MITSUBISHI ELECTRIC

Changes for the Better

MITSUBISHI CNC

Simple Programming Function

NAVI LATHE Instruction Manual

IB-1500146(ENG)-E

Introduction

This instruction manual describes how to use NAVI LATHE. Incorrect handling may lead to unforeseen accidents, so make sure to read this instruction manual thoroughly before operation to ensure correct usage.
NAVI LATHE supports the following NC series.

Written as in this manual	Appropriate NC
M7 series	M70/M70V/M700/M700V series
E70 series	E70 series

Notes on Reading This Manual

(1) This manual describes as many special operations as possible, but it should be kept in mind that operations not mentioned in this manual cannot be performed.

(2) For the specifications of individual machine tools, refer to the manuals issued by the respective machine tool builders. The "restrictions" and "available functions" described by the machine tool builders have precedence over this manual.

(3) This manual is written on the assumption that all option functions are added.
Confirm with the specifications issued by the machine tool builder before starting to use.

(4) Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.

(5) Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

Refer to the following documents.

MITSUBISHI CNC 700/70 Series Instruction Manual IB-1500042
MITSUBISHI CNC 700/70 Series Setup Manual IB-1500124
MITSUBISHI CNC 700/70 Series Programming Manual (Lathe System) IB-1500057
MITSUBISHI CNC M700V/M70V Series Instruction Manual.....IB-1500922
MITSUBISHI CNC M700VW Series Setup Manual..... IB-1500933
MITSUBISHI CNC M700VS Series Setup Manual..... IB-1500906
MITSUBISHI CNC M70V Series Setup Manual..... IB-1500958
MITSUBISHI CNC M700V/M70V Series Programming Manual (Lathe System) IB-1500924
MITSUBISHI CNC E70 Series Instruction Manual IB-1501186
MITSUBISHI CNC E70 Series Setup Manual IB-1501158
MITSUBISHI CNC E70 Series Programming Manual (Lathe System) IB-1501193

Precautions for Safety

Always read the specifications issued by the machine tool builder, this manual, related manuals and attached documents before operation or programming to ensure correct use. Understand the NAVI LATHE, safety items and cautions before using the system.
This manual ranks the safety precautions into "DANGER", "WARNING" and "CAUTION".

⚠ **DANGER**	When the user may be subject to imminent fatalities or major injuries if handling is mistaken.
⚠ **WARNING**	When the user may be subject to fatalities or major injuries if handling is mistaken.
⚠ **CAUTION**	When the user may be subject to bodily injury or when property damage may occur if handling is mistaken.

Note that even items ranked as "⚠ **CAUTION**", may lead to serious consequences depending on the situation. In any case, important information that must always be observed is described.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

1. Items related to operation

⚠ If the operation start position is set in a block which is in the middle of the program and the program is started, the program before the set block is not executed. Please confirm that G and F modal and coordinate values are appropriate. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, there is a danger of interference with the machine or of machine operation at an unexpected speed, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ CAUTION

1. Items related to product and manual

⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.

⚠ Items not described in this manual must be interpreted as "not possible".

⚠ This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine tool builder before starting use.

⚠ Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.

⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

2. Items related to installation and assembly

Ground the signal cables to ensure stable system operation. Also ground the NC unit main frame, power distribution panel and machine to one point, so they all have the same potential.

3. Items related to preparation before use

Always set the stored stroke limit. Failure to set this could result in collision with the machine end.

Always turn the power OFF before connecting/disconnecting the I/O device cable. Failure to do so could damage the I/O device and NC unit.

4. Items related to screen operation

⚠ NAVI LATHE uses the following variables in order to operate the NC program.

NC program mode	Variables used by NAVI LATHE
User macro mode	#100 to #199
MTB macro mode	#450 to #499

When NC program mode is user macro mode, do not use common variables (#100 to #199). If those variables are written over, malfunction will be resulted. If mistakenly written them over, turn the NC power OFF after securing your safety. When starting NAVI LATHE by turning the NC power ON again, the system recovers the data. NC program mode is specified on the Preferences screen.

⚠ When either "TOOL REG No." or "CYCLE" is input in each machining process screen, the cutting speed and feedrate are automatically determined using the data in the tool file screen and the cutting condition file screen. Note that the cutting speed and feedrate of each process determined once will not be changed by changing the data in the tool file screen and the cutting condition file screen.

⚠ When starting NAVI LATHE by mistake while NAVI LATHE is not used, perform the operation after setting the variable value again and confirming the safety.

(Continued on next page)

⚠ CAUTION

(Continued from previous page)

5. Items related to operation

- ⚠ Stay out of the moveable range of the machine during automatic operation. During rotation, keep hands, feet and face away from the spindle.
- ⚠ Carry out dry operation before actually machining, and confirm the machining program, tool offset and workpiece coordinate system offset.
- ⚠ If the operation start position is set from a block in the program and the program is started, the program before the set block is not executed. If there are coordinate system shift commands or M, S, T, and B commands before the block set as the starting position, carry out the required commands using the MDI, etc. There is a danger of interference with the machine if the operation is started from the set starting position block without carrying out these operations.
- ⊘ Program so the mirror image function is turned ON/OFF at the mirror image center. The mirror image center will deviate if the function is turned ON/OFF at a position other than the mirror image center.

6. Items related to faults and abnormalities

- ❗ If the battery low warning is issued, save the machining programs, tool data and parameters in an input/output device, and then replace the battery. When the battery alarm is issued, the machining programs, tool data and parameters may be destroyed. Reload the data after replacing the battery.
- ❗ If the axis overruns or emits an abnormal noise, immediately press the emergency stop button and stop the axis movement.

(Continued on next page)

⚠ CAUTION

(Continued from previous page)

7. Items related to maintenance

- ⚠ Incorrect connections may damage the devices, so connect the cables to the specified connectors.
- ⚠ Do not apply voltages other than those indicated according to specification on the connector. Doing so may lead to destruction or damage.
- ⊘ Do not connect or disconnect the connection cables between each unit while the power is ON.
- ⊘ Do not connect or disconnect the PCBs while the power is ON.
- ⊘ Do not connect the cable by pulling on the cable wire.
- ⚠ Do not short circuit, charge, overheat, incinerate or disassemble the battery.
- ⚠ Dispose the spent battery according to local laws.
- ⚠ Dispose the spent cooling fan according to local laws.
- ⚠ Do not replace the control unit while the power is ON.
- ⚠ Do not replace the operation panel I/O unit while the power is ON.
- ⚠ Do not replace the control section power supply PCB while the power is ON.
- ⚠ Do not replace the expansion PCB while the power is ON.
- ⚠ Do not replace the memory cassette while the power is ON.
- ⚠ Do not replace the cooling fan while the power is ON.
- ⚠ Do not replace the battery while the power is ON.
- ⚠ Be careful that metal cutting chips, etc., do not come into contact with the connector contacts of the memory cassette.
- ⚠ Do not replace the high-speed program server unit while the power is ON.

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSOFT, GOT, CC-Link, CC-Link/LT and CC-Link IE are either trademarks or registered trademarks of Mitsubishi Electric Corporation in Japan and/or other countries.

Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.
Microsoft® and Windows® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
CompactFlash and CF are either trademarks or registered trademarks of SanDisk Corporation in the United States and/or other countries.
UNIX is a registered trademark of The Open Group in the United States and/or other countries.
Intel® and Pentium® are either trademarks or registered trademarks of Intel Corporation in the United States and/or other countries.
Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

Contents

1. OUTLINE

1.1 System Outline

This manual is an instruction manual for NAVI LATHE.
The part program for the turning center is created with the NAVI LATHE.

(1) The following machining processes can be edited.

Turning Processes

- Turning (Outer dia., inner dia., front face)
- Copy cutting (Outer dia., inner dia., front face)
- Threading (Outer dia., inner dia., front face)
- Grooving (Outer dia., inner dia., front face)
- Trapezoidal grooving (Outer dia., inner dia., front face)
- Hole drilling (Drilling, deep-hole drilling, step, tapping)
- EIA
- Cutting off

Milling Processes

- Milling hole drilling (Drilling, deep-hole drilling, boring, tapping)
  [Hole pattern]
  - Random (front face/outer surface/side surface)
  - Line (front face/outer surface/side surface)
  - Arc (front face/side surface)
  - Circle (front face/side surface)
  - Square (front face/side surface)
  - Grid (front face/side surface)
- Keyway cutting (Front face, outer surface, side surface)
- Contour cutting (Front face, outer surface, side surface)

Assist process

- Transfer

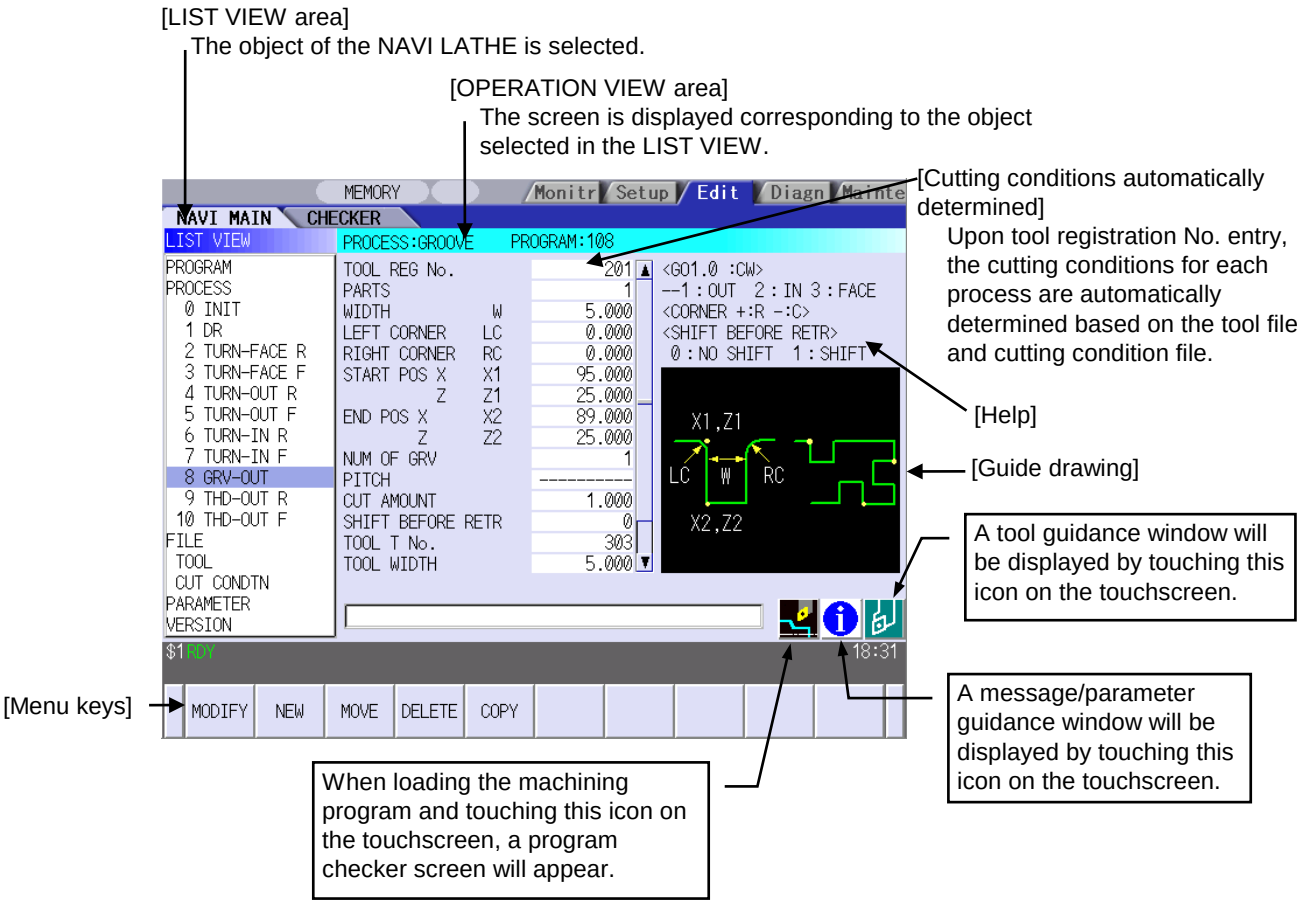
Balance cut

- Turning balance cut
- Copying balance cut
- Two-part system simultaneous thread cutting

(Note) Milling interporation specifications are required to edit the milling processes.

(2) The tool file (for the turning/milling machining) and the cutting condition file (for the turning/milling machining) are provided and the cutting conditions for each process are determined automatically.

(3) The operation screen consists of the LIST VIEW area and the OPERATION VIEW area. In the LIST VIEW area, the whole part program can be always viewed. In the OPERATION VIEW area, there are the guide drawings related to the input items, and the data can be easily input by using these guide drawings.



(4) Program Checker enables the machining shape of a part program to be graphically traced. With this function, errors in input data can be detected at an earlier stage.

(5) Guidance function provides an operator with error recovery information.

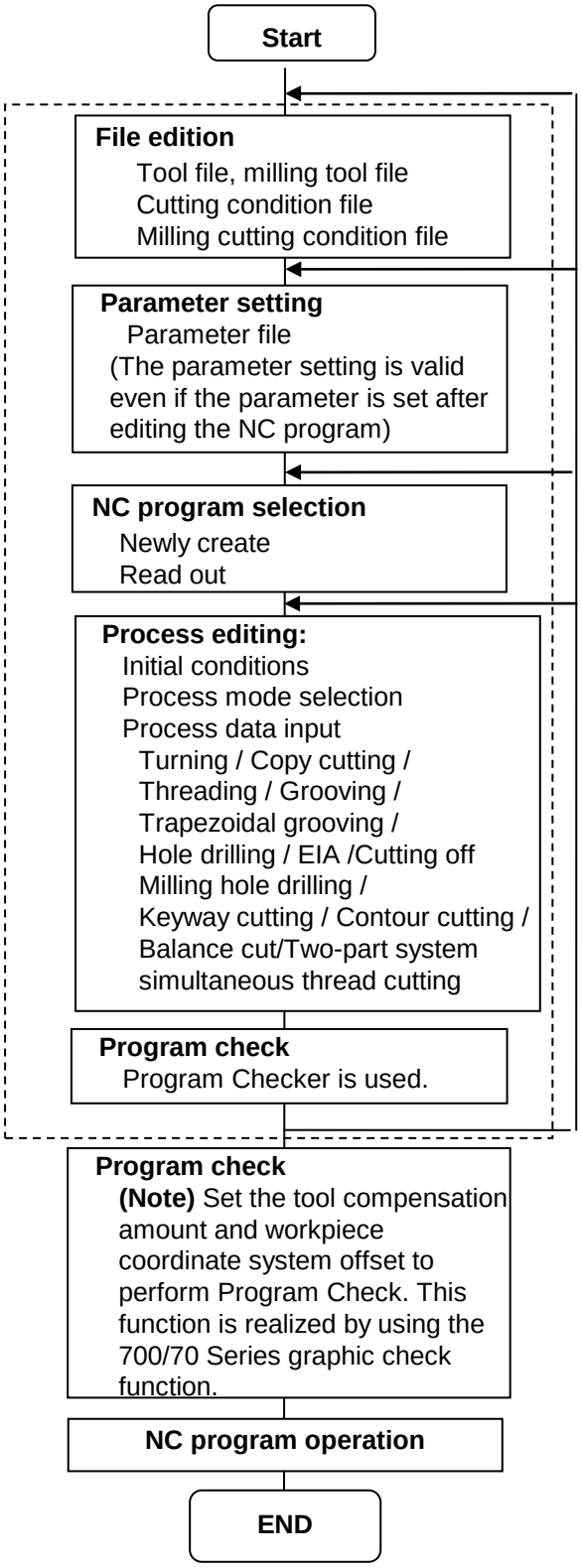
(6) Part program is a macro-program-based NC program. Commands can be added between processes from the edit screen of the standard MITSUBISHI CNC 700/70 Series.

(7) The macro program mentioned above can be customized by the machine tool builder.

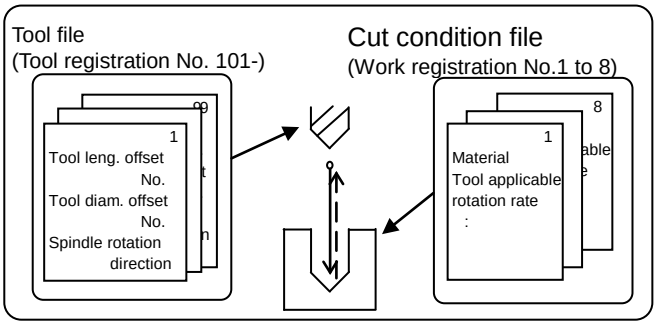
1.2 Input Procedures

The input procedure for the NAVI LATHE is shown below.

The part [ ] is operated on the NAVI LATHE's screen.

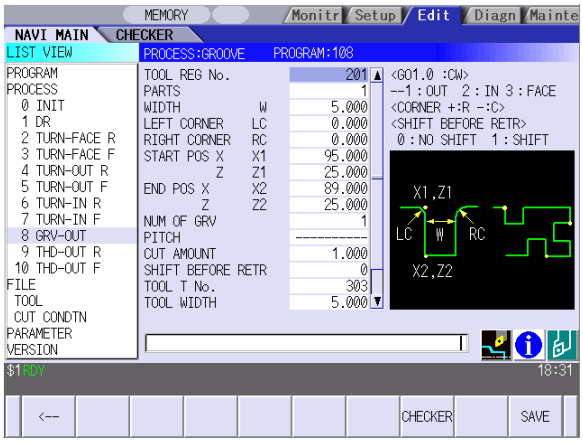


Supplements



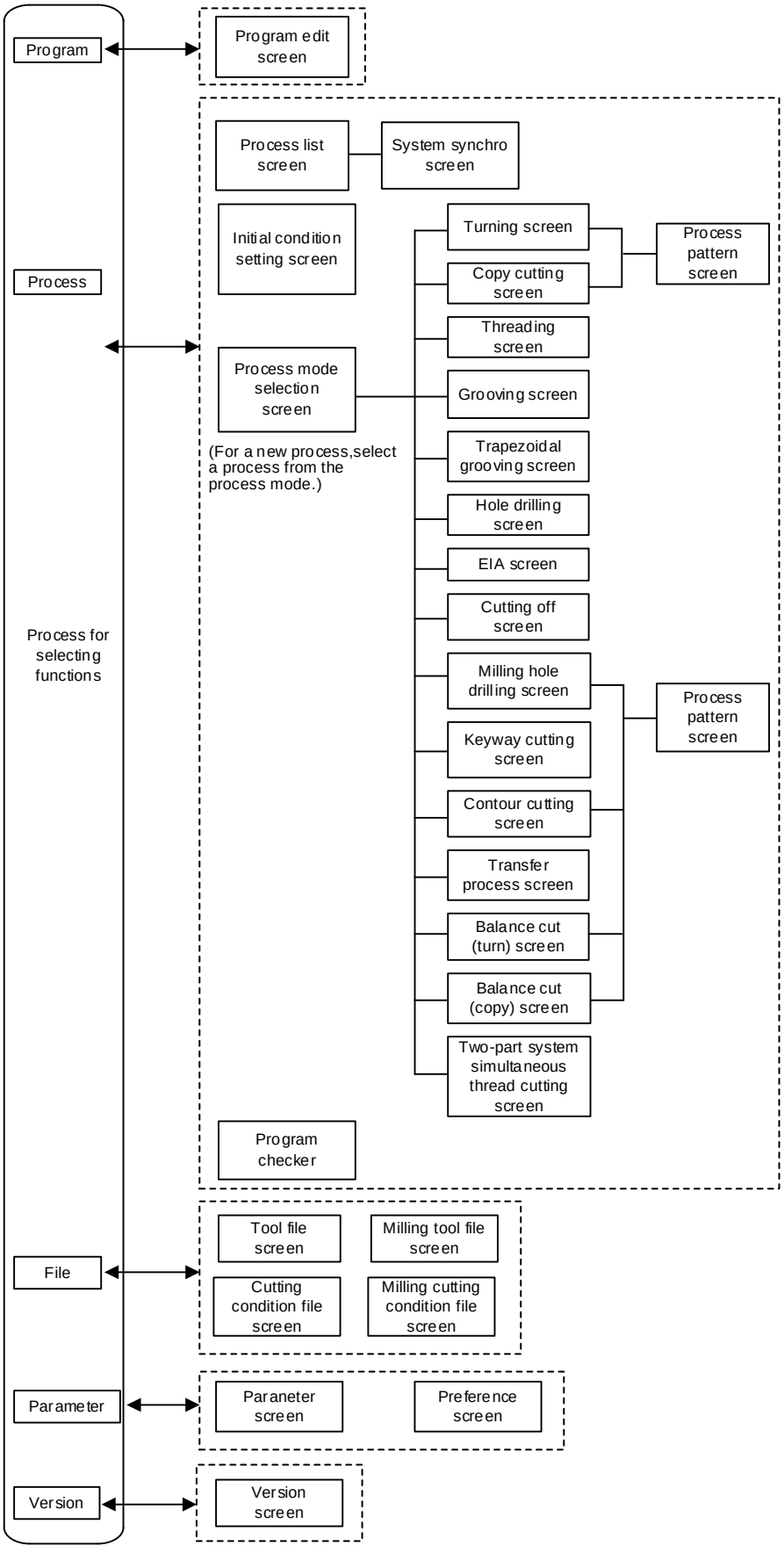
Parameter setting
- M0 output
- Maximum number of spindle rotations
- Clearance
- Tool return position
- Common parameters for threading process
- Common parameters for grooving process
- Common parameters for hole drilling process

Process editing



1.3 Screen Configuration

The screen configuration for the NAVI LATHE is shown below.



Screen name	Details
Program edit screen	NC program is newly created and read out, etc.
Process list screen	Tool information and cutting conditions for each process of a NC program are listed.
Process mode selection screen	The process mode (turning process, etc.) is selected.
System synchro screen	The order of the processes of the NC programs created for each part system is edited.
Initial conditions setting screen	The initial conditions for a NC program are set.
Turning screen	Various parameters for turning process are input.
Turning pattern screen	The machining patterns for turning process are input.
Copy cutting screen	Various parameters for copy cutting process are input.
Copy cutting pattern screen	Machining patterns for copy cutting process are input.
Threading screen	Various parameters for threading process are input.
Grooving screen	Various parameters for grooving process are input.
Trapezoidal grooving screen	Various parameters for trapezoidal grooving process are input.
Hole drilling screen	Various parameters for hole drilling process are input.
EIA screen	The EIA process is input.
Cutting off screen	Various parameters for cutting-off process are input.
Milling hole drilling screen	Various parameters for milling hole drilling process are input.
Milling hole drilling pattern screen	The machining patterns for milling hole drilling process are input.
Keyway cutting screen	Various parameters for keyway cutting process are input.
Contour cutting screen	Various parameters for contour cutting process are input.
Contour cutting pattern screen	The machining patterns for contour cutting process are input.
Transfer screen	Various parameters for transfer process are input.
Balance cut (turn) screen	Various parameters for balance cut (turn) process are input.
Balance cut (turn) machining pattern screen	Various parameters for balance cut (turn) pattern are input.
Balance cut (copy) screen	Various parameters for balance cut (copy) process are input.
Balance cut (copy) pattern screen	Various parameters for balance cut (copy) pattern are input.
Two-part systems simultaneous thread cutting (identical screw) screen	Various parameters for two-part system simultaneous thread cutting (identical screw) process are input.
Tool file screen	The tool data by each tool is registered.
Milling tool file screen	The tool data for milling machining is registered.
Cutting condition file screen	The cutting conditions (cutting speed, feedrate) by each process are input, corresponding to tip material. Also, the cutting conditions (speed rate) by each process are input, corresponding to workpiece material.
Milling cutting condition file screen	The cutting conditions (the cutting speed and the feedrate) by each process for the tip materials of the milling machining and the cutting condition (speed ratio) for the workpiece materials are registered.
Parameter screen	The parameters for a NC program are set.
Preference screen	The system is set up.
Version screen	The version data of the NAVI LATHE is displayed.
Program checker	The machining shape of a NC program is graphically displayed.

1.4 Starting NAVI LATHE

Select EDIT function, then the lathe menu to display NAVI LATHE screen.
Program edit screen is displayed once when the power is turned ON. Then, whatever the screen previously selected with NAVI LATHE is displayed thereafter.

1.5 Setting up NAVI LATHE

Part program output from NAVI LATHE is a macro-program-based NC program. Thus, macro programs have to be registered in the NC system in advance. Also, the destinations where NC programs or NAVI LATHE's reference files are saved, as well as the unit for data input, have to be specified prior to NAVI LATHE operations.

NAVI LATHE setup items

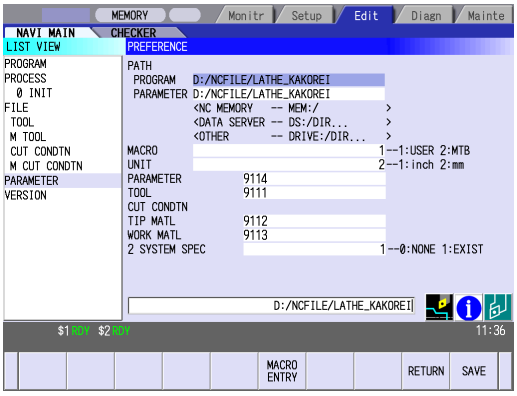
Item	Details	Standard value
PATH PROGRAM	Path to the folder in which NC program is saved.	MEM:/
PATH PARAMETER	Path to the folder in which tool file, cutting condition file and parameter file are saved.	In M700/M700VM: D:/NCFILE/NAVI Other than those above: MEM:/
MACRO	Macro program mode 1: User macro mode 2: MTB macro mode	1 (User Macro)
UNIT	Unit for data input 1: inch 2: mm	2 (mm)
Parameter	Name of parameter file	9114
Tool file	Name of tool file	9111
Cutting condition file tip material	Name of cutting condition file (tip material)	9112
Cutting condition file workpiece material	Name of cutting condition file (workpiece material)	9113
2-part system specification	Whether 2-part system specification is provided or not. (0: NONE, 1: EXIST)	0 (NONE)

NAVI LATHE setup procedures

(1) Open PARAMETER screen.

(2) Set "999 MAINTE" to 1. ➡ [PREFERENCE] menu is displayed.

(3) Press [PREFERENCE] menu. ➡ PREFERENCE screen is displayed.



(4) Select the macro type.
(1:User macro 2:MTB macro)

(5) Press [MACRO ENTRY] menu. ➡ "OK?(Y/N)" message is displayed.

(6) Press [Y] key. ➡ Macro program is registered in NC system.

(7) Enter the program path.

(8) Enter the parameter path.

(9) Select the unit.
(1:inch, 2:mm) ➡ When the unit is changed, turn the power OFF and ON again.

(10) Enter the name of parameter file

(11) Enter the name of tool file

(12) Enter the name of cutting condition file
(tip material and workpiece material)

(13) Enter 0 or 1 according to whether the 2-part system specification is provided or not. ➡ When the setting value for 2-part system specification is changed, turn the power OFF and ON again.

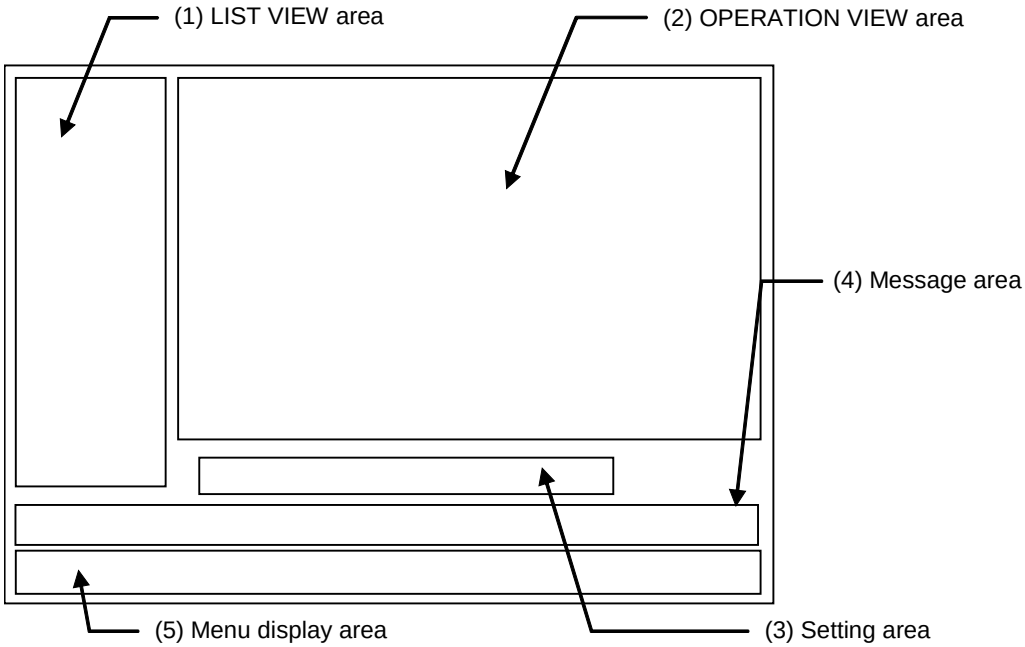
(Addendum)

- Always carry out a macro program registration when setting up NAVI LATHE or switching "MACRO" types.
- Change "PROGRAM PATH" and "PARAMETER PATH" when necessary.
- When "UNIT" is changed, turn the power OFF and ON again.
- If the tool file, cutting condition file and parameter file do not exist in "PARAMETER PATH" folder when the power is turned ON, the system creates them.
- When the value for 2-part system specification is changed, turn the power OFF and ON again.

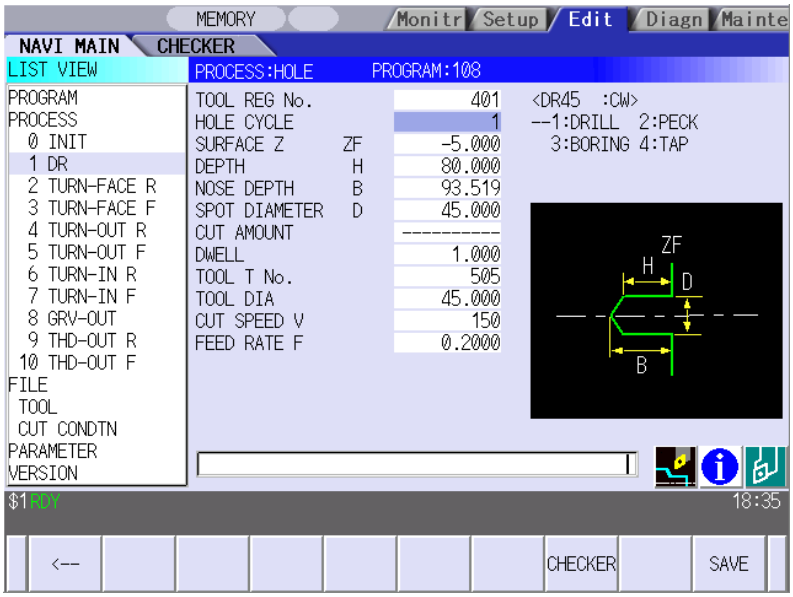
2. FUNCTIONS OF DISPLAY AREA

The screen of the NAVI LATHE is divided into the following five areas.

