cannot be specified on blank figure of embossing (figure type of start point element is concave free figure at embossing). Input item [Side Finish Feed Rate] is also not displayed.

- **Pocketing**
  ・Side Finishing (G1042)

- **Grooving**
  ・Side Finishing (G1052)

5.3. Output Side Finish Feed Rate

When output a program of figure element, entered value to [Side Finish Feed Rate] is output as a following format continuing to address F.
When [Side Finish Feed Rate] is “blank”, address F is not output.

（Example of line of XY plane） G1201 H-10. V-20. K5. C-10. L0. M0. **F50.** ;

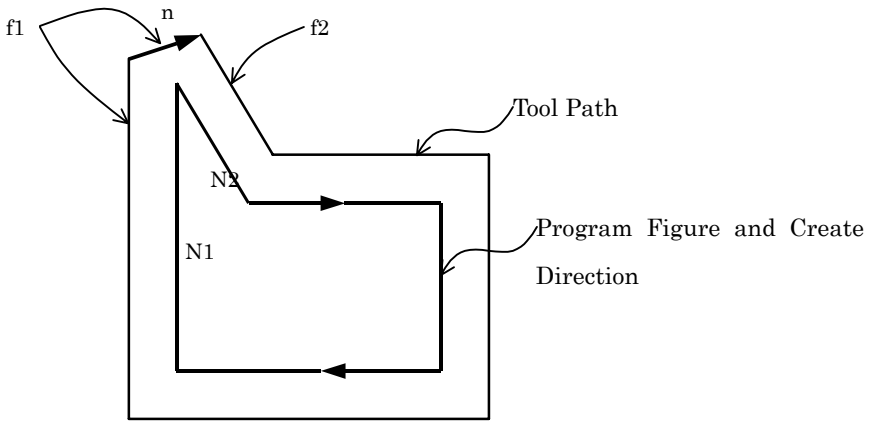
Element that a setting value [Side Finish Feed Rate] is “0” or ”blank”, it is cut by a value [Single Cut Feed Rate] which is set by machining type block.

5.4. Number of Finishing

[Side Finishing] at contouring and grooving, [Cut Number (cut number at finishing)] is possible to set, and [Side finish feed rate] is available at each cutting.

5.5. Feed Rate of Block Added when Machining

When programming a figure of acute angle as figure below, in some case a new block is added for route of tool at a corner of acute angle. At this event, a feed rate of added block is set as a federate specified at element right before creating a figure. For example, when f1 is set for a value [Side Finish Feed Rate] to program figure block N1 as the figure below, a feed rate of added block n is set to f1.



5.6. Figure Copying

When operating a figure copy, a value of [Side Finish Feed Rate] of copied parts element is set “blank”. When [Side Finish Feed Rate] is need to set, alter by entering a value at [Alter] soft key at free figure entry screen.
At a figure copy, it is converted that a corner is for element of arc, and chamfering is for element of line. At this time, a value of [Side Finish Feed Rate] is set to “blank”.

5.7. NC Program Conversion

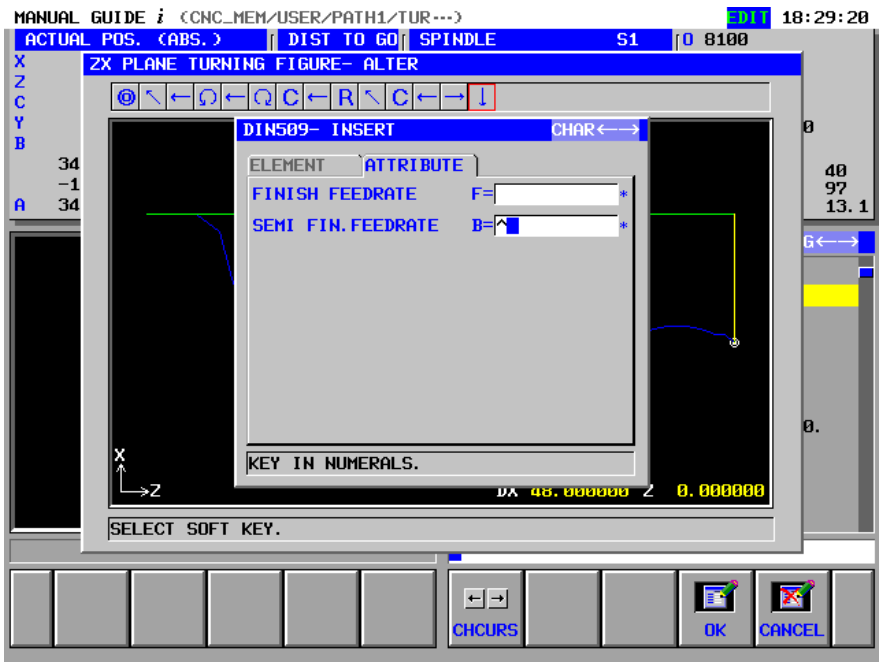
At NC program conversion, a value specified to [Side Finish Feed Rate] at converted cutting block is output continued by address F.
When a setting value of [Side Finish Feed Rate] is set to “0” or “blank”, a value of [Single Cut Feed Rate] is output.

6 Specify Semi Finish Feed Rate of Free Figure for Turning (ZX Plane)

6.1. Outline

On free figure at turning, semi finish feed rate per element can be set. This function is available with setting parameter No27102<#4> = 1. When semi finish feed rate is available, semi finish feed rate at 2nd row of [Attribute] tab of the following figure element or necking figure are displayed.

- Line G1451
- Arc(CW) G1452
- Arc(CCW) G1453
- Chamfer G1454
- Corner R G1455
- DIN509
- DIN509F
- DIN76



6.2. Machining Type that Semi Finish Feed Rate is Available

Cycle that setting of semi finish feed rate is available.

- Outer Semi Finishing G1123
- Inner Semi Finishing G1124
- Face Semi Finishing G1125

6.3. Output a Program of Semi Finish Feed Rate

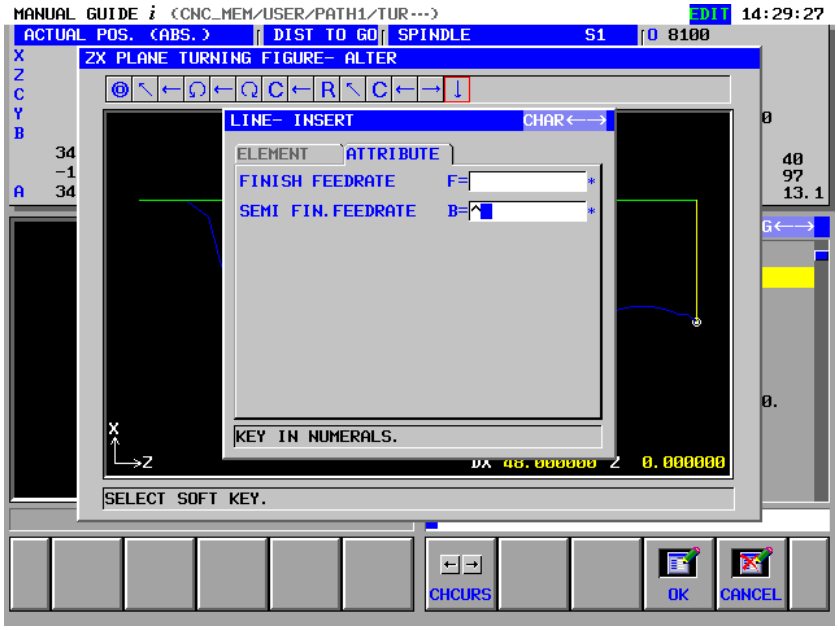
When output a program of figure element, a value entered to [Semi Finish Feed Rate] of figure element is output as a following format continuing to address B. When a value of [Semi Finish Feed Rate] is “0” or ”blank”, address B is not output.

（Example：If it is a line. G1451 H-2. V17.5 K4. D17.5 A45. L0. M0. T1. **Bxx** ;）

When element that a setting value of [Semi Finish Feed Rate] is “0” or “blank”, it is cut by [Feed Rate] specified at machining block of semi finishing.

6.4. Necking Figure

As the same as figure element, semi finish feed rate can set at [Element Attribute] of necking figure. Set value is automatically entered the same value to each element (Line, Arc, Corner R). At free figure entry screen, open an element created by soft key [Alter] to change the setting value.



6.5. Figure Copying

When operating a figure copy, a value of [semi finish feed rate] of copied part element is set “blank”. When [semi finish feed rate] is required to set, change the value by a soft key [Alter] at free figure entry screen.
At figure copy, corner R is converted to an element of arc, chamfering is converted to line element. At this event, a value of [semi finish feed rate] is set to “blank”.

6.6. NC Program Conversion

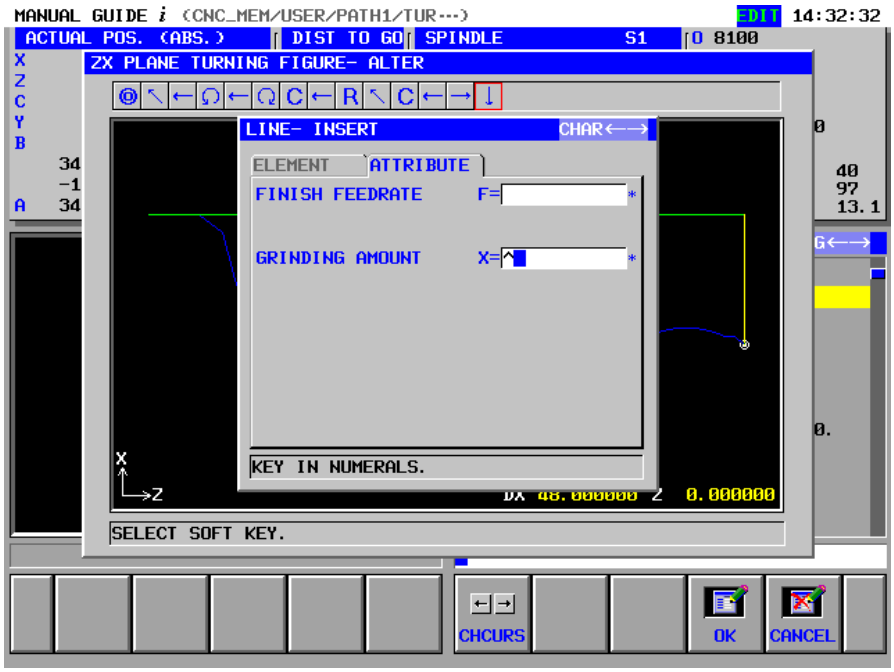
On NC program conversion, a value specified at [Semi Finish Feed Rate] at converted cutting block is output continuing to address F.
When set value of [Semi Finish Feed Rate] is set to “0” or “blank”, a value of [Feed Rate] set at machining block of semi finishing is output.

7 Specify a Griding Amount of Free Figure for Turning (ZX Plane)

7.1. Outline

At a free figure for turning, grinding amount can be set per element. This function is available by setting a parameter No.27102<#3> is set to “1”. When a grinding amount is available, Input item of grinding amount is displayed at 3rd row of [Attribute] tab of figure element listed below. At turning grooving, input item of grinding amount is displayed; however, the entered grinding amount is invalid.

- Line G1451
- Arc (CW) G1452
- Arc (CCW) G1453
- Chamfer G1454
- Corner R G1455



7.2. Machining Grinding Amount is Available

Cycle that a setting of grinding amount is available.

- Outer Roughing G1120
- Inner Roughing G1121
- Face Roughing G1122
- Outer Semi Finishing G1123
- Inner Semi Finishing G1124
- Face Semi Finishing G1125
- Outer finishing G1126
- Inner finishing G1127
- Face finishing G1128

7.3. Output a Program of Grinding Amount

When output a program of figure element, a value entered to [Grinding Amount] is output by a following format continuing to the address X. When a value of [Grinding Amount] is set "blank", address X is not output.

（Example：When Line G1451 H-2. V17.5 K4. D17.5 A45. L0. M0. T1. **Xxx**；）

7.4. Unit of Grinding Amount and Setting Range

Setting unit is input unit. Value of setting range is as follows.

IS-A: -999999.99～999999.99mm, -99999.999～99999.999inch
IS-B: -99999.999～99999.999mm, -9999.9999～9999.9999inch
IS-C: -9999.9999～9999.9999mm, -999.99999～999.99999inch
IS-D: -999.99999～999.99999mm, -99.999999～99.999999inch
IS-E: -99.999999～99.999999mm, -9.9999999～9.9999999inch

7.5. Use with a Finish Amount Specified at Machining Block.

When [Grinding Amount] is set, a specified value is reflected and corrected to the last part figure. Machining cycle considered a value of [X axis direction Finish Amount] and [Z axis direction Finish Amount] specified at machining block for corrected last part figure.

7.6. Necking Figure Inputting

At necking figure, [Grinding Amount] is not specified. However created necking figure is spread for matching element of line, arc, and corner R, and it is possible to input [Grinding Amount] by opening an element spread by soft key [Alter] at free figure entry screen.

7.7. Figure Copying

When operating a figure copy, a value of [Grinding Amount] of copied part element is set “blank”. When it is required to set, input a value by soft key [Alter] at free figure entry screen to change. At figure copy, chamfering is converted to line element; however, a value of [Grinding Amount] is set “blank”.

7.8. Direction that Grinding Amount is Available

Direction of specified [Grinding Amount] is a normal direction of each element.

8 Combining Tool Databese and Tool Management Function

When a option of tool management function is available, data table on tool management function can be displayed at MANUAL GUIDE *i* screen. In this instance, setting screen of tool database is added to the table of tool management function.

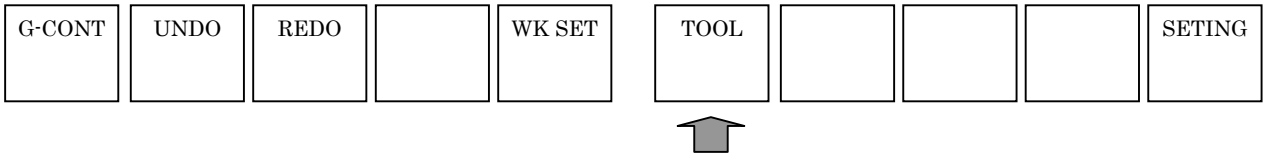
NOTE

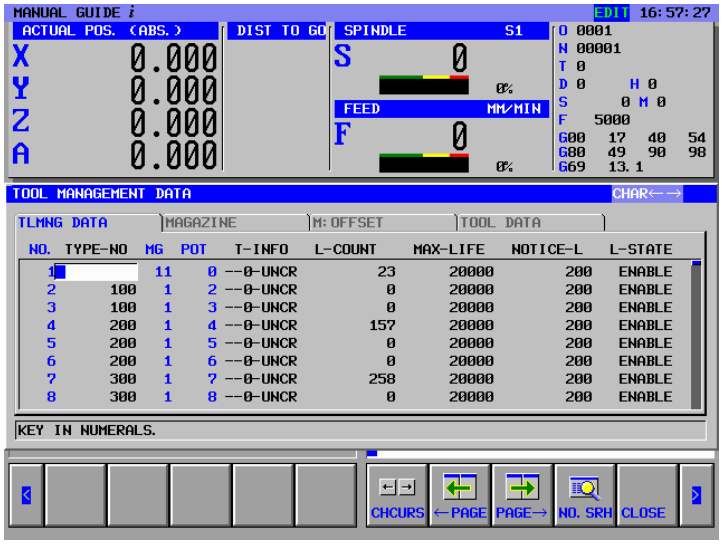
1 The option of tool management function is needed to use this function.
2 This function is available by setting a parameter No.14705#5 = 1
3 Please refer [A-78614 Specifications of Tool Management Function] for details of tool management function.

8.1. Setting of Tool Management Data

8.1.1 Tool Management Data Screen

Press ［TOOL］ to display tool management data window.
In this window, all of tool management data, magazine data, tool offset and tool data can be set.

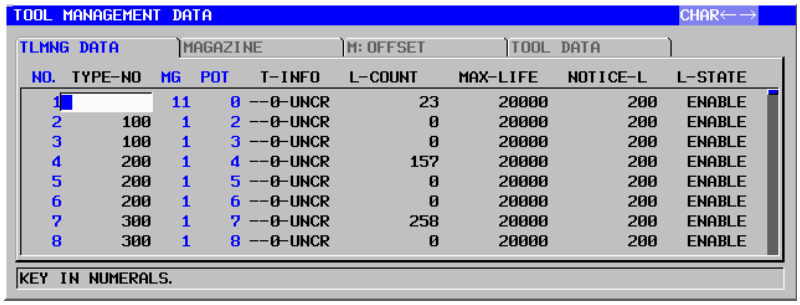




8.1.2 Tool Management Data Tab

At the tool data management data tab, offset number, life state, spindle speed, feedrate, customize information and tool information are possible to set.

At the below screen, life state can be set.



The following data are possible to set.

NO.（Tool Management Data Number）

Display a tool kind number（T code）. It only displays, and cannot be set.

TYPE-NO

Display a tool type number（T code）.

MG（Magazine Number）

Indicate a magazine number. It only displays, and cannot be set.

POT

Indicate a pot number. It only displays, and cannot be set.

T-INFO

Indicate the 6 kind of information listed below in order from the right.

2 bit from the left are not used.

・Tool Management Data : Enable（R）／Unable（-）

・Life Count Type : Time（T）／Count（C）

・Tool Type : Large diameter tool（B）／Regular Tool（N）

・Access to the Data : Locked（L）／Unlocked（U）

・Include / exclude a subject for search when Life State [Unable] : Include（-）／Exclude（S）

・Life Count cycle : 1 second（0）／8 millisecond（1）

L-COUNT

Count used a tool / time is displayed.

MAX-LIFE

Maximum tool life / time is displayed.

NOTICE-L

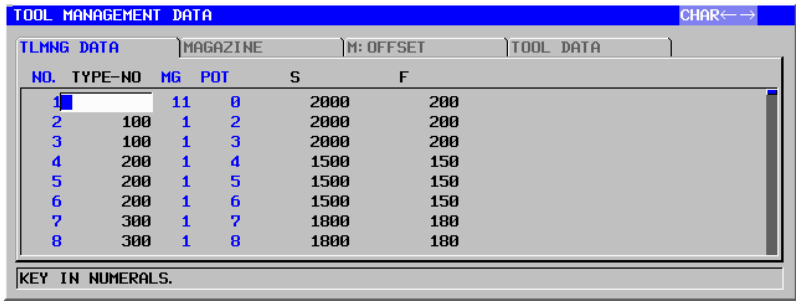
Notice life value of a tool / time is displayed.

L-STATE

Current condition of a tool is displayed.

4 conditions of NO-MNG, ENABLE, OVER and DAMAGE are displayed.

Pressing [←PAGE] or [PAGE→] several times to display the following spindle rotation number or setting screen of feed rate are displayed.



The following data can be set.

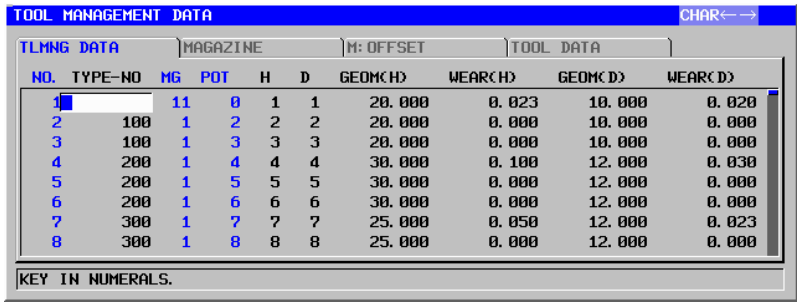
S（Spindle Speed）

Display a spindle speed.

F（Feedrate）

Display a feedrate.

Press [←PAGE] or [PAGE→] several times, the following setting screen of tool offset is displayed.



The following data is possible to set.

H（Tool Length Compensation Number）, D（Tool Diameter Compensation Number）

Tool length compensation number and tool diameter compensation number are displayed. It does not display on lathes.

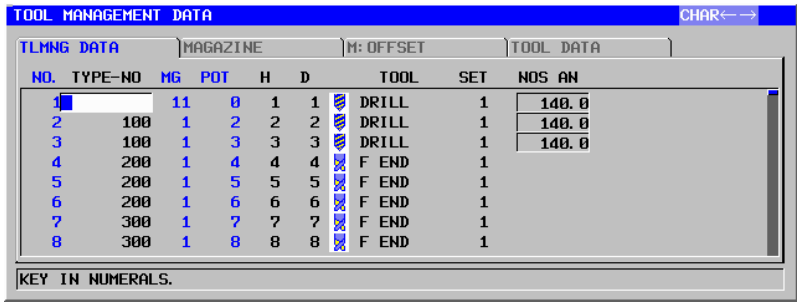
TG（Geometry offset Number）, TW（Wear offset Number）

Tool geometry compensation number and tool wear offset number are displayed. It does not display on machining centers.

Figure, Wear

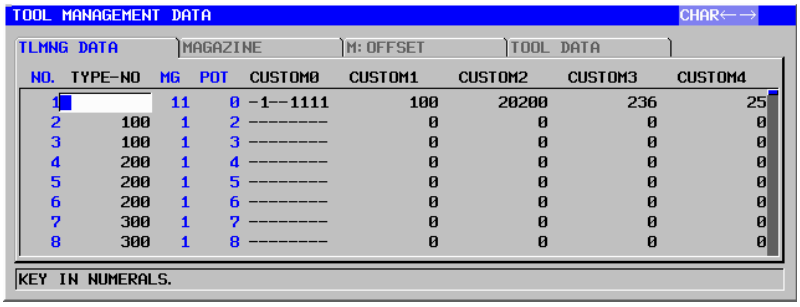
Offset value of compensation number set to H, D, TG and TW are displayed.

Press [←PAGE] or [PAGE→] several times to display a setting screen of the following tool information is displayed.



Refer a chapter of tool database function for details of data contents.

Press [←PAGE] or [PAGE→] several times to display the following setting screen of customize data.



The following data can be set.

CUSTOM 0

Customize information by bit type

1 or 0 is possible to enter per bit

CUSTOM 1～4

Customize information

CUSTOM 5～20

Customize information. Displayed only when a option of adding customize data of tool management function is available.

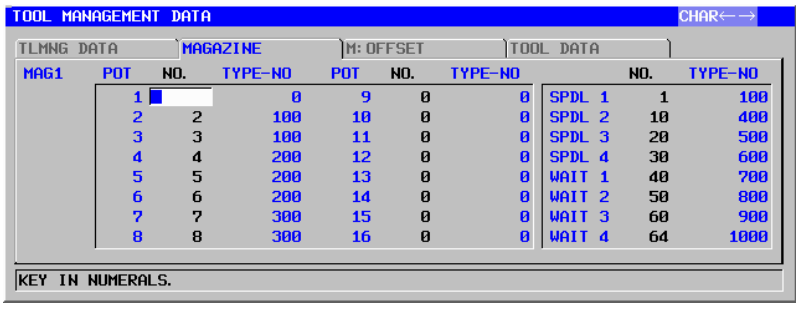
8.1.3 Customize of Tool Management Data Display

Changing displaying position of screen element at tool management data screen(Type Number, Tool Information, and Life Counter etc.), or selection of display/ undisplay is possible to operate.

Refer [A-78614 Tool Management Function specifications manual] for details.

8.1.4 Magazine Management Table Tab

At magazine data tab, tool management data number and tool type number corresponding to pot of magazine, spindle, and wait position are displayed.



POT

Pot number is displayed.

NO.

Display a tool management data number.

TYPE-NO.

Tool type number corresponding to tool management data number is displayed.

SPDL

Tool management number of spindle position and tool type number are displayed.

WAIT

Tool management number of wait position and tool type number are displayed.

8.1.5 Tool Data Tab

Display a tool information screen.
Operation is the same as existing tool data screen.

8.1.6 Tool Offset Tab

Display tool offset screen.
Operation is the same as tool offset screen.

8.2. Memory card Input / Output

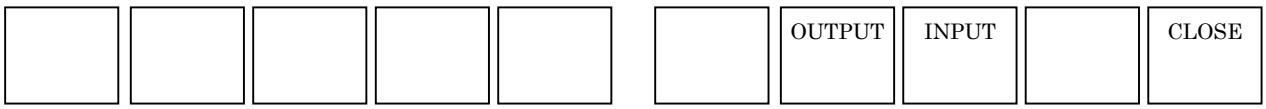
By following a below operation, it is possible to input / output tool management data to memory card at MANUAL GUIDE *i* screen.

8.2.1 Output Tool Management Data

Follow a below operation to output tool management data.

(1) Select EDIT mode on machine control board. And display a screen of tool management function.

(2) After display a tool management function screen by pressing [Tool] and [<] or [>] is pressed several times, the following soft key is displayed.



(3) A below softkey row is displayed by pressing [Output], and output data can be select.

TOOL	MAG	STATUS	CSTM		DISPCT	P- NAME	POINT		CANCEL

[TOOL] : Output a tool management data and tool database data.
[MAG] : Output a data of magazine management table.
[STATUS] : Output a name data of tool life status.
[CSTM] : Output a name data of customize data

To display the following 3 items, an option of tool management function expansion is required.

[DISPCT] : Output screen customize data.
[P-NAME] : Output a name data of spindle/wait position.
[POINT] : Output a position of decimal point of customize data.

(4) After pressing a soft key of tool management data that you want to output, check message is disaplayed. Press [Yes] to output tool management data they are output to memory card.

8.2.2 Input Tool Management Data

To input tool management data, follow the operation below.

(1) Select edit mode by control panel to display the screen of tool management function.
(2) Press [TOOL] to display the tool management function screen, and press[<] or [>] several times to display the below softkey.

						OUTPUT	INPUT		CLOSE

(3) press [INPUT] to display a below soft key row, and input data can select.

TOOL	MAG	STATUS	CSTM		DSPCT	P- NAME	POINT		CANCEL

[TOOL] : Input a tool management information and tool database data.
[MAG] : Input a data of magazine management table.
[STATUS] : Input a name data of tool life information.
[CSTM] : Input a name data of customize data

To display the following 3 items, an option of tool management function expansion is required.

[DSPCT] : Input screen customize data.
[P-NAME] : Input a name data of spindle/wait position.
[POINT] : Input a position of decimal point of customize data.

(4) After pressing a soft key of tool management data that you want to input, check message is disaplayed. Press [EXEC] to input tool management data they are input to memory card.

8.3. Parameters

14845	TLTAB

Set the display position of each tab of tool management data, magazine information, tool offset, and tool information.

1digit	Tool Management Data Tab
10 place	Magazine Data Tab
100 place	Tool Offset Tab
1000 place	Tool Data Tab

Set 1～4 to each value to set a displaying order of a tab.
Tab is no displayed when 0 is set. When 0 is set to all, it is displayed by default sorting.
Default sorting display is as follows.

1st	Tool Management Data Tab
2nd	Magazine Data Tab
3rd	Tool Offset Tab
4th	Tool Data Tab

Please input the figure of four digits to this parameter. And the figure of each digit must not overlap.

	#7	#6	#5	#4	#3	#2	#1	#0
14705			#5	#4				

Softkey [T-OFS]

#4 0： Display.
1： Undisplay.

Softkey [TOOL]

#5 0： Undisplay.
1： Display.

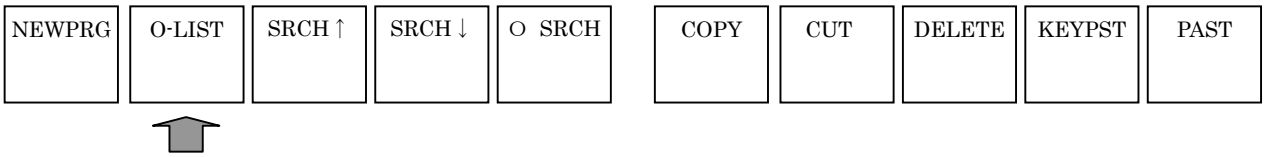
9 Simultaneous Delete of Multiple Programs

Select a several programs, and it is possible to delete at once. Using the three of individual selection, range selection, and all selection to select the programs to delete.
It is also possible to remove from the selection individually using unselect. For these selected multiple programs, deleting is performed.

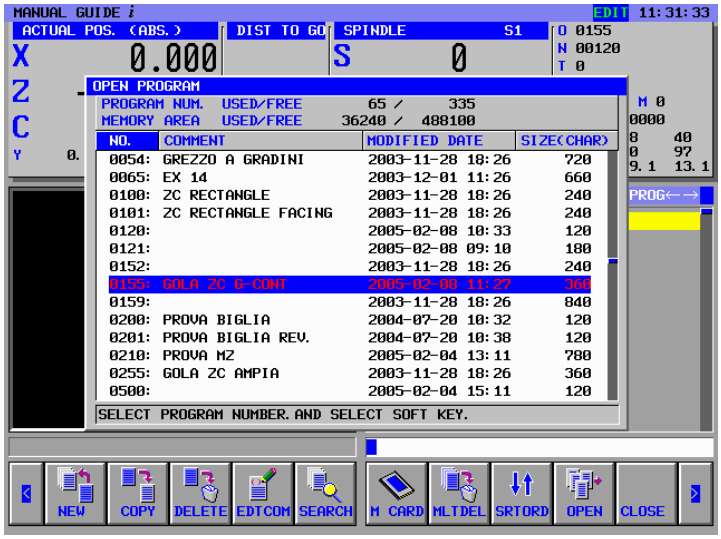
NOTE

1 Conventionally, deleting all registered program in a folder is performed by pressing [ALLDEL] soft key. That conventional function is included to this function. All delete operation is converted to the method of deleting after selecting the all programs with this function of multi programs simultaneous deletion. Therefore, unnecessary [ALLDEL] soft key is deleted, and newly added the [MLTDEL] soft key to operate this function for same position.

9.1. Details



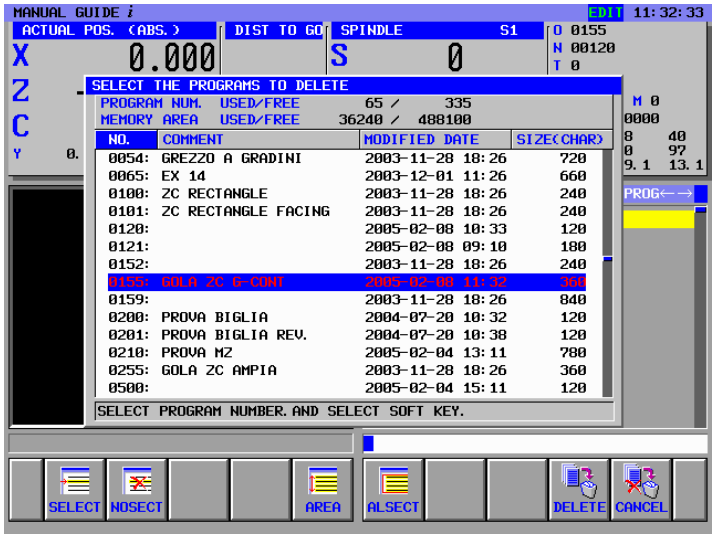
Press [O-List] to display a window to open program.



In this event, below soft keys are displayed.
A position [ALLDEL] were displayed in conventional version, [MULDEL] is displayed.

NEW	COPY	DELETE	EDTCOM	SEARCH	M CARD	MLTDEL	SRTORD	OPEN	CLOSE

At a window to open program, press [MLTDEL] to open a window to select the programs to delete.



In this event, below soft keys are displayed.

SELECT	NOSECT			AREA		ALSECT			DELETE	CANCEL

[SELECT] Use to select a program individually. Move a cursor to a program selected, and press [SELECT]. Background color of selected program is changed to yellow.

[NOSECT] Individually remove from a selection. Move to a cursor to a program to unselect, and press [NOSECT]. Background color of unselected program is changed back from yellow to white.

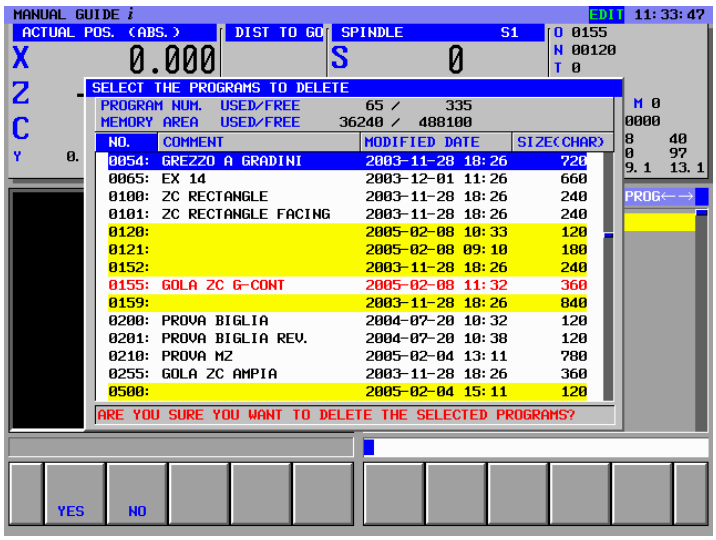
[AREA] Use to select a range. Move a cursor to a top of selected program, and press [AREA]. Move a cursor to the end program to be selected. At this time, background color of specified area changed to purple. Then, press [SELECT] to determine. Background color of selected program is changed from purple to yellow.

[ALSECT] Use to select all programs. All programs in a folder are selected, and background color of these programs is changed to yellow.

[DELETE] Delete the selected programs.

[CANCEL] All programs in a folder are unselected, and background color of program change back from yellow to white.

By press [DELETE], a message is displayed whether to delete the program. Press [YES] to Delete the program, and press [NO] to cancel the deletion. However, when [NO] is pressed, it keeps a selection for deleting range, and range selection screen is displayed.



YES	NO								

10 Invalidate of Shortcut Function

This function is to invalidate the shortcut function (pressing **NUMERIC** + **INPUT**) in base window.

10.1. Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
14705								#0

Make the shortcut function of to soft key

#0 0： available.

1： invalid.

11 Shortcut Function to Tool Database Screen

When tool database function is available, move a cursor to tool offset command, and press [ALTER] or INPUT key to display tool database screen.
At this event, cursor move on the data of tool number corresponding to tool geometry offset number (Lathe) or tool radius compensation number (machining center)

NOTE

1 This function is unavailable when compound machine function is available.
2 This function is unavailable when changing offset memory function is available.
3 This function is unavailable when tool database function is unavailable. (No.14850#0=1).
4 This function is available with setting a parameter No. 14705#7=1
5 Operate this function by pressing INPUT key. On MDI mode, machining simulation, and NC program conversion, [ALTER] is not displayed.
6 When move a cursor on tool offset command by block unit, and there are multiple tool offset command in a block, the last command is used to operate this function.
7 On machining center with tool management function is available, D99 is used to command a offset number on tool at spindle position. Therefore, when move a cursor on D99 to use this function, warning is displayed and invalid this function.
8 On a lathe with tool management function is available, D9/D99/D999 are used to command a offset number on tool at spindle position. Therefore, when move a cursor on D9/D99/D999 to use this function, warning is displayed and invalid this function.
9 D code to command a offset number on tool at spindle position is changed by set a parameter No.13265.

11.1. Shortcut Function to Tool Database Screen

11.1.1 Use Condition

This function is available on the status listed below.

1. Editing foreground in EDIT mode.
2. Editing background in MEM mode.
3. Editing MDI program in MDI mode.
4. Not operating drawing at machining simulation.
5. Not operating conversion at NC program conversion.

11.1.2 Tool Offset Command

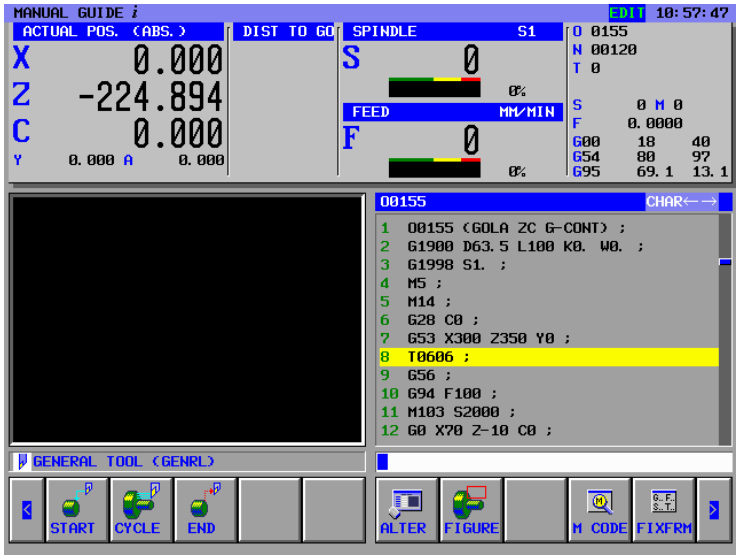
Whether tool offset command operated this function is T code or D code, are depend on 3 cases listed below.

(1) When a lathe which tool management function is unavailable.
Tool offset command is T code.
(2) When a machining center which tool management function is unavailable.
Tool offset command is D code.
(3) When a lather and a machining center which tool management function is available.
Tool offset command is D code.

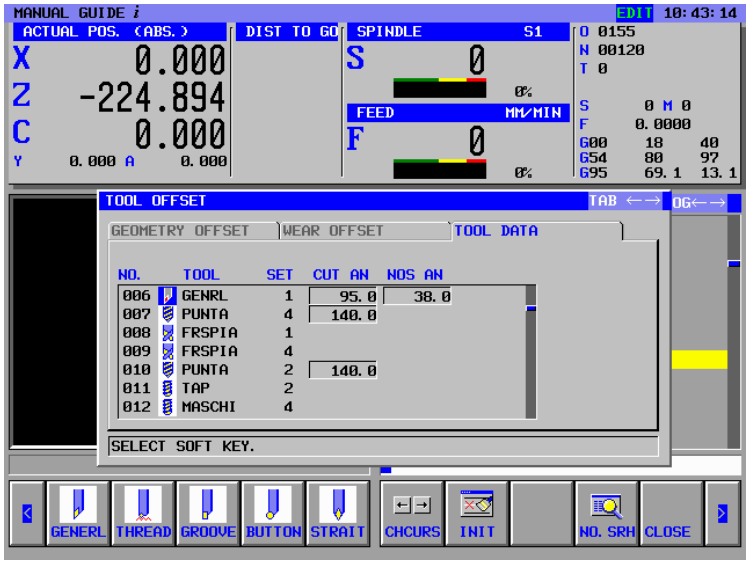
11.1.3 Operation Procedure

Operation procedure of this function is as follows.

(1) When this function is available to use (Refer 11.1.1).
(2) Move a cursor on tool offset command (Refer 11.1.2). Place a cursor either at a word or at a block unit. When a cursor is placed on a block, it operates by searching tool offset command in a block.



(3) Press [ALTER] or INPUT key. However, in MDI mode, machining simulation, and NC conversion, [ALTER] is not displayed, so press INPUT key.



(4) Tool database screen (tool information tab at tool offset screen) is displayed. At this event, a cursor move to the data of tool offset number corresponding to tool geometry offset number (lathe), or tool radius compensation number (machining center)

(5) Confirm or input a tool information.

(6) By pressing [CLOSE] to close a tool database screen. At this event, a cursor return to a tool offset command.

12 Display Tool Icon Function

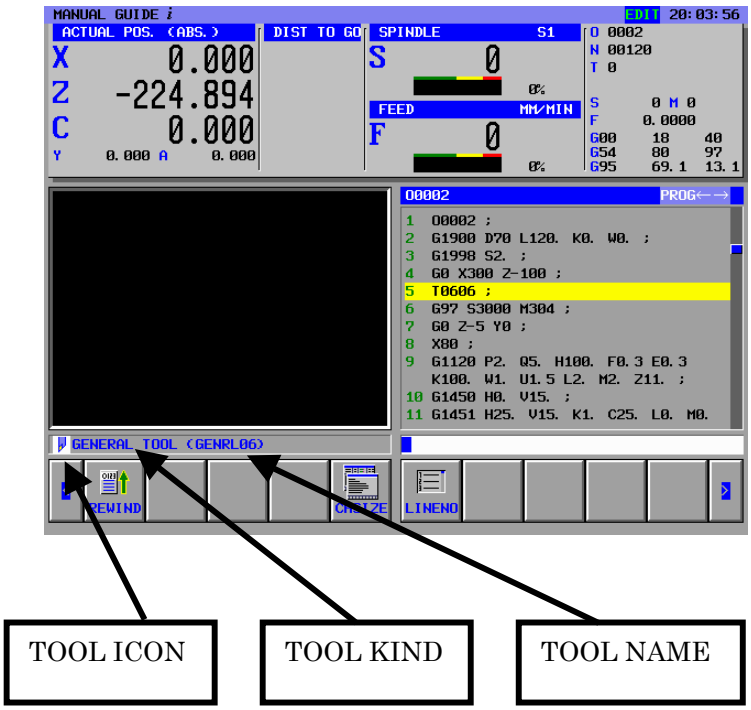
When tool database function is available, if the cursor is on the offset command, tool icon and tool name corresponding to the number is displayed on the guidance message screen.
The offset command is T code on lathes without tool management function. And it is D code on lathes with tool management function or machining centers.