(1) LIST VIEW area (Refer to "2.1 LIST VIEW Area")
(2) OPERATION VIEW area (Refer to "2.2 OPERATION VIEW Area")
(3) Setting area (Refer to "2.3 Setting Area")
(4) Message area (Refer to "2.4 Message Area")
(5) Menu display area (Refer to "2.5 Menu Display Area")

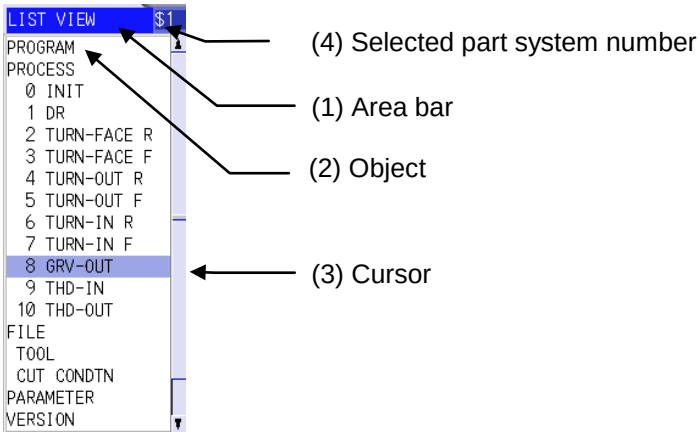


<Screen example>



2.1 LIST VIEW Area

The object of the NAVI LATHE is selected in this area.



(1) Area bar

When the LIST VIEW area is active, the area bar is highlighted.

(2) Objects

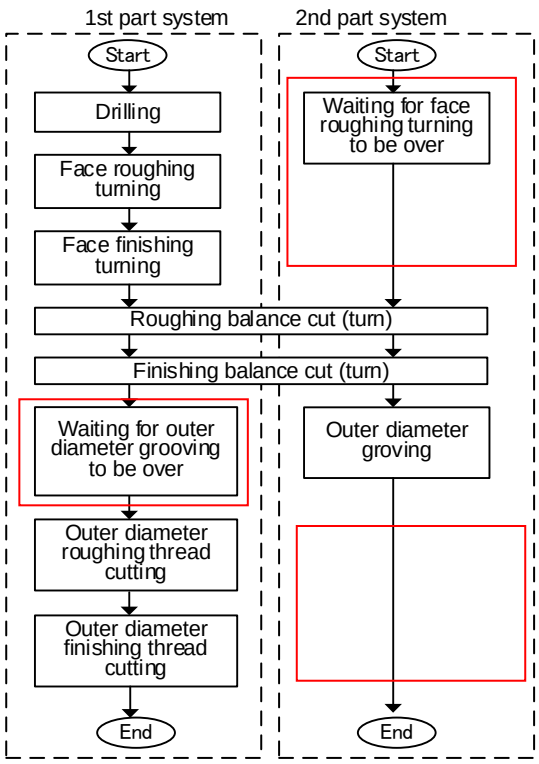
The list of objects that can be selected are displayed. The object is composed of the main object and the sub object, which is a specification of the main object. The details of each object are as follows.

Main object	Sub object	Details
PROGRAM	-	Newly creates, reads out, and deletes, etc. the NC program.
PROCESS	0 INIT 1 DR :	Displays the currently edited process list. The settings of the selected process can be displayed and changed. When the 2-part system specification is set to "1: EXIST", the process list of the currently edited part system is displayed. If you select a waiting part system during a process that is carried out just by the other part system, this view shows the process being carried out by the other part system (shows the process number, but no process name). (*1)
FILE	TOOL	Displays and changes the tool file.
	M TOOL	Displays and changes the tool file for the milling machining. **(Note)** This is valid when the milling interporation specifications are provided.
	CUT CONDTN	Displays and changes the cutting conditions for each process per tip material or workpiece material.
	M CUT CONDTN	Displays and changes the cutting conditions for each process per tip material or workpiece material for the milling machining. **(Note)** This file is valid when the milling interporation specifications are provided.
PARAMETER	-	Displays the tool option and the miscellaneous parameter to be used in each process. Those can be changed.
VERSION	-	Displays the version data of the NAVI LATHE.

(Note) If too many processes are registered and all the objects cannot be displayed, a scroll bar will be displayed. In this case, change display of the list by pressing cursor key or page key down, or by clicking on the scroll bar.

*1 For the following machining case, the process list of the LIST VIEW is displayed as below.

<Flow for machining>

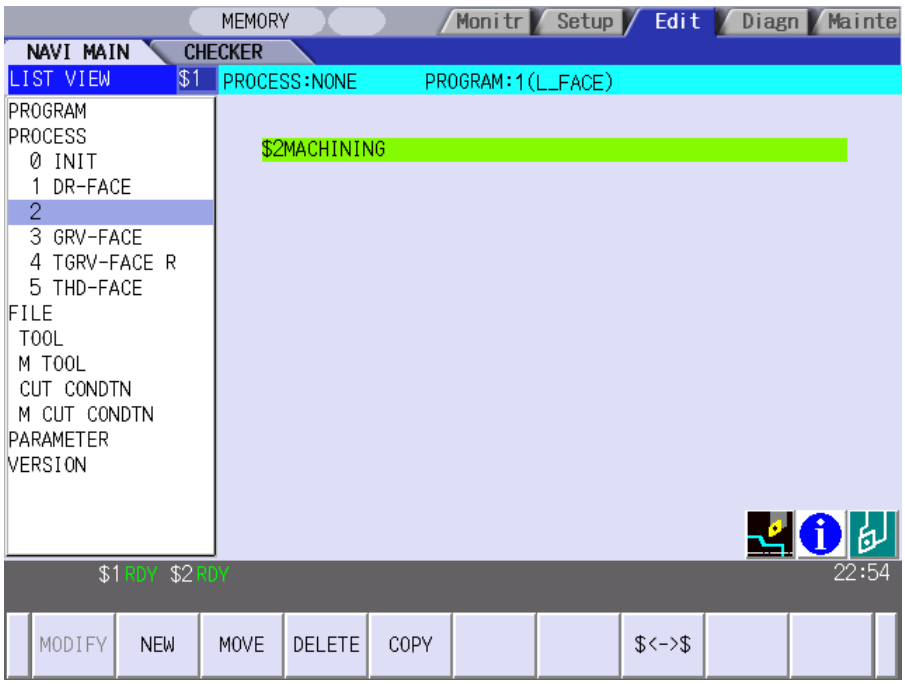


Process List	
Editing 1st part system	Editing 2nd part system
Process	Process
0 INIT	0 INIT
1 DR-FACE	1
2 TURN-FACE R	2
3 TURN-FACE F	3
4 !TURN-OUT R	4 !TURN-OUT R
5 !TURN-OUT F	5 !TURN-OUT F
6	6 GRV-OUT
7 THD-OUT R	7
8 THD-OUT F	8

The processes that are being run by the other system

<Addendum>

If a selected process shows no process name in the LIST VIEW (a process being run by the other part system), "$nMACHINING" (n: part system number) is displayed In the OPERATION VIEW area.



(3) Cursors

When the LIST VIEW area is active and the object is selected with the cursor, the display in the OPERATION VIEW area and the menu display area will be changed.

<Cursor movement>
The cursor is moved using the cursor keys or a pointing device.

Key type	Operation of cursor
[↑] Cursor key	Moves the cursor one field up regardless of the main object or sub object. Note that if the ↑ cursor is pressed when the cursor is at the top, the cursor does not move.
[↓] Cursor key	Moves the cursor one field down regardless of the main object or sub object. Note that if the ↓ cursor is pressed when the cursor is at the bottom, the cursor does not move.
[←] Cursor key	When the cursor is at the sub object, moves the cursor to the previous main object.
[→] Cursor key	When the cursor is at the sub object, moves the cursor to the next main object.
[Page Up] key	Moves the displayed data toward the top.
[Page Down] key	Moves the displayed data toward the bottom.
Pointing device	Cursor jumps to the spot where clicked with a pointing device. If an object not selectable is clicked, cursor does not jump.

(4) The part system number being selected

When the 2-part system specification is "1: EXIST" and the multi-part system program management is ON, the part system number being selected at the NAVI LATHE is displayed.
If the program is not opened, the selected part number cannot be displayed.

- When selecting the 1st part system: 

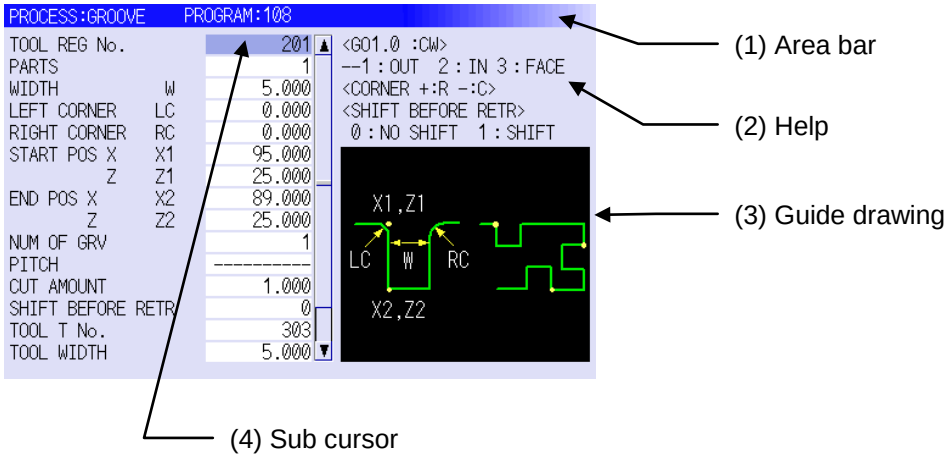
- When selecting the 2nd part system: 

This display is changed by the menu [$<->$].

* The menu [$<->$] is displayed in the condition that the LIST VIEW is active and the cursor is on the machining process or the process name.

2.2 OPERATION VIEW Area

The various data are displayed in this area. Selecting the object in the LIST VIEW area changes the contents displayed in the OPERATION VIEW area.



(1) Area bar

When the OPERATION VIEW area is active, the area bar is highlighted.
The name of the currently edited program is displayed.

(2) Help

Quick reference on the setting items is displayed.

(3) Guide drawing

When the process is edited, a guide drawing according to the currently edited machining mode is displayed.

(4) Sub cursor

Key type	Operation of cursor
[↑] Cursor key	Moves the cursor one field up. Note that if the ↑ cursor is pressed when the cursor is at the top, the cursor does not move.
[↓] Cursor key	Moves the cursor one field down. Note that if the ↓ cursor is pressed when the cursor is at the bottom, the cursor does not move.
[Page Up] key	Moves the displayed data toward the top.
[Page Down] key	Moves the displayed data toward the bottom.

2.3 Setting Area

The value to be set to data is input.

2.4 Message Area

An error message or operation message, etc. during operation is displayed.

2.5 Menu Display Area

The screen operation is selected, and the screen is changed.
The different menus are displayed in each screen. (Refer to the chapter 4.)

3. BASIC OPERATIONS

3.1 Changing Active View

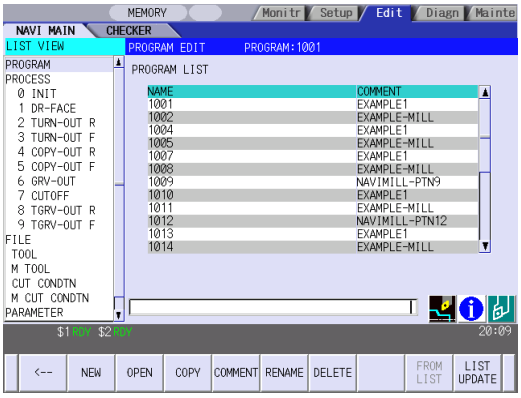
To operate NAVI LATHE, activate either LIST VIEW area or OPERATION VIEW area. When the VIEW is active, the area bar is highlighted and data can be input. Use menu keys [←] and [→] or a pointing device to switch either one of the VIEWs to be activated.

3.2 Changing Screen

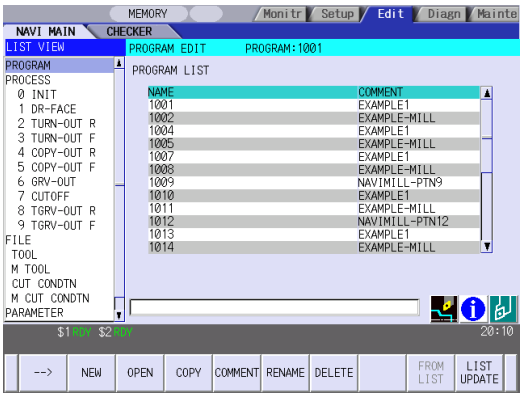
When the object is selected in the LIST VIEW area, the screen (contents in the OPERATION VIEW area) changes. (Refer to the section 2.1 LIST VIEW Area.)
Note that the screen cannot be changed while the OPERATION VIEW area is active.
In such a case, press the [←] menu key or click "LIST VIEW" with a pointing device to turn the LIST VIEW area active.

Operation example

(1) Open the program edit screen. ➡ The OPERATION VIEW area is active.

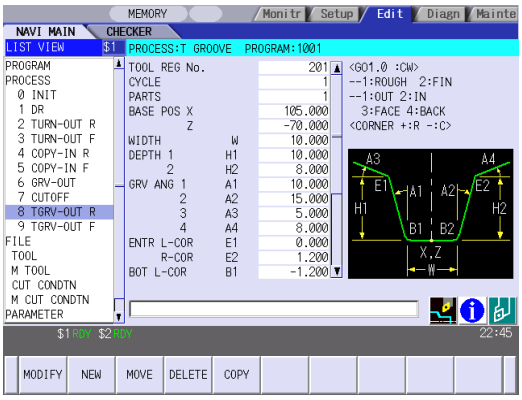


(2) Press the [←] menu key. ➡ The LIST VIEW area will turn active.



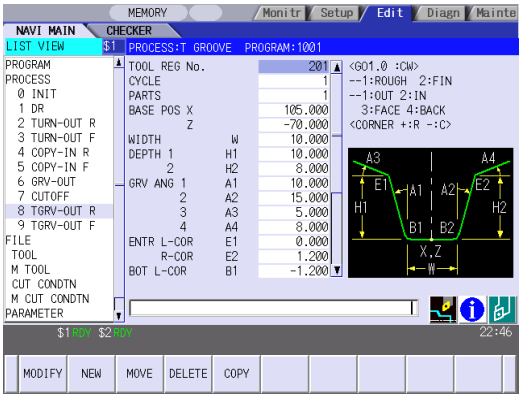
(3) Select the object with the cursor key.

➡ The OPERATION VIEW area will change into the screen corresponding to the selected object.



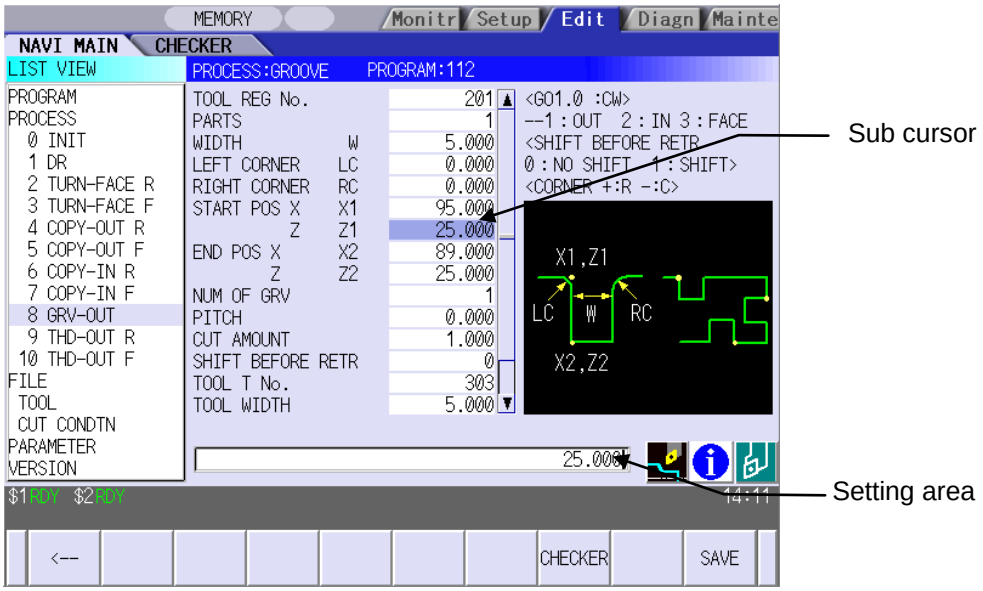
(4) Press the [MODIFY] menu key.

➡ The OPERATION VIEW area will turn active.



3.3 Setting Data

After moving the sub cursor, input the data into the setting area and then press the [INPUT] key, and the data will be set. (The sub cursor is displayed only when the OPERATION VIEW area is active.)



Operation method

An example for setting the data on the hole drilling screen is shown below.

(1) Screen selection

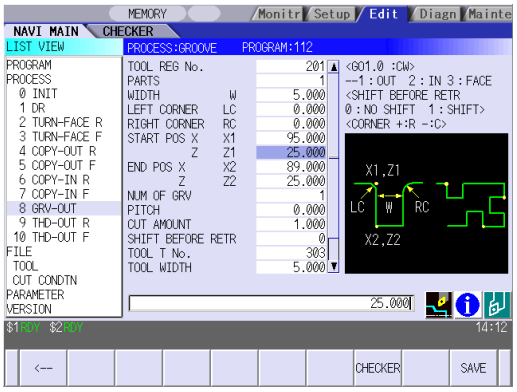
Select the object to be changed from the LIST VIEW and press [MODIFY] menu key.

➡ The OPERATION VIEW area will turn active.
(Refer to the section 3.2 "Changing screen".)

(2) Setting item selection

Move the sub cursor with cursor keys.

➡ This is an example of the sub cursor movement on the hole drilling screen.



(3) Data key input

Set data with the numeral keys or alphabet keys, etc.
[1] [8] [.] [0] [0] [0]

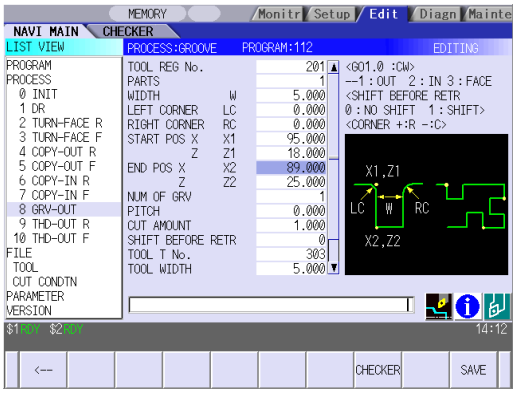
➡ The data is set in the data setting area.

18. 000

(4) [INPUT] key input

Press the [INPUT] key.

➡ Data for the selected setting item is set.
The sub cursor moves to the next position.



(Note 1) The contents in the data setting area are only displayed when [INPUT] key is not pressed and will be invalidated if the screen is changed at this time. Data for the currently selected setting item will be set when [INPUT] key is pressed.

(Note 2) If illegal data is set, an error occurs when [INPUT] is pressed. Set the correct data again.

Operations in the data setting area

The key is input at the position where the cursor is displayed. If a cursor is not displayed, the key input is invalid.
When a key is input, the data appears at the cursor position, and the cursor moves one character space to the right.

■ [→] / [←] keys: Moves the cursor one character to the left or right.

(1) The cursor is at the position shown on the right. ➡ 1 2 3 7 7 7 **|** 4 5 6

(2) Press the [→] key. ➡ The cursor moves one character space to the right.
1 2 3 7 7 7 4 **|** 5 6

■ [DETETE] key: Deletes the character in front of the cursor.

(1) Move the cursor to the position where the data is to be deleted. ➡ The cursor in the data setting area moves.
1 2 3 4 **|** 5 6

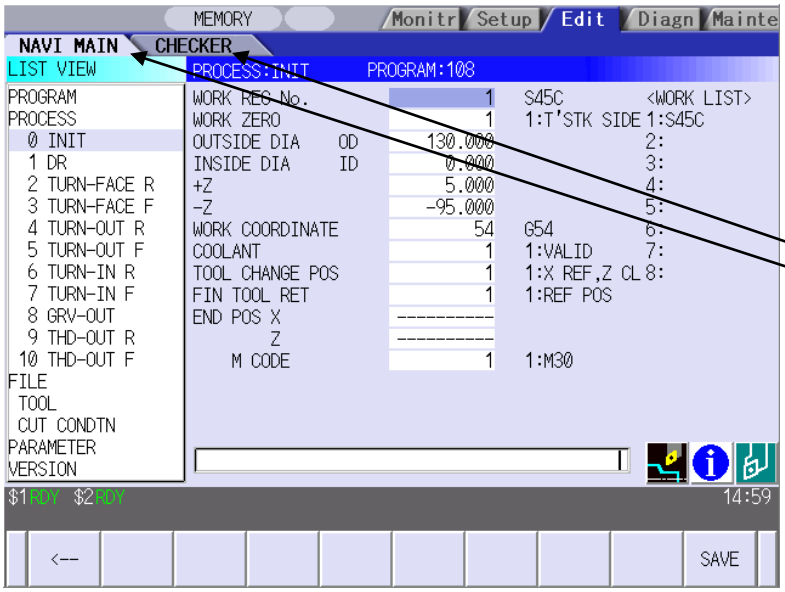
(2) Press the [DETETE] key. ➡ The character in front of the cursor is deleted.
1 2 3 **|** 5 6

3.4 Switching Windows

When a shortcut button on the keyboard is pressed, its corresponding window is displayed.

Button	Application
	Displays the tool guidance window.
	Displays the message guidance window.
	Displays the checker window.

3.5 Switching Selection Tags



When a tag button on the keyboard is pressed, the main window and checker window can be switched over.

Button	Application
	Selects the tag on the left.
	Selects the tag on the right.

(Note 1) Depending on the keyboard specifications, tab button may not be available.

3.6 Inputting Operations

In addition to the method of directly inputting numeric data for specific data settings, a method to input the operation results using four rules operators and function symbols can be used.

Input method

Numeric values, function symbols, operators and parentheses ( ) are combined and set in the data setting area.
The operation results appear when the [INPUT] key is pressed. Data for the currently selected setting item will be set when [INPUT] key is pressed again.
The contents in the data setting area are erased.

Examples of operator settings, and results

Operation	Setting example	Operation results
Addition	=100+50	150.000
Subtraction	=100–50	50.000
Multiplication	=12.3∗4	49.200
Division	=100/3	33.333
Function	=1.2∗ (2.5+SQRT(4))	5.400

Function symbols, setting examples and results

Function	Function symbol	Setting example	Operation results
Absolute value	ABS	=ABS (50–60)	10.000
Square root	SQRT	=SQRT (3)	1.732
Sine	SIN	=SIN (30)	0.5
Cosine	COS	=COS (15)	0.966
Tangent	TAN	=TAN (45)	1
Arc tangent	ATAN	=ATAN (1.3)	52.431
Circle ratio	PAI	=PAI*10	31.415
Inch	INCH	=INCH/10	2.54

Operation examples

(1) Set as shown below, and press the [INPUT] key.
=12∗20 [INPUT]

➡ The operation results appear in the data setting area.

240 |

(2) Press the [INPUT] key again.

➡ Data for the selected setting item is set. The cursor moves to the next position.

Notes for using operators and functions

Division:	Zero division causes an error.
Square root:	If the value in the parentheses is negative, an error occurs.
Triangle function:	The unit of angle θ is degree (°).
Arc tangent:	–90 < operation results < 90.

Restrictions

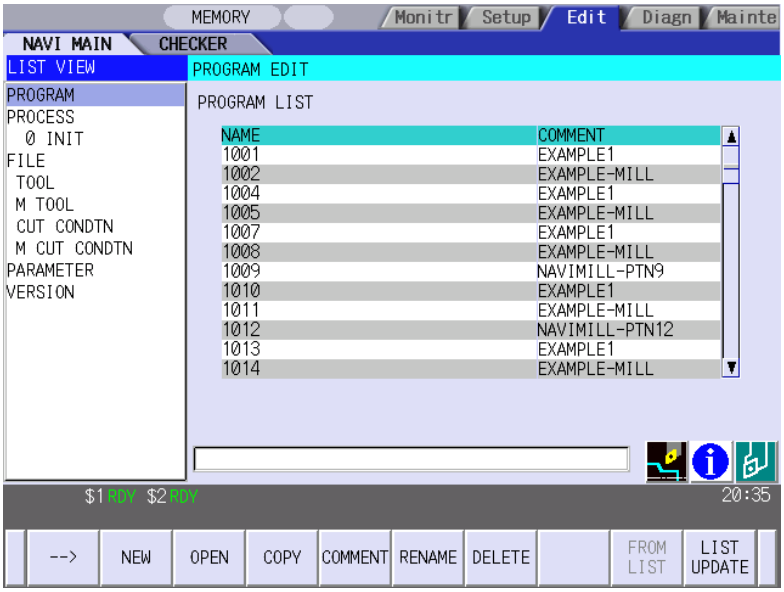
- Always use "=" for the first character.
- Do not use the following characters as the second character or last character.
  Invalid as second character: ∗, /,)
  Invalid as last character: ∗, /, (, +, -
- Make sure that the left parentheses and right parentheses are balanced.
- The 360° limit does not apply on the angle. SIN (500) is interpreted as SIN (140).

4. SCREEN SPECIFICATIONS

4.1 Starting NAVI LATHE

When NAVI LATHE is started, the program edit screen will be displayed.

Screen layout



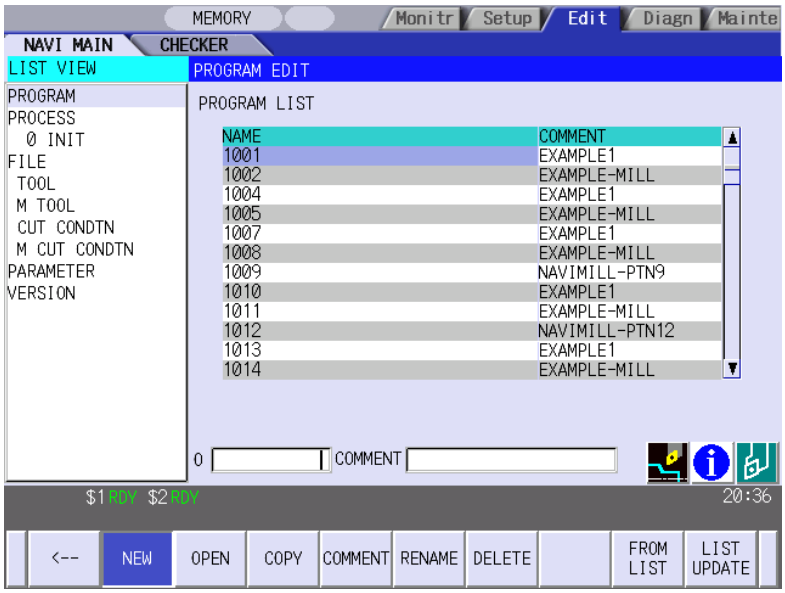
At the initial start up of NAVI LATHE, the cursor is displayed at the position of [PROGRAM] in the LIST VIEW area, and the program edit screen is displayed in the OPERATION VIEW area.
The LIST VIEW area is active.
The process program is not selected.

4.2 Screen Related to the Program

4.2.1 Program Edit Screen

The NC program is newly created and read out, etc. on this screen. When [PROGRAM] is selected in the LIST VIEW area, this screen is displayed.

Screen layout



The process list of the currently selected program is displayed in the LIST VIEW area.

<Turning process displays>

Process name		Display character	Remarks
Turning	OD OPEN	TURN-OUT ?	A symbol that indicates the machining type (rough/finishing) is put at ?. • Rough machining: R • Finishing machining: F
	OD CLOSE	TURN-OUT ?	
	ID OPEN	TURN-IN ?	
	ID CLOSE	TURN-IN ?	
	FACE OPEN	TURN-FACE ?	
	FACE CLOSE	TURN-FACE ?	
	BACK OPEN	TURN-BACK ?	
	BACK CLOSE	TURN-BACK ?	
Copy cutting	Outer diameter	COPY OUT ?	A symbol that indicates the machining type (rough/finishing) is put at ?. • Rough machining: R • Finishing machining: F
	Inner diameter	COPY-IN ?	
Thread	Outer diameter	THD-OUT ?	A symbol that indicates the machining type (rough/finishing) is put at ?. • Rough machining: R • Finishing machining: F • Rough + finishing: No symbol
	Inner diameter	THD-IN ?	
	Face	THD-FACE ?	
	Back	THD-BACK ?	
Groove	Outer diameter	GRV-OUT ?	
	Inner diameter	GRV-IN ?	
	Face	GRV-FACE ?	
	Back	GRV-BACK ?	
Trapezoidal grooving	Outer diameter	TGRV-OUT ?	A symbol that indicates the machining type (rough/finishing) is put at ?. • Rough machining: R • Finishing machining: F
	Inner diameter	TGRV-IN ?	
	Face	TGRV-FACE ?	
	Back	TGRV-BACK ?	
Hole drilling	Drill	DR-****	Symbols that indicate the machining area (front face/back surface) are put at ****. (When the process is created with the parameter "#1001 SUB SPINDLE SPED" set to "1: EXIST".) - FACE - BACK
	Deep hole	PECK-****	
	Bore	BORE-****	
	Tapping	TAP-****	
EIA		EIA	
Cutting off		CUTOFF	

<Milling process displays>

Process name		Display character	Remarks
Milling hole drilling	Drilling	M DR-****	Symbols that indicate the machining area (front face/outer surface/side surface) are put at ****. ・FACE ・OUT ・SIDE ・BACK
	Deep hole drilling	M PECK-****	
	Step	M BORE-****	
	Tapping	M TAP-****	
Keyway cutting	Front face	K WAY-FACE ?	A symbol that indicates machining type (rough/finishing) is put at ?. ・Rough machining: R ・Finishing machining: F
	Outer surface	K WAY-OUT ?	
	Side surface	K WAY-SIDE ?	
	Back surface	K WAY-BACK ?	
Contour cutting	Front face	CONT-FACE ?	
	Outer surface	CONT-OUT ?	
	Side surface	CONT-SIDE ?	
	Back surface	CONT-BACK?	

<Assist process displays>

Process name		Display character	Remarks
Transfer	MAIN -> SUB	TRS-SUB	
	SUB -> MAIN	TRS-MAIN	
	SYNC	TRS-SYNC	

<Balance cut process displays>

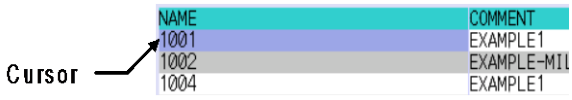
Process name		Display character	Remarks
Balance cut (turn)	Outer diameter	! TURN-OUT ?	A symbol that indicates the machining type (rough/finishing) is put at ?. - Rough machining: R - Finishing machining: F
	Inner diameter	! TURN-IN ?	
	Face	! TURN-FACE ?	
	Back	! TURN-BACK ?	
Balance cut (copy)	Outer diameter	! COPY-OUT ?	
	Inner diameter	! COPY-IN ?	
Two-part system simultaneous thread cutting (identical screw)	Outer diameter	! THD1-OUT	
	Inner diameter	! THD1-IN	

Screen display item

No.	Display item	Details	Setting range
1	PROGRAM LIST	Displays the names and comments of the NC program that can be currently read out. The program name can be displayed up to 32 characters. When the 2-part system specification is "1: EXIST", the program names and comments to be displayed are switched by the parameter and program path. For the details of the PROGRAM LIST, refer to " 4.2.1.1 Program Editing and Part Systems".	-

(Note 1) The program list displays the files stored under the directory which you designated in the preference screen. The directory is not displayed.
(Note 2) The maximum length of program name for display is 32 characters. Any exceeded part is not displayed in the list. If you move the cursor left or right in the program setting area, you can browse the exceeded part.
(Note 3) For the multi-part system, a file name can be set up to 29 characters.
(Note 4) The program list shows up to 120 files in the numerical order (ascending). Any file after the 120th file is not displayed in that order.
(Note 5) If the first character of the program name is 0, it is treated as a character string, and is sorted.
(Note 6) If the number of the program name is larger than "2147483647", it is treated as a character string, and is sorted.