NOTE

1 This function is unavailable on compound machines.
2 This function is unavailable when there is the offset memory switching function.
3 Set a parameter No.14705#7=1 to use this function.

12.1. DETAILS

When place a cursor to tool offset command defined a data of tool database tool icon, tool type, and tool name are displayed.



NOTE

1 On the lathe, tool data that is registered a same number to tool geometry offset command are displayed. It is not a offset number of tool wear.

2 On machining center, tool data registered to the same number of tool diameter compensation command is displayed.

13 Machining Time Display Function for Series 30*i*

13.1. Details

For Series 30*i*, Machining time display function is available.
The difference from Series 16*i* are as followings.

1. Inserting a machining time is possible in EDIT mode.
2. In MEM mode, a machining time cannot be inserted to a file selecting at foreground.

In details of this function, refer to [Chapter 23 "Machining Time Display Function"] of [II. Operations] on [B-63874JA/05 FANUC MANUAL GUIDE *i* operator's manual].

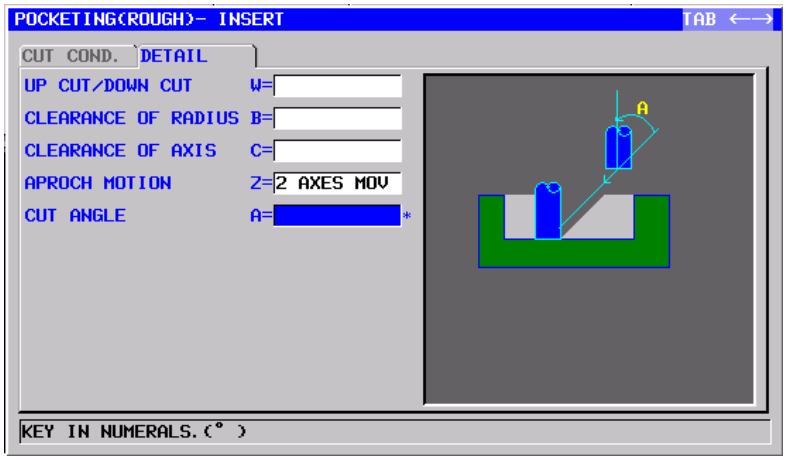
14 Improvement of Input Item for Milling

14.1. Outline

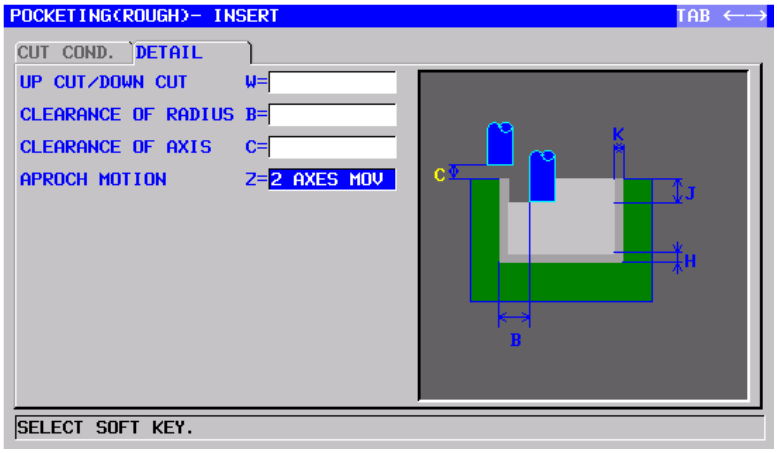
[Approach Motion] is undisplayed when a option of control axis number expansion is unavailable. [Cut Angle] can be set by a parameter. Undisplay [Approach Motion] when there is no option for simultaneous control axis number expansion.

14.2. Details

The below screen is [Detail] tab at pocketing (Rough).
A following is displaying the input items [Approach Motion] and [Cut Angle].



The screen below is the condition to undisplay the input item [Cut Angle] by setting the parameter 27061<#5> to 1.



14.3. Related Cycles

Improved cycle menus are listed below.

● Machining block related to approach motion.

Hole Machining

Center drilling	: G1000
Drilling	: G1001
Tapping	: G1002
Reaming	: G1003
Boring	: G1004
Fine Boring	: G1005
Back Boring	: G1006

Facing

Rough	: G1020
Finishing	: G1021

Contouring

Outer Wall Rough	: G1060
Outer Wall Bottom Finish	: G1061
Outer Wall Side Finish	: G1062
Outer Wall Chamfer	: G1063
Inner Wall Rough	: G1064
Inner Wall Bottom Finish	: G1065
Inner Wall Side Finish	: G1066
Inner Wall Chamfer	: G1067
Partial Wall Rough	: G1068
Partial Wall Bottom Finish	: G1069

Partial Wall Side Finish : G1070
Partial Wall Chamfer : G1071

Embossing
Rough : G1080
Bottom Finish : G1081
Side Finish : G1082
Chamfer : G1083

Pocketing
Rough : G1040
Bottom Finish : G1041
Side Finish : G1042
Chamfer : G1043

Grooving
Rough : G1050
Bottom Finish : G1051
Side Finish : G1052
Chamfer : G1053

A-Axis Grooving
Rough : G1058
B-Axis Grooving
Rough : G1054
C-Axis Grooving
Rough : G1056

● Machining block related to cut angle

Pocketing
Rough : G1040
Bottom Finish : G1041

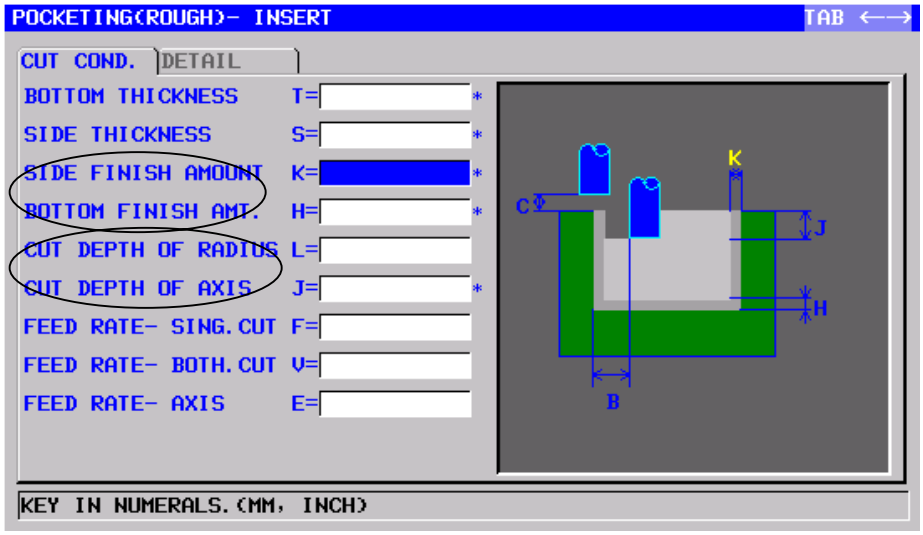
15 Imprvement of Displaying Position of Input Items [SIDE FINISH AMOUNT] and [BOTTOM FINISH AMT.]

15.1. Outline

Switch the order of input items [Cut Depth of Radius], [Cut Depth of Axis], [Side Finish Amount], and [Bottom Finish Amount] by setting the parameter 27002<#5> to 1.

15.2. Details

The following screen is a new spec pocketing (Rough) screen. The input items [Side Finish Amount], [Bottom Finish Amount] were placed upper part than the input items [Cut Depth of Radius], [Cut Depth of Axis].



15.3. Related Cycles

Improved cycle menus are listed below.

- Each Roughing of Contouring (G1060,G1064,G1068)
- Pocketing Rough (G1040)
- Grooving Rough (G1050)
- Embossing Rough (G1080)

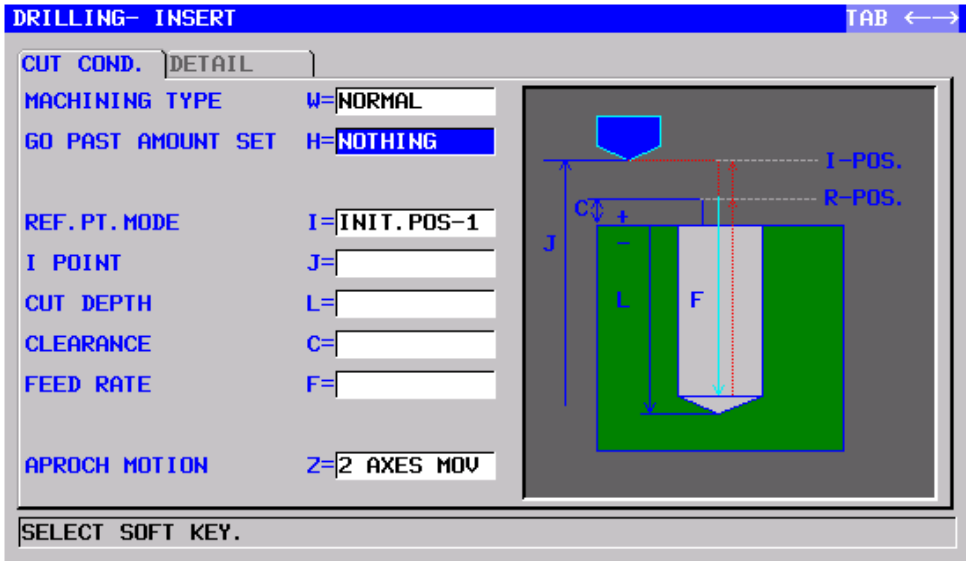
16 Improvement of the Input Item for Drilling

16.1. Outline

Whether to set or to unset the input item [GO PAST AMOUNT] at drilling can be set by the input item [GO PAST AMOUNT SET]. When the parameter 27020<#0> is set to 0, the input item [GO PAST AMOUNT SET] is not displayed.
[TOOL DIAMETER] and [NOSE ANGLE] are also added, so that the [GO PAST AMOUNT] can calculated by pressing [CALCULATE] soft key.

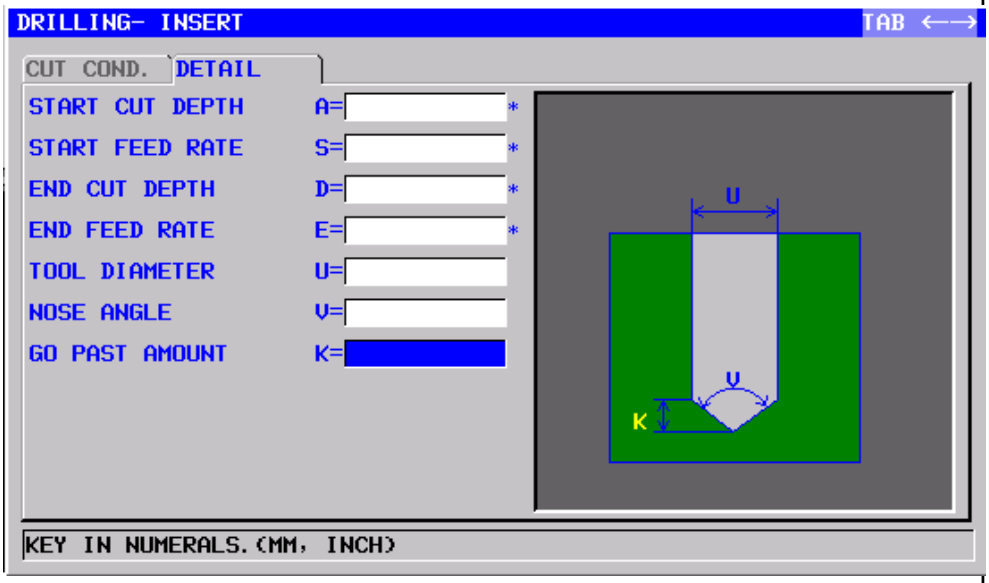
16.2. [Cut Condition] Tab

A below figure is a [CUT COND.] tab. Input item [GO PAST AMOUNT SET] is added, and [NOTHING] and [SETTING] can be selectable.



16.3. [Detail] Tab

The below figure is a [DETAIL] tab. When selecting [SETTING] in input item [GO PAST AMOUNT SET] at [Cut Condition] tab, input items [TOOL DIAMETER], [NOSE ANGLE], and [GO PAST AMOUNT] at [DETAIL] tab.



16.4. Calculation Function

When enter the data to input items [TOOL DIAMETER] and [NOSE ANGLE] at [DETAIL] tab, and press [CALCULATE] soft key, input item [GO PAST AMOUNT] is set the value automatically calculated.

16.5. Related Cycles

Improved Machining Types are listed below.

- Drilling :G1001 (Tool Rotation M Type)
- Drilling :G1111 (Tool Rotation T Type)
- Drilling :G1101 (Work Rotation)

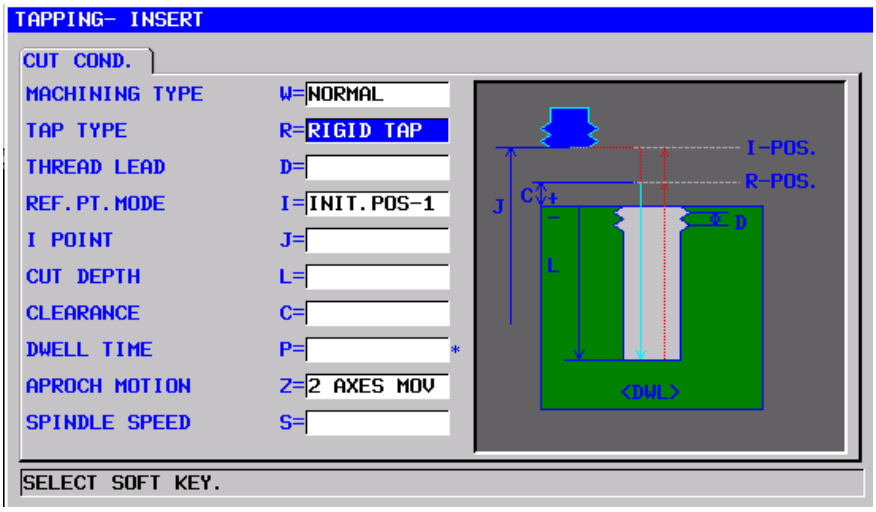
17 Improvement of Input Item for Tapping

17.1. Outline

At tapping cycle input screen, input items [TAP TYPE] and [SPINDLE SPEED] are moved to [CUT CONDITION] tab.
A default of input item [TAP TYPE] is also changed by parameter.

17.2. New Tapping Cycle G1002 Screen

For a result [RIGIT TAP] tab are deleted, and these input items are moved to a front tab, an operator can consider about tap type. Input item [TAP TYPE] are consistently displayed.



17.3. Default of [Tap Type]

The following soft key and default displayed when place a cursor to [TAP TYPE].

1 . When there is no option of [RIGIT TAP]

Default Value : [FLOAT TAP]

2 . When there is an option of [RIGIT TAP] and parameter No.5200 #0=0

Default Value : [RIGIT TAP]

3 . When there is an option of [RIGIT TAP] and parameter No.5200 #0=1

Default Value : [RIGIT TAP]

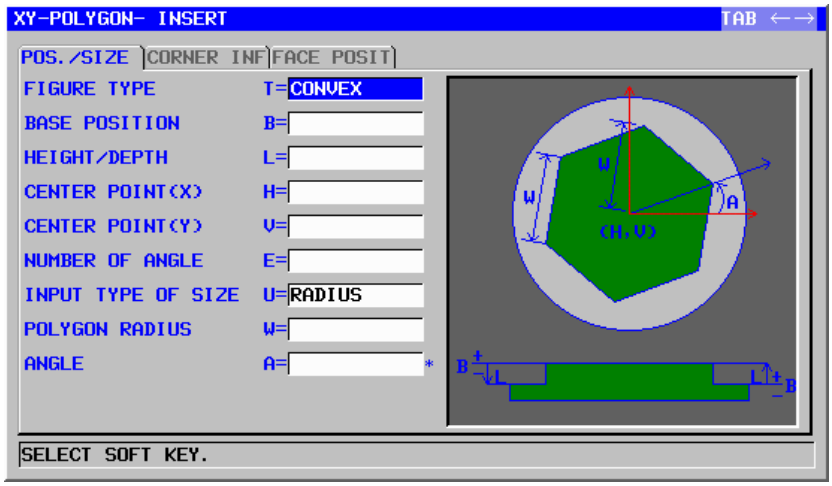
17.4. Related Cycles

Improved cycle machining types are as followings.

- Tapping : G1002 (Tool Rotation M Type)
- Tapping : G1112 (Tool Rotation T Type)
- Tapping : G1102 (Work Rotation)

18 Polygon Fixed Figure

”Polygon” can be used at fixed figure on XY, YZ, and XC plane for milling.
This function is available by set the parameter No.27095#2 = 1.



Example： XY plane polygon convex figure of Contouring

18.1. Details of Fixed Figure (Polygon) Cycle（G1x25）

18.1.1 Available Combination of Machining

Matching machining type block is as followings

Machining Type	Availability
Hole Machining (M Type)	×
Facing	○
Contouring (exclude parts machining)	○
Pocketing	○
Grooving	○
Hole Machining (T Type：Tool Rotation)	×
Hole Machining (work Rotation)	×
Turning	×
Turning Groove	×
Threading	×

18.1.2 G-Code

G code of polygon fixed figure at each machining plane are as follows.

Plane	G code
XY	G1225
YZ	G1325
XC	G1525

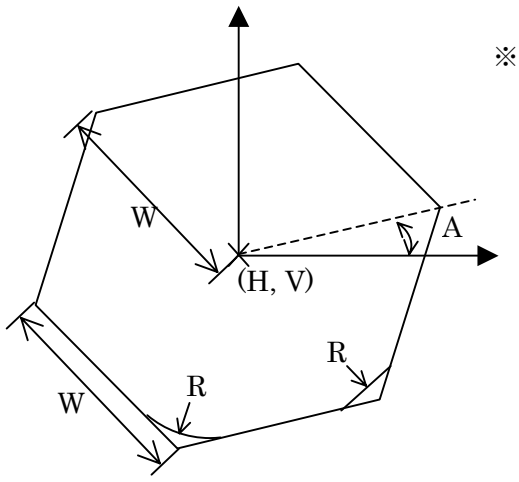
18.1.3 Format

G1x25 Tt Bb Ll Hh Vv Ee Uu Ww Aa Dd Cc Rr Zz Yy;

Meanings	Address	Explanation
FIGURE TYPE	T	T1: FACE, T2: CONVEX, T3: CONCAVE, T4: GROOVE
BASE POSITION	B	
HEIGHT/DEPTH	L	Undisplay at Facing
CENTER POINT (THE FIRST AXIS)	H	
CENTER POINT (THE SECOND AXIS)	V	
NUMBER OF ANGLE	E	Positive integer value from 3 to 99.
INPUT TYPE OF SIZE	U	U1: POLYGON RADIUS, U2: SIDE LENGTH
POLYGON RADIUS/ SIDE LENGTH	W	
ANGLE	A	Inclination from the first axis.
GROOVE WIDTH	D	Specify only used at Grooving.
CORNER TYPE	C	C1: NOTHING, C2: CHAMFER, C3: CORNER R
CORNER SIZE	R	
FACE POSITION	Z	Z1:＋FACE, Z2:－FACE,
ROTATION AXIS NAME	Y	

※ “Corner Position” is displayed with added tab by set parameter No.27000#4 = 1.

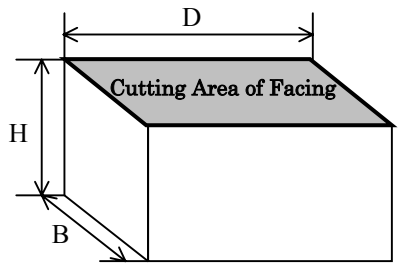
※ “Rotation Axis Name” is displayed on XC plane, by set parameter No.27001#0 = 1.



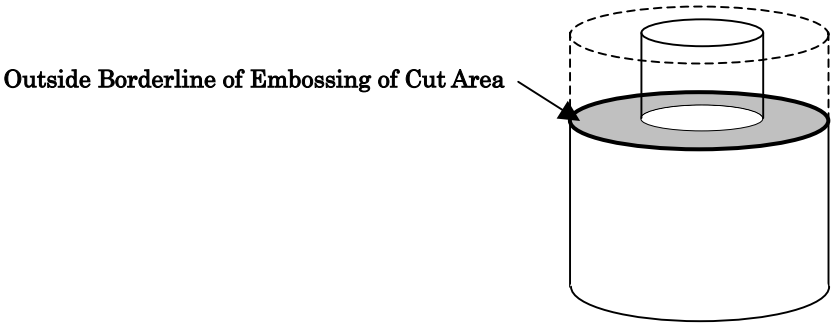
19 Reuse of Blank Form Data at Machining Figure Entry

Blank form data can reuse at machining figure entry. Measurement data (radius or length of each lines) entered at element figure is automatically copied to input items of fixed form figure or free figure. This function is available by setting a parameter No.27095#1=1.

1. At facing, blank form data of cutting face can be reused as cutting area.



2. At Embossing, blank form data can be reused as outside borderline of embossing.

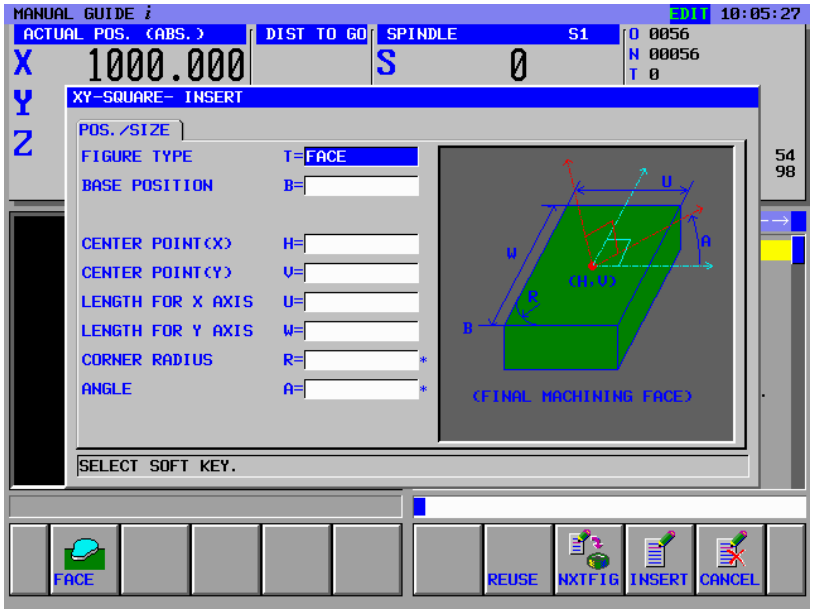


At other machining cycle, it is easy to enter the input item automatically copied by using calculating function.

19.1. Operation

(1) Enter Fixed Figure

When reuse of blank form data for machining figure data is available (parameter No.27095#1 = 1), it automatically copied to each items at figure input screen by pressing [REUSE] soft key.

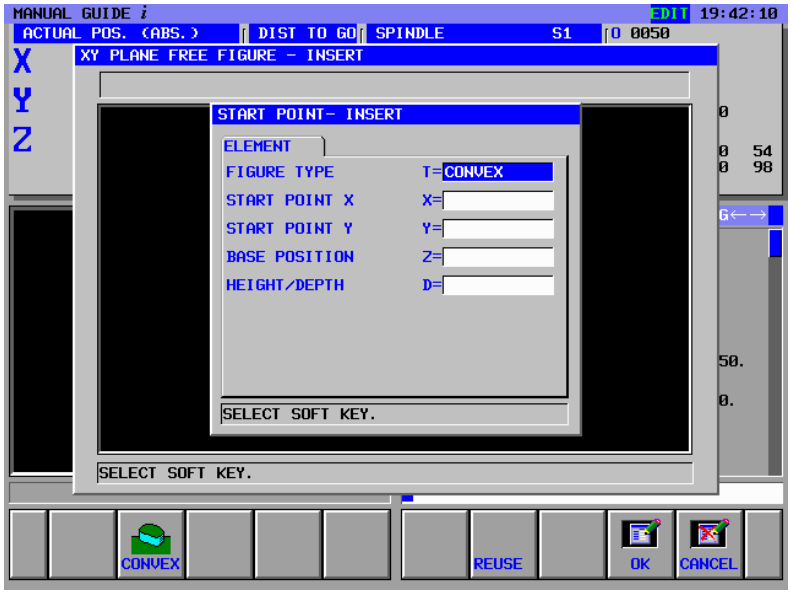


Note: When the plane of blank is circle figure, even if the figure data is tried to copy to rectangle, an alarm message “FIGURE DATA IS WRONG” is displayed.

(1) Enter Free Figure

When reuse of blank form data for machining figure data is available (parameter No.27095#1 = 1), it automatically copied to each items at figure input screen by pressing [REUSE] soft key.

After the entry of requirement items of start point (Standard position, Height/ Depth), press [REUSE] soft key. Free figure is not drew by pressing [INSERT] soft key followed by pressing [REUSE] soft key without entered a standard position, or height/depth.



19.2. Reuse Condition

19.2.1 Blank and Available Machining Figure/Plane

At each blank form, the relation of reusable machining figure and plane are listed below.

(1) Fixed Figure

	Rectangular			Arc			Polygon		
	XY	YZ	XC	XY	YZ	XC	XY	YZ	XC
Rectangular	○	○	○						
Cylinder				○		○			
Cylinder (X axis rotation)					○				
Hollow Cylinder				○		○			
Hollow Cylinder (X axis rotation)					○				
Prism							○		○
Hollow Prism							○		○

Reuse is available only at machining figure corresponding to blank form data.

(2) Free Figure

	XY	YZ	XC
Rectangular	○	○	○
Cylinder	○		○
Cylinder (X axis rotation)		○	
Hollow Cylinder	○		○
Hollow Cylinder (X axis rotation)		○	
Prism	○		○
Hollow Prism	○		○

19.2.2 Blank Figure Data to be Used

(1) Timing to update a blank form data to reuse.

1) When alter or create a blank and register, the blank form data is saved and used as reuse data at machining figure entry screen.
2) When machining program is opened at O list, program from a top of to a maximum block set by a parameter No. 27018(SRCHBLNK) are analyzed, and save an blank form data placed from the beginning, and used as reuse data at machining figure entry screen.

(2) Timing to clear the blank form data to reuse.

1) Saved blank form data is cleared when powered off.
2) A part of the first blank form data in a block set by a parameter NO.27018 (SRCHBLNK) is deleted (Example: Only an element width B of rectangular is deleted), saved blank form data to reuse is cleared.

Note 1: Reuse function is assuming only one blank form block existing at one machining. When there are multiple blank form blocks in a machining program, the corresponding blank form block is register again to edit a machining figure block for 2nd blank for normal operation.

Note 2: When there is a sub program in a block range set by parameter No.27018 (SRCHBLNK), it does not analyze in the sub program.

20 Changing Input Screen of Thickness Item

Input items [Bottom Thickness] and [Side Thickness] can be entered at figure entry screen instead of cycle entry screen. With using this function, there are some advantages as following.

1: [Bottom Thickness] and [Side Thickness] at cycle entry screen and [Base Position] and [Height/Depth] at figure entry screen are closely related. These items can enter at the same block.

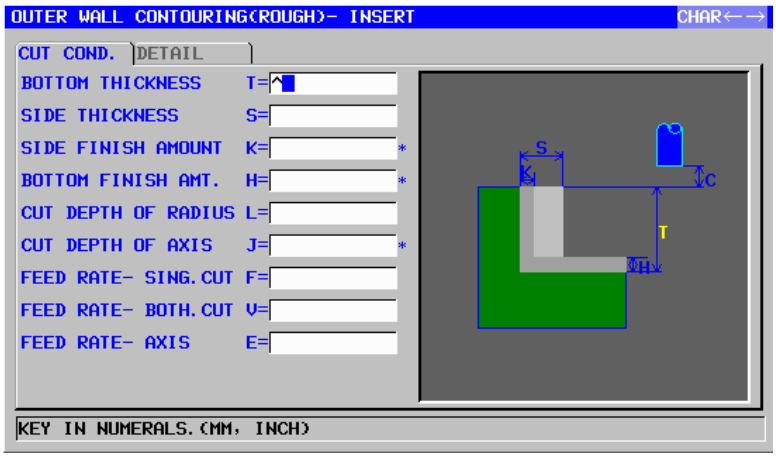
2: Multiple figure blocks can enter to one processing. By entering [Bottom Thickness] and [Side Thickness] at figure block, [Thickness] can be set to multiple machining figure individually.

This function is available by set a parameter No.27095#0=1.

20.1. Cycle Entry Screen

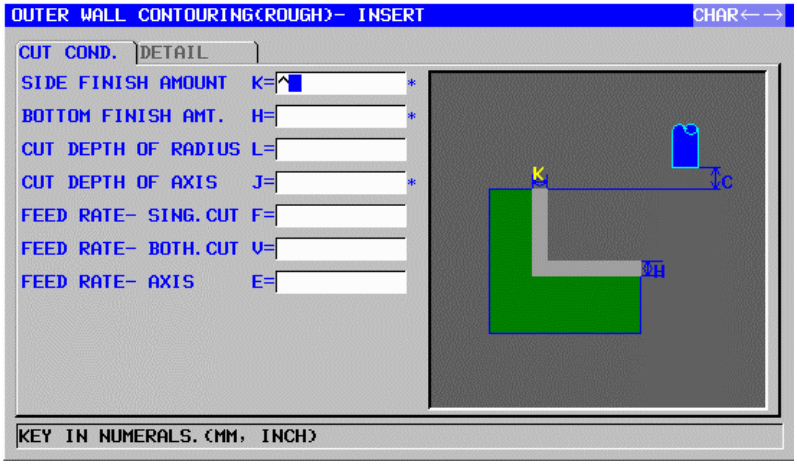
Cycle entry screen is displayed as figure below by setting a parameter No.27095#0

(1) NO.27095#0=0



Example : Outer Wall Contouring (Rough)

(2) NO.27095#0=1

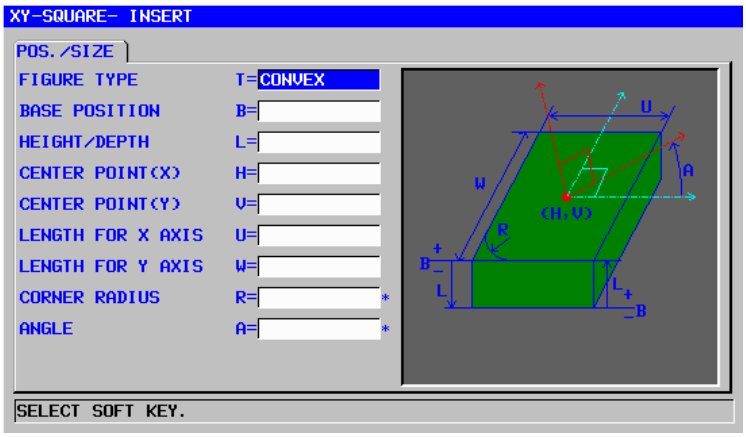


Example : Outer Wall Contouring(Rough)

20.2. Figure Entry Screen

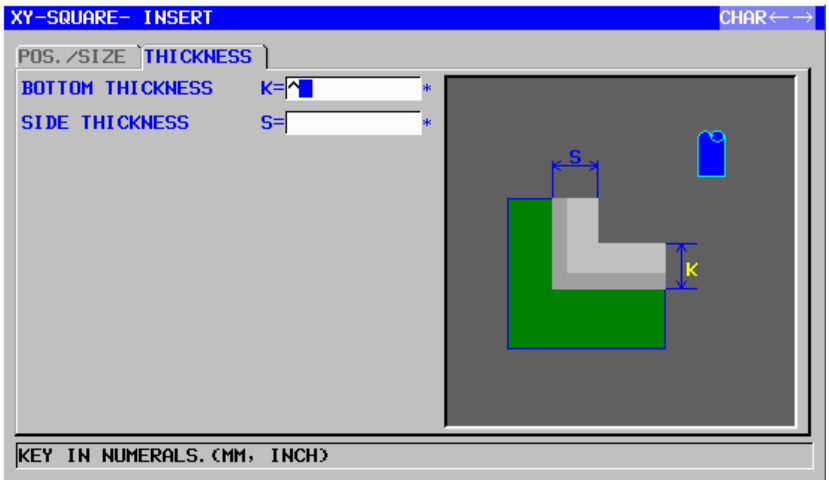
Figure entry screen is displayed as a figure below by setting a parameter No.27095#0. When a parameter No.27095#0=1, <Thickness> tab is displayed, and [Base Thickness] and [Side Thickness] can be input.

(1) NO.27095#0=0



Example : XY plane Square

(2) NO.27095#0=1



Example : XY plane Square

20.3. Mutuality

When enable a setting of input items [Bottom Thickness] and [Side Thickness] at figure entry screen, machining program created at a setting of [bottom thickness] and [Side Thickness] at cycle entry screen cannot be used as it stands. It needs to alter partially.
Depending on a setting, alarms listed below would occur.

1. When input bottom / side thickness at figure entry screen (Parameter No.27095#0=1).

・ At machining type block, a bottom thickness or side thickness exists.

→ Alarm Number 3043(16*i*), 3543(30*i*) "WRONG THICKNESS"

・ At machining figure block, even bottom thickness or side thickness is required item, a return value does not exist.

→ Alarm Number 3016(16*i*), 3516(30*i*) "NO NECESSARY ADDRESS"

2. When input bottom / side thickness at cycle entry screen. (Parameter No.27095#0=0)

・ At machining figure block, a bottom thickness or side thickness exists

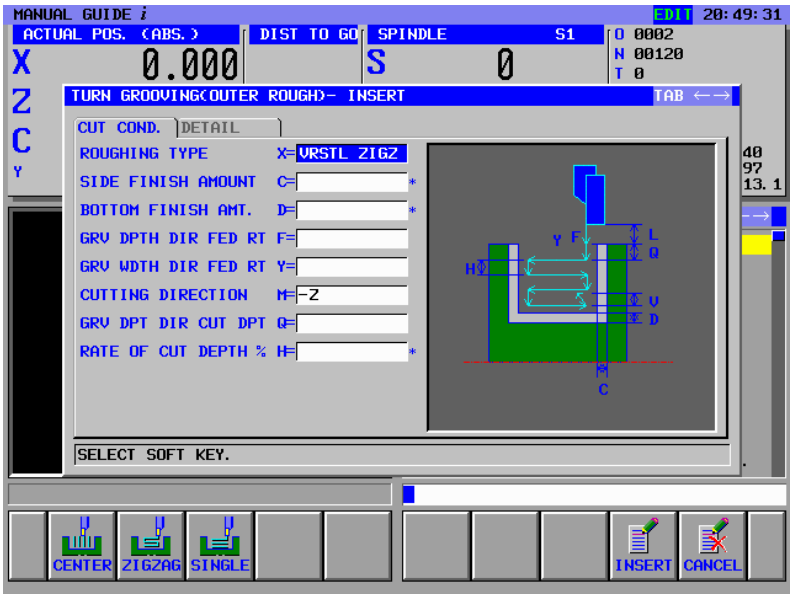
→ Alarm Number 3043(16*i*), 3543(30*i*) "WRONG THICKNESS"

Note: When bottom / side thickness is omissible at machining type block, it won't alarm out with leaving a blank to the item of bottom/side thickness.
When bottom/side thickness is required item (a part of plane contouring), it alarm out as "NO NECESSARY ADDRESS".

21 Turning Grooving by Versatile Tool

At turning grooving, input item [Roughing Type] is added.. When select versatile ZIGZAG, or versatile one direction by roughing type, turning grooving by versatile tool can be operated.

21.1. Data Entry Screen of Turning Grooving

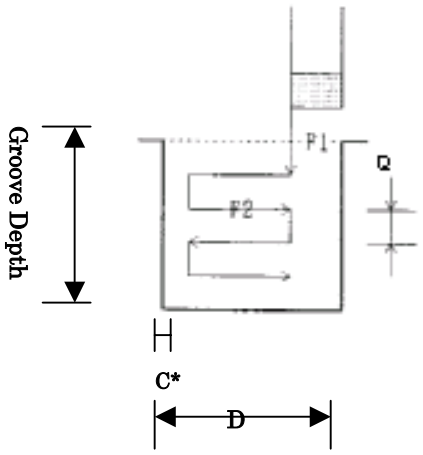


Input item [Roughing Type] is added to the following machining type block screen.
[Versatile ZIGZAG] and [Versatile 1 Direction] is added to [Roughing Type].
By selecting these, turning grooving by versatile tool is available.

(1) Outer Rough of Turning Grooving（G1130）
(2) Inner Rough of Turning Grooving（G1131）
(3) Facing Rough of Turning Grooving（G1132）
(4) Outer Rough and Finishing of Turning Grooving（G1133）
(5) Inner Rough and Finishing of Turning Grooving（G1134）
(6) Facing Rough and Finishing of Turning Grooving（G1135）

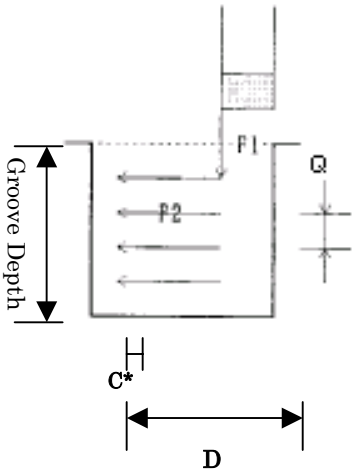
21.2. Machining Operation

21.2.1 Versatile ZIGZAG



F1 : Groove Depth Direction Feed Rate
F2 : Groove Width Direction Feed Rate
D : Groove Width
Q : Pecking Cut Depth（X axis direction）
C* : Side Finish Amount

21.2.2 Versatile 1 Direction



F1 : Groove Depth Direction Feed Rate
F2 : Groove Width Direction Feed Rate
D : Groove Width
Q : Pecking Cut Depth（X axis direction）

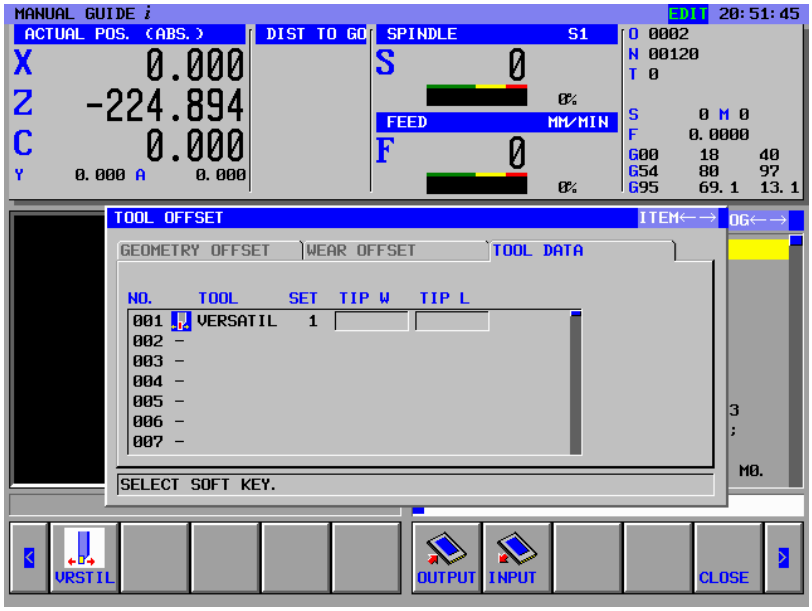
C* : Side Finish Amount

21.3. Improvement of Tool Database Function

Versatile tool is added to tool kind of tool database function.

21.3.1 Tool Offset Screen

[Tool Data] tab at tool offset screen is displayed. Then, press right page soft key several times to display [VERSTL] soft key. By pressing this soft key to select a versatile tool.



21.4. Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
27170							#1	

#1 0 : Undisplay Roughing Type of Turning Grooving.
1 : Display Roughing Type of Turning Grooving.
(Roughing by versatile tool is available)

27178	

Depth direction Minimum cut depth of Turning Grooving (Roughing) by versatile tool.