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	NEW	Newly creates the NC program. (Note 1) < Display in the setting area when pressing the menu > O( ) COMMENT( )
3	OPEN	Reads out the existing NC program. (Note 1) (Note 2) < Display in the setting area when pressing the menu > O( ) When this menu is pressed, the cursor appears at the program list's name section. When the setting area is empty, select a program with the cursor and press the [INPUT] key to read the program. 
4	COPY	Copies the existing NC program to another program. (Note 1) < Display in the setting area when pressing the menu > O( ) → O( )
5	COMMENT	Edits the comment in the NC program. (Note 1) < Display in the setting area when pressing the menu > O( ) COMMENT( )
6	RENAME	Renames the existing NC program. (Note 1) < Display in the setting area when pressing the menu > O( ) → O( )

No.	Menu	Details
7	DELETE	Deletes the NC program. < Display in the setting area when pressing the menu > O( ) to O( )
9	FROM LIST	Load the list contents to the setting area by pressing the [INPUT] key. <The Display when the menu is pressed.> O( ) to O( )
10	LIST UPDATE	Updates the list display. The program names are displayed in the numerical ascending order.

(Note 1) The restrictions on the program name are as follows.
(1) Program names can be displayed up to 32 characters including the extension of the file. For the multi-part system, up to 29 characters are able to be set.
(2) The available characters are one byte numbers, one byte upper-case alphabets and one-byte symbols. However, there are some exceptions as follows.
Cannot be used: "¥", "/", ":", ",", "*", "?", """, "<", ">", "|", lower-case alphabets from a to z and a space (blank).
(3) The following programs are not treated as a program name.
- Extension: "$$$", "$$0", "$$1", "$$2", "$$3", "$$4", "$$5", "$$6", "$$7", "$$8" and "$$9".
- Program Name: Any of the following numbers from 9100 to 9199 and from 100019100 to 100019199.

(Note 2) NC program mode includes user macro mode and MTB mode. (This is specified in the preferences screen.) When user macro mode is active and an NC program created with MTB mode is opened, the NC program is converted into user macro mode. When MTB mode is active and an NC program created with user macro mode is opened, the NC program is converted into MTB mode.

(Note 3) The multiple program deletion is not available except those program names are only with numbers. When the deletion is not available, an error message "E002 DATA RANGE OVER" will appear.
Multiple deletion NG ...O(AAA) to O(1)
O(AAA) to O(BBCB)
Multiple deletion OK ...O(1) to O(50)

(Supplement)

1. The following is the operation to enable the import from the program list.
   INPUT key …[NEW] and [OPEN]
   FROM LIST menu …[NEW], [OPEN], [COMMENT], [COPY], [RENAME] and [DELETE]

2. When [FROM LIST] menu is pressed, contents are not echoed back to the setting section.

3. When [FROM LIST] menu is pressed, the cursor is displayed only in the name field of the list. The cursor is not displayed in the comment field.

4. Program list is the numerical priority (ascending order). The following is the priority order.

Priority order
1. The numerical value only program (excluding the case which "0" is put at the beginning) ascending order
2. The program name character code ascending order

(Note) "The character code order" is the method that the file names are compared one by one using the ASCII code.
If the ascending order is applied, they are listed from 1 to A because "1" is "0x31" and "A" is "0x41".

The following is an example of sorting.

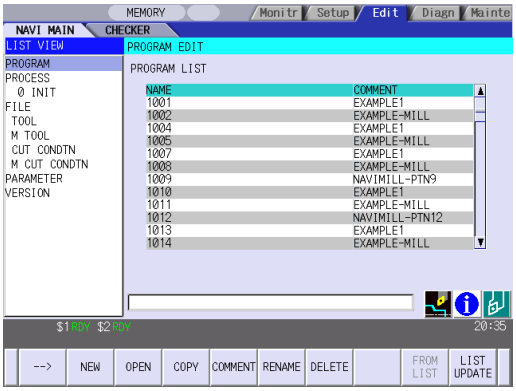
No.1 Program name, Numerical priority, Ascending order

Program name	Date/comment
1	
2	SAMPLE
3	2005-04-01
211	
1000	MAIN
1002	SUB2
01	COLOR_CHECK
1001.PRG	sub1
1003A12	
2.PRG	
A	DATAFILE
A.TXT	COLOR_CHECK
ABCD	AAA
PROTOTYPE	

Operation example (Opening the existing NC program)

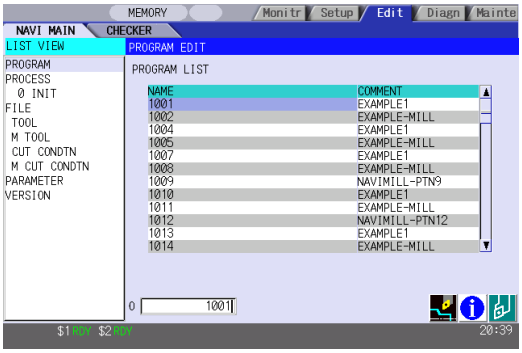
(1) Select the [PROGRAM] in the LIST VIEW area.

➡ The program edit screen will be displayed. The list of the NC program that can be read out will be displayed.



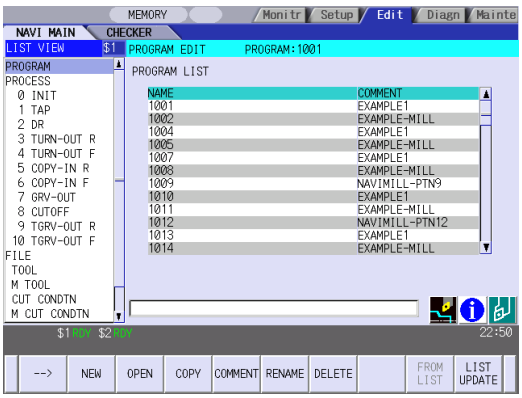
(2) Press the [OPEN] menu key, and input the NC program No. to be read out.

➡ The [OPEN] menu will be highlighted, and the setting area will be displayed.
The cursor appears on the program name field of the list.



(3) Press the [INPUT] key.

➡ The highlight of the [OPEN] menu will turn OFF, and the setting area will disappear.
The process of the NC program read out will be displayed in the LIST VIEW area.
The NC program No. read out will be displayed on the area bar of the OPERATION VIEW area.



4.2.1.1 Program Edit and Part System

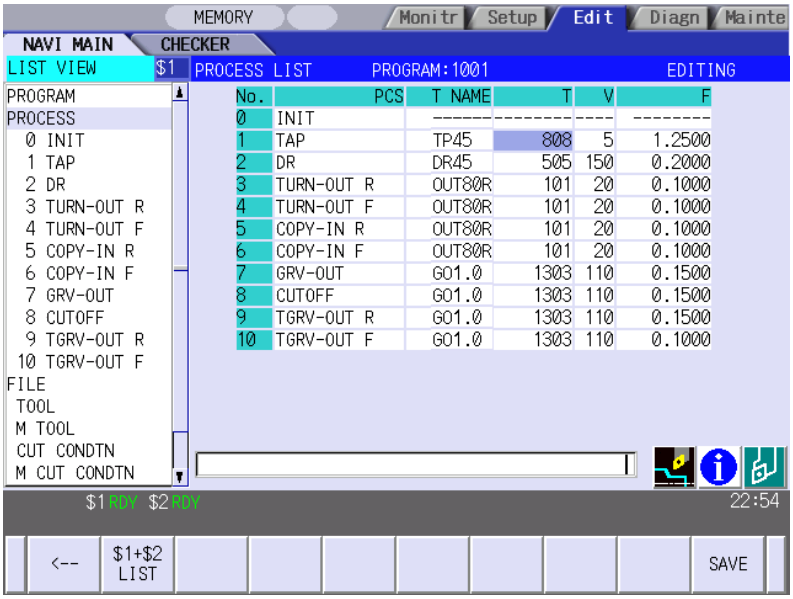
Program path	#1285 ext21 bit0: Multi-part system program management	#1285 ext21 bit2: Multi-system program generation and operation	Program list display pattern	[NEW]	[OPEN]	[COPY] [COMMENT] [RENAME] [DELETE]	Remarks
NC memory	OFF	-	Part system none specified.	$1	Type 1 [Program existed] Open the 1st part system program. [No program] Error "E01 Designated file does not exist."	Per file	
	ON	OFF	All part systems (for the number of the part system)	All part systems (for the number of the part system) 1st and 2nd part system generate the INIT process.	Type 2 [Program existed] Open the 1st part system program. [No program] Error "E01 Designated file does not exist."	All part systems at a time (For the comments, only $1 and $2 are applicable.)	
	ON	ON	$1	$1	Type 1 [Program existed] Open the 1st part system program. [No program] Error "E01 Designated file does not exist."	Per file	The 2-part system spec. cannot be set to "1: EXIST".
Excluding NC memory	-	-	All files	$1	Type 1 [Program existed] Open the 1st part system program. [No program] Error "E01 Designated file does not exist."	Per file	

4.3 Screens Related to the Process Edit Functions

4.3.1 Process List Screen

The tool information and cutting conditions for each process are displayed on this screen. When [PROCESS] is selected in the LIST VIEW area, this screen is displayed.
When the NC program is not selected, this screen is not displayed.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	PCS	The process name is displayed. **(Note)** This name is same as the name displayed in the LIST VIEW area.	-
2	T NAME	The name of tool to be used is displayed.	-
3	T	The tool No. and compensation No. are displayed. The tool No. can be changed. T-command will not be output if the tool No. is set to "0". Set the tool No. to "0" unless T-command needs to be output, such as when the same tool is used for the multiple consecutive processes.	0 to 99999999
4	V	The cutting speed is displayed. The cutting speed can be changed.	1 to 9999 m/min 1 to 9999 feet/min
5	F	The feedrate is displayed. The feedrate can be changed. When TAP or THREAD process is applied, the pitch (mm/rev) is displayed.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev

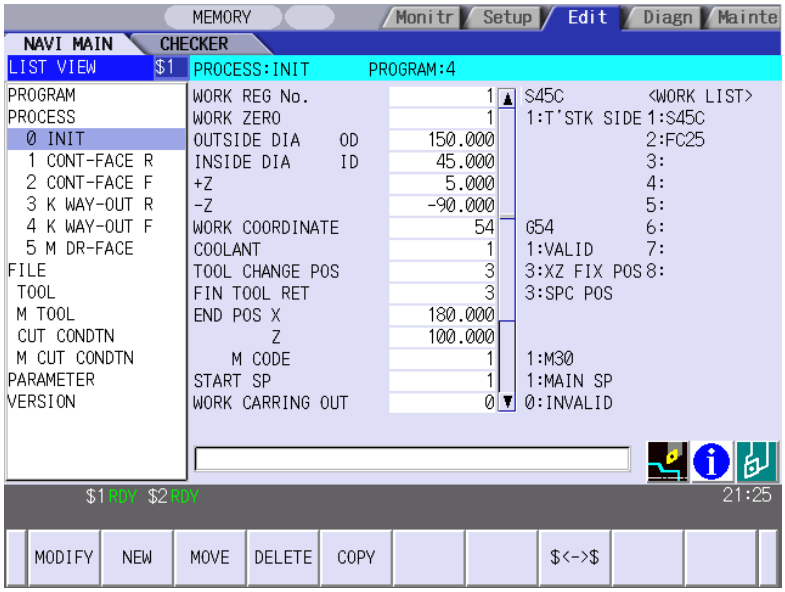
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	$1+$2 LIST	System synchro screen is displayed. When setting the timing synchronization in each process of the programs created for each part system, press this menu. This menu is displayed only when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.

4.3.2 Operating Process

When the cursor is moved to the sub-object of [PROCESS] in the LIST VIEW area, a menu for editing the process is displayed, and the process can be operated.

Screen layout



Menus

No.	Menu	Details
1	MODIFY	The OPERATION VIEW area turns active, and the process parameters of the part system being edited can be changed. When selecting a process with no name (a process being run by other part system), this menu turns gray and cannot be selected.
2	NEW	The mode selection screen is displayed, and add the selected process. The process will be inserted into the cursor position. When the 2-part system specification is "1: EXIST", and the multi-part system program management is ON, the selected process is added to the part system being edited. For the other part system (not edited), the process currently being run by the part system is added. If the selected process is either balance cut or the two-part system simultaneous thread cutting, the process is added to the both part systems.

<table>
<tr><th>No.</th><th>Menu</th><th>Details</th></tr>
<tr><td>3</td><td>MOVE</td><td>Changes the process position.

The process can be moved between the part systems when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON. (The movement between the part systems cannot be performed in balance cut machining process and the two-part system simultaneous thread cutting process.)
Change the tool No. when the process is moved between the part systems.
The other part system corresponding to the process is interchanged with the part system of the process in operation when the process is moved between the part systems.

Example) Move "2 TURN-FACE R" of $1 to $2

<Before moving>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 DR-FACE
2 TURN-FACE R
3 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2
3
:</td></tr></table>
<After moving>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 DR-FACE
2
3 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2 TURN-FACE R
3
:</td></tr></table></td></tr>
<tr><td>4</td><td>DELETE</td><td>Deletes the process at the cursor position.
When performing the deletion, the process under the deleted process will be moved up.
The processes corresponding to each part system are deleted together when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.

Example) Delete "1 DR-FACE" of $1

< Before deleting>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 DR-FACE
2 TURN-FACE R
3 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2
3
:</td></tr></table>
< After deleting>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 TURN-FACE R
2 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2
:</td></tr></table></td></tr>
</table>

<table>
<tr><th>No.</th><th>Menu</th><th>Details</th></tr>
<tr><td>5</td><td>COPY</td><td>Copies the process at the cursor position.
The copied process will be inserted under the cursor position.
The processes corresponding to each part system are copied together when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.

Example) Copy "1 DR-FACE" in $1

< Before copying>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 DR-FACE
2 TURN-FACE R
3 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2
3
:</td></tr></table>⇨
< After copying>
<table><tr><th colspan="2">Process list</th></tr><tr><th>$1</th><th>$2</th></tr><tr><td>PROCESS
0 INIT
1 DR-FACE
2 DR-FACE
3 TURN-FACE R
4 TURN-FACE F
:</td><td>PROCESS
0 INIT
1
2
3
4
:</td></tr></table></td></tr>
<tr><td>8</td><td>$<->$</td><td>Switches a part system to be edited.
Pressing this menu, the process data of the next part system is displayed in the LIST VIEW.
The part system is switched in the order of $1, $2 and $1.

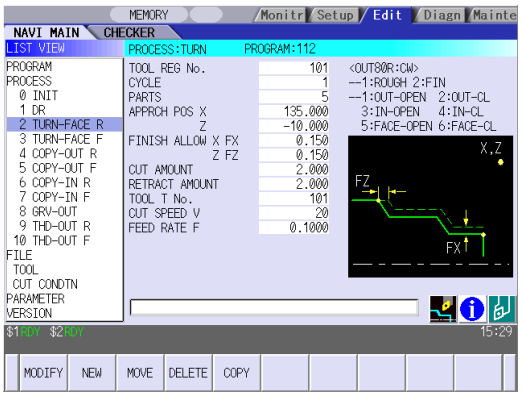
After switching the part system, the cursor is displayed in the same process position as before the switch.

* When the 2-part system specification is "0: NONE", or when the 2-part system specification is "1: EXIST" and the multi-part system program management is OFF, this menu is not displayed.</td></tr>
</table>

Operation example (Selecting the process)

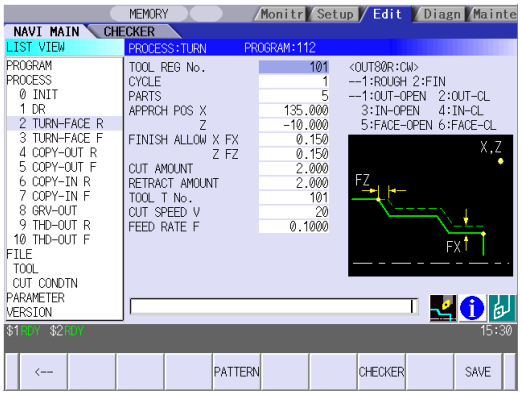
(1) Validate the LIST VIEW area, select the process with the cursor key.

➡ The contents of the OPERATION VIEW area will change to those of the selected process.



(2) Press the [MODIFY] menu key.

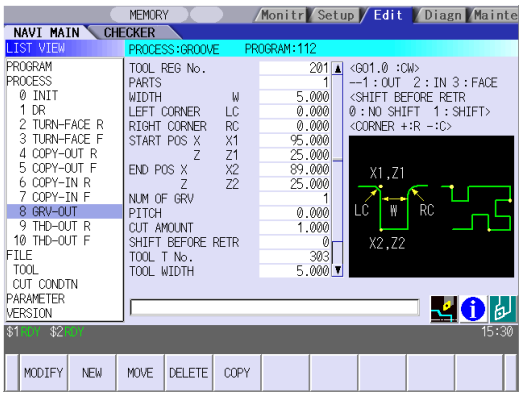
➡ The OPERATION VIEW area will turn active.



Operation example (Deleting the process)

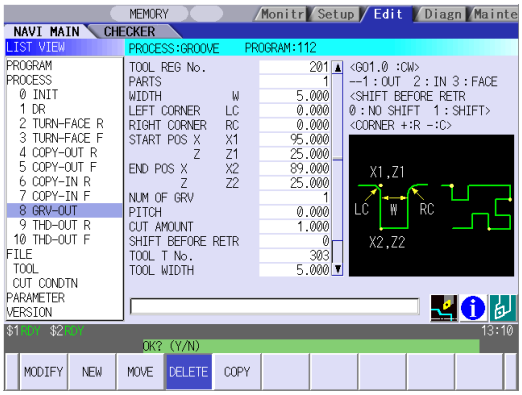
(1) Validate the LIST VIEW area, select the process to be deleted with the cursor key.

➡ The contents of the OPERATION VIEW area will change to those of the selected process.



(2) Press the [DELETE] menu key.

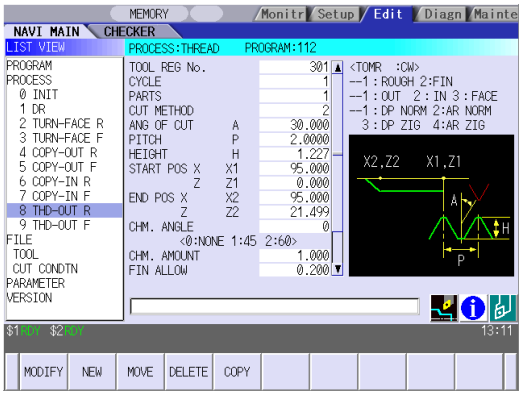
➡ The [DELETE] menu will be highlighted, and a massage confirming the deletion will appear.



(3) Press the [Y] key.

When not deleting the process, press the [N] key

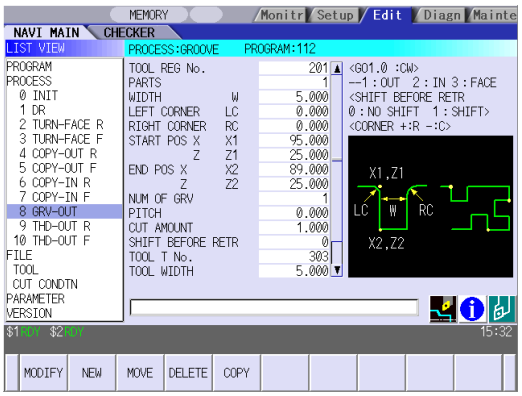
➡ The highlight of the [DELETE] menu will turn OFF, and the process at the cursor position will be deleted.
The process under the deleted process will be moved up one.
The contents in the OPERATION VIEW area will change to those of the process at the cursor position.



Operation example (Copying the process)

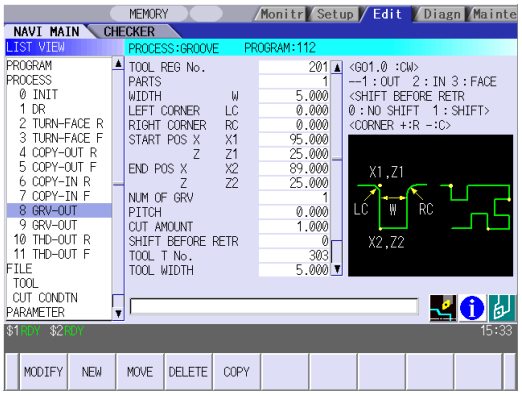
(1) Validate the LIST VIEW area, select the process of the copy source with the cursor key.

➡ The contents of the OPERATION VIEW area will change to those of the selected process.



(2) Press the [COPY] menu key.

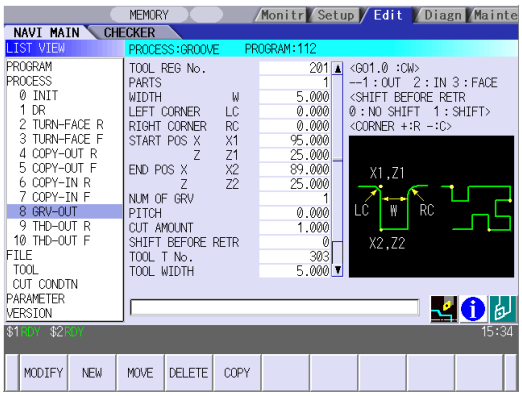
➡ The copied process will be inserted under the cursor position.



Operation example (Moving the process)

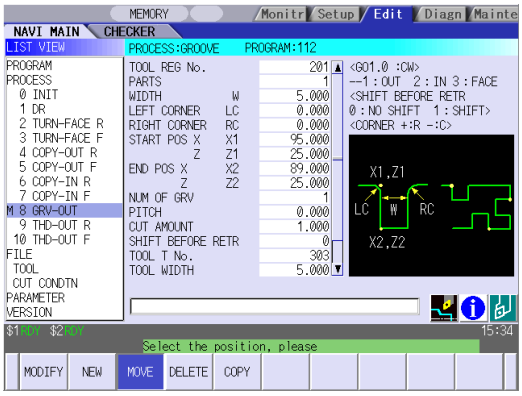
(1) Validate the LIST VIEW area, select the process to be moved with the cursor key.

The contents of the OPERATION VIEW area will change to those of the selected process.

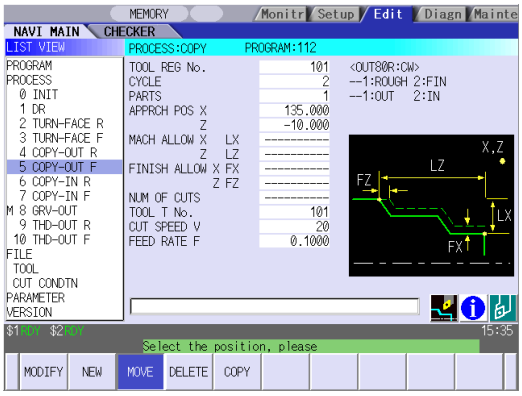


(2) Press the [MOVE] menu key.

The [MOVE] menu will be highlighted.
The mark "M" will be displayed beside the process to be moved.



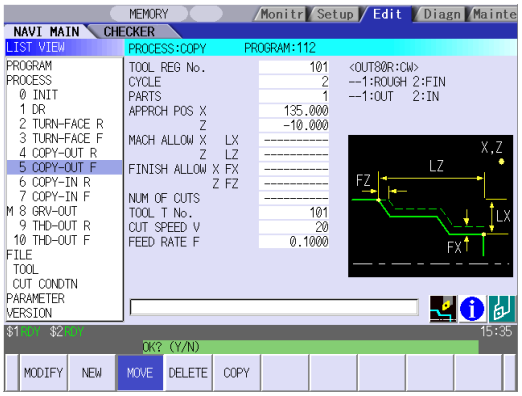
(3) Select the position of the movement destination with the cursor key.



(4) Press the [INPUT] key.

If the [MOVE] menu key is pressed again during the movement operation, the movement operation will be canceled.

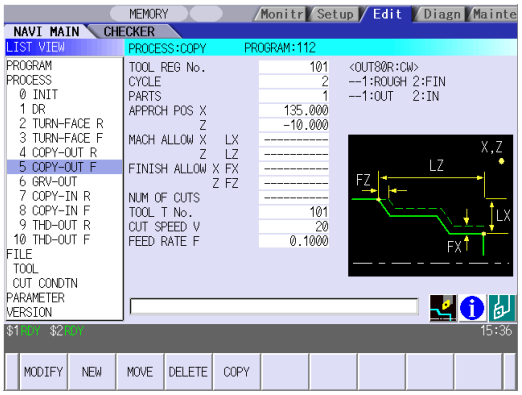
➡ The message to confirm a movement is displayed.



(5) Press the [Y] key.

When not moving the process, press the [N] key

➡ The process of the movement source will be moved to the cursor position.
The highlight of the [MOVE] menu will turn OFF.

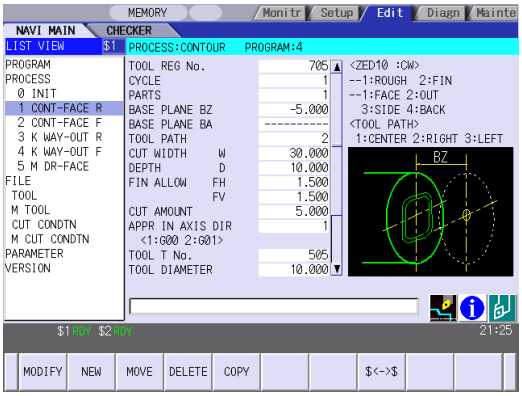


(Note) For the [NEW] menu, refer to the next section.

Operation example (Part system changeover)

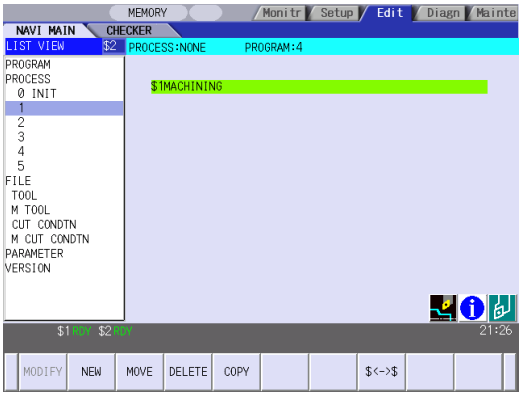
(1) Validate the LIST VIEW area, and select the process to be changed with the cursor key.

➡ The contents of the OPERATION VIEW area will change to those of the selected process.



(2) Press the [$<->$] menu key.

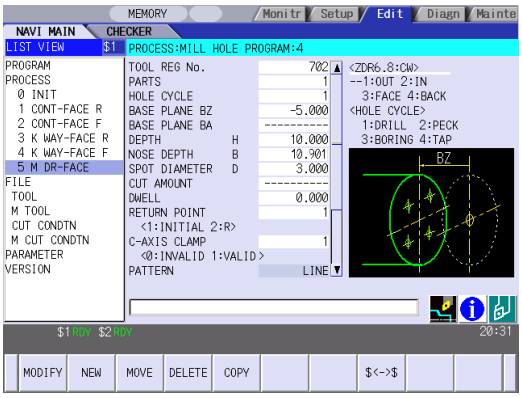
➡ The process data of the part system after the changeover is displayed in the LIST VIEW area.
The cursor position is not moved.
The contents in the OPERATION VIEW area will change to those of the process at the cursor position.
The currently selected part system number will change.



Operation example (Moving process (between the part systems))

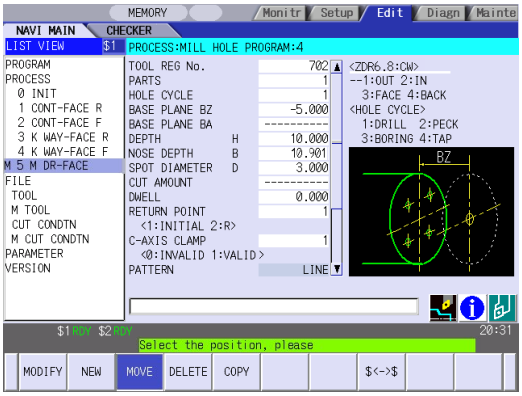
(1) Validate the LIST VIEW area, and select the process to be changed with the cursor key.

➡ The contents of the OPERATION VIEW area will change to those of the selected process.



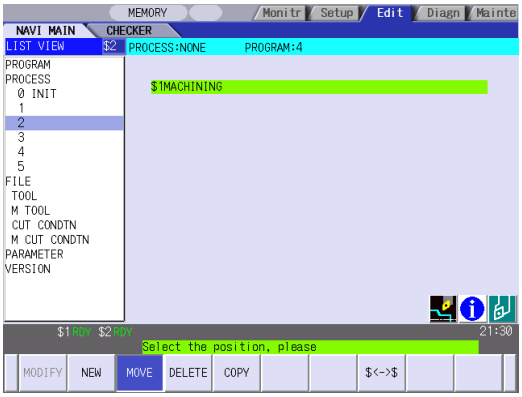
(2) Press the [MOVE] menu key.

➡ The [MOVE] menu will be highlighted. The "M" mark will be displayed beside the process to be moved.



(3) Select the position of the movement destination with the cursor key and [$<->$] menu.

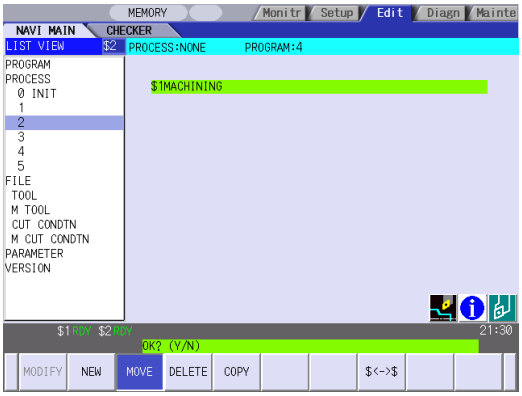
➡ The message "Select the position, please." is not deleted even if the currently selected part system is switched.



(4) Press the [INPUT] key.

If the [MOVE] menu key is pressed again during the movement operation, the operation will be canceled.

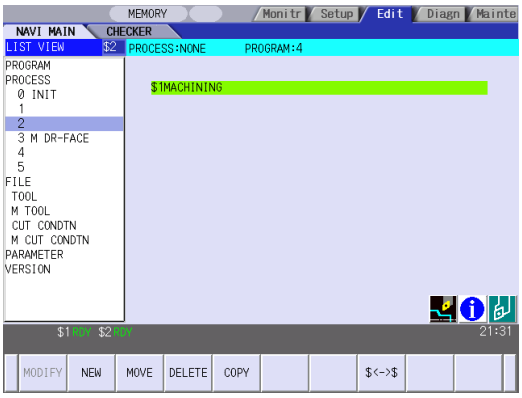
➡ The message to confirm a movement appears.



(5) Press the [Y] key.

Press the [N] key in order not to move.

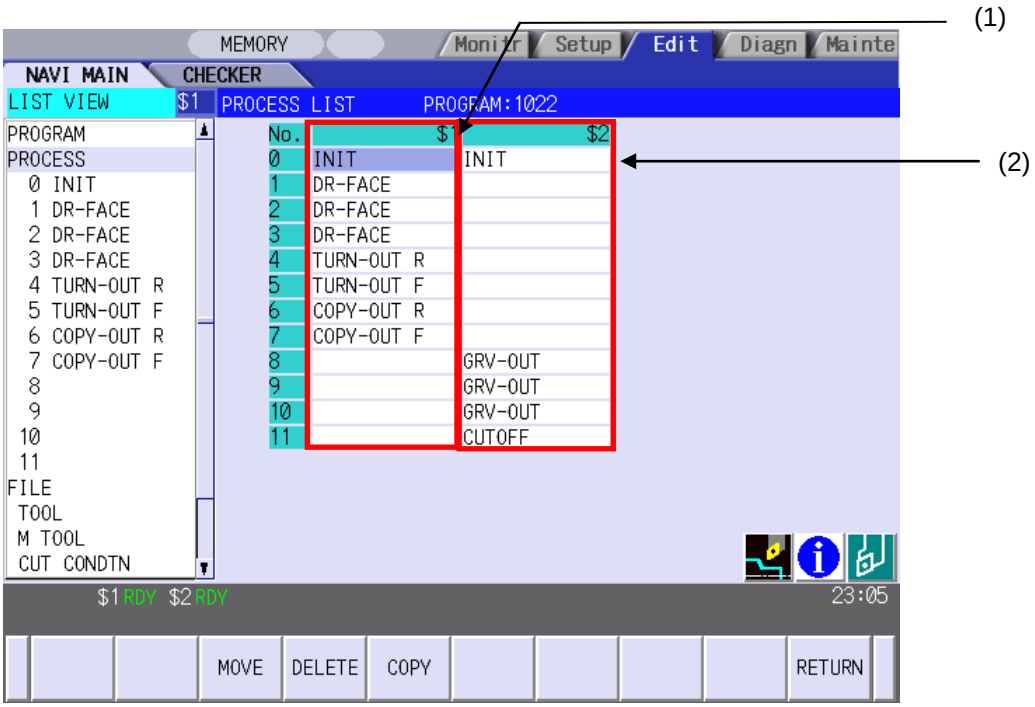
➡ The process of the movement source will be moved to the cursor position.
The highlight of the [MOVE] menu will turn OFF.