	#7	#6	#5	#4	#3	#2	#1	#0
27384								#0

#0 0 : When versatile tool is drew by animation, tip position is drew in front.

1 : When versatile tool is drew by animation, tip position is drew in rear.

27385

Holder length when versatile tool is drew by animation.

Data Unit : When input unit by milli (0000#2=0) : 0.001 mm

When input unit by inch (0000#2=1) : 0.0001 inch

When input 0 by millimeter input unit (0000#2=0), it automatically set to 50mm

When input 0 by inch (0000#2=1), it automatically set to 1.9685inch.

27386

Holder width when versatile tool is drew by animation.

Data Unit : When input unit by milli (0000#2=0) : 0.001 mm

When input unit by inch (0000#2=1) : 0.0001 inch

When input 0 by millimeter input unit (0000#2=0), it automatically set to 14mm.

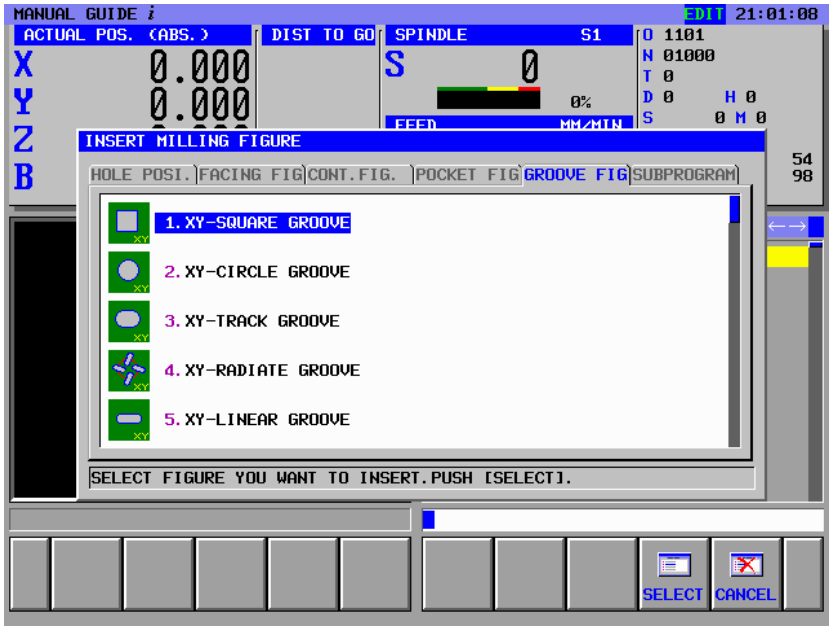
When input 0 by inch (0000#2=1), it automatically set to 0.5512inch.

22 Linear Groove for XY Plane

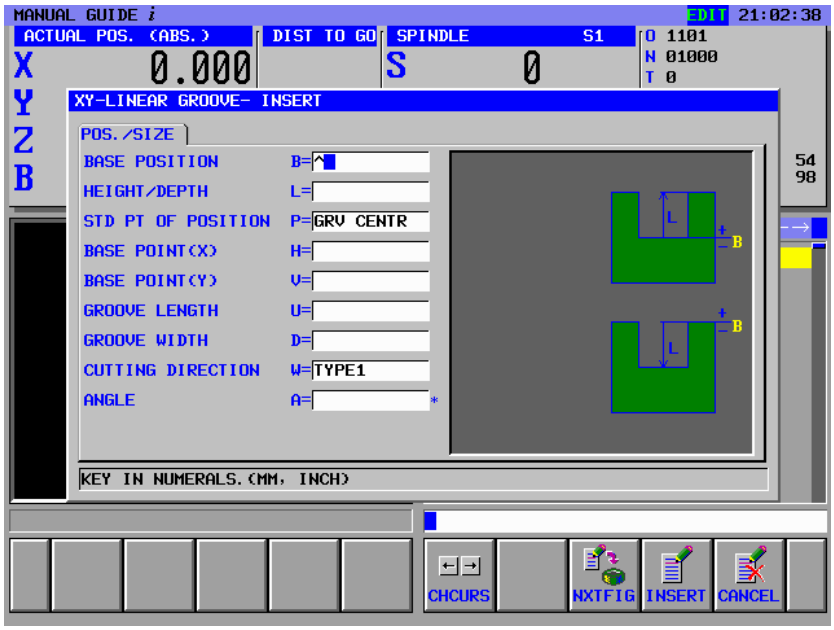
Add [XY plane linear groove] on grooving figure at milling cycle. By adding this, linear groove is possible to operate.

22.1. Cycle Input Screen

22.1.1 Figure Selection Screen



22.1.2 Linear Groove for XY Plane Input Screen



	Input Item	Meaning
B	Base Position	Z-axis coordination at either upper face of work to groove, or bottom of groove.
L	Height / Depth	Upper face of work is selected as a standard position, and distance to the bottom of groove is set by negative value. →Depth Bottom of groove is selected as a standard position, and distance to upper face of groove is set by positive value. →Height
P	Standard Point of Position	Position of standard position on coordination X and coordination Y 1:Center of Groove 2:Corner of Groove 3:Center circle of corner groove
H	Base Point (X)	Value of coordination X at standard position
V	Base Point (Y)	Value of coordination Y at standard position
U	Grooving Length	Length of a grooving
D	Grooving Width	Width of a grooving
W	Cutting Direction	Direction of cutting
A	Angle	Slope angle from a position of standard point.

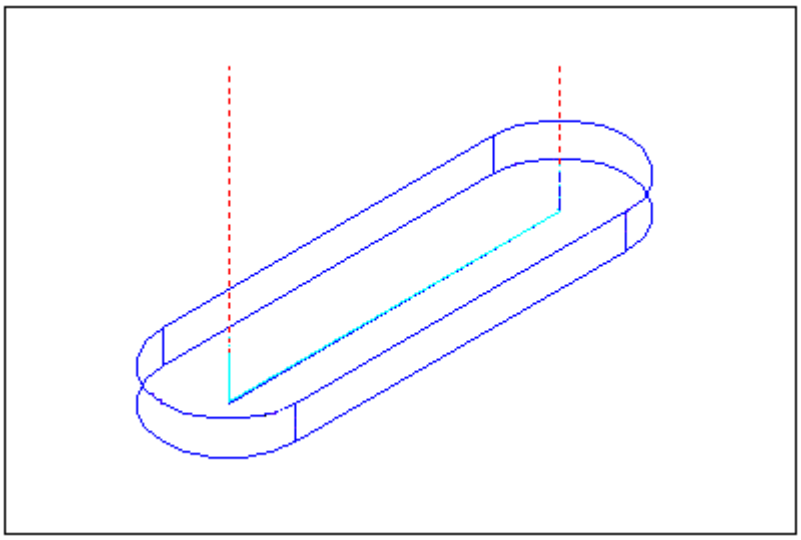
22.2. Format of Program

22.2.1 Roughing of Linear Groove

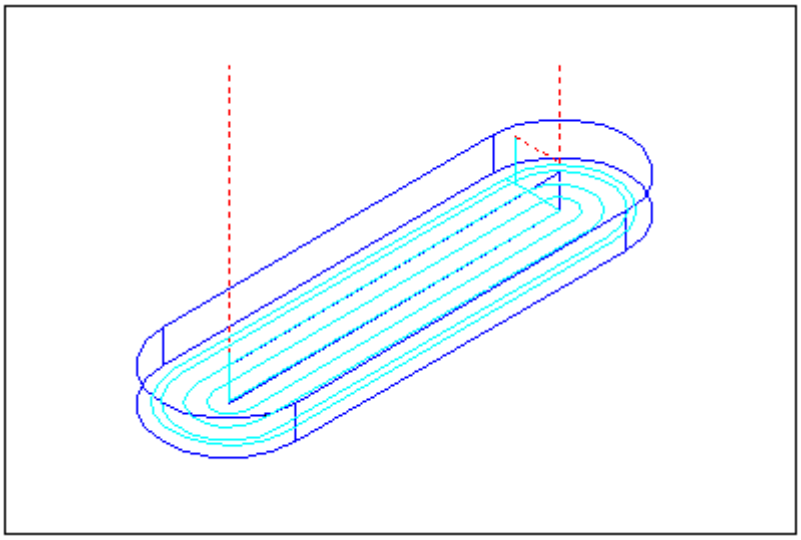
When creating a program of linear groove roughing, enter a grooving process block and linear groove block.

（Example Program）
G1050 (Grooving Rough)
G1224 (Linear Groove)

Tool path when a groove width of linear groove roughing is the same as tool diameter.



Tool path when a groove width of linear groove roughing is larger than tool diameter.



22.2.2 Bottom Finishing of Linear Groove

When creating a program of linear groove bottom finishing, enter a grooving process block and linear groove block.
Tool path is the same as linear groove roughing.
（Example Program）
G1051 (Bottom Finish Grooving)
G1224 (Linear Groove)

22.2.3 Side Finishing of Linear Groove

When creating a program of linear groove side finishing, enter a grooving process block and linear groove block.
Tool path is the same when linear groove is defined on free figure.
（Example Program）
G1052 (Side Finish Grooving))
G1224 (Linear Groove)

22.2.4 Chamfering of Linear Groove

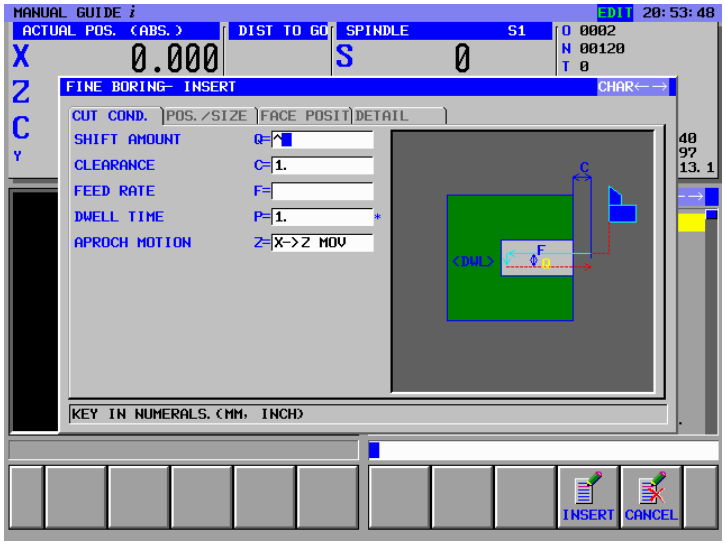
When creating a program of linear groove chamfering, enter a grooving process block and linear groove block.
Tool path is the same when linear groove is defined on free figure.
（Example Program）
G1053 (Chamfering Grooving)
G1224 (Linear Groove)

23 Fine Boring for Turning

Fine Boring is added for turning.

(1) Fine Boring (Work Rotation for Turning) : G1105
(2) Fine Boring (Tool Rotation for Turning) : G1115

23.1. Fine Boring (Work Rotation for Turning) : G1105



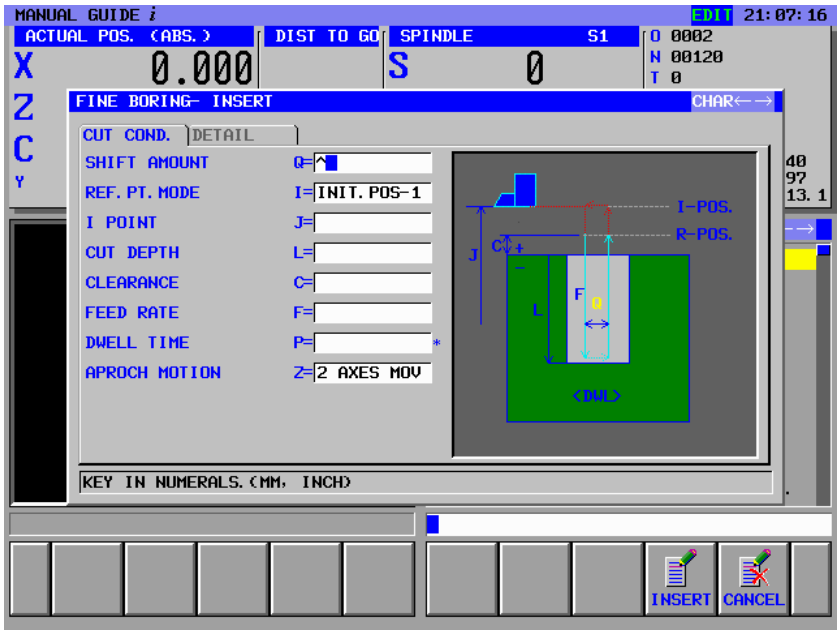
Cut Condition		
Data Items		Meanings
Q	Shift Amount	Bottom Shift Amount (Radius Value) (Copy)
C	Clearance	Distance between work surface and R point. (Radius Value, Positive Value) (Copy)
F	Feed Rate	Feed Rate at Cutting (Positive Value) (Copy)
P*	Dwell Time	Dwell time at the Bottom (Second Bit, Positive Value) (Copy)
Z	Approach Motion	[Z→X]:When moving from current position to start point, X axis moves after Z axis movement. [X→Z]:When moving from current position to start point, X axis moves after Z axis movement.(Default)

		[2 axis]:When moving from current position to start point, X/Z axis move at once.

Position / Measurement		
Data Items		Meanings
B	Base Position	Z axis coordination value at work surface
L	Cut Depth	Available depth of hole（Radius Value, Negative Value）

Details		
Data Items		Meanings
D	Cut Depth at Ending	Cut depth when cutting with federate changed at cut ending.
E	Feed rate at Ending	Cut feed rate when cutting with federate changed at cut ending.

23.2. Fine Boring（Tool Rotation for Turning）: G1115



Cut Condition		
Data Items		Meanings
Q	Shift Amount	Shift Amount after the Spindle orientation at the Bottom（Radius Value） (Copy)
I	Return Mode	[Point I－1]:Move between holes by R point Return,

		and move to the last by I point.（Default） [Point I－２]:All movement between holes by I point return including a last return mode. [Point R]: All movement between holes by R point return including a last return mode.
J	Point I Coordination	Coordination of Point I（Copy）
L	Available Depth	Available Depth of Hole（Radius Value, Negative Value）
C	Clearance	Distance between work surface and R point（Radius Value, Positive Value）（Copy）
F	Feed Rate	Feed Rate at Cutting（Positive Value）（Copy）
P*	Dwell Time	Dwell Time at the Bottom（Second Bit, Positive Value）（Copy）
Z	Approach Motion	[Z→X]:When moving from current position to start point, X axis moves after Z axis movement. [X→Z]:When moving from current position to start point, X axis moves after Z axis movement.(Default) [2 axis]:When moving from current position to start point, X/Z axis move at once.

Details		
	Data Items	Meanings
D*	Cut Depth at Ending	Cut depth when cutting with feed rate changed at cut ending.
E*	Feed rate at Ending	Cut feed rate when cutting with feed rate changed at cut ending.

23.3. Parameters

27013	MSTOP

MSTOP : M code for Spindle stop in Fine Boring

＝0 Output M05 for Spindle stop

≠0 Set M code number for Spindle stop

For Fine Boring for M series with End Feedrate (G1005) and Fine Boring for T series (G1105, G1115)

27014	MORIE

MORIE : M code for Spindle orientation in Fine Boring

＝0 Output M19 for Spindle orientation

≠0 Set M code number for Spindle orientation

For Fine Boring for M series with End Feedrate (G1005) and Fine Boring for T series (G1115)

	#7	#6	#5	#4	#3	#2	#1	#0
27020		#6	#5	#4	#3			

In Fine Boring for T series:

#3 0 : Output M code for spindle stop before an oriented spindle stop in Fine Boring (G1115).

Output M code for spindle stop in Fine Boring (G1105).

1 : Not output M code for spindle stop in Fine Boring (G1105, G1115).

#5/ #4 : Set the axis and direction in which the tool in Fine Boring for T series (G1115) is got free. #5 and #4 are set as shown below by plane selection.

#5	#4	G17	G18	G19
0	0	+X	+Z	+Y
0	1	-X	-Z	-Y
1	0	+Y	+X	+Z
1	1	-Y	-X	-Z

Set the direction in which the tool in Fine Boring for T series (G1105) is got free.

#6 0 : +X direction is free.

1 : -X direction is free.

24 Milling Machining of Workpiece Rotating round Y-axis

24.1. Outline

This function is complied with the Milling machine which machines a workpiece rotating round Y-axis.

NOTE
1 This function is available for only Milling machine.
2 Set the number of B axis as the rotational axis of workpiece.

24.1.1 Applied Milling Cycle

In Hole machining and Grooving, this function is applied as follows.

(1) Hole Machining
Only B axis hole machining on cylindrical face (YB) are applied.
(2) Grooving
B axis grooving and Y-axis grooving on cylindrical face are appplied.

NOTE
1 Turning Cycle cannot be used.
2 Polar Coordinate Interpolation cannot be used.
3 Cylindrical Interpolation cannot be used.

24.1.2 Machining Simulation

Machining simulation of workpiece rotating round Y-axis is available.

NOTE
1 Turning Simulation cannot be operated.
2 Simulation with Polar Coordinate Interpolation cannot be operated.
3 Simulation with Cylindrical Interpolation cannot be operated.
4 Since rotational axis of workpiece is decided by parameter, the rotational axis cannot be changed while machining simulation.

5 While machining simulation of the workpiece rotating around Y-axis, the axis name is displayed as X, Y, and –Z

24.2. Details

24.2.1 Machining Type Block

The following menu and G code are added for Milling.

1. Grooving Block
   Groove (B axis) : G1054

24.2.2 Figure Type Block

The following menu and G code are added for each Milling.

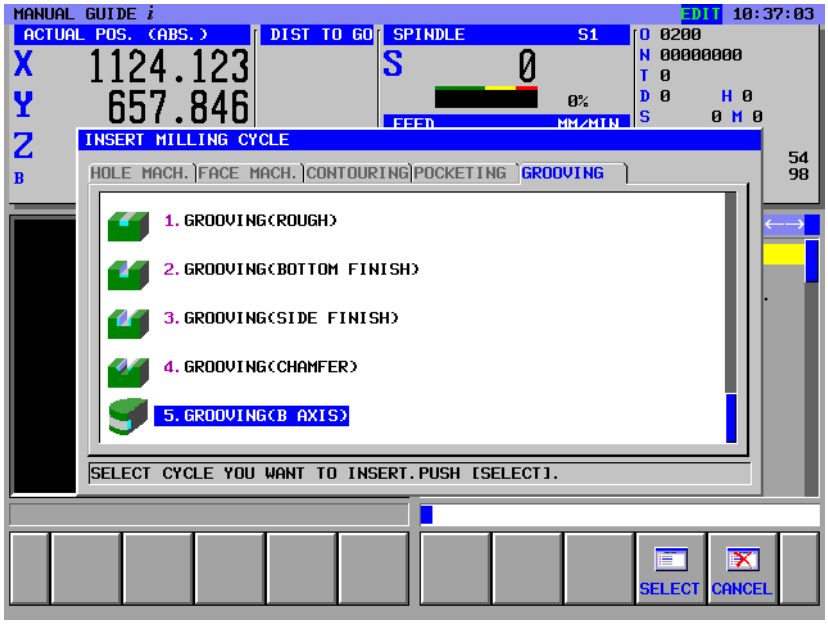
1. Hole position block
   B axis hole (arc) figure of cylindrical face (YB) : G1777
   B axis hole (random points) figure of cylindrical face (YB) : G1778
2. Groove figure block
   B axis groove figure of cylindrical face: G1775
   Y axis groove figure of cylindrical face: G1776

NOTE

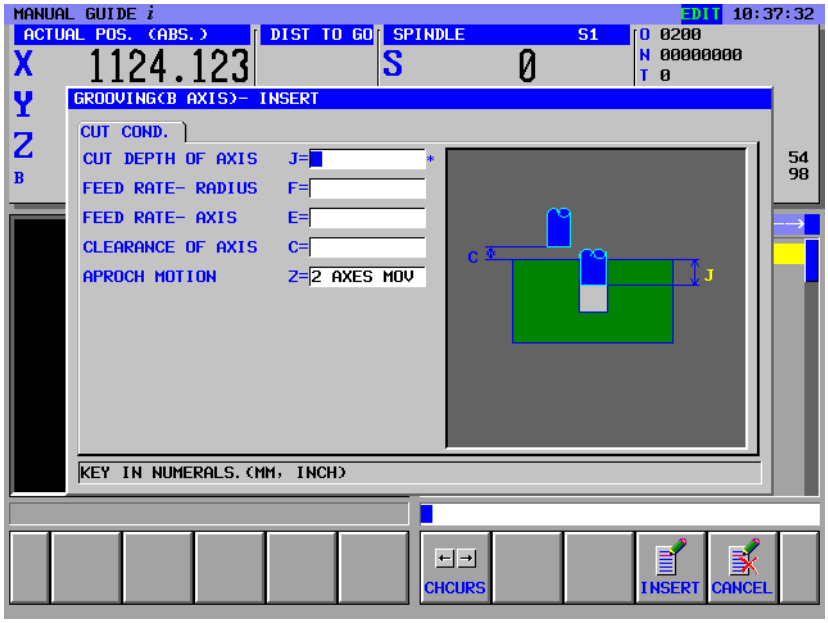
1 When the parameter P27004#3 = 1, the items of YZ plane is not displayed.