4.3.3 System Synchro Screen

The machining processes order of NC program created by each part system are edited on this screen.
The screen is displayed by pressing [$1+$2 LIST] menu key on the Process list screen.

Screen layout



Screen display item

No.	Display item	Details	Setting range
1	$1	The process list generated for the 1st part system is displayed.	
2	$2	The process list generated for the 2nd part system is displayed.	

Menus

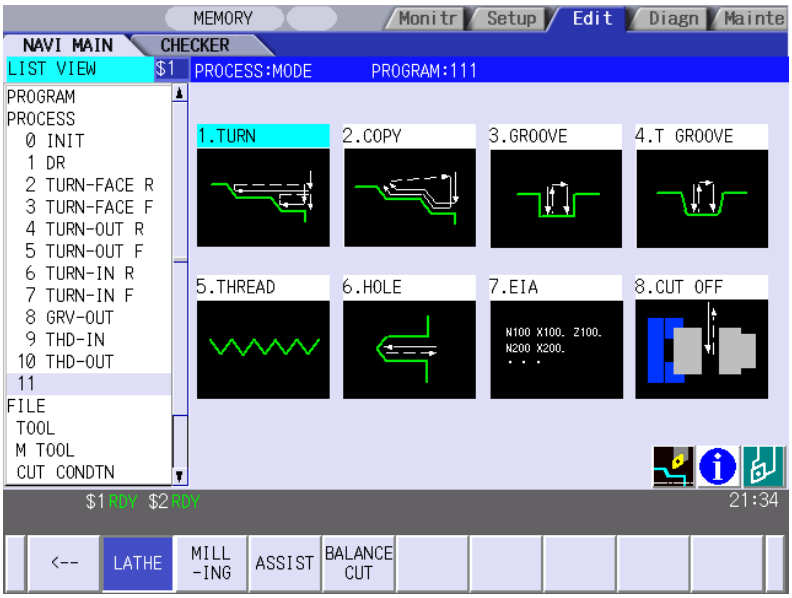
No.	Display item	Details	Remarks
3	MOVE	Changes the process position. The process can be moved between the part systems when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON. (The movement between the part systems cannot be performed in balance cut machining process and the two-part system simultaneous thread cutting process.) Change the tool No. when the process is moved between the part systems. The other part system corresponding to the process is interchanged with the part system of the process in operation when the process is moved between the part systems.	
4	DELETE	Deletes the process at the cursor position. When deleting a process, the process under the deleted process will be moved up. The processes corresponding to each part system are deleted together when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.	
5	COPY	Copies the process at the cursor position. The copied process will be inserted under the cursor position. The processes corresponding to each part system are copied together when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.	
10	RETURN	Return to the Process List Screen.	

4.3.4 Process Mode Selection Screen

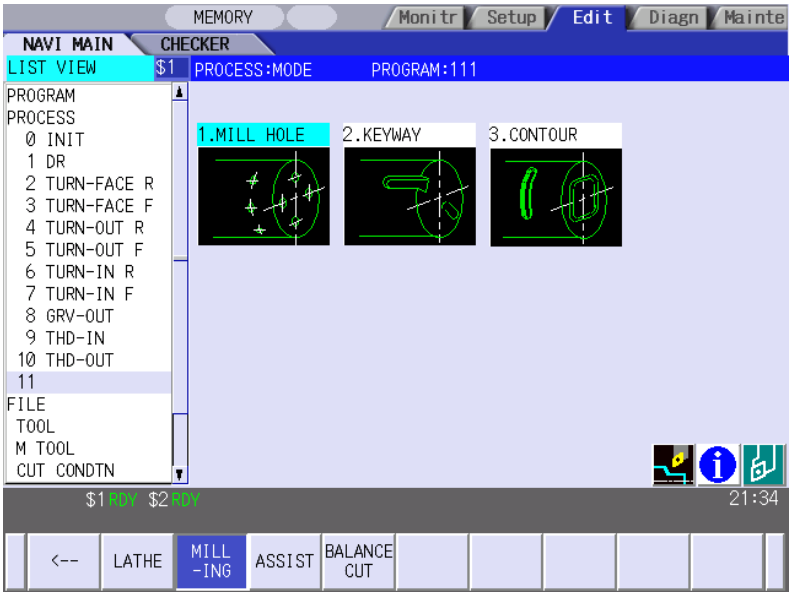
When a new process is added, the process mode is selected on this screen.
This screen is displayed by pressing the [NEW] menu key with the cursor positioned on [PROCESS] in the LIST VIEW.

Screen layout

• Turning

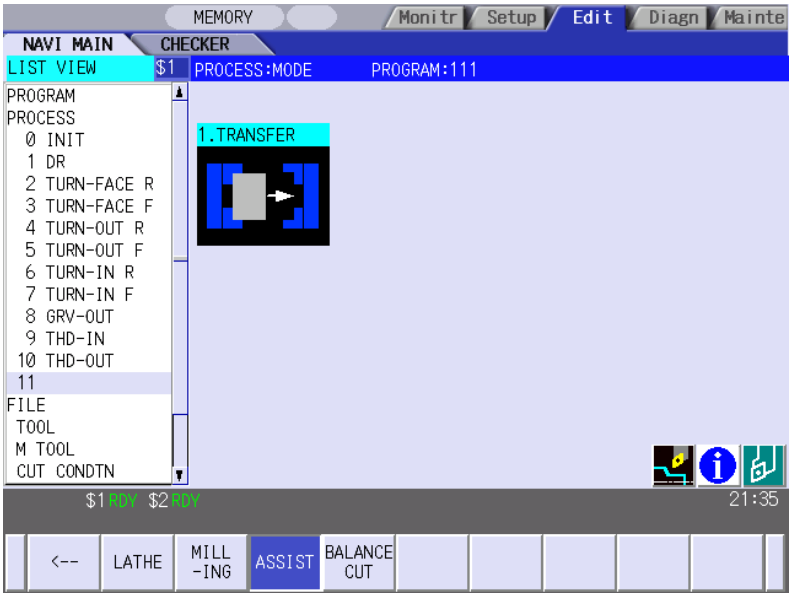


• Milling

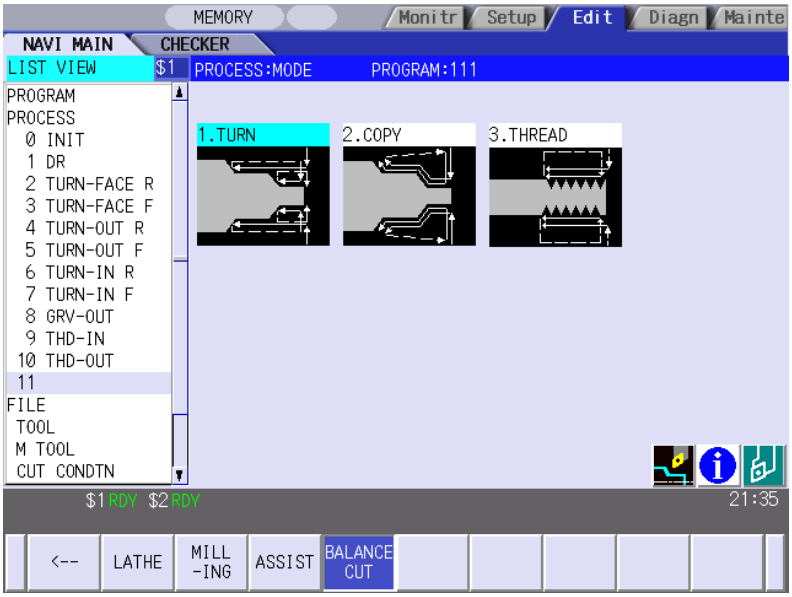


(Note) Milling process is available only when the milling interporation specifications are provided.

• Assist



•Balance cut



Screen display item

• Turning process

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected for the turning machining. Select the process mode by moving the sub cursor or inputting numerical values.	1: TURN 2: COPY 3: GROOVE 4: T GROOVE 5: THREAD 6: HOLE 7: EIA 8: CUTOFF

• Milling Process

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected for milling. Select the process mode by moving the sub cursor or inputting numerical values.	1: MILL HOLE 2: KEYWAY 3: CONTOUR

• Assist process

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected for assist process. Select the process mode by moving the sub cursor or inputting numerical values. (Note) The transfer process is available only when the parameter "#1001 SUB SPINDLE SPEC" is "1: EXIST".	1: TRANSFER

• Balance cut process

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected for balance cut machining. Select the process mode by moving the sub cursor or inputting numerical values. (Note) The balance cut process is available only when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.	1: TURN 2: COPY 3: THREAD

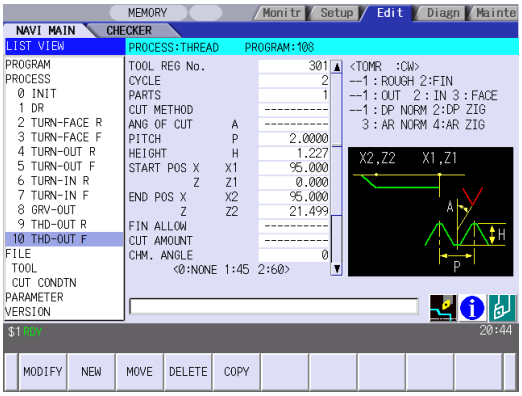
Menu

No.	Menu	Details
1	←	Cancels adding a new process. The LIST VIEW area will turn active after cancel.
2	LATHE	Displays the process mode for the turning machining.
3	MILLING	Displays the process mode for milling. (Note) This is valid when the milling interporation specifications are provided.
4	ASSIST	Displays the process mode for assist process. (Note) This menu is available only when the parameter "#1001 SUB SPINDLE SPEC" is "1: EXIST".
5	BALANCE CUT	Displays the process mode for balance cut machining. (Note) This menu is available only when the 2-part system specification is "1: EXIST" and the multi-part system program management is ON.

(Note) The process insertion position for the second part system is the same as the process number position of the first part system.

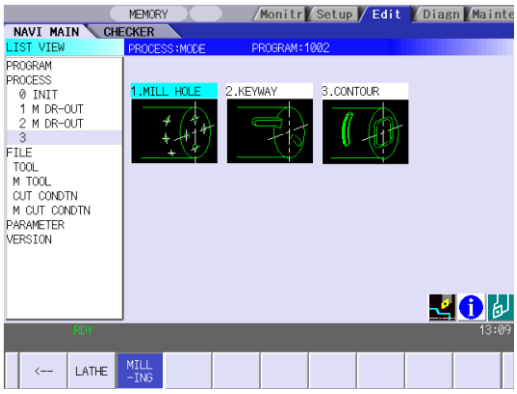
Operation example (Adding a new process)

(1) Validate the LIST VIEW area, and select the position where the process is added with the cursor key.

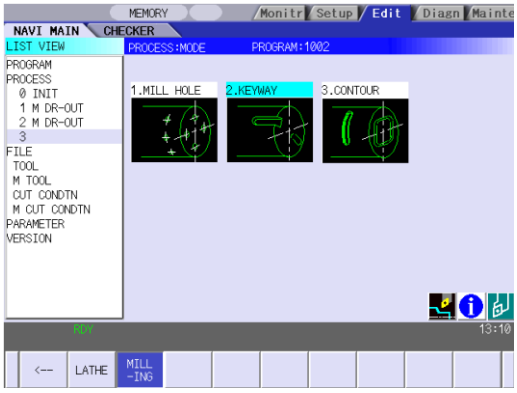


(2) Press the [NEW] menu key.

A blank process will be inserted into the cursor position.
The process mode selection screen will be displayed in the OPERATION VIEW area, and the OPERATION VIEW area will turn active.

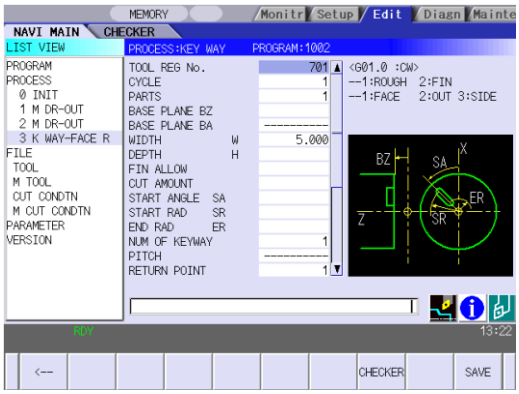


(3) Select the process mode with the cursor or the numerical value input.



(4) Press the [INPUT] key.

The contents in the OPERATION VIEW area will change into those of the selected process mode.
The selected process mode will be displayed at the cursor position in the LIST VIEW area.

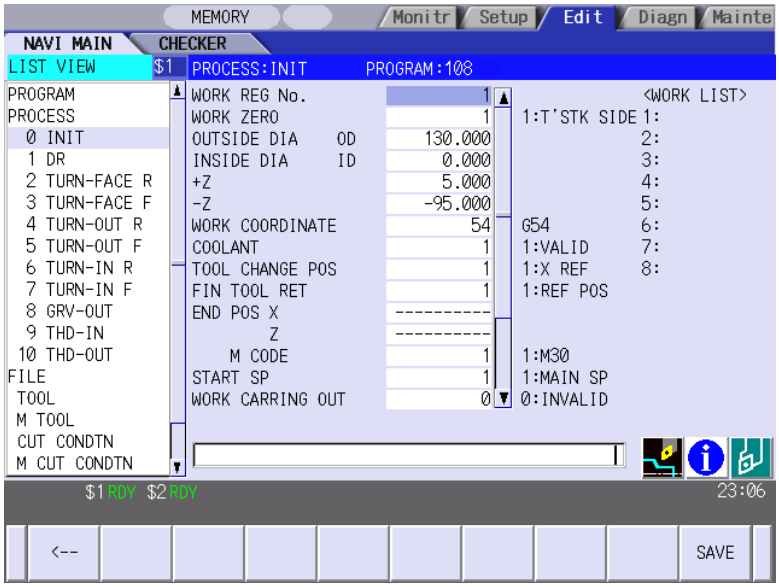


(Note) If the [←] menu key is pressed during adding the process, the screen will return to the state before pressing the [NEW] menu key (state of the 1).

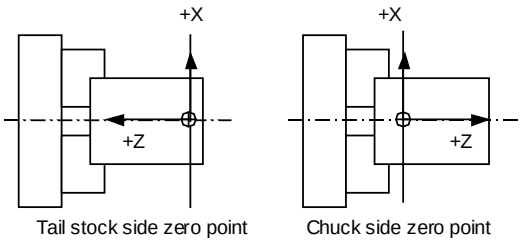
4.3.5 Initial Condition Setting Screen

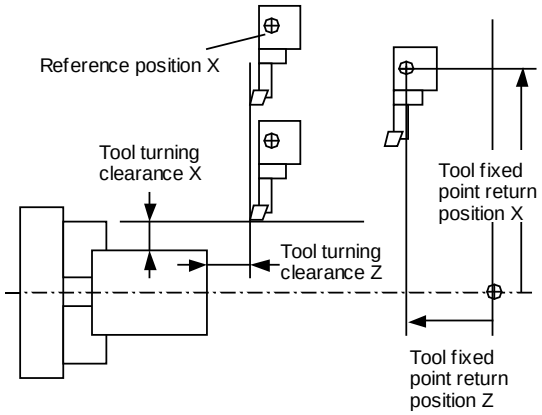
The initial conditions for the program are set on this screen. When the [INIT] is selected in the LIST VIEW area, this screen is displayed.

Screen layout

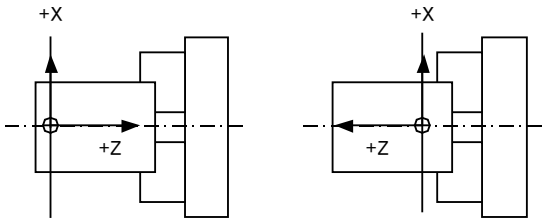


Screen display items

No.	Display item	Details	Setting range
1	WORK REG No.	Input the registration No. of the workpiece material to be cut. Specify it with the No. registered in the cutting condition file. (The list of material names set on the cutting condition file screen will be displayed. Input the corresponding No. based on the list.)	1 to 8
2	WORK ZERO	Input the program zero point. Depending on the program zero point selection, the program coordinate system is determined. 1: Tailstock side zero point 2: Chuck side zero point  Tail stock side zero point Chuck side zero point	1 to 2
3	OUTSIDE DIA	Input the workpiece outer diameter.	0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
4	INSIDE DIA	Input the workpiece inner diameter.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
5	+Z	Input the workpiece face position looking from the program zero point.	-99999.999 to 99999.999mm
6	-Z	Input the workpiece backside position looking from the program zero point.	-9999.9999 to 9999.9999inch
7	WORK COORDINATE	Specify the workpiece coordinate system to be used. 54 : G54 : 59 : G59 P1 : G54.1 P1 : P48 : G54.1 P48 (Note1) If WORK COORDINATE and WORK COORD. SUB SP are set to the same value, P1 to P48 (extended workpiece coordinate system) are not available. An error message "E283 Work coordinate setting error" will be displayed when storing the data.	54 to 59 P1 to P48
8	COOLANT	Select valid/invalid of the coolant. 0: Coolant invalid 1: Coolant valid	0 to 1
9	TOOL CHANGE POS	Select the tool change position. 1: X axis: Reference position Z axis: Tool turning clearance position 2: X axis, Z axis: Tool turning clearance position 3: X axis, Z axis: Tool fixed point return position	1 to 3
10	FIN TOOL RET	Select the tool return type after the program end. 1: Reference position 2: Machining end position 3: Specified position 	1 to 3
11	END POS X	Input the tool return position after the program end by using machine coordinate system. This is valid when end tool return type 3 (specified position) is selected.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
12	END POS Z		
13	END M CODE	At the program end, select the M command to be output. 1 : M30 2 : M02 3 : M99	1 to 3

No.	Display item	Details	Setting range
14	START SP	Select the spindle that performs machining at the start of the program. 1: MAIN SP 2: SUB SP (Note) This can be set only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1,2 (Default: 1)
15	WORK CARRING OUT	Select valid/invalid of the workpiece delivery to the parts catcher when the machining is completed. 0: INVALID 1: VALID (Note 1) This can be set only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". (Note 2) A workpiece delivery device is necessary for the machine specifications.	0,1 (Default: 0)
16	CARRING OUT POS Z	Set the workpiece carrying out position with the Z coordinate 0: INVALID 1: VALID (Note) This can be set only when the WORK CARRING OUT is set to "1: VALID".	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
17	WORK COORD. SUB SP	Specify the workpiece coordinate system to be used with the sub spindle. (Note 1) This can be set only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". (Note 2) If WORK COORDINATE and WORK COORD. SUB SP are set to the same value, P1 to P48 (extended workpiece coordinate system) are not available. An error message "E283 Work coordinate setting error" will be displayed when storing the data.	54 to 59 P1 to P48

No.	Display item	Details	Setting range
18	WORK ZERO SUB SP	Input the program zero point to be used with the sub spindle. Depending on the program zero point selection, the program coordinate system is determined. 1: Tailstock side zero point 2: Chuck side zero point  Tail stock side zero point Chuck side zero point (Note 1) This can be set only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". (Note 2) This can be set only when the workpiece coordinate system used for the main spindle is different from that for the sub spindle. If the workpiece coordinate system used for the main and sub spindles are the same, the zero point will be determined as follows. <When the main spindle's program zero point is the tail stock side zero point> The sub spindle's program zero point is the chuck side zero point. <When the main spindle's program zero point is the chuck side zero point> The sub spindle's program zero point is the tail stock side zero point.	1 to 2
19	Z SUB SP	Input the workpiece sub position looking from the program zero point to be used with the sub spindle. (Note 1) This can be set only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". (Note 2) This can be set only when the workpiece coordinate system used for the main spindle is different from that for the sub spindle.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
20	PART SYSTEM SEL.	When the multi-part system program management is invalid, specify for which part system you create a program. 1: $1 2: $2 (Note) This can be set only when the 2-part system specification is "1: EXIST" and the multi-part system program management is OFF (parameter #1285 ext21/bit0).	1,2

(Note) If the workpiece coordinate system used by the main spindle is the same as that of the sub spindle, there is no need to consider the workpiece movement amount for workpiece transfer in machining with the sub spindle. For the sub spindle's machining process data, set the values of the state when the workpiece is mounted to the main spindle.

Menus

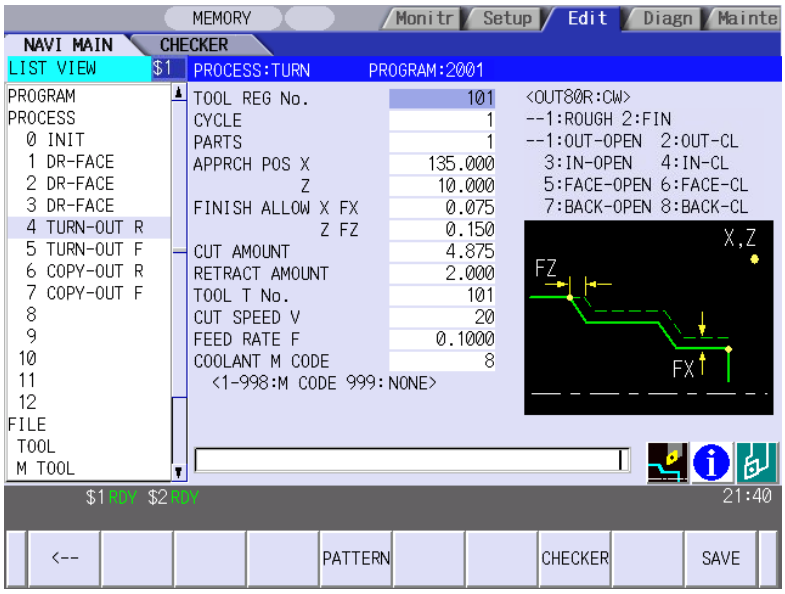
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
10	SAVE	Saves the changes in the initial conditions. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.6 Turning Screen

(1) Turning screen

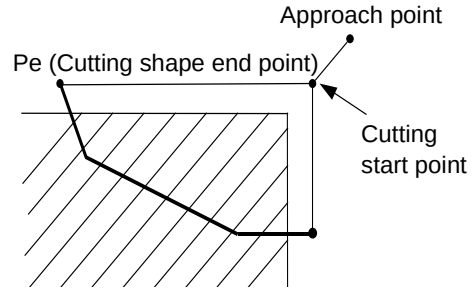
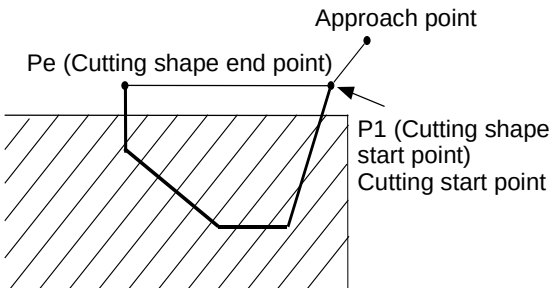
The parameters for the turning process are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:101 to 150 601 to 650 (Default: 101) $2: 1101 to 1150 1601 to 1650 (Default: 1101)
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the cutting area gradually. Leaves the finishing allowance for the cutting shape. <2: Finishing machining> Machines the cutting shape in one cycle.	1,2 (Default: 1)

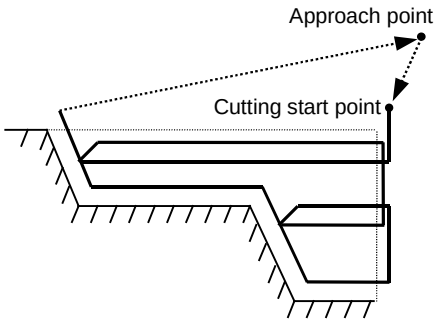
No.	Display item	Details	Setting range
3	PARTS	Input the machining area. <1: OUT-OPEN> Machines the outer diameter area from the front face of workpiece. <2: OUT-CL> Machines the outer diameter area from the halfway of workpiece. <3: IN-OPEN> Machines the inner diameter area from the front face of workpiece. <4: IN-CL> Machines inner area from the halfway of workpiece. <5: FACE-OPEN> Machines the front face of workpiece. <6: FACE-CL> Machines the front face from the halfway of workpiece. <7: BACK-OPEN> Machines the back side of workpiece. <8: BACK-CL> Machines the back side of workpiece from the halfway of workpiece. (Note 1) BACK-OPEN and BACK-CL are available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". [OPEN type]  [CLOSE type]  When the cutting shape is not incremented or decremented monotonously, CLOSE type is selected.	1 to 8 (Default: 1)
4	APPRCH POS X	Input the approach point. After machining, the tool returns to the approach point.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
5	APPRCH POS Z		

No.	Display item	Details	Setting range
6	FINISH ALLOW X (FX)	Input the finishing allowance for the rough machining. Input both FX and FZ with radius value.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
7	FINISH ALLOW Z (FZ)		
8	CUT AMOUNT	Input the cut amount for the rough machining.	0.001 to 99.999mm
9	RETRACT AMOUNT	Input the retract amount for the rough machining.	0.0001 to 9.9999inch
10	TOOL T No.	Specify the tool No. and the compensation No. to be used. (T function code data being output as the NC data) When tool registration No. is specified, tool No. registered in the tool file is automatically set. If this is set to "0", T function code is not output.	0 to 99999999
11	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
12	FEEDRATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
13	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

(Addendum) The tool is retracted as shown below during rough machining.

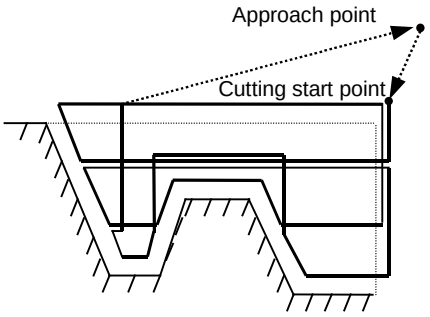
[OPEN type]

The tool is retracted in 45˚ direction in respect to the cutting shape.



[CLOSE type]

The tool is retracted tracing the cutting shape.



(Note) Tool path is not provided based on the tool shape (tool nose angle, front edge angle, etc.) Therefore, when the cutting shape is not incremented or decremented monotonously, take the tool shape into consideration to input the cutting shape.

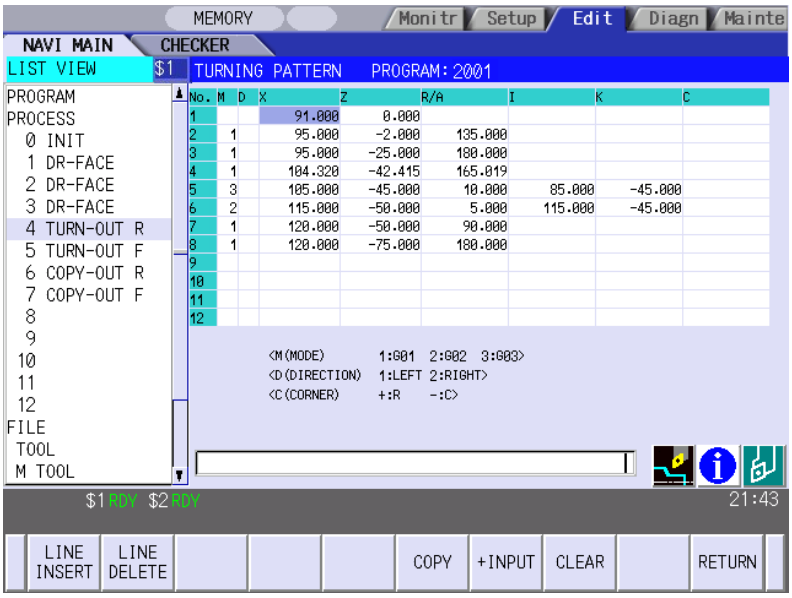
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	Machining pattern selection screen is displayed.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position. If illegal parameters are found in the pattern input screen, the screen name and error will be displayed.

(2) Turning pattern screen

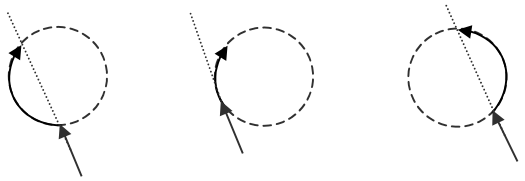
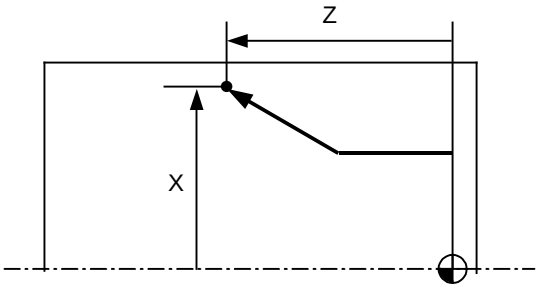
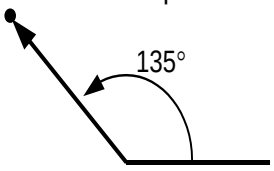
The cutting shapes for the turning process are input on this screen.