24.2.3 Machining Type Block Entry

Press [CYCLE] soft key, menu screen for selecting milling cycle is opened. And select tab “GROOVING”, the menu “Grooving (B axis)” are displayed as the following figure.

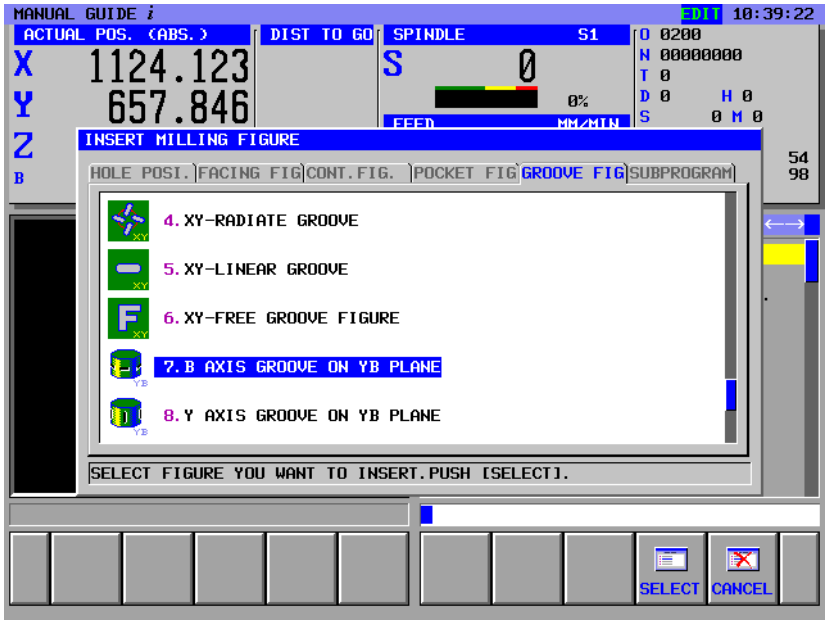


Then, select the menu to display data entry screen of grooving (B axis) as below figure. Input the data by following the entry items, and press [INSERT] soft key to create a program.

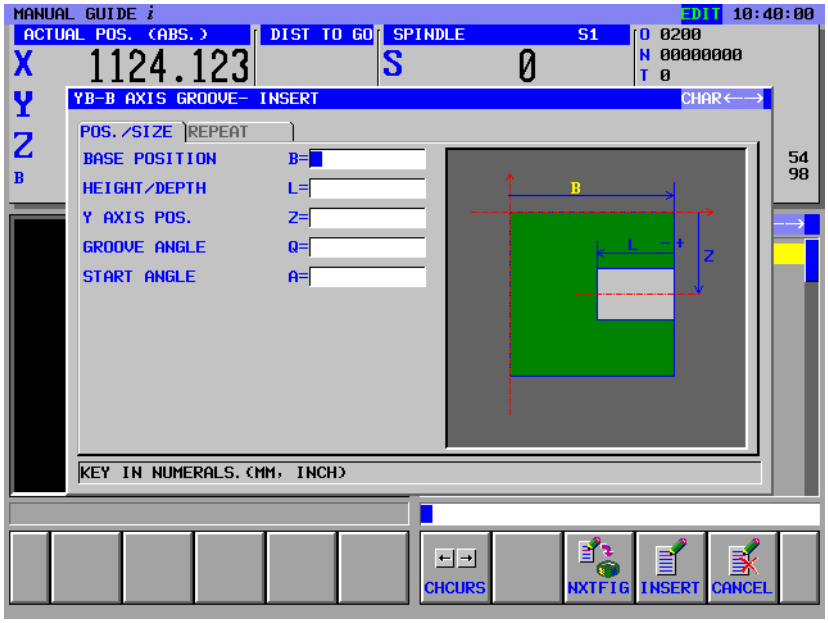


24.2.4 Figure Block Entry

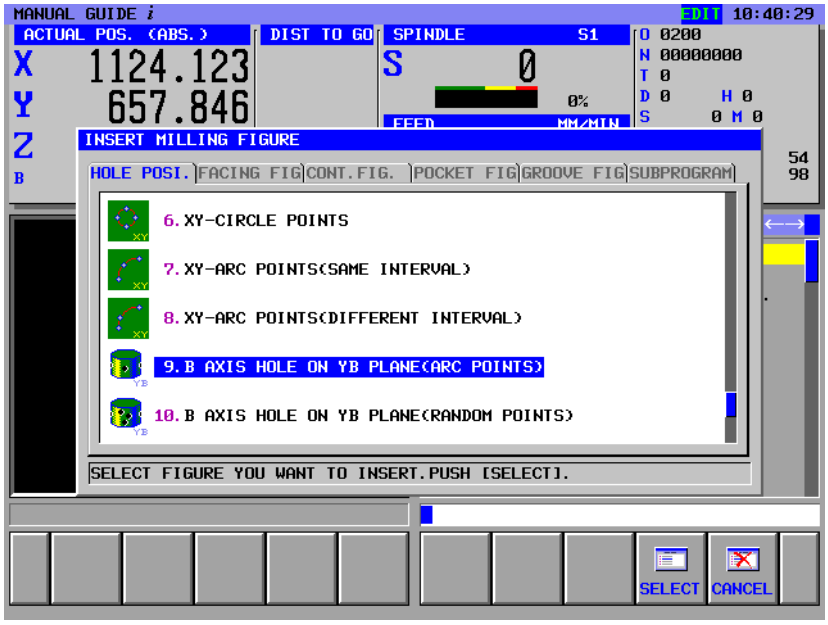
Press [FIGURE] soft key, the menu screen for selecting milling figure is opened. And select tab “GROOVE FIG”, the menus of the groove figure on YB plane are displayed.



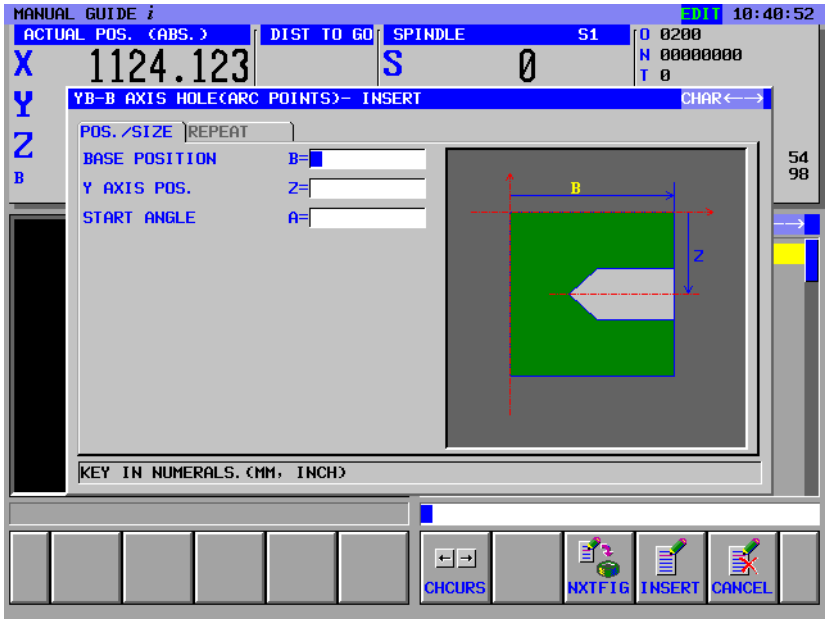
Select the menu to display the data entry screen of YB cylindrical face as below figure. Input the data by following the entry items, and press [INSERT] soft key to create a program.



Press [FIGURE] soft key, the screen for selecting milling figure is opened. And select [HOLE POSI.] tab, the menus of hole figure on YB plane are displayed as below figure.



Select the menu to display the figure data entry screen of YB cylinder face as below figure. Input the data by following the entry items, and press [INSERT] soft key to create a program.



24.2.5 Blank Data

The following G code is added as the Blank data.

(1) Cylinder Blank Figure (Round Y axis) : G1756

(2) Hollow Cylinder Blank Figure (Round Y axis) : G1757

(3) Prism Blank Figure (Round Y axis) : G1758

(4) Hollow Prism Blank Figure (Round Y axis) : G1759

(5) Free Figure Cylinder Blank Figure (Round Y axis)

Start Point (Round Y axis) : G1780

Line (Round Y axis) : G1781

Arc (CW) (Round Y axis) : G1782

Arc (CCW) (Round Y axis) : G1783

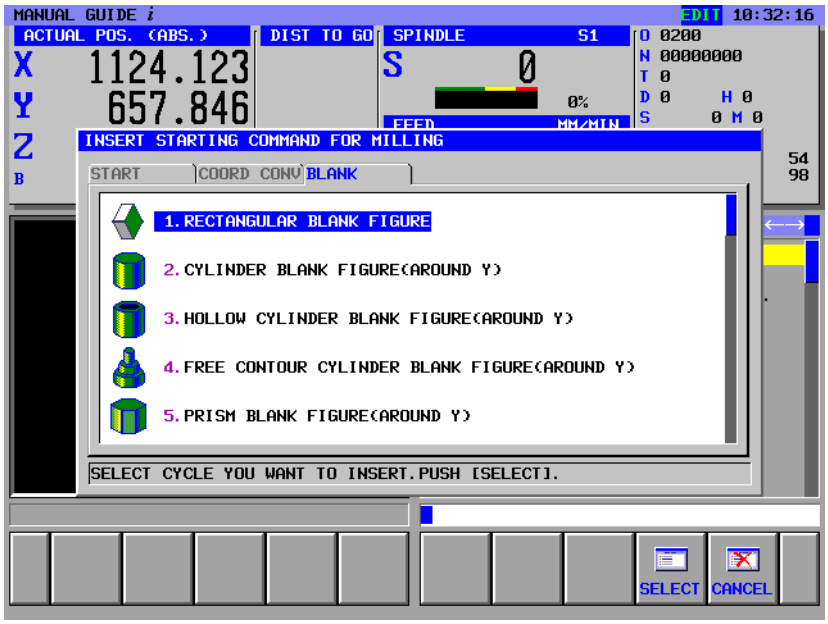
Face Removal (Round Y axis) : G1784

Corner R (Round Y axis) : G1785

End Point (Round Y axis) : G1786

24.2.6 Blank Data Entry

The menus of Blank Figure round Y axis are displayed for cylinder blank figure, hollow cylinder blank figure and free figure cylinder blank figure. By selecting the menu and enter the data according to a guidance message, define the blank figure.



24.3. Parameters

This function is available by setting the below parameters.

14717	

Axis Number of B Axis

Data Range : 0 ~ Number of control axis

14842	

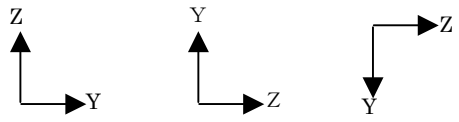
Drawing coordination on the blank figure screen of cylindrical free figure.

=0 Same as the data 6

=1 Plane figure Horizontal axis =+Y, Vertical axis =+Z

=5 Plane figure Horizontal =+Y, Vertical axis =+Z

=6 Plane figure Horizontal =+Y, Vertical axis =−Z



1 5 6

	#7	#6	#5	#4	#3	#2	#1	#0
27314							#1	#0

Whether to display a blank figure of Y-axis direction is depend on the following parameter settings.

#0 0： Display a blank figure of Z-axis direction at definition screen of blank figure.

1： Undisplay a blank figure of Z-axis direction at definition screen of blank figure.

#1 0： Undisplay a blank figure of Y-axis direction at definition screen of blank figure.

1： Display a blank figure of Y-axis direction at definition screen of blank figure.

* Because rectangular blank figure is common with each axis rotation, it displays always with no relation to the above-mentioned parameter.

27334	

Center axis of work rotation for rectangular blank figure at machining simulation.

=0 Round Z-axis

=1 Round Y-axis

	#7	#6	#5	#4	#3	#2	#1	#0
27003					#3	#2	#1	#0

27003#0=0, 27003#1=0, 27003#2=0, 27003#3=1,
Axis structure of machining center is X/Y/Z/B.
Displaying a menu of blank round Y-axis direction in blank definition.

	#7	#6	#5	#4	#3	#2	#1	#0
27004					#3	#2	#1	#0

P27004#0=1, 27004#1=0, 27004#2=0, 27004#3=1,
The following menus are displayed.

Grooving : Grooving (B axis) --- G1054
Hole Figure : B axis Hole figure (on arc) YB Plane --- G1777
B axis Hole figure (random points) YB Plane --- G1778
Groove Figure : B axis groove figure of YB Plane --- G1775
Y axis groove figure of YB Plane --- G1776

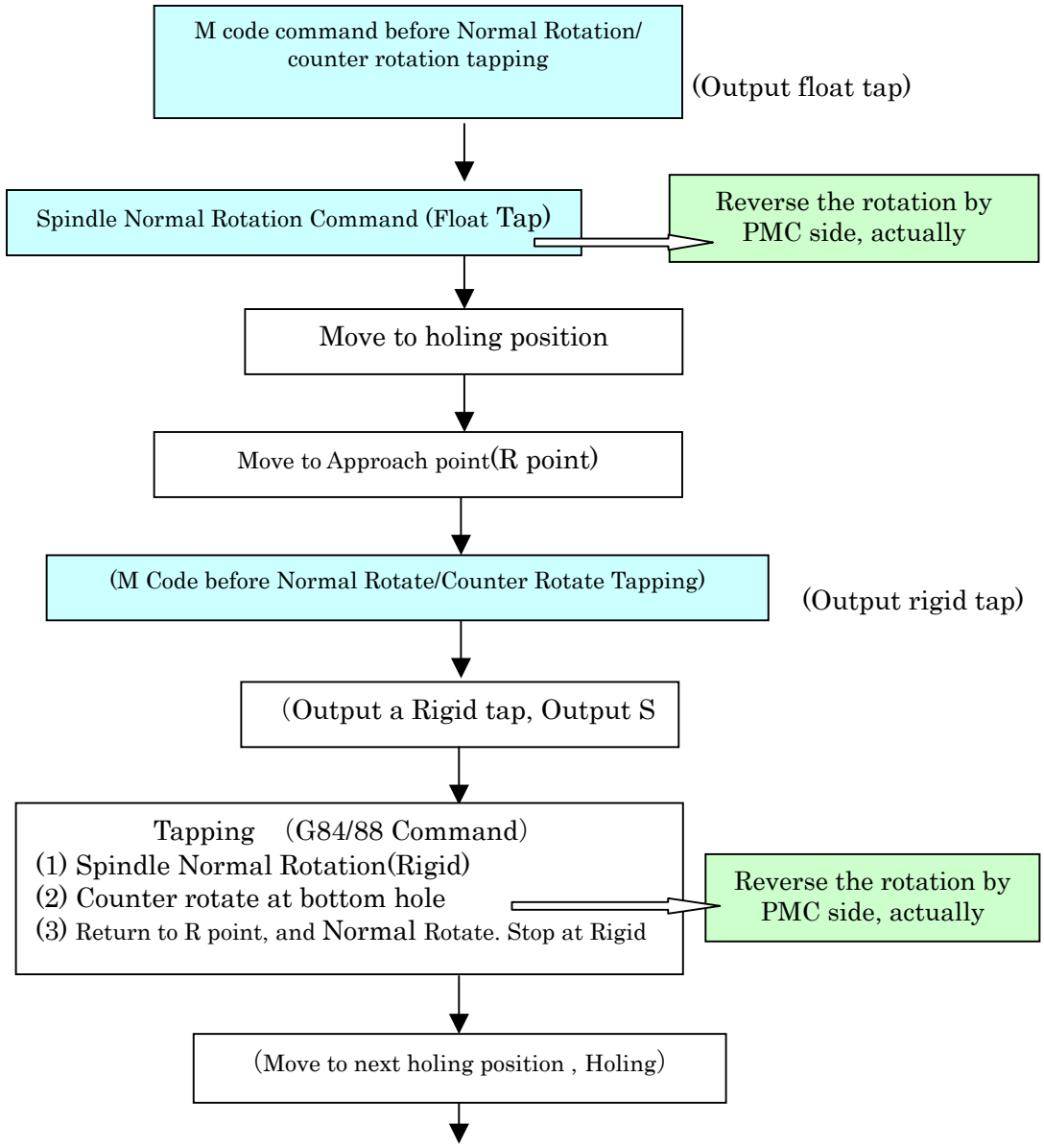
NOTE
1 After set the parameter No.27003, [F] key must be pressed at a screen of NOW LOADING after power has been restored. Required parameter is automatically set. (When parameter is set automatically, a message "NOW SETTING PARAMETERS" is displayed at bottom left corner.)
2 By set the No.27003, the following parameter related to display are automatically set.

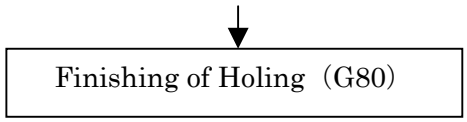
	No.14702#1 Select Spindle	No.27000#1 Change Hole Machining	No.27000#4 Face Position-M	No.27100#4 Face Position -T	No.27000#6 Convert Coordination	No.27001#0,#1 Rotation Axis Name	No.27060#1 PR1
ML0	0	0	0	0	1	0	1
ML1	0	0	0	0	0	0	1
ML2	0	0	0	0	0	0	1
ML3	0	0	0	0	0	0	1

	No.27004#0 Menu 1	No.27004#1 Menu 2	No.27004#2 OP 1	No.27004#3 OP 2	No.27314#0,#1 Set Blank Figure	No.27334 Set Rectangular Blank
ML0	1	0	0	0	0	0
ML1	0	0	0	0	0	0
ML2	1	0	1	0	0	0
ML3	1	0	0	1	1	1

25 Counter Tapping Cycle

At tapping cycle for turning (work rotation, tapping by rotating tool), counter tapping can be operated. This function is available by set the parameter 27195<#1 : TRV>, and <#0:WRV> to 1.
When this function is available, output [M-code to notice tapping / counter tapping], counter rotation of spindle is controlled by PMC with the M-code, as below.





Tapping cycle G 84 for turning, M04 is always output when spindle rotation command is weather normal rotation or counter rotation tapping. When tool is returned to R point, M 03 is output.（At rigid tapping, it output as M03 after moving to R point.→M05 at the bottom →M04,→M05 after returning.）

Because same M code is output by NC at normal rotation tapping or counter rotation tapping, actual spindle rotation is reversed at PMC side when operating counter tapping.

To reverse the spindle rotation by PMC side, [M code to notice Normal Rotation / Counter Rotation] is used to operate.

On PMC, provide a sign to reverse the rotation of spindle rotation command after this M code, and reverse the spindle rotation command by this sign. The sign newly provided is operate cancellation of this sign by PMC ladder side, such as canceling at TAP signal （F0002<#5:TAP>）.

25.1. Specify Normal Tapping / Counter Tapping

Parameter No.27195<#1 : TRV>, <#0:WRV> are set to 1, [Machining Type W] is displayed at deleting condition menu of tapping.

When a cursor is placed to [Machining Type W] , soft key [Normal] or [Counter Tap] is displayed. Select either one.

Specified upper machining type is programmed as below.