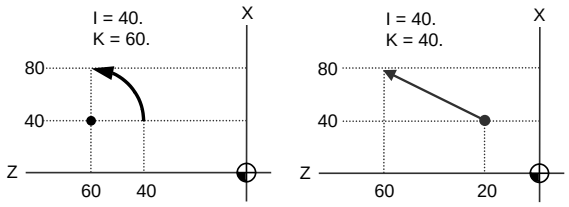
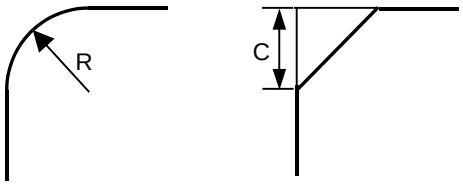
Screen layout



Screen display items

No.	Display item	Details	Setting range
1	No.	Shape No.	1 to 50
2	M	Input the shape. <1> Linear (G01) machining <2> CW circular (G02) machining <3> CCW circular (G03) machining (Note) Not omittable.	1 to 3

No.	Display item	Details	Setting range
3	D	Input right turn or left turn in respect to the vector at the end of the previous shape. 1: Left turn 2: Right turn (Note 1) When nothing is input, it is regarded as "contacting". (Note 2) Omittable. However, when the end point of the previous line, X and Z, is uncertain, always input.  Turn to left Tangent Turn to right	1,2
4	X Z	Input the start point of a shape in the line No.1 and the end point of each shape in the line No.2 and after. Specify with diameter value of the program coordinate system for X and with radius value for Z.  (Note 1) Always input the coordinate in the final line. Omittable except for the line No.1 and the last one. (Note 2) Always input when the corner shape dimension is input in the previous line.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
5	R/A	• When the shape is arc, input the radius of arc. Positive value: Arc command smaller than 180° Negative value: Arc command larger than 180° • When the shape is linear, input the angle.  (Note 1) Always input when the shape is arc. (Note 2) When the shape is linear and the coordinate X, Z or vector I, K is input, this data is invalid.	Radius: 0.001 to 999999.999mm, -999999.999 to -0.001mm Angle: -359.999 to 360.000°

No.	Display item	Details	Setting range
6	I K	• When the shape is arc, input the arc center coordinate. • When the shape is linear, input the gradient (vector).  (Note 1) When the shape is arc and only one of either I or K is input, the other one is regarded as "0". (Note 2) When the shape is linear and the coordinate X, Z or angle is input, this data is invalid.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
7	C	Input the corner dimension. Positive value: Corner R Negative value: Corner C  (Note 1) When corner dimension is specified, input the end point X, Y in the next line in principle.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch

Menus

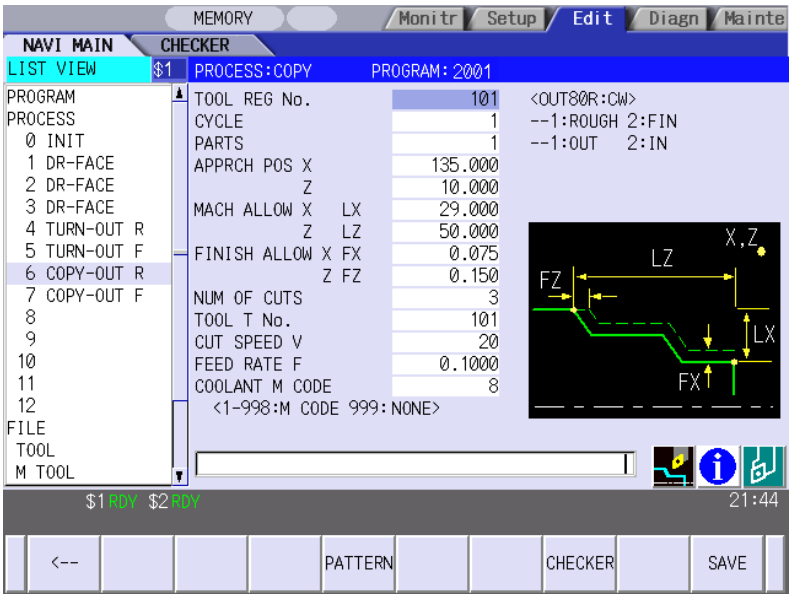
No.	Menu	Details
1	LINE INSERT	Inserts the shape data in front of the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
2	LINE DELETE	Deletes the shape data at the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
6	COPY	Copies the previous line data at the cursor position.
7	+INPUT	Inputs data at the cursor position with the data in the previous line added. (Note) This is valid only when inputting the coordinate X and Z.
8	CLEAR	Clears the data at the cursor position.
10	RETURN	Returns to the turning screen.

4.3.7 Copy Cutting Screen

(1) Copy cutting screen

The parameters for the copy cutting process are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:101 to 150 601 to 650 (Default: 101) $2: 1101 to 1150 1601 to 1650 (Default: 1101)
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the cutting area gradually. Leaves the finishing allowance for the cutting shape. <2: Finishing machining> Machines the cutting shape in one cycle.	1,2 (Default: 1)
3	PARTS	Input the machining area. <1: Outer diameter> Machine the outer diameter section of the workpiece. <2: Inner diameter> Machine the inner diameter section of the workpiece.	1 to 2 (Default: 1)
4	APPRCH POS X	Input the approach point. After machining, the tool returns to the approach point.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
5	APPRCH POS Z		

No.	Display item	Details	Setting range
6	MACH ALLOW X (LX)	Input the allowance in X axis direction with the radius value for the rough machining.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
7	MACH ALLOW Z (LZ)	Input the allowance in Z axis direction for the rough machining.	
8	FINISH ALLOW X (FX)	Input the finishing allowance for the rough machining. Input both FX and FZ with radius value.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
9	FINISH ALLOW FZ (FZ)		
10	NUM OF CUTS	Input the number of cuts for the rough machining.	1 to 99
11	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
12	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
13	FEED RATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
14	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

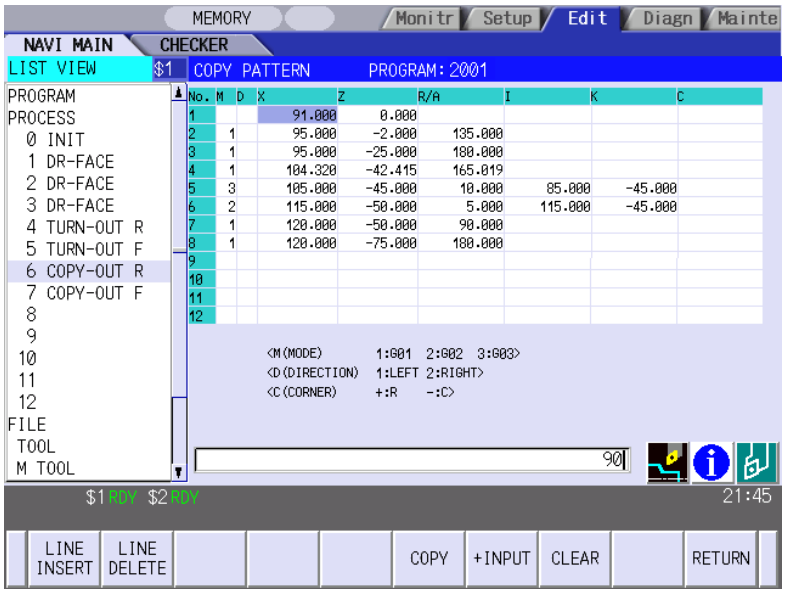
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	Displays the machining pattern selection screen.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position. If illegal parameters are input in the pattern input screen, the screen name and error will be displayed.

(2) Copy cutting pattern screen

The cutting shapes for the turning process are input on this screen.

Screen layout



Screen display items

Refer to the section "4.3.5 Turning Screen (2) Turning pattern screen".

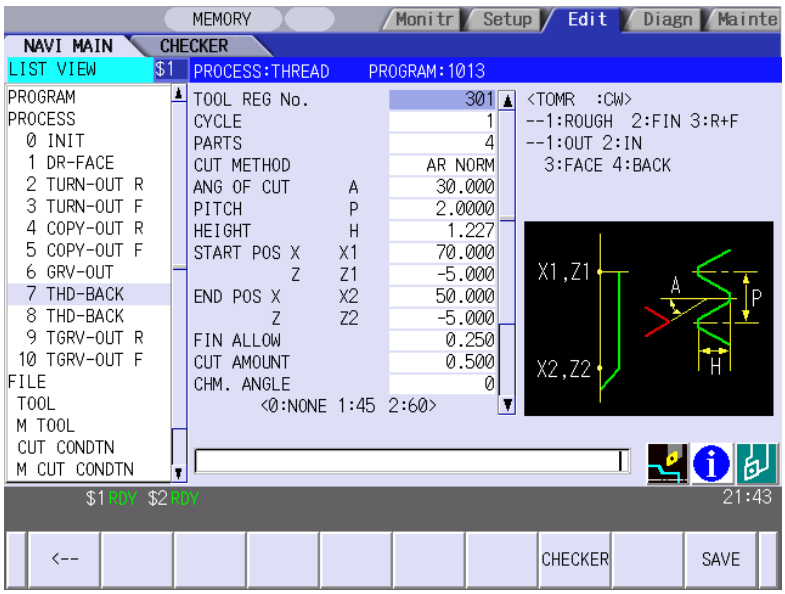
Menus

No.	Menu	Details
1	LINE INSERT	Inserts the shape data in front of the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
2	LINE DELETE	Deletes the shape data at the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
6	COPY	Copies the previous line data at the cursor position.
7	+INPUT	Input data at the cursor position with the data in the previous line added. (Note) This is valid only when inputting the coordinate X and Z.
8	CLEAR	Clears the data at the cursor position.
10	RETURN	Returns to the copy cutting screen.

4.3.8 Threading Screen

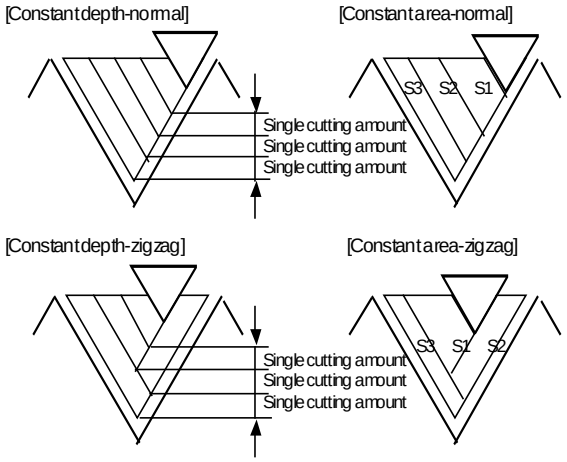
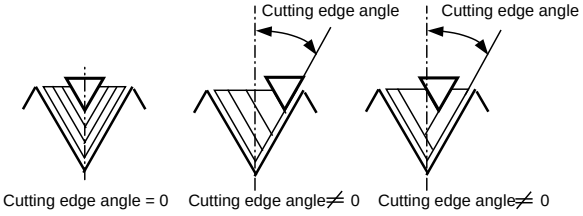
The parameters for the thread process are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:301 to 350 (Default: 301) $2: 1301 to 1350 (Default: 1301)
2	CYCLE	Input the machining method. <1: ROUGH (Rough machining)> Cuts into the thread shape gradually. Leaves the finishing allowance for the thread shape. <2: FIN (Finishing machining)> Machines the thread shape in one cycle. <3: R+F (Rough machining and Finishing machining)> Do the rough machining first before the finishing machining.	1 to 3 (Default: 1)
3	PARTS	Input the machining area. <1: OUT (Outer diameter)> Thread the outer diameter area of the workpiece. <2: IN (inner diameter)> Thread the inner diameter area of the workpiece. <3: Face> Thread the front area of the workpiece. <4: BACK> Thread the back side of the workpiece. (Note) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1 to 4 (Default: 1)

No.	Display item	Details	Setting range
4	CUT METHOD	Select the threading cutting pattern for the rough machining. 1: Constant area-normal 2: Constant area-zigzag 3: Constant depth-normal 4: Constant depth-zigzag 	1 to 4 (Default: 1)
5	ANG OF CUT (A)	Input the cutting edge angle for the rough machining. When the cutting edge angle is set to 0, the zigzag cutting pattern will be invalid. 	0.000 to 60.000°
6	PITCH (P)	Input the screw pitch.	0.0001 to 999.9999mm 0.00001 to 99.99999inch
7	HEIGHT (H)	Input the thread height. When selecting a thread type from the menu, thread height can be input automatically based on the pitch. M METER | UN UNIFY | W WIT | PF PT PS PIPING | NPT PIPING | TM TRAP.30° | TW TRAP.29°	0.001 to 999.999mm 0.0001 to 9999.9999mm
8	START POS X (X1)	Input the X coordinate of the threading start point in the diameter value.	-99999.999 to 99999.999mm
9	START POS Z (Z1)	Input the Z coordinate of the threading start point.	-9999.9999 to 9999.9999inch
10	END POS X (X2)	Input the X coordinate of the threading end point in the diameter value.	-99999.999 to 99999.999mm

No.	Display item	Details	Setting range
11	END POS Z (Z2)	Input the Z coordinate of the threading end point.	-9999.9999 to 9999.9999inch
12	FIN ALLOW	Input the threading finishing allowance for the rough machining. Chamfered section is machined as continuous thread.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
13	CUT AMOUNT	Input the cutting amount corresponding the respective methods below for the rough machining. <Constant cutting amount method> Maximum cutting amount per cut is input. Cutting amount is calculated according to the following formula, and the average is taken. Number of cutting cycles = ((Thread height - Threading finishing allowance)/Cutting amount) ↑ ↑: Rounded up Actual cutting amount = (Thread height – Threading finishing allowance)/Number of cutting cycles <Constant area method> Initial cutting amount is input. "n" th cutting amount (dn) is calculated according to the following formula. dn = d1($\sqrt{}$n - $\sqrt{}$(n-1)) d1: Initial cutting amount	0.001 to 99999.999mm 0.0001 to 9999.9999inch
14	CHM. ANGLE	Input the chamfering angle. 0: No chamfering 1: 45° 2: 60° Chamfering is not carried out when: Thread angle + chamfering angle > 90°	0 to 2
15	CHM. AMOUNT	Input the chamfering amount. Chamfered section is machined as continuous thread.	0.1 to 9.9 (Number of threads)
16	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
17	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
18	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

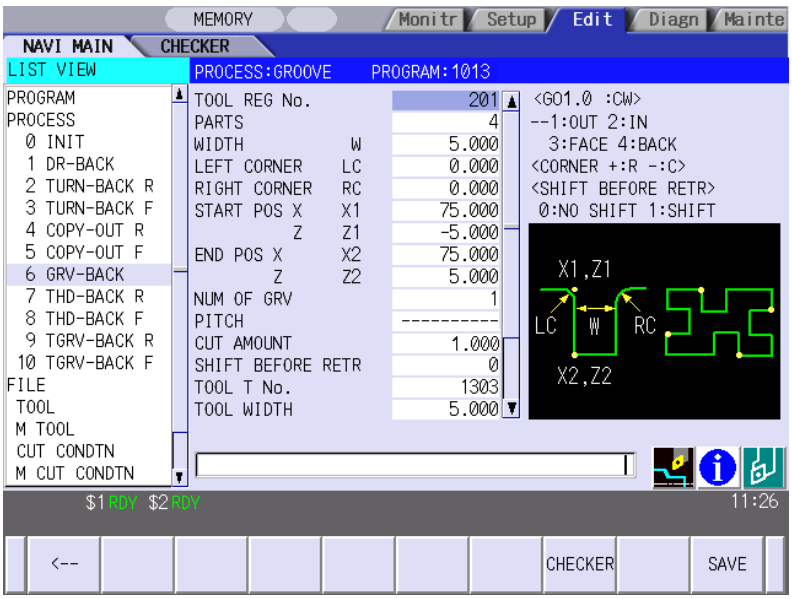
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.9 Grooving Screen

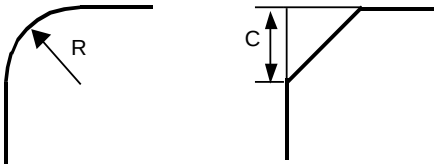
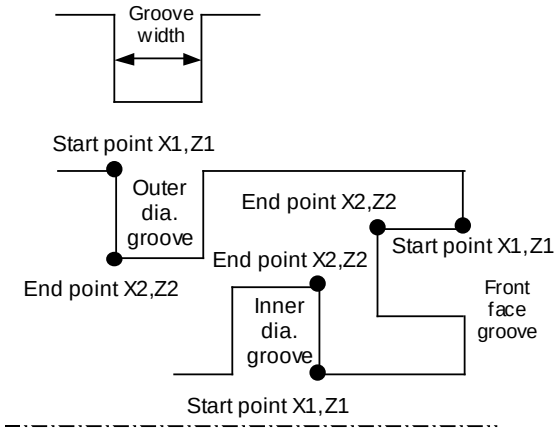
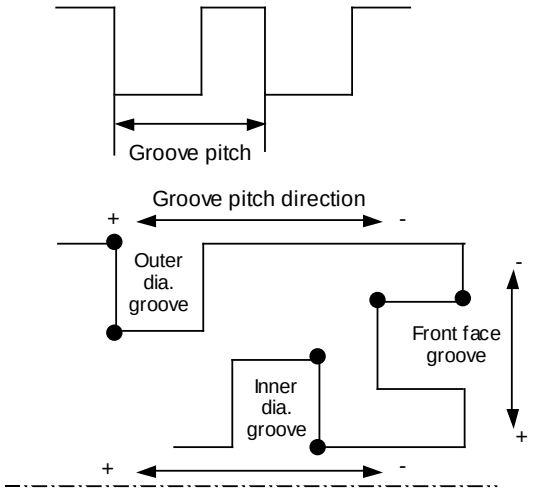
The parameters for the groove process are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:201 to 250 (Default: 201) $2: 1201 to 1250 (Default: 1201)
2	PARTS	Input the machining area. <1: Outer diameter> Groove the outer diameter area of the workpiece. <2: Inner diameter> Groove the inner diameter area of the workpiece. <3: Face> Groove the front area of the workpiece. <4: BACK> Groove the back area of the workpiece. (Note) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1 to 4 (Default: 1)
3	WIDTH (W)	Input the groove width.	0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
4	LEFT CORNER (LC)	Input the dimension of the left groove corner. Positive value: Corner R Negative value: Corner C  Corner R/C cannot be specified for taper grooving.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
5	RIGHT CORNER (RC)	Input the dimension of the right groove corner. Positive value: Corner R Negative value: Corner C Corner R/C cannot be specified for taper grooving.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
6	START POS X (X1)	Input the X coordinate of the grooving start point in the diameter value.	-99999.999 to 99999.999mm
7	START POS Z (Z1)	Input the Z coordinate of the grooving start point.	-9999.9999 to 9999.9999inch
8	END POS X (X2)	Input the X coordinate of the grooving end point in the diameter value.	
9	END POS Z (Z2)	Input the Z coordinate of the grooving end point. 	
10	NUM OF GRV	Input the number of grooves to be machined.	1 to 99
11	PITCH		-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch

No.	Display item	Details	Setting range
12	CUT AMOUNT	Input the cut amount.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
13	SHIFT BEFORE RETR	Specify whether to shift the tool with cutting feed toward the machined area after reaching the groove bottom second or more time. 0: Not shifted 1: Shifted	0 to 1
14	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
15	TOOL WIDTH	Input the tool width of the respective tool. When tool registration No. is specified, tool width registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999 inch
16	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
17	FEED RATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
18	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

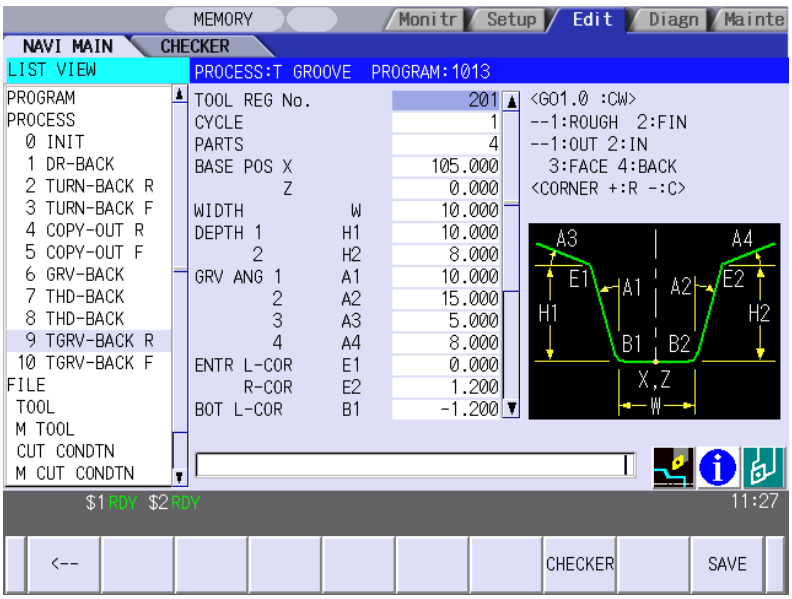
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.10 Trapezoidal Grooving Screen

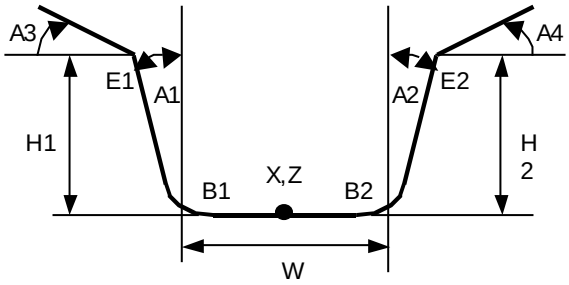
The parameters for the trapezoidal groove process are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:201 to 250 (Default: 201) $2: 1201 to 1250 (Default: 1201)
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the trapezoidal groove shape gradually. Leaves the finishing allowance for the trapezoidal groove shape. <2: Finishing machining> Machines the trapezoidal groove shape in one cycle.	1,2 (Default: 1)
3	PARTS	Input the machining area. <1: Outer diameter> Groove the outer diameter area of the workpiece. <2: Inner diameter> Groove the inner diameter area of the workpiece. <3: Face> Groove the front area of the workpiece. <4: BACK> Groove the back area of the workpiece. (Note) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1 to 4 (Default: 1)

No.	Display item	Details	Setting range
4	BASE POS X	Input the X coordinate, basic point of the trapezoidal groove (the bottom center of the trapezoidal groove), in the diameter value.	-99999.999 to 99999.999mm
5	BASE POS Z	Input the Z coordinate, basic point of the trapezoidal groove (the bottom center of the trapezoidal groove), in the diameter value.	-9999.9999 to 9999.9999inch 0.001 to 99999.999mm
6	WIDTH (W)	Input the groove width.	
7	DEPTH 1 (H1)	Input the left-side depth of the groove.	0.0001 to 9999.9999inch
8	DEPTH 2 (H2)	Input the right-side depth of the groove.	
9	GRV ANG 1 (A1)	Input the angle between the bottom and left-side surface of the groove.	0.000 to 89.999°
10	GRV ANG 2 (A2)	Input the angle between the bottom and right-side surface of the groove.	0.000 to 89.999°
11	GRV ANG 3 (A3)	Input the angle between the left-side of the groove and the workpiece surface.	-89.999 to 89.999°
12	GRV ANG 4 (A4)	Input the angle between the right-side of the groove and the workpiece surface. 	-89.999 to 89.999°
13	ENTR L-COR (E1)	Input the left corner amount of trapezoidal groove entrance. Positive value: Corner R Negative value: Corner C	-99999.999 to 99999.999mm
14	ENTR R-COR (E2)	Input the right corner amount of trapezoidal groove entrance. Positive value: Corner R Negative value: Corner C	-9999.9999 to 9999.9999inch
15	BOT L-COR (B1)	Input the left corner amount of trapezoidal groove bottom. Positive value: Corner R Negative value: Corner C	
16	BOT R-COR (B2)	Input the right corner amount of trapezoidal groove bottom. Positive value: Corner R Negative value: Corner C	

No.	Display item	Details	Setting range
17	FIN ALLOW	Input the finishing allowance of the groove for the rough machining.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
18	CUT AMOUNT	Input the cut amount.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
19	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
20	TOOL WIDTH	Input the tool width of the respective tool. When tool registration No. is specified, tool width registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
21	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
22	FEED RATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
23	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

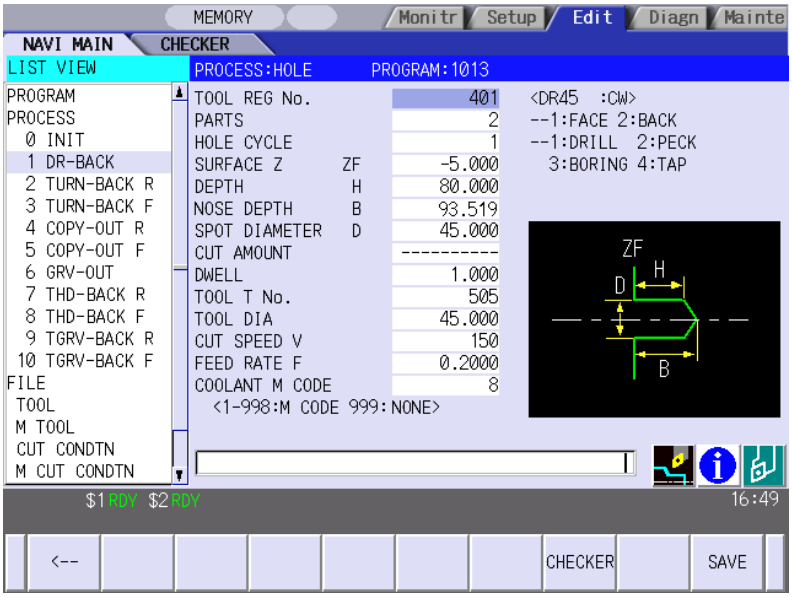
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.11 Hole Drilling Screen

Miscellaneous parameters related to the hole drilling process patterns are input on this screen. This is displayed when PATTERN menu is pressed on the hole drilling screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:401 to 450 501 to 550 (Default: 401) $2: 1401 to 1450 1501 to 1550 (Default: 1401)
2	PARTS	Input the machining area. <1: FACE> Hole drilling the face area of the workpiece. <2: BACK> Hole drilling the back area of the workpiece. (Note) This item is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1,2 (Default: 1)

No.	Display item	Details	Setting range
3	HOLE CYCLE	Input the type of hole machining cycle. <1: DRILL> (G83) The machining is performed as far as the hole bottom at a stretch, and the tool is lifted up after the hole bottom dwell has been executed. <2: PECK> (G83) The machining is performed halfway of the hole, and the tool is returned to the higher than the hole top position each time. The machining is performed as far as the hole bottom by repeating such operations. <3: BORING> (G85) The machining is performed as far as the hole bottom at a stretch, and the tool is lifted up with the cutting feedrate after the hole bottom dwell has been executed. <4: TAP> (G84,G84.1) The tap machining is performed as far as the hole bottom, and the tool is lifted up with the reversed rotation after the hole bottom dwell has been executed.	1 to 4 (Default: 1)
4	SURFACE Z (ZF)	Input the top surface position of the hole.	-99999.999 to 99999.999mm
5	DEPTH (H)	Input the hole depth from the workpiece top surface with the addition input method. When the hole depth is changed, tool nose depth will be automatically updated. If the calculated nose depth is 0 or below, the data range will be over.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
6	NOSE DEPTH (B)	Input the nose depth from the workpiece top surface with the addition input method. When the nose depth is changed, hole depth will be automatically updated.	0.001 to 99999.999mm
7	SPOT DIAMETER (D)	Input the spot diameter. When inputting the spot diameter, hole depth and nose depth are automatically changed.	0.001 to Tool diameter
8	CUT AMOUNT	When selecting the hole cycle type C=2(deep hole), input the cut amount per cut.	0.001 to 99999.999mm
9	DWELL	Input the dwell time at the bottom of the hole.	0.0 to 99.999sec
10	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
11	TOOL DIA	Input the tool radius of the respective tool. When tool registration No. is specified, tool radius registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
12	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min

No.	Display item	Details	Setting range
13	FEED RATE F	Input the feedrate. When the type of hole machining cycle is TAP, the pitch (mm/rev) is displayed. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
14	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

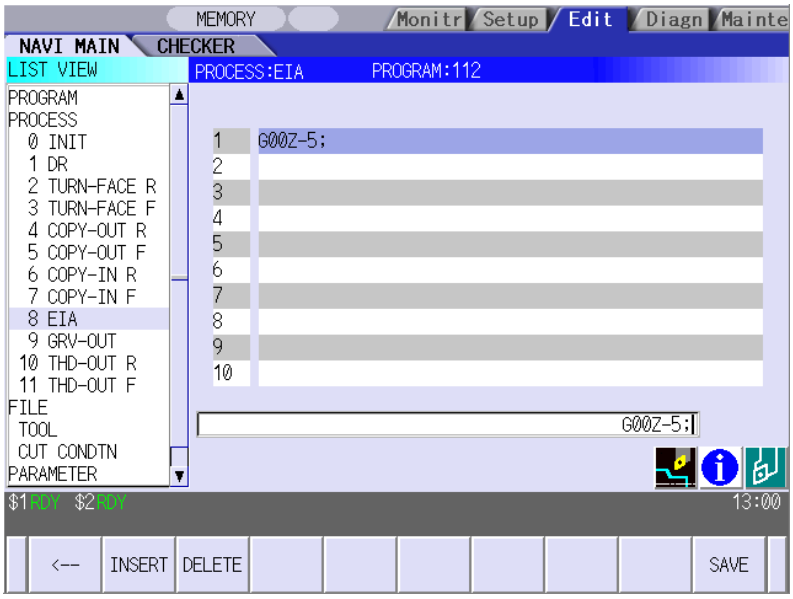
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.12 EIA Screen

The EIA process is input on this screen.

Screen layout



Screen display item

No.	Display item	Details	Setting range
1	EIA BLOCK	The current contents of the EIA block are displayed. Register the EIA by inputting the EIA from the setting area. Note that there is the following restriction. <Restriction> • Characters that can be input into the EIA block are up to 50 characters.	EIA code Max. 10 blocks

Menus

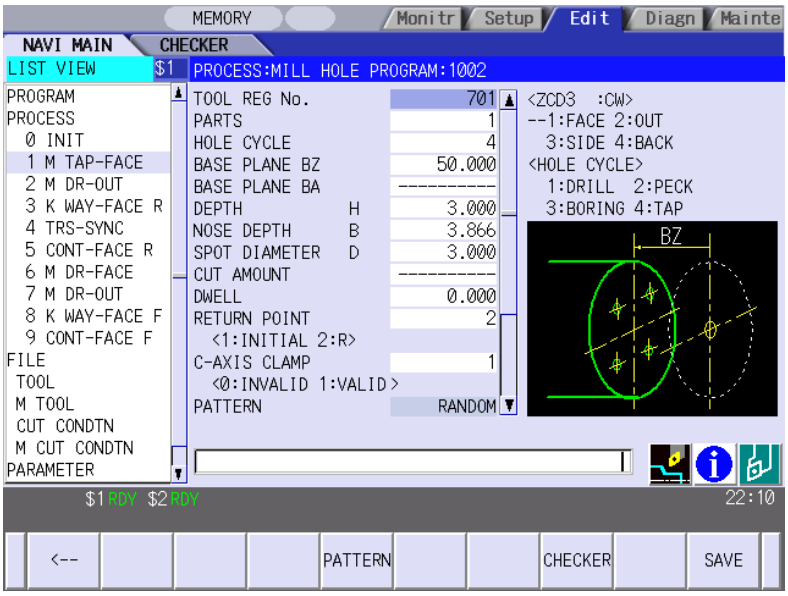
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	INSERT	Inserts a blank block before the block where the cursor exists.
3	DELETE	Deletes the data of the block where the cursor exists.
10	SAVE	Saves the changes in the process.

4.3.13 Milling Hole Drilling Screen

(1) Milling hole drilling screen

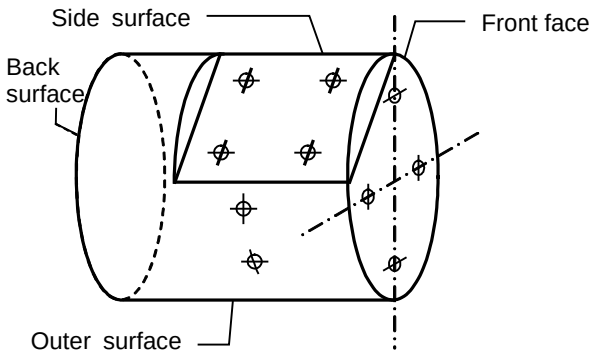
The parameters for the milling hole drilling are input on this screen.