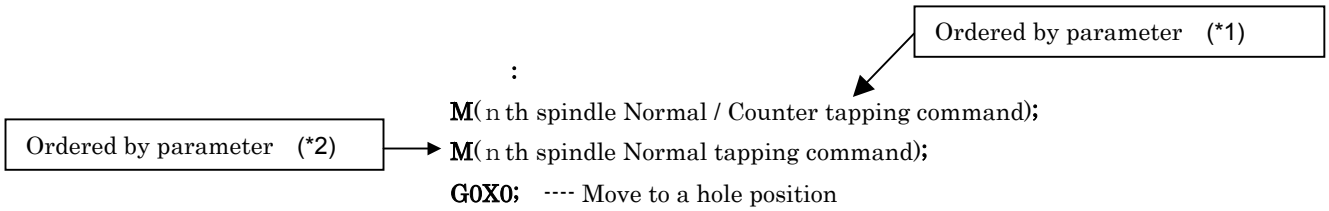
G1102 W1/W2 ……..; (Work Rotation) W1: Normal W2: Counter Tap

G1112 W1/W2 ……..; (Tool Rotation) W1: Normal W2: Counter Tap

25.2. Machining Operation

25.2.1 Float Tapping by work rotation

When this function is available, operate G1102 to operate a command as followings. (G code Type A)



```
Z___ _;  ----  Move to approach point
G84 Z_____R_____P____F____;  ----  Tapping Command
G80;
      :
```

When a parameter of counter tapping command M code is unset at operating a counter tapping command, P/S alarm [3017 NO M-CODE BEFORE TAP CYCL] is occurred.
When a parameter of normal tapping command M code is unset at operating a normal tapping command, no data is output with off alarm.

(*1) Parameter No.27196 ～ 27201
(*2) Parameter No.27202,27204,27206
Please refer the chapter of parameter description for details for the parameters listed above.

25.2.2 Rigid Tapping by work rotation

When this function is available, operate G1102 to operate a command as followings.(G code Type A)

```
      :
G0X0;  ----  Move to a hole position
Z____;  ----  Move to approach point
M(n th spindle Normal/Counter tapping command);
M__S____;  ----  Command of Rigid tapping mode
G84 Z_____R_____P____F____; ---- Command of Rigid tapping mode
G80;
      :
```

Ordered by parameter (*1)

When a parameter of counter tapping command M code is unset at operating a counter tapping command, P/S alarm [3017 NO M-CODE BEFORE TAP CYCL] is occurred.
When a parameter of normal tapping command M code is unset at operating a normal tapping command, no data is output with off alarm.

(*1) Parameter No.27196 ～ 27201
Please refer the chapter of parameter description for details for the parameters listed above.

25.2.3 Selecting Spindle

As explaining below, M code before tapping or M code for spindle normal rotation are required to output by following a spindle currently selected. Specify a selecting spindle number from a screen as below.

・When this function is available with a spindle is1(parameter No.14702#1=0), [spindle number Q] is displayed.
Spindle Number Q Spindle 1

・When a cursor is placed at [Spindle Number Q], soft key [Spindle 1] and [Spindle 2] are displayed. Spindle number selected at [Spindle number Q] is displayed by pressing soft key.

・Specified upper spindle number is programmed a below.
G1102 W1……Q n … ; (Work Rotation Tap)
Qn = Q1:Select a Spindle 1, Q2: Select a Spindle 2

NOTE) Operating G 1102 with this function is available, a movement when returned value Q of spindle number is unset is as follows.

・When using more than 2 spindle(parameter No14702#1=1), it alarm out as [WRONG SPINDLE NUMBER].
・When using spindle is one(parameter No. 14702#1=0), it operates as spindle number 1. (Refer a parameter for spindle number 1, for parameter set as referring spindle parameter)

25.2.4 Float Tapping by Tool Rotation

Operating G1112 when this function is available, the following command is operated. (Listed as G code Type A)

```
      :
G0_____;  ----  Move to 1st hole position
Z___;  ----  Move to approach point
M(For Each Turrets Pre command to rotating tool /
      command before counter tapping);
M(For Each Turrets Pre command to rotating tool);
G84/88 Z_____R_____P____F____;  ----  Tapping Command
_____;  ----  Move to 2nd hole position, and holing
      :
G80 ;
      :
```

Ordered by parameter (*1) → M(For Each Turrets Pre command to rotating tool / command before counter tapping);

Ordered by parameter (*2) → M(For Each Turrets Pre command to rotating tool);

When a parameter of M code command before a counter tapping is unset, P/S alarm occur.[3017 NO M-CODE BEFORE TAP CYCL]
When a parameter of normal tapping command M code is unset at operating a normal tapping command, no data is output with off alarm.

(*1) Parameter No.27208,27209
(*2) Parameter No.27210
Please refer the chapter of parameter description for details for the parameters listed above.

25.2.5 Rigid Tapping by Tool Rotation

Operating G1112 when this function is available, the following command is operated. (Listed as G code Type A)

```
      :
G0_____;  ----  Move to 1st hole position
Z___;  ----  Move to approach point
M(For Each Turrets Pre command to rotating tool /
      command before counter tapping);
M__ S____;  ----  Rigid Tapping mode Command
G84/88 Z_____R_____P____F____;  ----  Rigid Tapping Command
_____;  ----  Move to 2nd hole position, and holing
      :
G80 ;
      :
```

Ordered by parameter (*1) →

When a parameter of counter tapping command M code is unset at operating a counter tapping command, P/S alarm[3017 NO M-CODE BEFORE TAP CYCL]is occurred.
When a parameter of normal tapping command M code is unset at operating a normal tapping command, no data is output with off alarm.

(*1) Parameter No.27208,27209
Please refer the chapter of parameter description for details for the parameters listed above.

25.3. Parameters

About the details of the parameters for this function, refer to the chapter "30 Parameters". And there is the following CNC Parameters related to this function.

	#7	#6	#5	#4	#3	#2	#1	#0
5200								G84

Command method of rigid tapping.

G84 Command method of rigid tapping.

0 : command M code of rigid tapping mode command preceding G84command.

1 : Not command M code of rigid tapping mode command (G84command)

(It cannot command G84 for float tapping as G code.)

5210	M code command in rigid tapping mode

M code command in rigid tapping mode

Setting Range 0 ～255

M29 is output when 0 is set

When M code over 256 is needed to use, set a parameter No5212.

5212	M code command in rigid tapping mode

M code command in rigid tapping mode

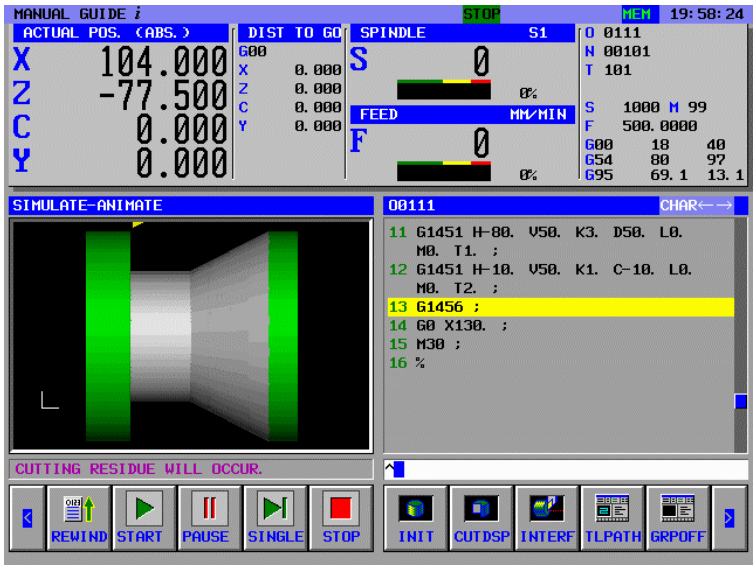
Setting Range 0 ～255

When M code over 256 is needed to use, set this parameter.

26 A warning message when a residual cutting part remains

26.1. Details

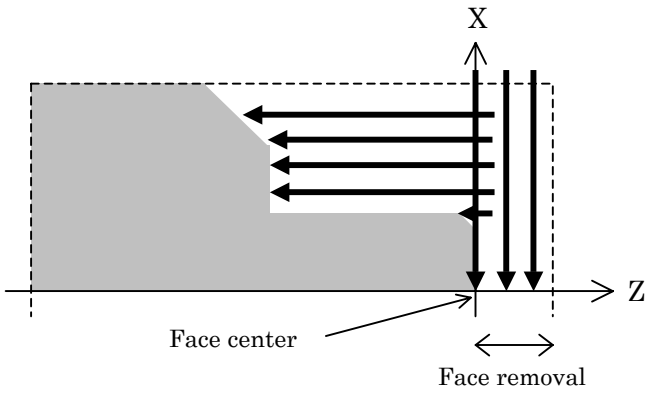
When a residual cutting part remains in turning cycle, a warning message ”CUTTING RESIDUE WILL OCCUR” is displayed. And, it is not necessary to set any parameter for this function.



27 Simultaneous Facing at Turning Cycle

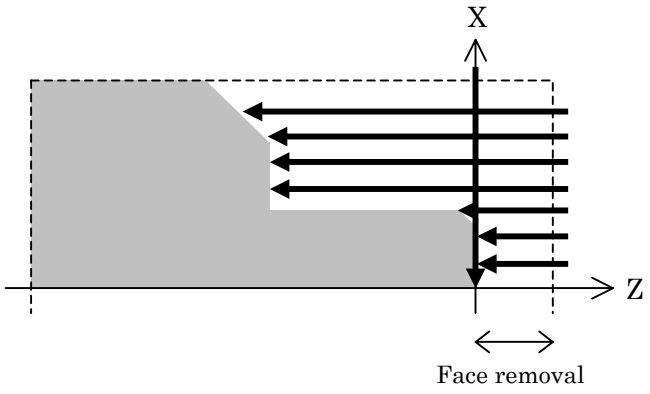
For blank outer or inner machining, facing can be operated at the same process (Simultaneous Facing) as follows.
This function is available when a parameter No. 27120#2 = 1.
When this function is available, by selecting the turning (outer rough) or the turning (inner rough) menu, [Facing] tab is displayed and possible to specify a simultaneous facing of turning.
This function is available only at turning rough.

< Roughing with using a simultaneous facing of turning >



(Operating outer machining after cutting the facing removal part)

< Roughing without using a simultaneous facing of turning >



27.1. Operations

27.1.1 Specify the Simultaneous Facing

When this function is available, [Facing] tab is displayed at [Turning (Outer Rough)] or [Turning (Inner Rough)].
Select [Facing] tab to display the following items.

・Facing availability N

Selection of facing availability is selected by the soft key [Facing] and [None] displayed at this tab.
On outer rough, the following entry items are displayed by selecting [Facing]. On inner rough , it is not displayed.

・Face Removal I

Cut depth and feed rate at facing is decided as following.

Facing cut depth is automatically set by face removal amount and outer/inner cut depth amount. Please refer a chapter on machining operations for in details.

Facing cut feed rate is used cut feed rate of outer/inner cut. Please refer a chapter on machining operations for in details.

27.1.2 Settings of Face Removal

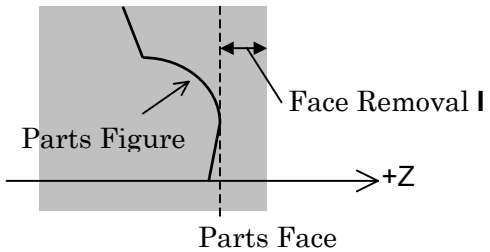
Face removal amount is used either [Face Removal I] that is entered on the screen, or face removal set by a parts figure and a blank figure.
However, a face removal cannot specify at inner rough and it should specify a blank figure at face removal parts.
When any removal amount isn't entered, simultaneous facing cannot operated.

- When face removal I is not entered, face removal amount set by parts figure, blank figure is used.
- When face removal I is entered, a larger amount of each removal amount is used.

Settings of [Face Removal]

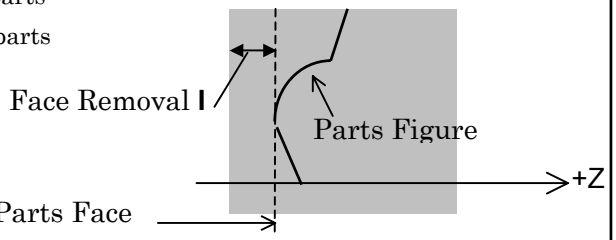
<When cut direction is －Z>

Specify the amount from parts Face(Max z point of parts figure) as right figure.



<When cut direction is ＋Z>

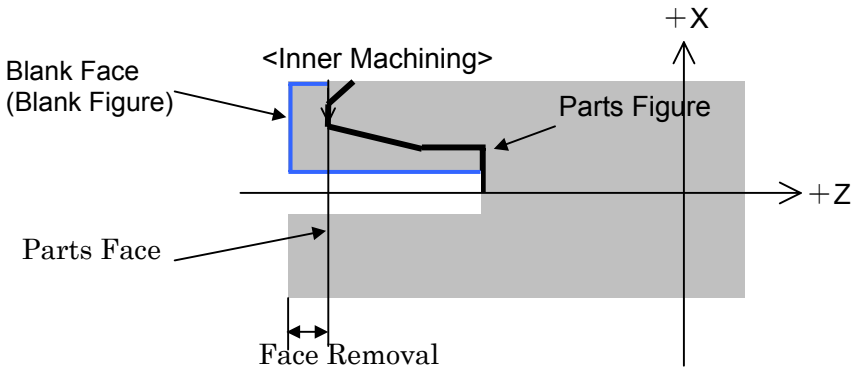
Specify the amount from parts Face(Minimum z point of parts figure) as right figure.



Face Removal set from Parts Figure, Blank Figure

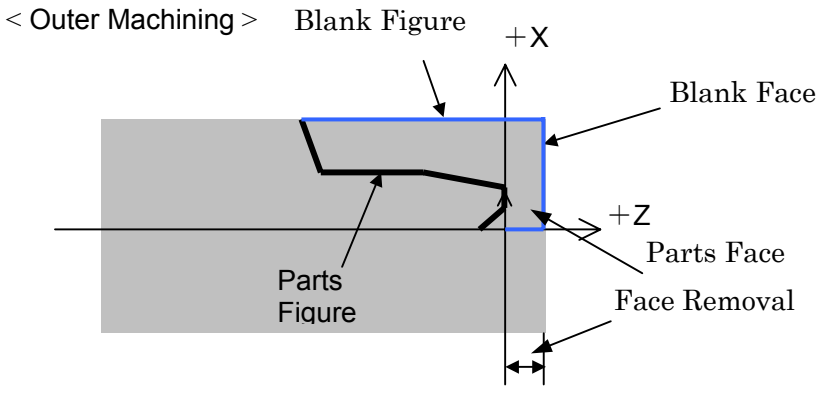
＜When cut direction is －Z＞

As the below figure, either at outer or inner, the amount from parts face(Maximum Z of parts figure) to blank figure(Maximum Z of blank figure) is the face removal amount.



＜When cut direction is －Z＞

As the below figure, either at outer or inner, the amount from parts face(Minimum Z of parts figure) to blank figure(Minimum Z of blank figure) is the face removal amount.

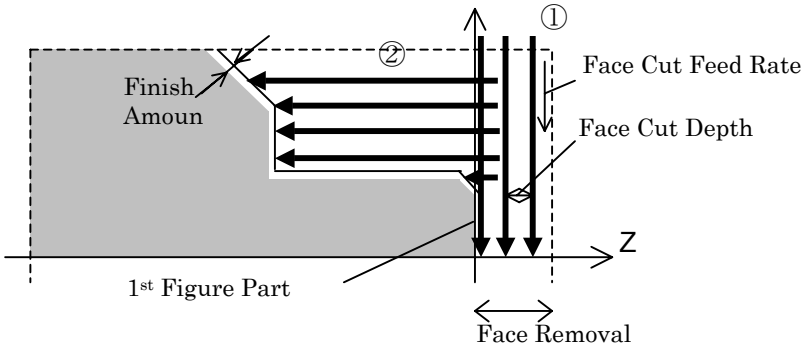


Note: Cut direction of Facing is follow the cut direction of outer/inner machining.

27.2. Machining Operations

27.2.1 Outer Facing Rough

At outer facing rough, as the figure below cut off the face part first(①), then outer machining (②) is operated. Cut direction of face part is always X minus direction as the figure below.



Face cut depth is automatically set by face removal and outer/inner cut depth as follow.

(Face Removal)
---------- ==(round up after the decimal point)=>Number
(Outer or Inner Cut Depth) of Cutting N

(Face Removal)
---------- ==Face Cut Depth d'
(N)

Calculated face cut depth d +override, recalculate the face cut depth.

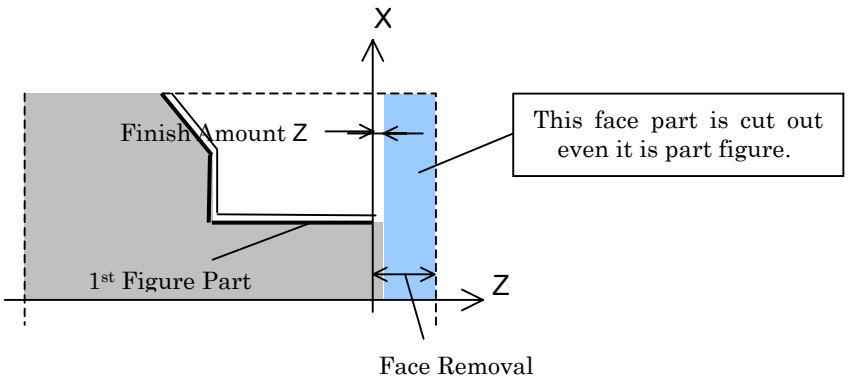
(Face Removal Amount)
---------- =(round up after the decimal point)=>Number
(d')x Override of Cutting N'

(Face Removal)
---------- == Face Cut Depth d
(N')

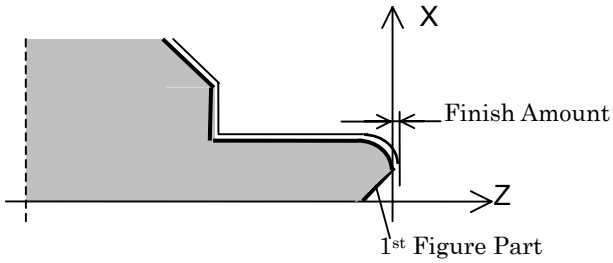
Specify the override by parameter 27131. When a set value is 0, the override is 100%

Face cut feed rate is the amount override(%) is added to outer/inner cut feedrate.
That override is specified by a parameter 27132, and a set value is 0,the override is 100%.

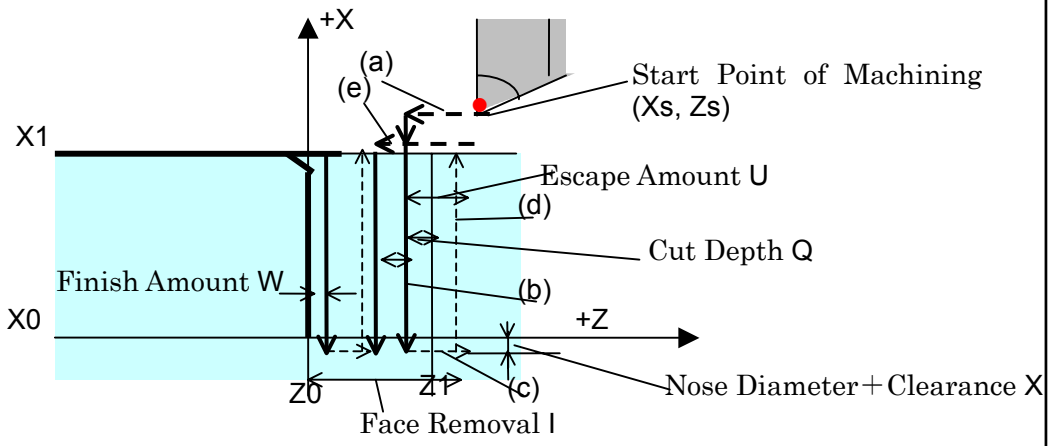
As the figure below, even though 1st figure part is started from a point X>0, face part will be cut when face removal is set. However, face part is cut off as below figure, even with these part figure is defined.



As upper description, when finish amount is set, finishing amount of face part remain after completing outer facing rough.