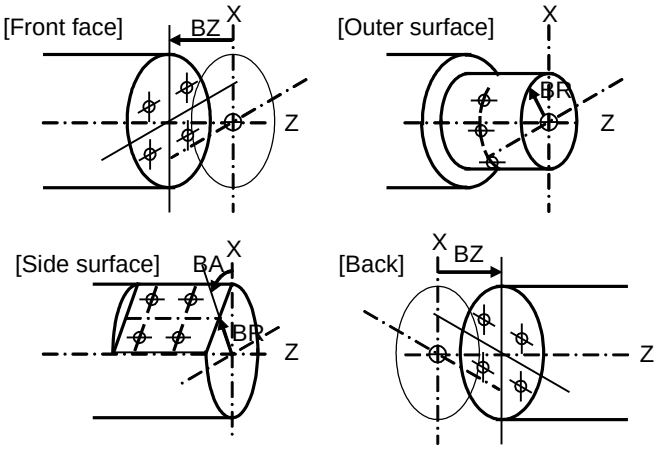
Screen layout

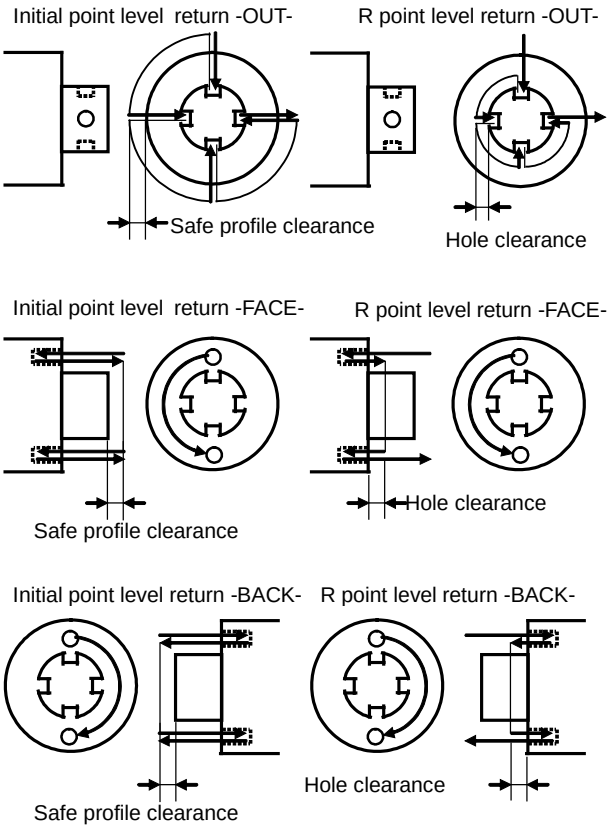


Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:701 to 799 (Default: 701) $2: 1701 to 1799 (Default: 1701)

No.	Display item	Details	Setting range
2	PARTS	Input the machining area. <1: FACE> Machines the front face of workpiece. <2: OUT> Machines the outer surface of workpiece. <3: SIDE> Machines the side surface of workpiece. <4: BACK> Machines the back surface of the workpiece. (Note 1) Y-axis specifications are required for the side cutting. (Note 2) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".  If any data is already registered in the hole drilling pattern screen when inputting the machining area, "Clear the pattern data? (Y/N)" will be displayed. (If the same value is input, the pattern data will not be cleared.)	1 to 4 (Default: 1)
3	HOLE CYCLE	Input the type of hole machining cycle. <1: DRILL>(G83,G87) The machining is performed as far as the hole bottom at a stretch, and the tool is lifted up after the hole bottom dwell has been executed. <2: PECK>(G83, G87) The machining is performed as far as the middle of the hole, and the tool is returned to the higher position than the hole top each time. The machining is performed as far as the hole bottom with such operation repeatedly executed. <3: BORING>(G85, G89) The machining is performed as far as the hole bottom at a stretch, and the tool is lifted up with cutting feed after the hole bottom dwell has been executed. <4: TAP>(G84, G84.1, G88, G88.1) The tap machining is performed as far as the hole bottom, and the tool is lifted up with reversed rotation after the hole bottom dwell has been executed.	1 to 4 (Default: 1)

No.	Display item	Details	Setting range
4	BASE PLANE BZ BASE PLANE BR BASE PLANE BA	Set the hole top position in respect to the machining area.  BASE PLANE BZ/BR are switched according to the machining area. BASE PLANE BA is set only for the side cutting.	Base plane BZ -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch Base plane BR 0.001 to 99999.999mm 0.0001 to 9999.9999inch Base plane BA -359.999 to 360.000°
5	DEPTH H	Input the hole depth from the workpiece top surface with an addition input method. When the hole depth is changed, nose depth is automatically updated. If the calculated nose depth is 0 or below, the data is out of the range.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
6	NOSE DEPTH B	Input the tool nose depth from the workpiece top surface with an addition input method. When the nose depth is changed, hole depth is automatically updated.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
7	SPOT DIAMETER D	Input the spot diameter. When inputting the spot diameter, hole depth and nose depth are automatically changed.	0.001 to Tool diameter (mm) 0.0001 to Tool diameter (inch)
8	CUT AMOUNT	Input the cutting amount per cut when the hole cycle type C=2 (PECK) is selected.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
9	DWELL	Input the dwell time at the bottom of the hole.	0.0 to 99.999sec

No.	Display item	Details	Setting range
10	RETURN POINT	When machining multiple holes, select the height of the tool movement to the next hole position. 1 : Initial point level return 2 : R point level return 	1,2 (Default: 1)
11	C-AXIS CLAMP	Select whether to clamp C axis or not in the machining. Select “Clamp C axis” for heavy load machining. 0 : Invalid 1 : Valid	0,1 (Default: 0)
12	PATTERN	The machining pattern is displayed. RANDOM LINE ARC CIRCLE SQUARE GRID Change the machining pattern on the machining pattern screen.	- (Default: LINE)
13	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, the tool No. registered in the tool file is automatically set.	0 to 99999999
14	DIA	Input the tool diameter. When tool registration No. is specified, the tool diameter registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
15	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min

No.	Display item	Details	Setting range
16	FEED RATE F	Input the feedrate. When the type of the hole machining cycle is TAP, the pitch (mm/rev) is displayed. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
17	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

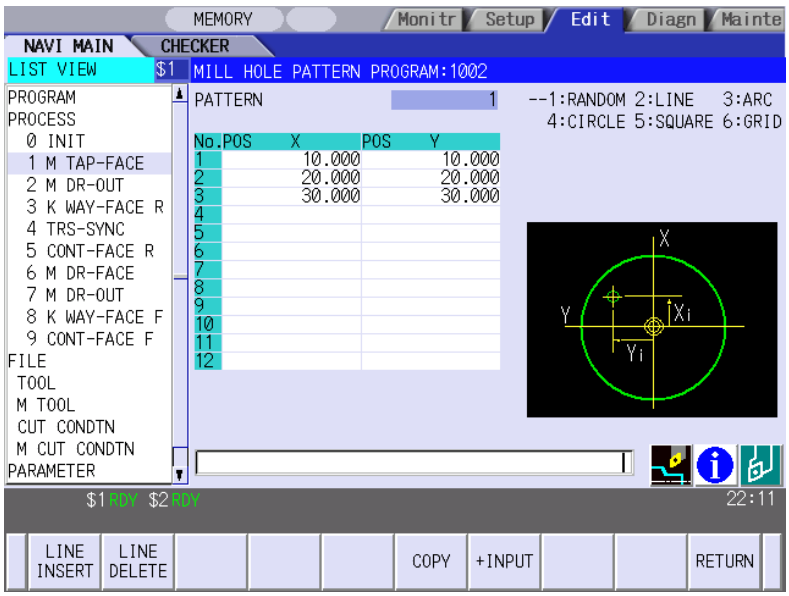
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	The machining pattern selection screen is displayed.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

(2) Hole Drilling Pattern Screen

Various parameters for hole drilling patterns are input on this screen. When the [PATTERN] menu is pressed on the hole drilling screen, this screen is displayed.

Screen layout



Machining area and hole machining pattern

The hole machining patterns selectable for each machining area are as follows.

Machining area \ Pattern	Random	Line	Arc	Circle	Square	Grid
Front face	○	○	○	○	○	○
Outer surface	○	○	×	×	×	×
Side surface	○	○	○	○	○	○
Back surface	○	○	○	○	○	○

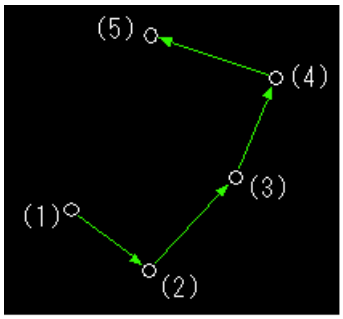
○: Selectable, ×: Not selectable

Screen display items

No.	Display item	Details	Setting range
1	PATTERN	Input the type of hole machining pattern. <1: RANDOM> The machining points are randomly arranged. <2: LINE> The machining points are equally spaced on a line. <3: ARC> The machining points are equally spaced on an arc. <4: CIRCLE> The machining points are equally spaced on a circle. <5: SQUARE> The machining points are squarely arranged. <6: GRID> The machining points are arranged in grid. **(Note)** If the pattern entered is not selectable for the machining area, the message “E002 Data range over” will appear.	1 to 6 (Default: 2)

(Note) The parameters of the second and subsequent lines differ according to the machining pattern setting. The displayed parameters for each pattern are as follows.

・Parameters for RANDOM

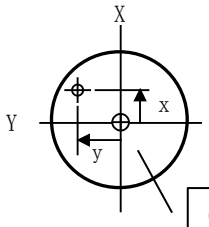


No.	Display item	Details	Setting range
2	HOLE No.	Input the hole No.	1 to 35
3	FACE: POS X POS Y OUT: POS C POS Z SIDE: POS Y POS Z	Input the hole position. [Front face] [Outer circumference] [Side face] [Back face]	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch C: -359.999 to 360.000

Input the hole position in tabular form for the random pattern.
The Image of the operation area

Pattern --1: RANDOM 2: LINE 3: ARC
Hole position 4: CIRCLE 5: SQUARE 6: GRID

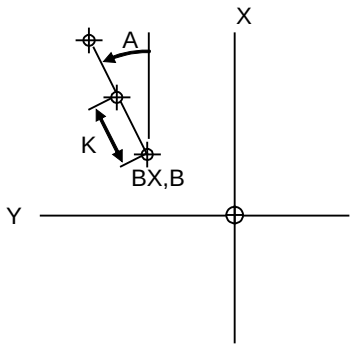
No.	X	Y
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		



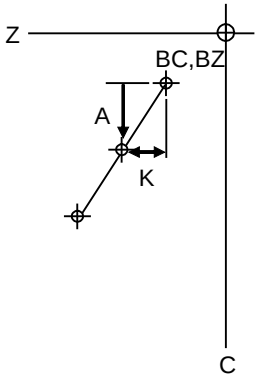
Guide drawing corresponds to the machining area.

・Parameters for LINE

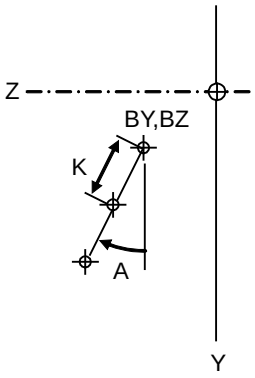
[Front face]



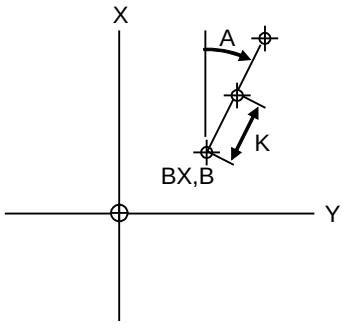
[Outer surface]



[Side surface]



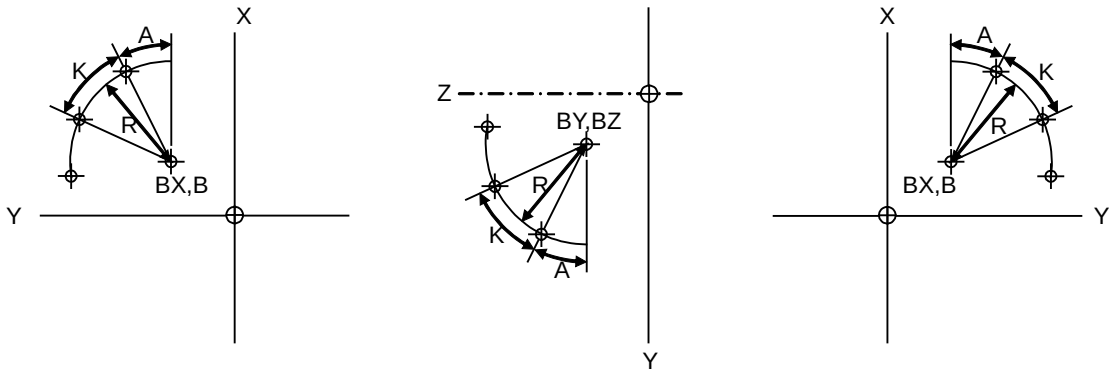
[Back surface]



No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y OUT: BASE POS C BASE POS Z SIDE: BASE POS Y BASE POS Z	Set the first hole position for the machining area.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch C: -359.999° to 360.000°
3	ANGLE (A) PITCH (A)	Front face: Input the angle formed with the machining direction and the positive direction of the X axis. Outer surface: Input the pitch angle in respect to the machining direction. Side surface: Input the angle formed with the machining direction and the positive direction of the Y axis.	-359.999° to 360.000°
4	PITCH (K)	Input the space from the machining point to the next machining point.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
5	NUM OF HOLES	Input the number of holes.	2 to 999
6	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	0 to number of holes (Default: 0)

・Parameters for ARC

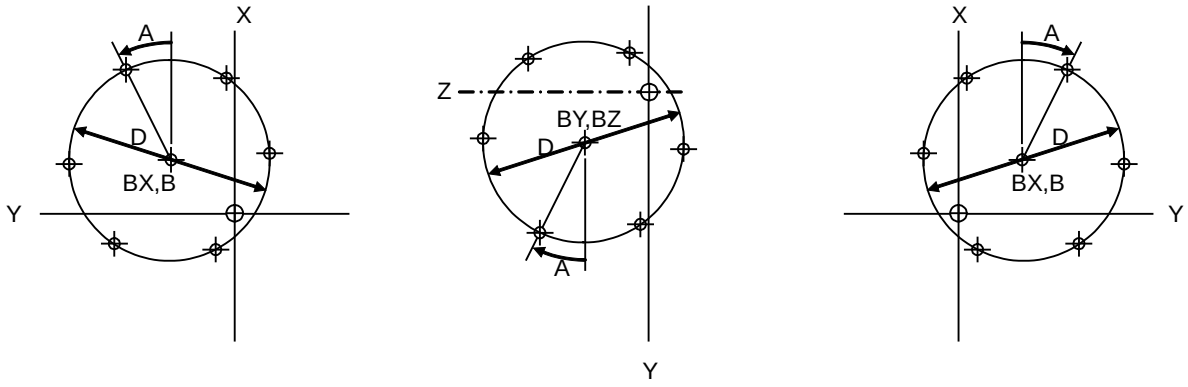
[Front face] [Side surface] [Back surface]



No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the arc center position.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
3	RADIUS R	Input the arc radius.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
4	START ANGLE A	Front face: Input the angle formed with the first machining point and the positive direction of the X axis. Side surface: Input the angle formed with the first machining point and the positive direction of the Y axis.	-359.999° to 360.000°
5	PITCH K	Input the angle from the previous machining point to the next machining point.	-359.999° to 360.000°
6	NUM OF HOLES	Input the number of holes.	2 to 999
7	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	0 to number of holes (Default: 0)

・Parameters for CIRCLE

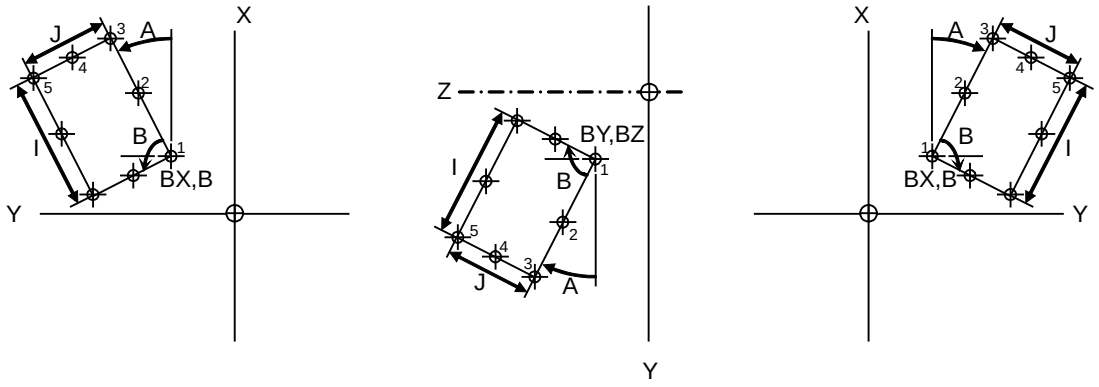
[Front face] [Side surface] [Back surface]



No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the circular center position.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
3	DIAMETER D	Input the circular diameter.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
4	START ANGLE A	Front face: Input the angle formed with the first machining point and the positive direction of the X axis. Side surface: Input the angle formed with the first machining point and the positive direction of the Y axis.	-359.999° to 360.000°
5	NUM OF HOLES	Input the number of holes.	1 to 999
6	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	0 to number of holes (Default: 0)

・Parameters for SQUARE

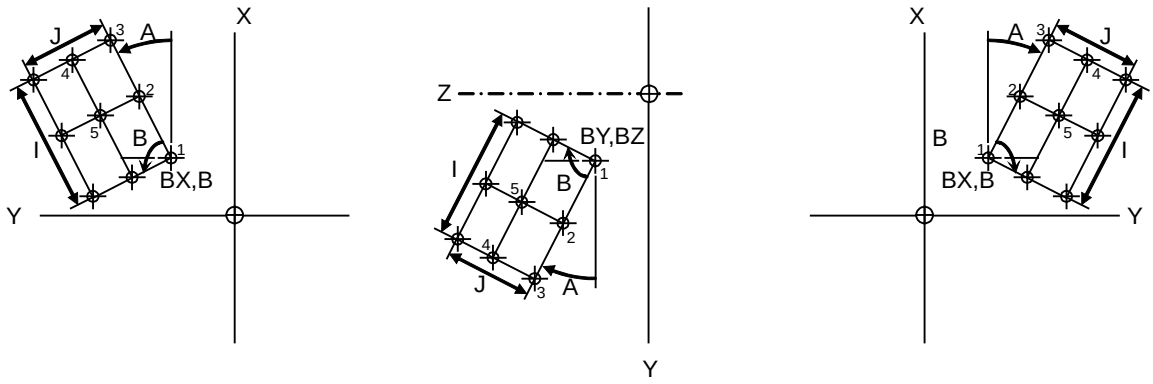
[Front face] [Side surface] [Back surface]



No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the position of the machining start point.	X,Y,Z: -99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
3	X WIDTH I	Input the width of the machining point in the X axis direction.	-99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
4	X NUM OF HOLES	Input the number of machining points in the X axis direction.	2 to 999
5	Y WIDTH J	Input the width of the machining point in the Y axis direction.	-99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
6	Y NUM OF HOLES	Input the number of machining points in the Y axis direction.	2 to999
7	ANGLE A	Front face: Input the angle formed with the machining start direction and the X axis. Side surface: Input the angle formed with the machining start direction and the Y axis.	-359.999° to 360.000°
8	ANGLE B	Input the interior angle. Default value is 90°.	0.001° to 179.999° (Default: 90°)
9	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	0 to number of holes (Default: 0)

・Parameters for GRID

[Front face] [Side surface] [Back surface]



No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the position of the machining start point.	X,Y,Z: -99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
3	X WIDTH I	Input the width of the machining point in the X axis direction.	-99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
4	X NUM OF HOLES	Input the number of machining points in the X axis direction.	2 to 999
5	Y WIDTH J	Input the width of the machining point in the Y axis direction.	-99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
6	Y NUM OF HOLES	Input the number of machining points in the Y axis direction.	2 to 999
7	ANGLE A	Front face: Input the angle formed with the machining start direction and the X axis. Side surface: Input the angle formed with the machining start direction and the Y axis.	-359.999° to 360.000°
8	ANGLE B	Input the interior angle. Default value is 90°.	0.001° to 179.999° (Default: 90°)
9	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	0 to number of holes (Default: 0)

Menus

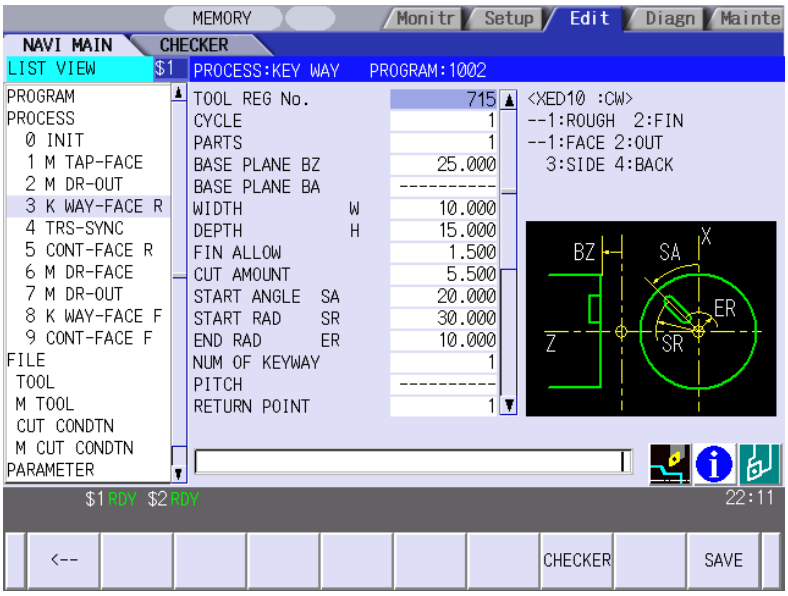
No.	Menu	Details
1	LINE INSERT	Inserts the hole position in front of the cursor position. This is available only for the RANDOM pattern.
2	LINE DELETE	Deletes the hole position at the cursor position. This is available only for the RANDOM pattern.
6	COPY	Copies the previous line data above cursor to the setting area. This is available only for the RANDOM pattern.
7	+INPUT	Adds the previous line data above cursor to the setting data and inputs the value to the setting area. This is available only for the RANDOM pattern.
10	RETURN	Returns to the hole drilling screen.

4.3.14 Keyway Cutting Screen

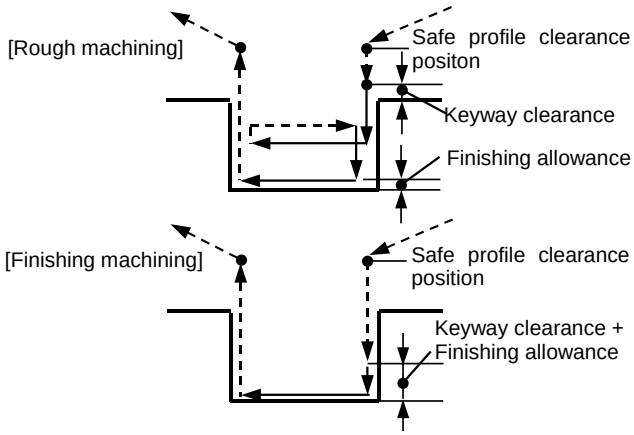
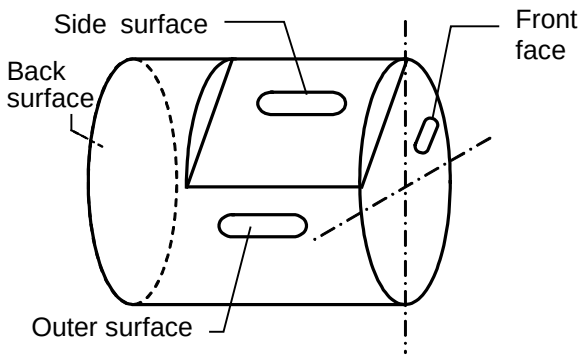
(1) Keyway Cutting Screen

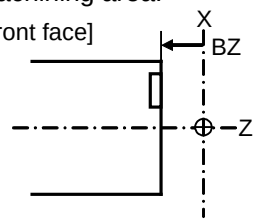
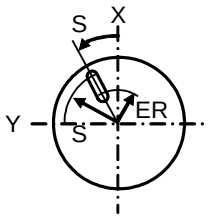
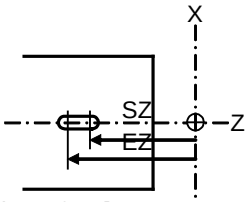
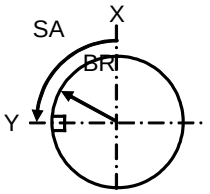
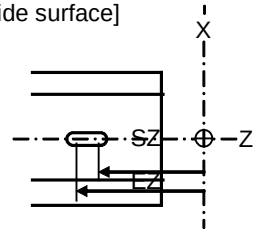
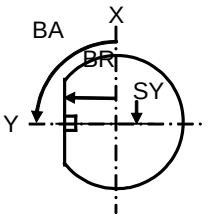
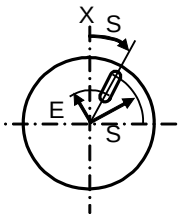
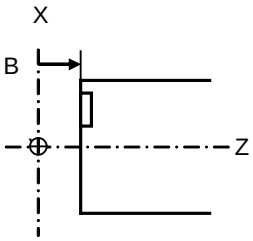
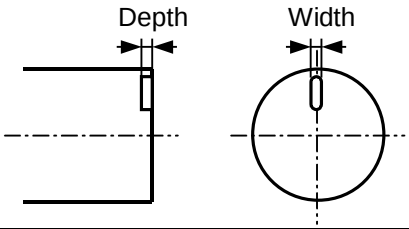
The parameters for the keyway cutting are input on this screen.

Screen layout

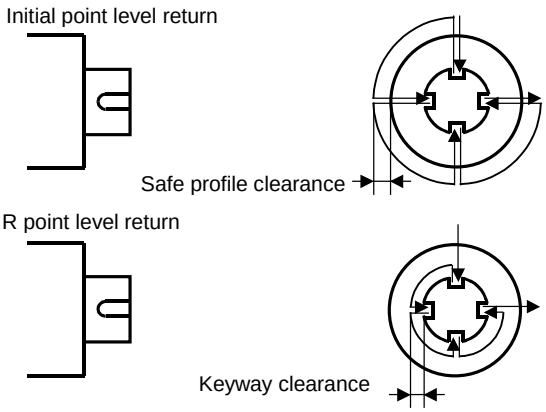
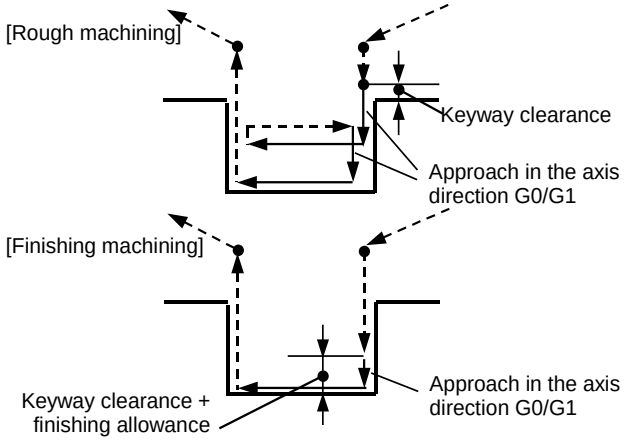


Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:701 to 799 (Default: 701) $2: 1701 to 1799 (Default: 1701)
2	CYCLE	Input the machining method. <1: ROUGH (rough machining)> Cuts into the keyway shape gradually. Leaves the finishing allowance in respect to the keyway shape. <2: FIN (finishing machining)> Machines the keyway shape in one cycle.  	1,2 (Default: 1)
3	PARTS	Input the machining area. <1: FACE> Machines the front face of workpiece. <2: OUT> Machines the outer surface of workpiece. <3: SIDE> Machines the side surface of workpiece. <4: BACK> Machines the back surface of the workpiece. (Note 1) Y-axis specifications are required for the side cutting. (Note 2) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".  	1 to 4 (Default: 1)

No.	Display item	Details	Setting range
4	BASE PLANE BZ BASE PLANE BR BASE PLANE BA	Set the machining base plane in respect to the machining area. [Front face]   [Outer surface]   [Side surface]   [Back surface]   BASE PLANE BZ/BR are changed each other according to the machining area. BASE PLANE BA is set only for the side cutting.	Base plane BZ -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch Base plane BR 0.001 to 99999.999mm 0.0001 to 9999.9999inch Base plane BA -359.999 to 360.000°
5	WIDTH W	Input the width and depth of the keyway. An error will occur when the keyway width is smaller than the tool width. Machining path is determined as follows depending on whether Y-axis specifications are provided or not. Y-axis specifications provided: When the keyway width exceeds the tool width, cutting is performed with shifting the tool on Y axis. No Y-axis specifications provided: Cutting is only executed on the center line of the keyway.  	0.001 to 999.999mm 0.0001 to 99.9999inch
6	DEPTH H		

No.	Display item	Details	Setting range
7	FIN ALLOW	Set the finishing allowance in the depth of the keyway. Rough machining leaves the finishing allowance in respect to the bottom of the keyway.	0.000 to 999.999mm 0.0000 to 99.9999inch
8	CUT AMOUNT	Input the cutting depth amount of the keyway for the rough machining.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
9	START ANGLE SA SHIFT POS SY	Refer to the figure of base plane. START ANGLE SA and SHIFT POS SY are switched each other according to the machining area. START RAD SR and START POS SZ are switched each other according to the machining area. END RADIUS ER and END POS EZ are switched each other according to the machining area.	Start position BZ, end position EZ -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch Start radius SR, end radius ER, shift position SY 0.001 to 99999.999mm 0.0001 to 9999.9999inch Start angle SA -359.999 to 360.000°
10	START RAD SR START POS SZ		
11	END RAD ER END POS EZ		
12	NUM OF KEYWAY	Input the number of keyways.	1 to 9 (Default: 1)
13	PITCH	Input the pitch if the number of keyways is 2 or more.	Front face, outer surface -359.999 to 360.000° Side surface 0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
14	RETURN POINT	When the number of keyways is 2 or more, select the height of the tool movement to the next hole position. 1: Initial point level return 2: R point level return 	1,2 (Default: 1)
15	C-AXIS CLAMP	Select whether to clamp C axis or not in the machining. Select “Clamp C axis” for heavy load machining. 0: Not clamp C axis 1: Clamp C axis	0,1 (Default: 0)
16	APPROACH IN AXIS DIR	When the positioning is performed, the tool moves to the position set in the K-WAY CLEARANCE with rapid traverse. Set “rapid traverse” or “cutting feed” to be performed in the cutting from that set position to the axis direction. 1: RAPID (G00) 2: CUT (G01) 	1,2 (Default: 1)
17	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, the tool No. registered in the tool file is automatically set.	0 to 99999999
18	DIA	Input the tool diameter. When tool registration No. is specified, the tool diameter registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
19	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to9999 m/min 1 to 9999 feet/min

No.	Display item	Details	Setting range
20	FEED RATE F1	Input the feedrate in the width direction of the keyway. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
21	FEED RATE F2	Input the feedrate in the depth direction of the keyway. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
22	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

Menus

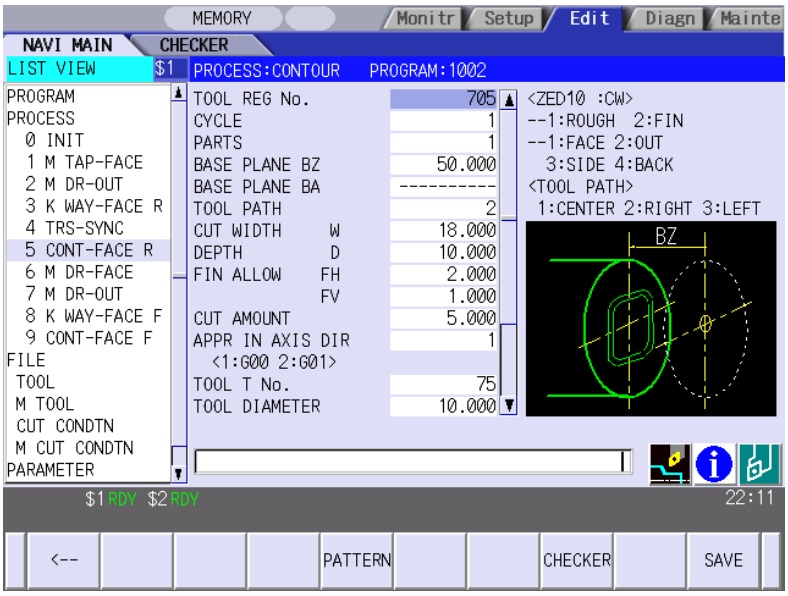
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.15 Contour Cutting Screen

(1) Contour Cutting Screen

The parameters for the contour cutting are input on this screen.

Screen layout

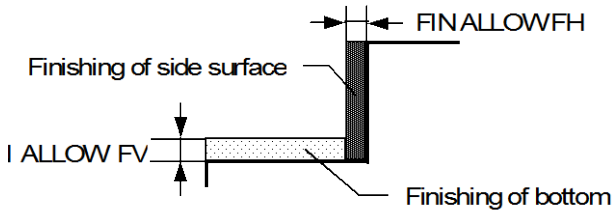
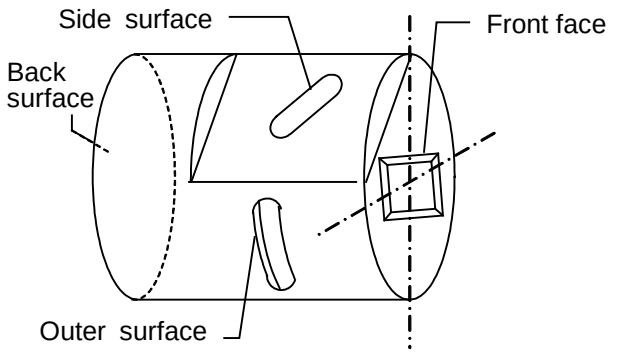


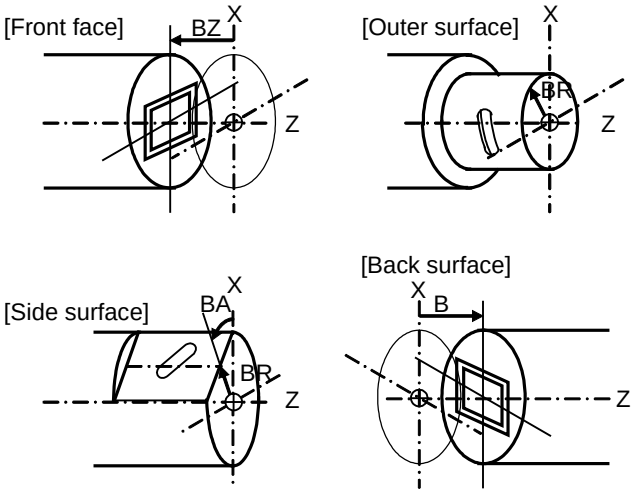
Screen display items

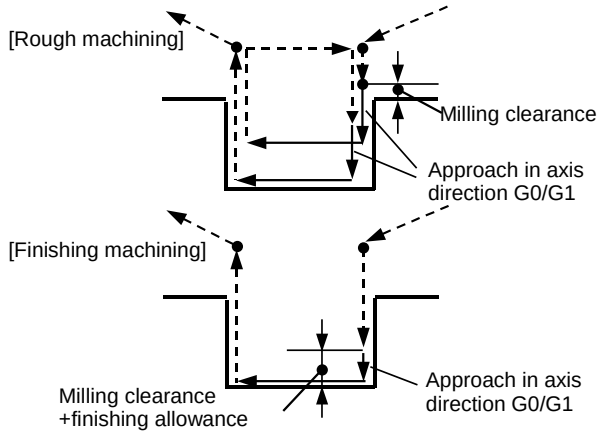
No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	$1:701 to 799 (Default: 701) $2: 1701 to 1799 (Default: 1701)

(Continued to the next page)

(Continued from the last page)

No.	Display item	Details	Setting range
2	CYCLE	Input the machining method. <1: ROUGH (rough machining)> In the axis direction: Machines with the tool cutting into the shape. FIN ALLOW FV is left. In the diameter direction: Machines with shifting the tool. FIN ALLOW FH is left. <2: FIN (finishing machining)> Finishes the bottom first and then the side surface.  [Finishing the bottom] In the axis direction: Machines the FIN ALLOW FV in one cycle. In the diameter direction: Machines with shifting the tool. FIN ALLOW FH is left. Finishing of bottom is not executed when FIN ALLOW FV is set to 0. [Finishing the side surface] In the axis direction: Machines with the tool cutting into the FIN ALLOW FH. In the diameter direction: Machines the FIN ALLOW FH in one cycle. Finishing of side surface cannot be executed when FIN ALLOW FH is set to 0	1,2 (Default: 1)
3	PARTS	Input the machining area. <1: FACE> Machines the front face of workpiece. <2: OUT> Machines the outer surface of workpiece. <3: SIDE> Machines the side surface of workpiece. <4: BACK> Machines the back surface of the workpiece. (Note 1) Y-axis specifications are required for the side cutting. (Note 2) BACK is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".  	1 to 4 (Default: 1)

No.	Display item	Details	Setting range
4	BASE PLANE BZ BASE PLANE BR BASE PLANE BA	Set the machining base plane in respect to the machining area.  BASE PLANE BZ/BR are switched according to the machining area. BASE PLANE BA is set only for the side cutting.	Base plane BZ -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch Base plane BR 0.001 to 99999.999mm 0.0001 to 9999.9999inch Base plane BA -359.999 to 360.000°
5	TOOL PATH	Input the tool path of the contour shape. <1: CENTER> Machines the center of the contour shape. <2: RIGHT> Machines the right side of the contour shape. <3: LEFT> Machines the left side of the contour shape.	1 to 3 (Default: 1)
6	WIDTH W	Input the machining width and depth of the contour shape. An error occurs when the machining width is smaller than the tool width. Machining width cannot be input when CENTER is set as tool path.	0.001 to 999.999mm 0.0001 to 99.9999inch
7	DEPTH D		
8	FIN ALLOW FH FIN ALLOW FV	Set the finishing allowance in the tool diameter direction and in the tool axis direction. FIN ALLOW FH cannot be input when CENTER is set as tool path.	0.000 to 999.999mm 0.0000 to 99.9999inch
9	CUT AMOUNT	Input the cutting amount to the tool axis direction. This is not available when CENTER is set as tool path for finishing machining.	0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
10	APPROACH IN AXIS DIR	When the positioning is performed, the tool moves to the position set in the E-ML CLEARANCE with rapid traverse. Set “rapid traverse” or “cutting feed” to be performed in the cutting from that set position to the axis direction. 1: RAPID (G00) 2: CUT (G01)  	1,2 (Default: 1)
11	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, the tool No. registered in the tool file is automatically set.	0 to 99999999
12	DIA	Input the tool diameter. When tool registration No. is specified, the tool diameter registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
13	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
14	FEED RATE F1	Input the feedrate in the width direction of the groove. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
15	FEED RATE F2	Input the feedrate in the depth direction of the groove. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
16	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

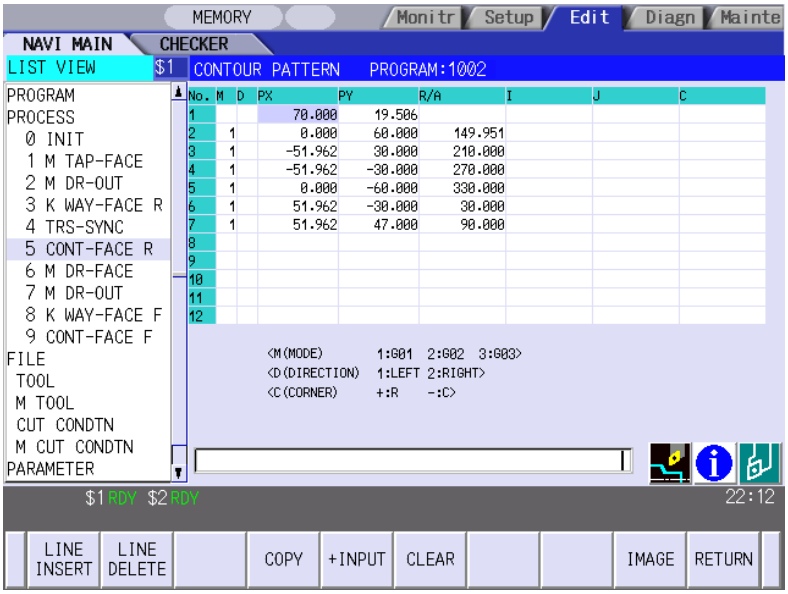
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	Displays the machining pattern selection screen.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

(2) Contour cutting pattern screen

The parameters for the contour cutting pattern are input on this screen.
When the [PATTERN] menu is pressed on the contour cutting screen, this screen is displayed.

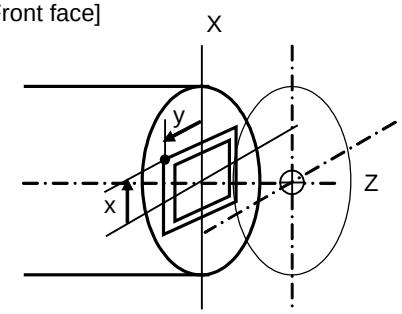
Screen layout



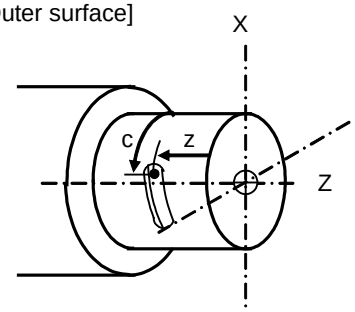
Input coordinate system of contour machining shape

Machining area	Input coordinate system	Remarks
Front face	X-Y	
Outer surface	C-Z, Y-Z	The input coordinate system can be changed with menu keys.
Side surface	Y-Z	
Back surface	X-Y	The sign of the Y coordinate is reversed from that of the front face.

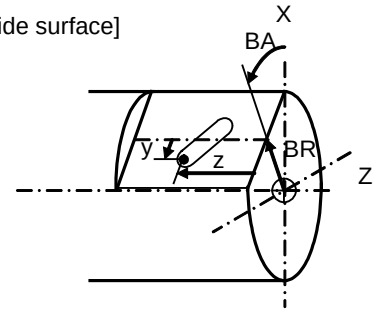
[Front face]



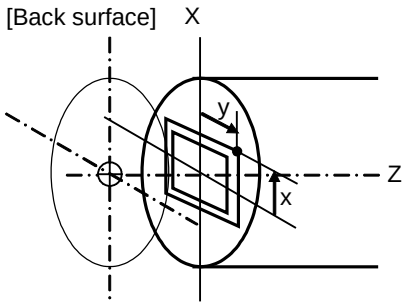
[Outer surface]



[Side surface]

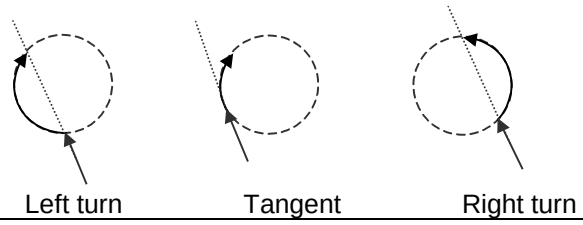
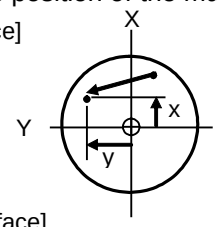


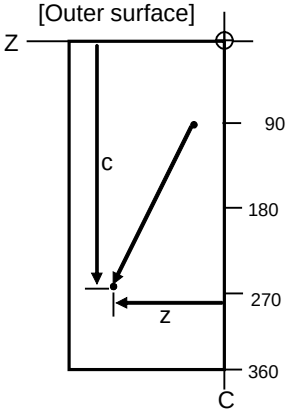
[Back surface]

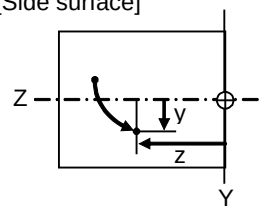


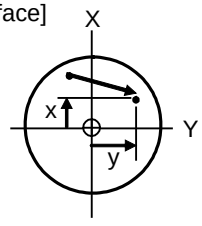
Screen display items

<table>
<tr><th>No.</th><th colspan="2">Display item</th><th>Details</th><th>Setting range</th></tr>
<tr><td>Shape</td><td colspan="2"></td><td></td><td>1 to 35</td></tr>
<tr><td rowspan="3">1</td><td>1</td><td>M</td><td>Input the shape.
<1>The linear (G01) machining is performed.
<2>The CW arc (G02) machining is performed.
<3>The CCW arc (G03) machining is performed.
(Note) This cannot be omitted.</td><td>1 to 3</td></tr>
<tr><td>2</td><td>D</td><td>Input right turn or left turn in respect to the vector at the end of the previous shape.
1: Left turn 2: Right turn

(Note 1) When nothing is input, it is regarded as “contacting”.
(Note 2) This data, although omittable, must be input when the end points X,Y of the previous line are uncertain.
 
</td><td>1,2</td></tr>
<tr><td>3</td><td>FACE:
PX,PY
OUT:
PC,PZ
PY,PZ
SIDE:
PY,PZ
BACK:
PX,PY</td><td>Input the position of the machining end point.
[Front face]
 

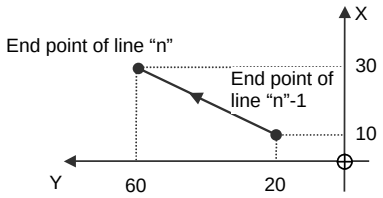
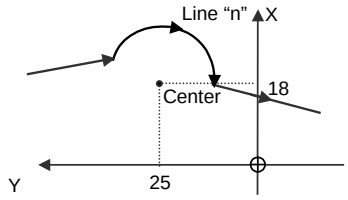
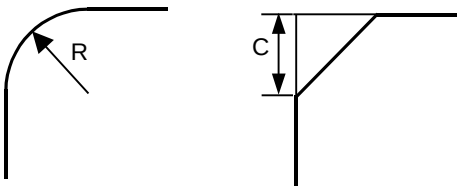
[Outer surface]
 

[Side surface]
 

[Back surface]
 

(Note 1) Input the end point PX, PY and PZ with radius value.
(Note 2) The input coordinate system C-Z and Y-Z can be changed each other when the machining area is set to outer surface.
(Note 3) This must be input if the line is the last one. This can be omitted unless it is the last one.
(Note 4) This must be input if the corner shape dimensions are set in the previous line.</td><td>X,Y,Z:
-99999.999 to 99999.999mm
-9999.9999 to 9999.9999inch

C:
-99999.999° to 99999.000°</td></tr>
</table>

No.	Display item		Details	Setting range
	4	R/A	• Input the radius when the shape is arc. Positive value: Arc command (less than 180°) Negative value: Arc command (more than 180°) • Input the angle when the shape is line. **(Note 1)** This must be input if the shape is arc. **(Note 2)** This data turns invalid when setting the position X,Y (C,Z/Y,Z) or vector I,J for the line shape. **(Note 3)** The radius R is specified by length even when machining outer surface.	Radius: -999999.999 to -0.001mm, 0.001 to 999999.999mm -99999.9999 to -0.0001inch, 0.0001 to 99999.9999 inch Angle: -359.999 to 360.000
	5	I J	• Input the gradient (vector) when the shape is line.  • Input the position of arc center when the shape is arc.  **(Note 1)** When either I or J is input in the arc shape, the other is regarded as 0. **(Note 2)** This data is invalid when setting the position X,Y (C,Z/Y,Z) or angle in the line shape.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
	6	C	Input the corner size. Positive value: Corner R Negative value: Corner C  **(Note 1)** When corner dimensions are specified, the end points X,Y (C,Z/Y,Z) are entered for the following line in principle	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch

(Note) The first point is a machining start point, so only the positions X,Y (C,Z/Y,Z) can be input.

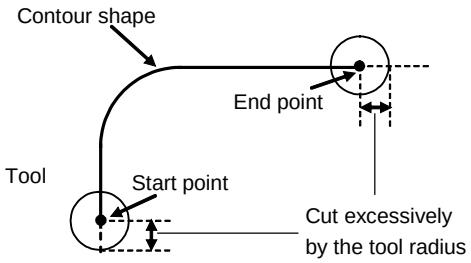
Menus

No.	Menu	Details
1	LINE INSERT	Inserts the shape data before the cursor position. **(Note)** This menu is not available when the cursor is at No.1 (machining start point).
2	LINE DELETE	Deletes the shape data at the cursor position. **(Note)** This menu is not available when the cursor is at No.1 (machining start point).
4	COPY	Copies the previous line data above cursor to the setting area.
5	+INPUT	Adds the previous line data above cursor to the setting data and inputs the value to the setting area. **(Note)** This is valid only when inputting the position X,Y (C,Z/Y,Z).
6	CLEAR	Clears the data at the cursor position.
8	C-Z INPUT	Changes the input coordinate system to C-Z. This menu is highlighted when the input coordinate system has been set to C-Z. This is available only when the machining area is set to outer surface.
9	Y-Z INPUT	Changes the input coordinate system to Y-Z. This menu is highlighted when the input coordinate system has been set to Y-Z. This is available only when the machining area is set to outer surface.
10	RETURN	Returns to the contour cutting screen.

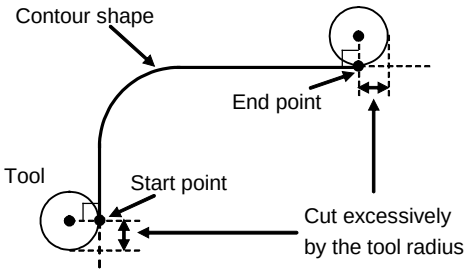
(3) Precautions for contour shape

A tool travels for the contour machining as follows. Thus the resulting cut shape is greater by the tool radius at the start and the end points.

[When the tool path is center]
The tool center travels from the start point to the end point of the contour.



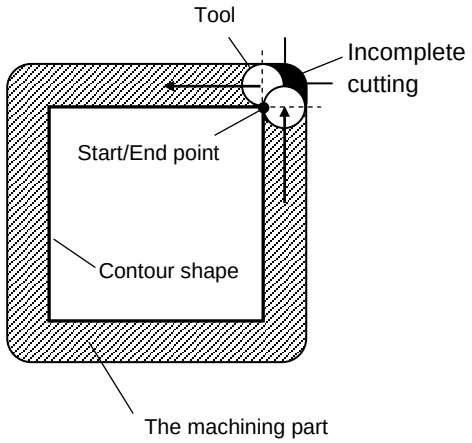
[When the tool path is the left, or the right]
The tool center travels in a position that is shifted by the tool radius in the vertical direction against the contour.



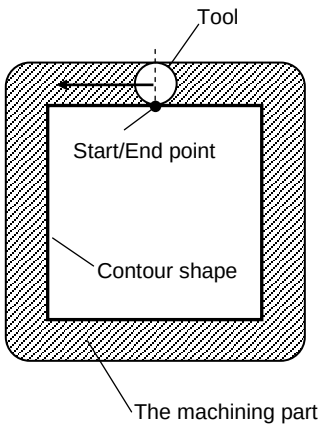
If the following three conditions are met, incomplete or excessive cutting may be caused, resulting in incorrect cutting:
- The tool path is left or right,
- The machining shape is an enclosed shape, and
- The start and end points are at a corner.

In that case, input a contour shape whose start and end points are in the middle of a side.

[When specifying the start and end points at the corner]



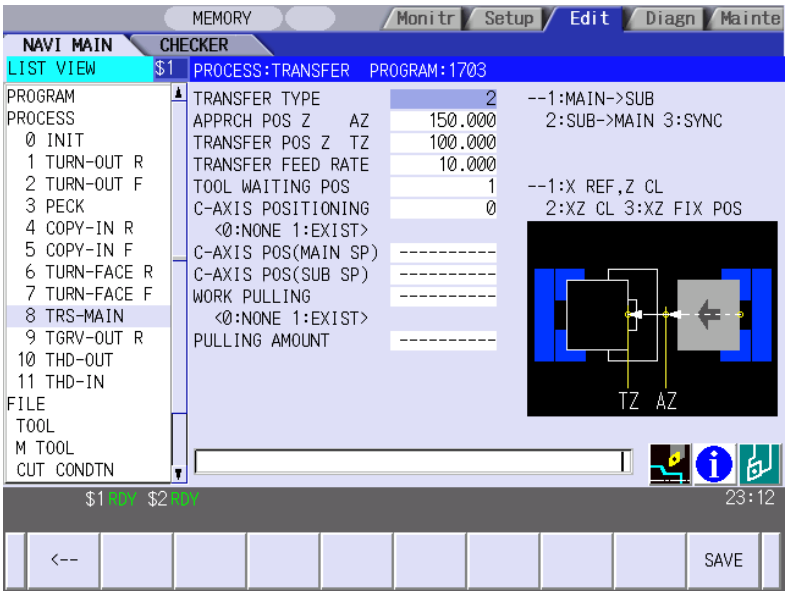
[When specifying the start and end points in the middle of a side]



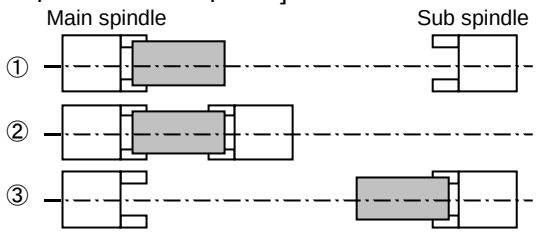
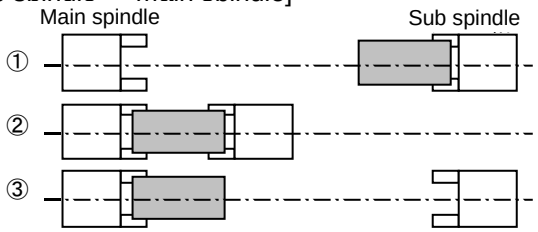
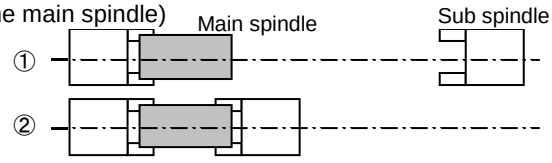
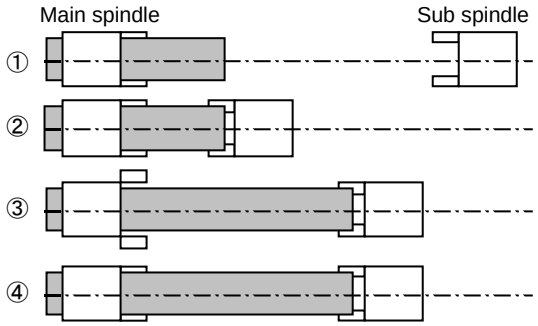
4.3.16 Transfer Screen

The parameters for the transfer process between the main and sub spindles and those for the workpiece transfer from the sub spindle to the parts catcher are input on this screen.
When there is a transfer process, the other processes are regarded as the transferred spindle side processes until another transfer process comes up.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TRANSFER TYPE	Select the transfer direction. 1: MAIN -> SUB (From main spindle to sub spindle) 2: SUB -> MAIN (From sub spindle to main spindle) 3: SYNC (Spindle synchronization) [Main spindle -> sub spindle]  [Sub spindle -> main spindle]  [Spindle synchronization] Without workpiece pulling (example: synchronized from the main spindle)  With workpiece pulling  (Note) When switching from the spindle synchronization to the main spindle, select "SUB -> MAIN", and when switching from the spindle synchronization to the sub spindle, select "MAIN -> SUB".	1 to 3 (Default: 1)
2	APPRCH POS Z AZ	Set the Z coordinate of the position in which the sub spindle approaches the main spindle at a rapid traverse rate for the transfer process. Set the position relative to the machine coordinate system.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch

No.	Display item	Details	Setting range
3	TRANSFER POS Z TZ	Set the Z coordinate of the transfer position relative to the machine coordinate system.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
4	TRANSFER FEED RATE	Set the movement feedrate from the approach position to the transfer position.	0.001 to 9999.999 mm/min 0.0001 to 999.9999 inch/min
5	TOOL WAITING POS	Select the waiting position of the tool for the transfer. 1: X REF, Z CL (X axis - reference position, Z axis - tool turning clearance position) 2: XZ CL (X and Z axes - tool turning clearance position) 3: XZ FIX POS (X and Z axes - tool fixed point return position)	1 to 3
6	C-AXIS POSITIONING	When transferring, select whether to perform the C-axis positioning for the both spindles. 0: NONE 1: EXIST	0, 1 (Default: 0)
7	C-AXIS POS (MAIN SP)	Set the C-axis position of the main spindle when transferring. (Note) This is available only when selecting "1: EXIST" at the C-AXIS POSITIONING.	-359.999 to 360.000°
8	C-AXIS POS (SUB SP)	Set the C-axis position of the sub spindle when transferring. (Note) This is available only when selecting "1: EXIST" at the C-AXIS POSITIONING.	-359.999 to 360.000°
9	WORK PULLING	Select whether to perform the workpiece pulling. 0: NONE 1: EXIST (Note 1) This is available only when selecting "3: SYNC" at the TRANSFER TYPE. (Note 2) This works only when the current machining spindle is the main or the spindle synchronization.	0, 1 (Default: 0)
10	PULLING AMOUNT	Set the pulling amount of the workpiece. (Note) This is available only when selecting "1: EXIST" at the WORK PULLING.	0.001 to 99999.999 mm 0.0001 to 9999.9999 inch

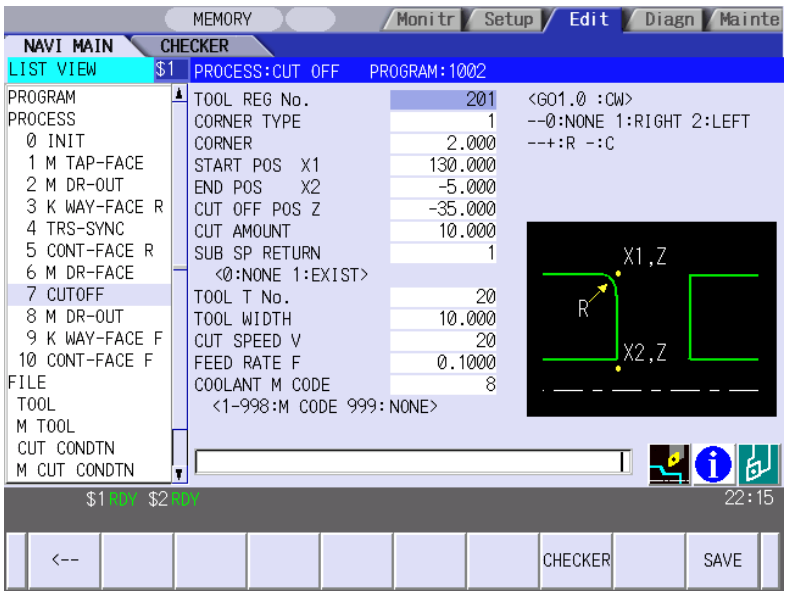
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

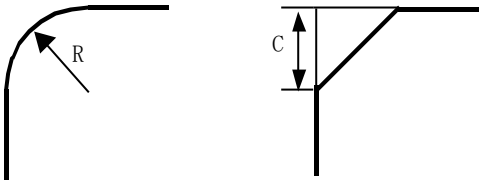
4.3.17 Cut Off Screen

The parameters for the cut off are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used for the cut off. Use the No. registered in the tool file. (Note) The tool for cutting off is used the grooving tools in the tool file.	$1: 201 to 250 $2: 1201 to 1250
2	CORNER TYPE	Specify the corner type 0: NONE (no corners) 1: RIGHT (right corner) 2: LEFT (left corner)	0 to 2 (Default: 0)
3	CORNER	Input the corner size. A positive value: corner R, a negative value: corner C  (Note) This is available only when selecting "1: RIGHT" or "2: LEFT" at the CORNER TYPE.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
4	START POS X1	Set the start position X for the cut off machining (cut off start diameter).	0.001 to 99999.999 mm 0.0001 to 9999.9999 inch
5	END POS X2	Set the end position X for the cut off machining (cut off end diameter). (Note) Set the position including the excessive amount of the machining.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch

No.	Display item	Details	Setting range
6	CUT OFF POS Z	Set the start position Z for the cut off machining. (Note) Set the position including the tool width.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
7	CUT AMOUNT	Input the cut amount.	0.001 to 99999.999 mm 0.0001 to 9999.9999 inch
8	SUB SP RETURN	Select whether to return the sub spindle to the original position after the cut off. (Note 1) This is available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST". (Note 2) When you select "1: EXIST" for this item, the sub spindle is selected as the machining spindle after the cut off. (Note 3) The machining spindle works only in "spindle synchronization", but does not work in the other cases.	0, 1 (Default: 0)
9	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	0 to 99999999
10	TOOL WIDTH	Input the tool width of the respective tool. When tool registration No. is specified, tool width registered in the tool file is automatically set.	0.001 to 999.999 mm 0.0001 to 99.9999 inch
11	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
12	FEED RATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
13	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set.	1 to 999

Menus

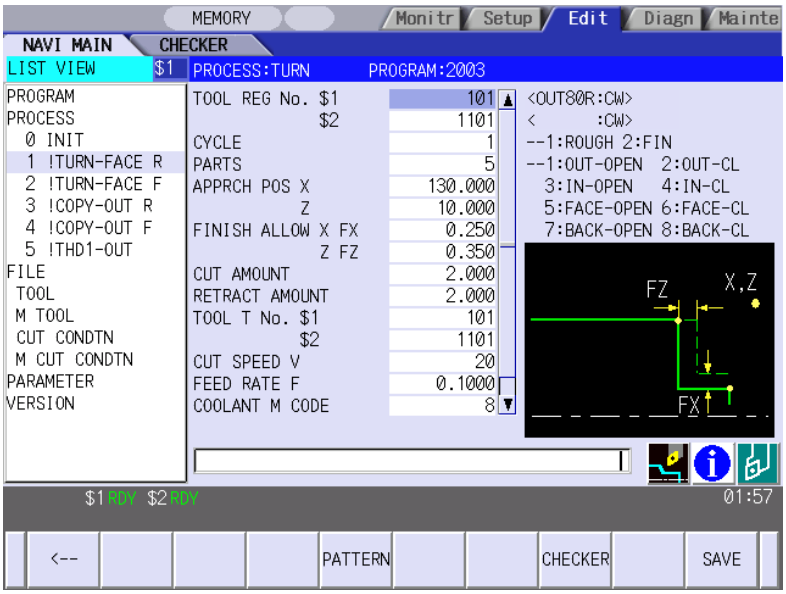
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

4.3.18 Balance Cut (Turn) Screen

(1) Balance Cut (Turn) Screen

The parameters for the balance cut (turn) are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No. $1	Input the registration No. of the tool to be used at the 1st part system ($1) or the 2nd part system ($2). Use the No. registered in the tool file.	101 to 150 601 to 650 (Default: 101)
2	TOOL REG No. $2		1101 to 1150 1601 to 1650 (Default: 1101)
3	CYCLE	Input the machining method. <1: ROUGH (rough machining)> Cuts into the cutting area gradually. Leaves the finishing allowance for the cutting shape. <2: FIN (finishing machining)> Machines the cutting shape in one cycle.	1, 2 (Default: 1)

No.	Display item	Details	Setting range
4	PARTS	Input the machining area. <1: OUT-OPEN> Machines the outer diameter area from the front face of workpiece. <2: OUT-CL> Machines the outer diameter area from the halfway of workpiece. <3: IN-OPEN> Machines the inner diameter area from the front face of workpiece. <4: IN-CL> Machines inner area from the halfway of workpiece. <5: FACE-OPEN> Machines the front face of workpiece. <6: FACE-CL> Machines the front face from the halfway of workpiece. <7: BACK-OPEN> Machines the back side of workpiece. <8: BACK-CL> Machines the back side of workpiece from the halfway of workpiece. (Note 1) BACK-OPEN and BACK-CL are available only when the parameter "1001 SUB SPINDLE SPEC" is "1: EXIST".	1 to 8 (Default: 1)
5	APPRCH POS X	Input the approach point. After machining, the tool returns to the approach point.	-99999.999 to 99999.999 mm
6	APPRCH POS Z		-9999.9999 to 9999.9999 inch
7	FINISH ALLOW X FX	Input the finishing allowance for the rough machining. Input both FX and FZ with radius value.	0.000 to 99999.999 mm
8	FINISH ALLOW Z FZ		0.0000 to 9999.9999 inch
9	CUT AMOUNT	Input the cut amount for the rough machining.	0.001 to 99.999 mm
10	RETRACT AMOUNT	Input the retract amount for the rough machining.	0.0001 to 9.9999 inch
11	TOOL T No. $1	Input the turret No. (or ATC No.) of the tool being set in the 1st part system or the 2nd part system, as well as the compensation No. When the tool registration No. $1 or $2 is specified, a tool No. registered in the tool file is automatically set.	0 to 99999999
12	TOOL T No. $2		0 to 99999999
13	CUT SPEED V	Input the cutting speed common to both part systems. When tool registration No. $1 is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file. The cut speed of the 1st part system is set.	1 to 9999 m/min 1 to 9999 feet/min
14	FEED RATE F	Input the feedrate common to both part systems. When tool registration No. $1 is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file. The feed rate of the 1st part system is set.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
15	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set. The coolant M code of the1st part system is set.	1 to 999

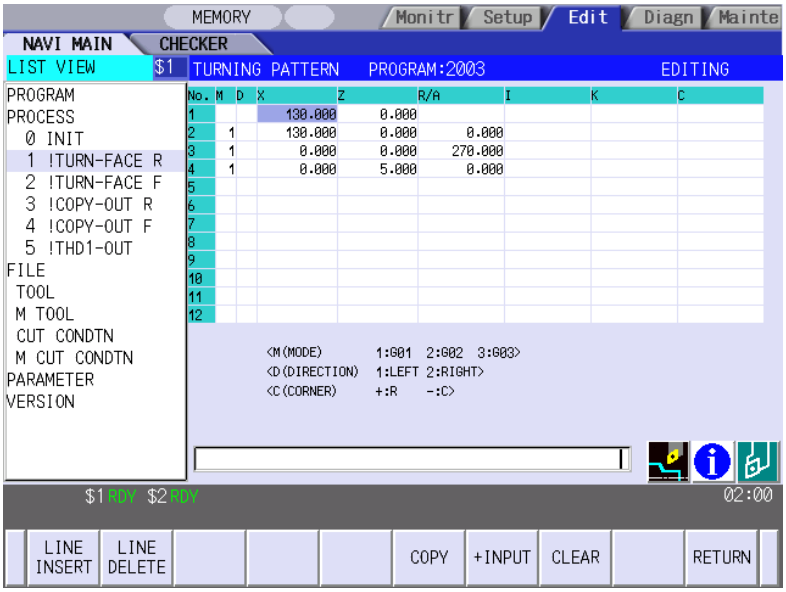
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	Machining pattern selection screen is displayed.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position. If illegal parameters are input in the pattern input screen, the screen name and error will be displayed.