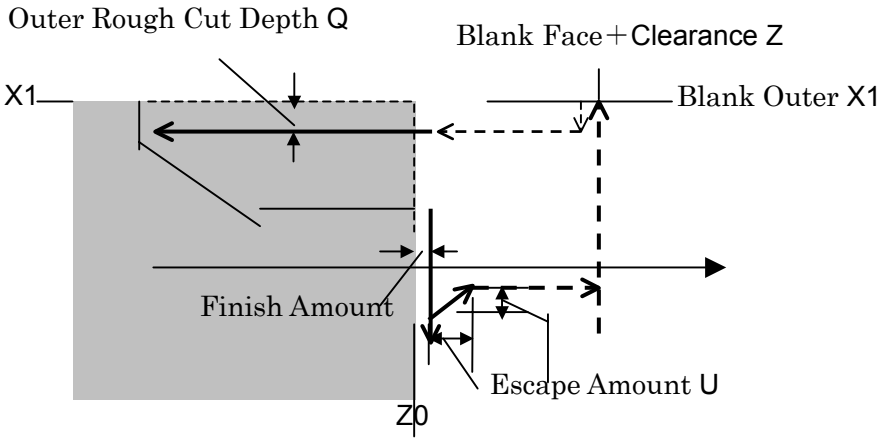
Cut off the face part operates as followings.



(a) Tool is moved to “Z1－Q” beside Z axis
Then, move to [X1 + Clearance] beside X axis
(b) Tool is cut “X0 – Nose Diameter*2 – Clearance*2” beside X axis.
(c) Tool is returned to “Z1 + Clearance” beside Z axis by G00.
(d) Tool is returned to X1＋ClearanceX*2 by G00.
(e) Tool is moved to “Z1－2*Q”, the point cut by amout Q from the last cut position, beside Z axis by G00
(f) Here in after, tool is cut to ”Z0+W ” by cut depth Q.

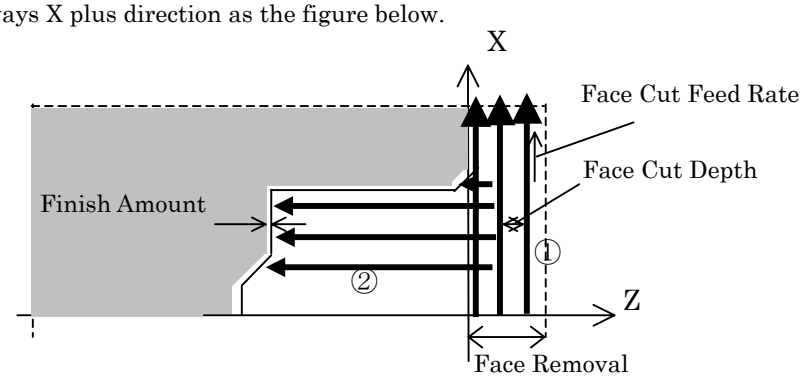
(g) At last returning motion, escape by escape amount to 45° direction as the following, and then move to "Blank Face + Clearance Z" beside Z axis by G00.

Move to +X direction "Blank outer point X1" by G00. Then, goes to outer rough operation.

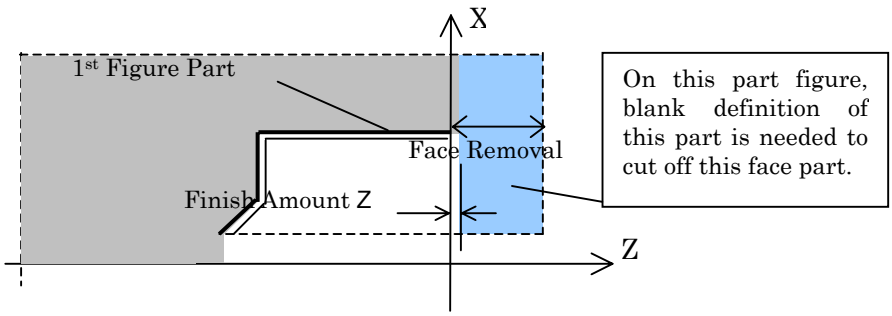


27.2.2 Inner Facing Rough

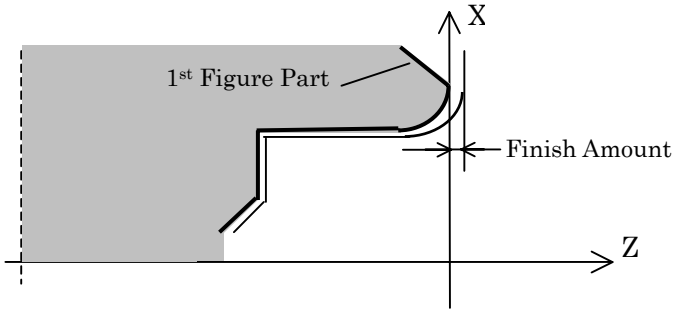
When inner rough machining, as the figure below cut off the face part first(①), then inner machining (②) is operated. Cut direction of face part is always X plus direction as the figure below.



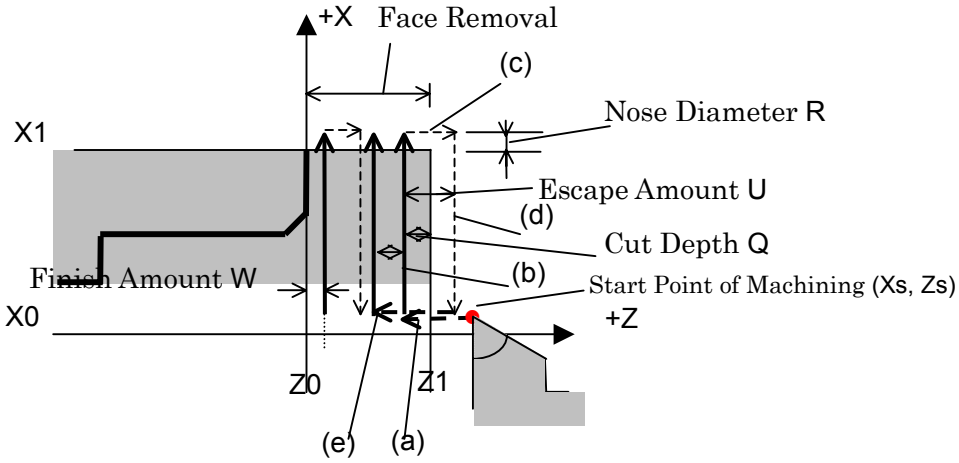
As the figure below, when 1st figure part is begin at X position inner from outer of blank, and to cut off a face part, make sure to define face part of blank figure.



As upper description, when finish amount is set, finishing amount of face part remain after completing outer facing rough as figure below.

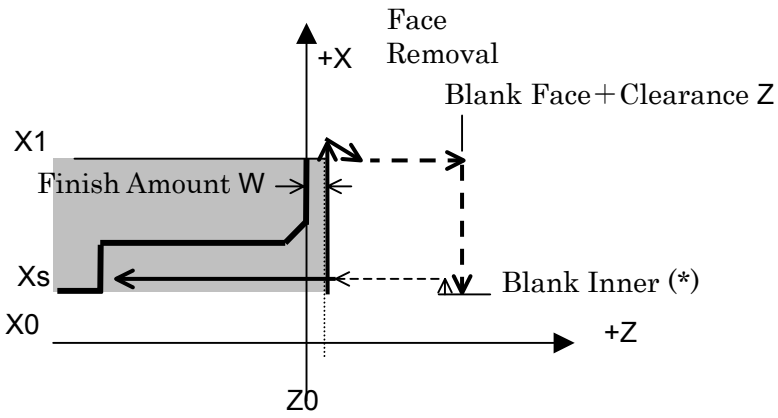


Cut off the face part ① operates as followings.



Each operation of face cut off (a)～(e) is the same as outer facing rough only move direction is changed.

Approach motion to inner rough after the completion of facing is as follows. As the same as outer facing, after escaping by escape amount to 45° direction, move to the point "Blank Face + Clearance Z" beside Z axis direction by G00. Then move to "Blank inner point Xs" by G00. Then, goes to inner rough operation.


Face
Removal
+X
Blank Face＋Clearance Z
X1
Finish Amount W
Xs
Blank Inner (*)
X0
+Z
Z0

28 Tool Management Function for Series 30*i*

Tool management function for Series 30*i* is available.
For details of this function, refer to [V. Tool Management Function] on [B-63874EN/05 FANUC MANUAL GUIDE *i* operator's manual].

	#7	#6	#5	#4	#3	#2	#1	#0
14705	#7		#5	#4			#1	#0

#0 Make the shortcut function of to soft key
0： available.
1： invalid.

#1 Cursor position after the copy are
0： Return to the top block of a selected range.
1： Keep a cursor to the end block of a selected range.

#4 Softkey [T-OFS]
0： Display.
1： Undisplay.

#5 Softkey [TOOL]
0： Undisplay.
1： Display.

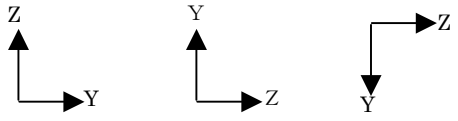
#7 Tool icon display function and shortcut function to tool database screen are
0： not enabled.
1： enabled.

14717	

Axis Number of B Axis
Data Range： 0 ～ Number of control axis

14842	

Drawing coordination of free figure cylinder blank figure screen.
=0 Same as 6
=1 Plane figure Horizontal axis =+Y, Vertical axis =+Z
=5 Plane figure Horizontal =+Y, Vertical axis =+Z
=6 Plane figure Horizontal =+Y, Vertical axis =−Z



1 5 6

14845	TLTAB

Set the display position of each tab of tool management data, magazine information, tool offset, and tool information.

1digit	Tool Management Data Tab
10 place	Magazine Data Tab
100 place	Tool Offset Tab
1000 place	Tool Data Tab

Set 1～4 to each value to set a displaying order of a tab.
Tab is no displayed when 0 is set. When 0 is set to all, it is displayed by default sorting.
Default sorting display is as follows.

1st	Tool Management Data Tab
2nd	Magazine Data Tab
3rd	Tool Offset Tab
4th	Tool Data Tab

Please input the figure of four digits to this parameter. And the figure of each digit must not overlap.

	#7	#6	#5	#4	#3	#2	#1	#0
27002			#5	#4				

#4 Display/Undisplay a Side Finish Feed Rate on free figure for milling.
0： Undisplay.
1： Display.
#5 At entry screen of Pocketing/Embossing/Contouring/Grooving cycle
0： Conventional order of tool cut depth and finish amount.
1： Switch the order of tool cut depth and finish amount.

	#7	#6	#5	#4	#3	#2	#1	#0
27003					#3	#2	#1	#0

27003#0=0, 27003#1=0, 27003#2=0, 27003#3=1,
Axis structure of machining center is X/Y/Z/B.
Displaying a menu of blank round Y-axis direction in blank definition.

	#7	#6	#5	#4	#3	#2	#1	#0
27004					#3	#2	#1	#0

P27004#0=1, 27004#1=0, 27004#2=0, 27004#3=1,
The following menus are displayed.

Grooving : Grooving (B axis) --- G1054
Hole Figure : B axis Hole figure (on arc) YB Plane --- G1777
B axis Hole figure (random points) YB Plane --- G1778
Groove Figure : B axis groove figure of YB Plane --- G1775
Y axis groove figure of YB Plane --- G1776

NOTE
3 After set the parameter No.27003, [F] key must be pressed at a screen of NOW LOADING after power has been restored. Required parameter is automatically set. (When parameter is set automatically, a message "NOW SETTING PARAMETERS" is displayed at bottom left corner.)
4 By set the No.27003, the following parameter related to display are automatically set.

	No.14702#1 Select Spindle	No.27000#1 Change Hole Machining	No.27000#4 Face Position-M	No.27100#4 Face Position -T	No.27000#6 Convert Coordination	No.27001#0,#1 Rotation Axis Name	No.27060#1 PR1
ML0	0	0	0	0	1	0	1
ML1	0	0	0	0	0	0	1
ML2	0	0	0	0	0	0	1
ML3	0	0	0	0	0	0	1

	No.27004#0 Menu 1	No.27004#1 Menu 2	No.27004#2 OP 1	No.27004#3 OP 2	No.27314#0,#1 Set Blank Figure	No.27334 Set Rectangular Blank
ML0	1	0	0	0	0	0
ML1	0	0	0	0	0	0
ML2	1	0	1	0	0	0
ML3	1	0	0	1	1	1

27013 **MSTOP**

MSTOP : M code for Spindle stop in Fine Boring
=0 Output M05 for Spindle stop
≠0 Set M code number for Spindle stop
For Fine Boring for M series with End Feedrate (G1005) and Fine Boring for T series (G1105, G1115)

27014 **MORIE**

MORIE : M code for Spindle orientation in Fine Boring
＝0 Output M19 for Spindle orientation
≠0 Set M code number for Spindle orientation

For Fine Boring for M series with End Feedrate (G1005) and Fine Boring for T series (G1115)

27018	SRCHBLNK

Blank form data is automatically copied at figure entry, reused function is available by setting the below parameter.

SRCHBLNK : Block number when searching a blank form block. When a standard value is (=0), block number to search is 100.

	#7	#6	#5	#4	#3	#2	#1	#0
27020		#6	#5	#4	#3			#0

#0 Input Item [GO PAST AMOUNT SET (H)] for drilling
0 : Undisplay.
1 : Display.

#3 In Fine Boring for T series:
0 : Output M code for spindle stop before an oriented spindle stop in Fine Boring (G1115).
Output M code for spindle stop in Fine Boring (G1105).
1 : Not output M code for spindle stop in Fine Boring (G1105, G1115).

#5/#4 : Set the axis and direction in which the tool in Fine Boring for T series (G1115) is got free. #5 and #4 are set as shown below by plane selection.

#5	#4	G17	G18	G19
0	0	+X	+Z	+Y
0	1	-X	-Z	-Y
1	0	+Y	+X	+Z
1	1	-Y	-X	-Z

#6 Set the direction in which the tool in Fine Boring for T series (G1105) is got free.
0 : +X direction is free.
1 : -X direction is free.

	#7	#6	#5	#4	#3	#2	#1	#0
27061			#5					

#5 Display / Undisplay an input item [Cut Angle] at pocketing.

0： Display.

1： Undisplay.

	#7	#6	#5	#4	#3	#2	#1	#0
27095						#2	#1	#0

#0 Set parameter No.27095#0 to exchange the input items [Bottom Thickness] and [Side Thickness] at cycle entry screen and figure entry screen.

0： Input bottom / side thickness at cycle entry screen.

1： Input bottom / side thickness at figure entry screen.

#1 Blank form data is automatically copied at figure entry, reused function is available by setting the below parameter.

0： Not reuse the blank form data for figure data at machining.
(Reuse soft key is not displayed at figure entry screen.)

1： Reuse the blank form data for figure data at machining.
(Reuse soft key is displayed at figure entry screen)

#2 Polygon pattern of fixed figure is

0： Undisplay.

1： Display.

	#7	#6	#5	#4	#3	#2	#1	#0
27102				#4	#3			

#3 Display / undisplay [Grinding Amount] at free figure of ZX plane for turning.

0： Undisplay.

1： Display.

#4 Display / undisplay a semi finish feed rate on a free figure of ZX plane at turning.

0： Undisplay.

1： Display.

	#7	#6	#5	#4	#3	#2	#1	#0
27120						#2		

#2 Simultaneous Facing of Turning at outer / inner machining

0： unavailable.

1： available.

27131	CDOVR

CDOVR Override of face cut depth with Simultaneous Facing of Turning
1～200%, 1 % Unit
Set 0 is considered as 100%

27132	CFOVR

CFOVR Override of face cut feed rate with Simultaneous Facing
1～200%, 1 % Unit
Set 0 is considered as 100%

	#7	#6	#5	#4	#3	#2	#1	#0
27140							#1	

#1 Display / Undisplay [Threading Direction]
0： Undisplay.
1： Display.

	#7	#6	#5	#4	#3	#2	#1	#0
27170							#1	

#1 0： Undisplay Roughing Type of Turning Grooving.
1： Display Roughing Type of Turning Grooving.
(Roughing by versatile tool is available)

27178	

Depth direction Minimum cut depth of Turning Grooving (Roughing) by versatile tool.

	#7	#6	#5	#4	#3	#2	#1	#0
27195							#1	#0

#0 Counter tapping by work rotation
0： Disabled.
1： Enabled.

#1 Counter tapping by rotating tool
0： Disabled.
1： Enabled.

The below parameter 27196～27211 are available only when counter tapping (both or either 27195<#0>, and <#1> is set to “1”) is available.

27196	S1TTMN

S1TTMN M code command before the normal tapping of spindle 1
Setting Range 0 ～65535
No output when 0 is set. This data is set individually per turret (system) in multi path system.

27197	S1TTMR

S1TTMR M code command before the counter tapping of spindle 1
Setting Range 0 ～65535
No output when 0 is set。 This data is set individually per turret (system) in multi path system.

27198	S2TTMN

S2TTMN M code command before the normal tapping of spindle 2
Setting Range 0 ～65535
No output when 0 is set This data is set individually per turret (system) in multi path system.

27199	S2TTMR

S2TTMR M code command before the counter tapping of spindle 2
Setting Range 0 ～65535
No output when 0 is set。 This data is set individually per turret (system) in multi path system.

27200	S3TTMN

S3TTMN M code command before the normal tapping of spindle 3
Setting Range 0 ～65535
No output when 0 is set This data is set individually per turret (system) in multi path system.

27201	S3TTMR

S3TTMR M code command before the counter tapping of spindle 3
Setting Range 0 ～65535
No output when 0 is set. This data is set individually per turret (system) in multi path system.

27202	S1NMLM

S1NMLM M code for rotating spindle 1.
Setting Range 0 ～65535

M3 is output when 0 is set. This data is set individually per turret (system) in multi path system.

27203	S1RVSM（Reserved for counter rotation of spindle 1）

(For each turret (system))

27204	S2NMLM

S2NMLM M code for rotating spindle 2.
Setting Range 0～65535
M3 is output when 0 is set This data is set individually per turret (system) in multi path system.

27205	S2RVSM（Reserved for counter rotation of spindle 2）

(For each turret (system))

27206	S3NMLM

S3NMLM M code for rotating spindle 3.
Setting Range 0～65535
M3 is output when 0 is set This data is set individually per turret (system) in multi path system.

27207	S3RVSM（Reserved for counter rotation of spindle 3）

(For each turret (system))

27208	MILTMN

MILTMN M code command before the normal tapping by rotating tool
Setting Range 0～65535
M3 is output when 0 is set This data is set individually per turret (system) in multi path system.

27209	MILTMR

MILTMR M code command before the counter tapping by rotating tool
Setting Range 0～65535
M3 is output when 0 is set This data is set individually per turret (system) in multi path system.

27210	MLNMLM

MLNMLM M code for rotating tool
Setting Range 0～65535
＜Related parameter：No.27195#3(MLT)＞

M3 is output when 0 is set This data is set individually per turret (system) in multi path system.

27211	MLRVSM（Reserved for counter rotation of tool rotation）

(For each turret (system))

	#7	#6	#5	#4	#3	#2	#1	#0
27314							#1	#0

Whether to display a blank figure of Y-axis direction is depend on the following parameter settings.

#0 0： Display a blank figure of Z-axis direction at definition screen of blank figure.

1： Undisplay a blank figure of Z-axis direction at definition screen of blank figure.

#1 0： Undisplay a blank figure of Y-axis direction at definition screen of blank figure.

1： Display a blank figure of Y-axis direction at definition screen of blank figure.

※Because rectangular blank figure is common with each axis rotation, it displays always with no relation to the parameter above.

27334	

Center axis of work rotation for rectangular blank figure at simulation.

=0 Z-axis direction

=1 Y-axis direction

	#7	#6	#5	#4	#3	#2	#1	#0
27384								#0

#0 0： When versatile tool is drew by animation, tip position is drew in front.

1： When versatile tool is drew by animation, tip position is drew in rear.

27385	

Holder length when versatile tool is drew by animation.

Data Unit： When input unit by milli (0000#2=0)：0.001 mm

When input unit by inch (0000#2=1)：0.0001 inch

When input 0 by millimeter input unit (0000#2=0), it automatically set to 50mm

When input 0 by inch (0000#2=1), it automatically set to 1.9685inch.

27386	

Holder width when versatile tool is drew by animation.

Data Unit : When input unit by milli (0000#2=0) : 0.001 mm
When input unit by inch (0000#2=1) : 0.0001 inch
When input 0 by millimeter input unit (0000#2=0), it automatically set to 14mm.
When input 0 by inch (0000#2=1), it automatically set to 0.5512inch.