(2) Balance Cut (Turn) pattern Screen

This screen is for entering the cutting shapes of balance cut (turn) process. The items to set through this screen are the same as of the turning pattern screen.

Screen layout



Screen display items

Refer to the section "4.3.6 Turning Screen (2) Turning pattern screen".

Menus

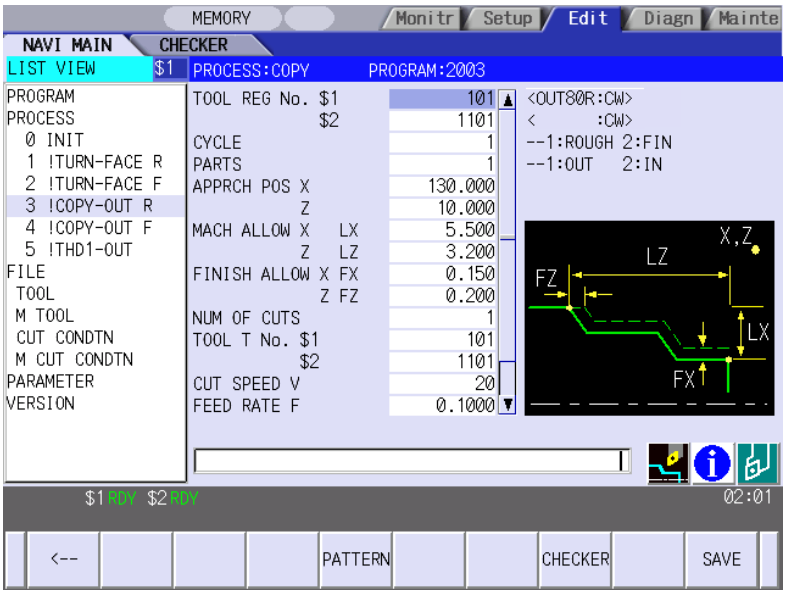
Refer to the section "4.3.6 Turning Screen (2) Turning pattern screen".

4.3.19 Balance Cut (Copy) Screen

(1) Balance Cut (Copy) Screen

The parameters for the balance cut (copy) are input on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No. $1	Input the registration No. of the tool to be used at the 1st part system ($1) or the 2nd part system ($2). Use the No. registered in the tool file.	101 to 150 601 to 650 (Default: 101)
2	TOOL REG No. $2		1101 to 1150 1601 to 1650 (Default: 1101)
3	CYCLE	Input the machining method. <1: ROUGH (rough machining)> Cuts into the cutting area gradually. Leaves the finishing allowance for the cutting shape. <2: FIN (finishing machining)> Machines the cutting shape in one cycle.	1, 2 (Default: 1)
4	PARTS	Input the machining area. <1:OUT (outer diameter)> Machine the outer diameter section of the workpiece. <2: IN (inner diameter)> Machine the inner diameter section of the workpiece.	1 to 2 (Default: 1)
5	APPRCH POS X	Input the approach point. After machining, the tool returns to the approach point.	-99999.999 to 99999.999 mm
6	APPRCH POS Z		-9999.9999 to 9999.9999 inch
7	MACH ALLOW X LX	Input the allowance in X axis direction with the radius value for the rough machining.	0.001 to 99999.999 mm
8	MACH ALLOW Z LZ	Input the allowance in Z axis direction for the rough machining.	0.0001 to 9999.9999 inch

No.	Display item	Details	Setting range
9	FINISH ALLOW X FX	Input the finishing allowance for the rough machining. Input both FX and FZ with radius value.	0.000 to 99999.999 mm
10	FINISH ALLOW Z FZ		0.0000 to 9999.9999 inch
11	NUM OF CUTS	Input the number of cuts for the rough machining.	1 to 99
12	TOOL T No. $1	Input the turret No. (or ATC No.) of the tool being set in the 1st part system or the 2nd part system, as well as the compensation No. When the tool registration No. $1 or $2 is specified, a tool No. registered in the tool file is automatically set.	0 to 99999999
13	TOOL T No. $2		0 to 99999999
14	CUT SPEED V	Input the cutting speed common to both part systems. When tool registration No. $1 is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file. The cut speed of the 1st part system is set.	1 to 9999 m/min 1 to 9999 feet/min
15	FEED RATE F	Input the feedrate common to both part systems. When tool registration No. $1 is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file. The feed rate of the 1st part system is set.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
16	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set. The coolant M code of the1st part system is set.	1 to 999

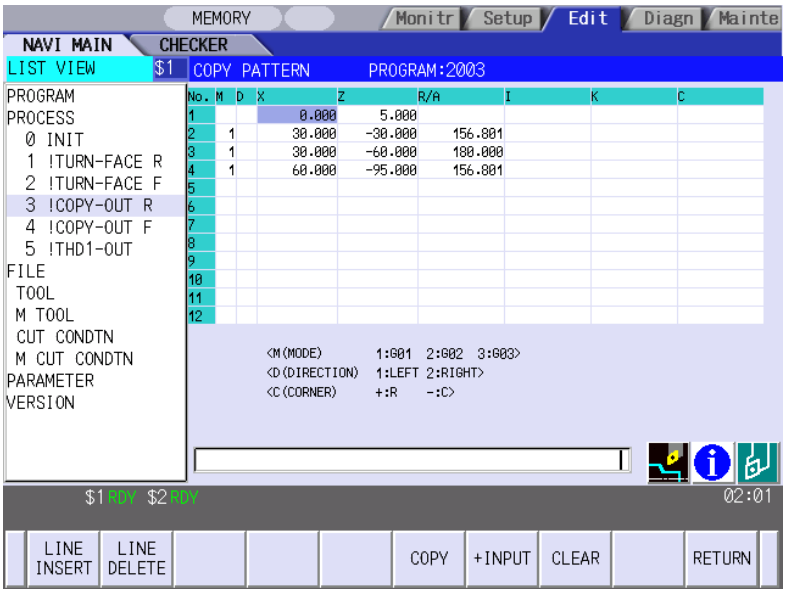
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
5	PATTERN	Machining pattern selection screen is displayed.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position. If illegal parameters are input in the pattern input screen, the screen name and error will be displayed.

(2) Balance Cut (Copy) Pattern Screen

This screen is for entering the cutting shapes of balance cut (copy) process. The items to set through this screen are the same as of the copy cutting pattern screen.

Screen layout



Screen display items

Refer to the section "4.3.7 Copy cutting screen (2) Copy cutting pattern screen".

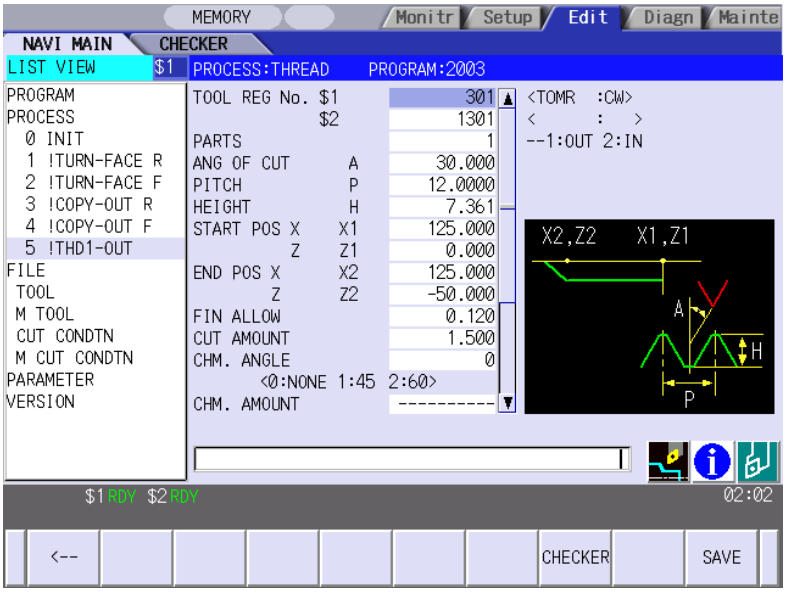
Menus

Refer to the section "4.3.7 Copy cutting screen (2) Copy cutting pattern screen".

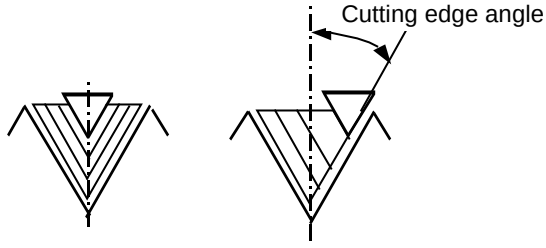
4.3.20 Two-part System Simultaneous Thread Cutting (identical screw) Screen

The parameters for the two-part system simultaneous thread cutting (identical screw) are input on this screen.

Screen layout

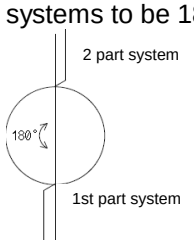


Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No. $1	Input the registration No. of the tool to be used at the 1st part system ($1) or the 2nd part system ($2). Use the No. registered in the tool file.	301 to 350 (Default: 301)
2	TOOL REG No. $2		1301 to 1350 (Default: 1301)
3	PARTS	Input the machining area. <1: OUT (outer diameter)> Thread the outer diameter area of the workpiece. <2: IN (inner diameter)> Thread the inner diameter area of the workpiece.	1 to 2 (Default: 1)
4	ANG OF CUT A	Input the cutting edge angle for the rough machining.  	0.000 to 60.000°

<table>
<tr><th>No.</th><th>Display item</th><th>Details</th><th>Setting range</th></tr>
<tr><td>5</td><td>PITCH P</td><td>Input the screw pitch.</td><td>0.0001 to 999.9999 mm
0.00001 to 99.99999 inch</td></tr>
<tr><td>6</td><td>HEIGHT H</td><td>Input the thread height.
When selecting a thread type from the menu, thread height can be input automatically based on the pitch.<table><tr><td>M
METER</td><td>UN
UNIFY</td><td>W
WIT</td><td>PF PT PS
PIPING</td><td>NPT
PIPING</td><td>TM
TRAP.30°</td><td>TW
TRAP.29°</td></tr></table></td><td>0.001 to 999.999 mm
0.0001 to 9999.9999 inch</td></tr>
<tr><td>7</td><td>START POS X
X1</td><td>Input the X coordinate of the threading start point in the diameter value.</td><td rowspan="2">-99999.999 to 99999.999 mm
-9999.9999 to 9999.9999 inch</td></tr>
<tr><td>8</td><td>START POS Z
Z1</td><td>Input the Z coordinate of the threading start point.</td></tr>
<tr><td>9</td><td>END POS X
X2</td><td>Input the X coordinate of the threading end point in the diameter value.</td><td rowspan="2">-99999.999 to 99999.999 mm
-9999.9999 to 9999.9999 inch</td></tr>
<tr><td>10</td><td>END POS Z
Z2</td><td>Input the Z coordinate of the threading end point.</td></tr>
<tr><td>11</td><td>FIN ALLOW</td><td>Input the threading finishing allowance for the rough machining.
The number of finishing times for two-part system simultaneous thread cutting is fixed to one.</td><td>0.000 to 99999.999 mm
0.0000 to 9999.9999 inch</td></tr>
<tr><td>12</td><td>CUT AMOUNT</td><td>Input the cutting amount corresponding the respective methods below for the rough machining.

Input the initial cutting amount.
The n-th cutting amount (dn) is calculated by the following formula.
dn=d1(√n-√(n-1))
d1 = initial cutting amount</td><td>0.001 to 99999.999 mm
0.0001 to 9999.9999 inch</td></tr>
<tr><td>13</td><td>CHM. ANGLE</td><td>Input the chamfering angle.
0: No chamfering
1: 45°
2: 60°
Chamfering is not carried out when:
Thread angle + chamfering angle > 90°</td><td>0 to 2
(Default: 0)</td></tr>
<tr><td>14</td><td>CHM.
AMOUNT</td><td>Input the chamfering amount.
Chamfered section is machined as continuous thread.</td><td>0.1 to 9.9
(Number of threads)</td></tr>
<tr><td>15

16</td><td>THD START ANG. $1
THD START ANG. $2</td><td>Specify the shift angle of the thread cutting start point for the 1st part system and the 2nd part system.
When the cutters of the 1st and 2nd part systems are opposing each other at 180 degrees as illustrated below, set the difference of the thread cut start shift angles of the 1st and 2nd part systems to be 180 degrees.
 
Example) Thread cutting start angle $1: 0.
Thread cutting start angle $2: 180.</td><td>$1:0 to 359.999°
(Default: 0)
$2:0 to 359.999°
(Default: 180)</td></tr>
</table>

No.	Display item	Details	Setting range
17 18	TOOL T No. $1 TOOL T No. $2	Input the turret No. (or ATC No.) of the tool being set in the 1st part system or the 2nd part system, as well as the compensation No. When the tool registration No. $1 or $2 is specified, a tool No. registered in the tool file is automatically set.	0 to 99999999
19	CUT SPEED V	Input the cutting speed common to both part systems. When tool registration No. $1 is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file. The cut speed of the 1st part system is set.	1 to 9999 m/min 1 to 9999 feet/min
20	COOLANT M CODE	Input the tool coolant M code. When there is no coolant, input 999. When tool registration No. is specified, the coolant M code registered in the tool file is automatically set. The coolant M code of the1st part system is set.	1 to 999

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
8	CHECKER	Displays the checker screen. Select this to check the set data.
10	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that parameter position.

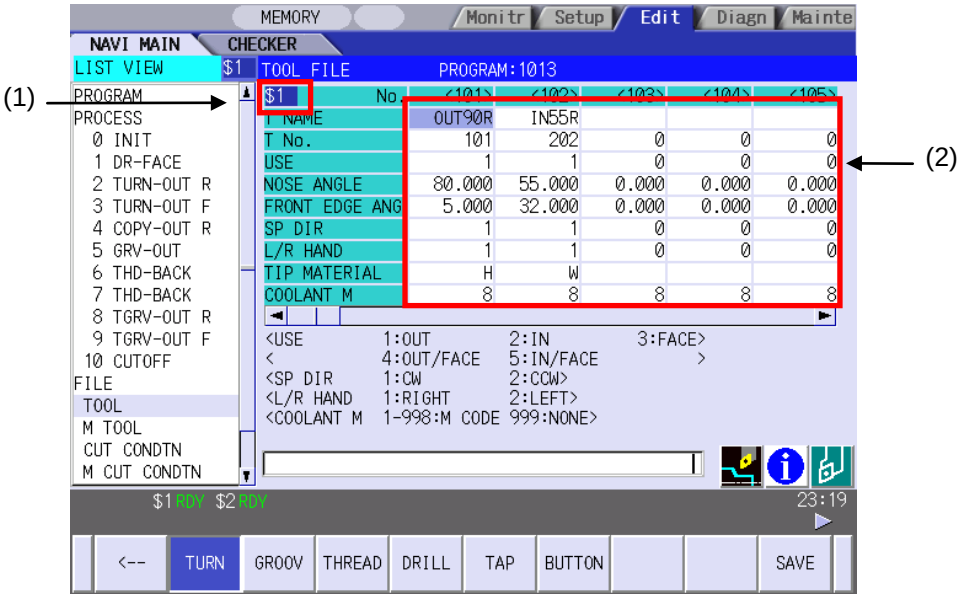
4.4 Screens Related to File Editing

4.4.1 Tool File Screen for Turning

The tool data for turning is registered on this screen. When [TOOL] is selected in the LIST VIEW area, this screen is displayed. The tool data for turning includes the followings.
Use the menu key to select one.

- TURNING TOOLS
- GROOVING TOOLS
- THREADING TOOLS
- DRILLS
- TAPS
- BUTTON TOOLS

Screen layout

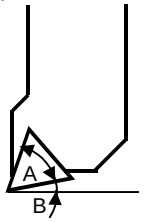


(Note) Menu for the currently selected tool is highlighted.

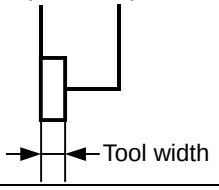
Screen display items

No.	Display item	Details	Remarks
1	Part system of tool data displayed	Indicates the part system number of the tool data displayed. (Note) This is available only when the 2-part system specification is "1: EXIST".	$1 $2
2	Tool file list	Indicates the tool data of the part system displayed on the screen. The tool No. of the 2nd part system is represented with the tool No. of the 1st part system plus 1000. Tool No. of 1st part system: 101 to 650 Tool No. of 2nd part system: 1101 to 1650 (Note) The tool data of the 2nd part system is available only when the 2-part system specification is "1: EXIST".	

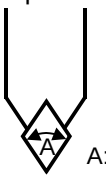
• TURNING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:101 to 150 $2:1101 to 1150
2	T NAME	Specify the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
4	USE	Input the application of the tool. 1: for outer diameter 2: for inner diameter 3: for face 4: for outer diameter/face 5: for inner diameter/face	1 to 5
5	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000°
6	FRONT EDGE ANG	Input the front edge angle of the tool.  A: Nose angle B: Front edge angle	0.001 to 180.000°
7	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
8	L/R HAND	Input left/right hand for the tool.	1: Right 2: Left
9	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
10	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

• GROOVING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:201 to 250 $2:1201 to 1250
2	T NAME	Input the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
4	USE	Input the application of the tool. 1: for outer diameter 2: for inner diameter 3: for face	1 to 3
5	TOOL WIDTH	Input the tip width.  	0.001 to 999.999mm 0.0001 to 99.9999inch
6	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
7	L/R HAND	Input left/right hand for the tool.	1: Right 2: Left
8	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
9	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

• THREADING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:301 to 350 $2:1301 to 1350
2	T NAME	Input the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
4	USE	Input the application of the tool. 1: for outer diameter 2: for inner diameter 3: for face	1 to 3
5	NOSE ANGLE	Input the tool nose angle.  A: Nose angle	0.001 to 180.000°
6	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
7	L/R HAND	Input left/right hand for the tool.	1: Right 2: Left
8	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
9	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

• DRILLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:401 to 450 $2:1401 to 1450
2	T NAME	Input the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
4	DIA	Input the tool radius.	0.001 to 999.999mm 0.0001 to 99.9999inch
5	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000˚
6	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
7	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
8	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

• TAPS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:501 to 550 $2:1501 to 1550
2	T NAME	Input the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
4	DIA	Input the tool radius.	0.001 to 999.999mm 0.0001 to 99.9999inch
5	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000°
6	PITCH	Input the pitch.	0.0001 to 999.9999mm/rev 0.00001 to 99.99999inch/rev
7	SP DIR	Input the spindle rotation direction.	1:CW 2:CCW
8	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
9	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

• BUTTON TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	$1:601 to 650 $2:1601 to 1650
2	T NAME	Input the tool name.	Max. 6 alphanumerical characters
3	T No.	Input the No. of the tool to be used. (T function code data output as the NC data)	1 to 999999
4	USE	Input the application of the tool. 1: for outer diameter 3. for face	1, 3
5	TIP DIA	Input the tip diameter.	0.001 to 999.999mm 0.001 to 99.9999inch
6	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
7	L/R HAND	Input left/right hand for the tool.	1: Right 2: Left
8	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters
9	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

Menus

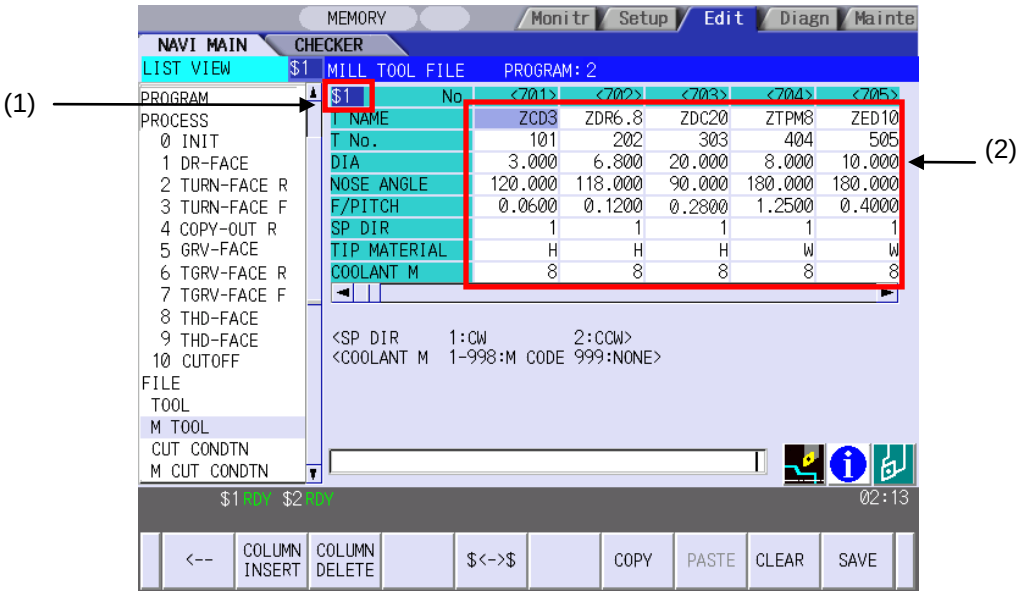
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	TURN	Displays the turning tool input screen.
3	GROOV	Displays the grooving tool input screen.
4	THREAD	Displays the threading tool input screen.
5	DRILL	Displays the drilling input screen.
6	TAP	Displays the tapping input screen.
7	BUTTON	Displays the button tool input screen.
10	SAVE	Saves the changes in the tool file.
12	COLUMN INSERT	Inserts the tool data (one column) in the column before the cursor.
13	COLUMN DELETE	Deletes the tool data (one column) in the column of the cursor position.
15	$<->$	Switches the part system of the tool data displayed. Pressing this menu key, the tool data of the next part system is displayed on the tool file list. The part system switches in this order of $1, $2 and $1. After the switch, the cursor moves to the retained position. The cursor position is retained while the tool data of the same process is displayed. (Note) This menu is available only when the 2-part system specification is "1: EXIST".
17	COPY	Copies the tool data contents of the cursor position (one column).
18	PASTE	Pastes the copied tool data contents (one column) in the column of the cursor position.
19	CLEAR	Clears the tool data (one column) in the column of the cursor position.

(Note) The cursor position is retained until the following actions are taken.
- Until activating the LIST VIEW (until pressing the menu [<-] or clicking the LIST VIEW.)

4.4.2 Tool File Screen for Milling

The tool data for milling is registered on this screen. When [M TOOL] is selected in the LIST VIEW area, this screen is displayed.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	Part system of tool data displayed	Indicates the part system number of the tool data displayed. (Note) This is available only when the 2-part system specification is "1: EXIST".	$1 $2
2	Tool file list	Indicates the tool data of the part system displayed on the screen. The tool No. of the 2nd part system is represented with the tool No. of the 1st part system plus 1000. Tool No. of 1st part system: 101 to 650 Tool No. of 2nd part system: 1101 to 1650 (Note) The tool data of the 2nd part system is available only when the 2-part system specification is "1: EXIST".	
3	No.	Tool registration No.	$1:701 to 799 $2:1701 to 1799
4	T NAME	Input the tool name.	Max. 6 alphanumeric characters
5	T NO.	Input the No. of the tool to be used. (T function code data output as the NC data)	0 to 99999999
6	DIA	Input the tool diameter.	0.001 to 999.999mm 0.0001 to 99.9999 inch
7	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000°

No.	Display item	Details	Setting range
8	F/PITCH	Input the feedrate of the tool. Input the pitch when performing tapping.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
9	SP DIR	Input the spindle rotation direction.	1:CW 2:CCW
10	TIP MATERIAL	Input the tip material.	Max. 4 alphanumeric characters
11	COOLANT M	Input the tool coolant M code. When there is no coolant, input 999.	1 to 999

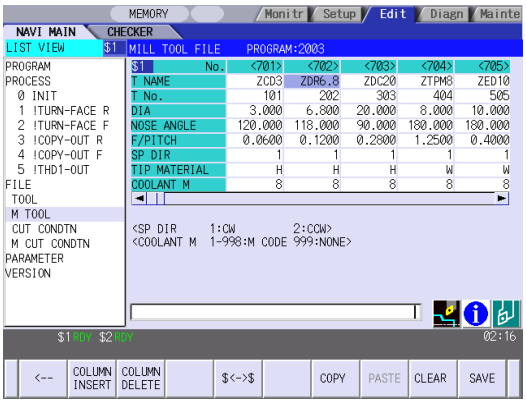
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	COLUMN INSERT	Inserts the tool data (one column) in the column before the cursor.
3	COLUMN DELETE	Deletes the tool data (one column) in the column of the cursor position.
5	$<->$	Switches the part system of the tool data displayed. Pressing this menu key, the tool data of the next part system is displayed on the tool file list. The part system switches in this order of $1, $2 and $1. After the switch, the cursor moves to the retained position. The cursor position is retained while the tool data of the same process is displayed. (Note) This menu is available only when the 2-part system specification is "1: EXIST".
7	COPY	Copies the tool data contents of the cursor position (one column).
8	PASTE	Pastes the copied tool data contents (one column) in the column of the cursor position.
9	CLEAR	Clears the tool data (one column) in the column of the cursor position.
10	SAVE	Saves the changes in the tool file.

(Note) The cursor position is retained until the following actions are taken.
- Until activating the LIST VIEW (until pressing the menu [<-] or clicking the LIST VIEW

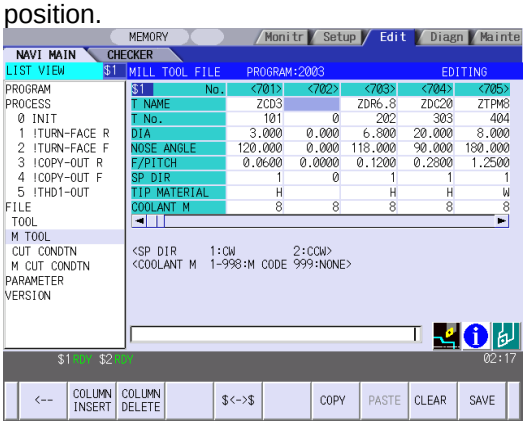
Operation method ([COLUMN INSERT])

(1) Display the tool file screen and move the cursor to the position where the column is to be inserted.



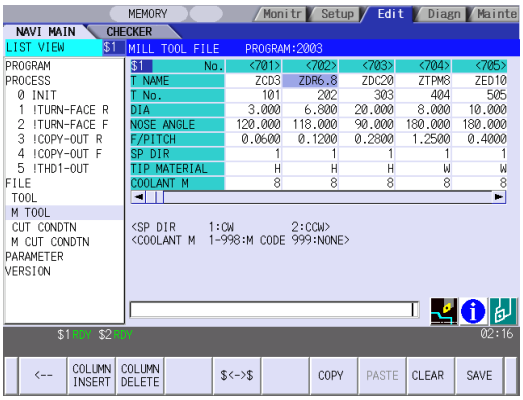
(2) Press the [COLUMN INSERT] menu key.

The blank column is inserted in the cursor position.



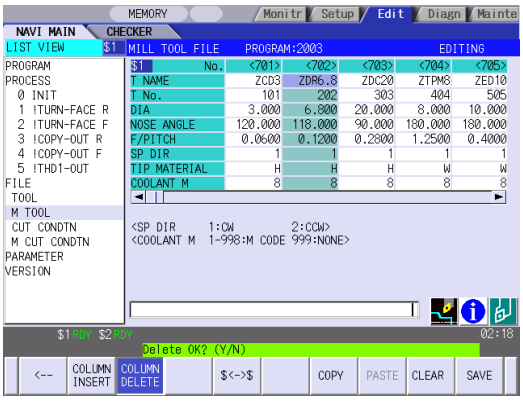
Operation method ([COLUMN DELETE])

(1) Display the tool file screen and move the cursor to the position where the column is to be deleted.



(2) Press the [COLUMN DELETE] menu key.

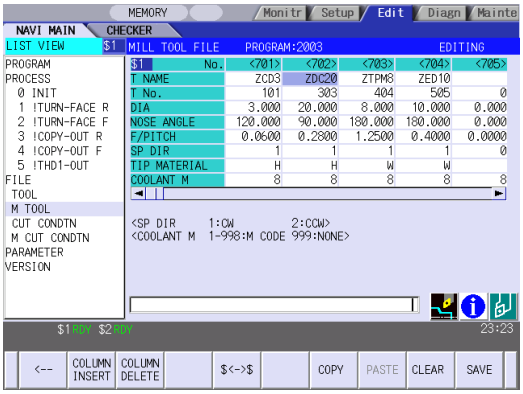
The column to be deleted is highlighted and the confirmation message appears.



(3) Press the [Y] key.

Press the [N] key in order not to erase the column.

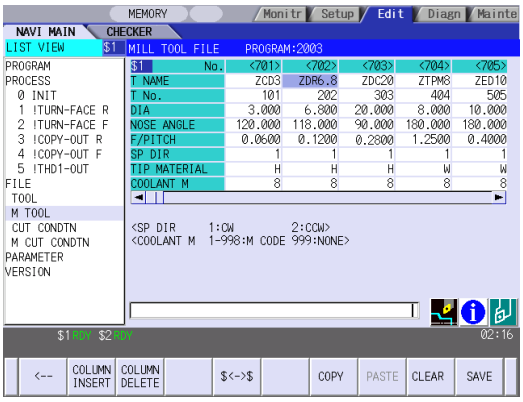
The data in the cursor position is deleted.



(Note) After the deletion, the data will be aligned.

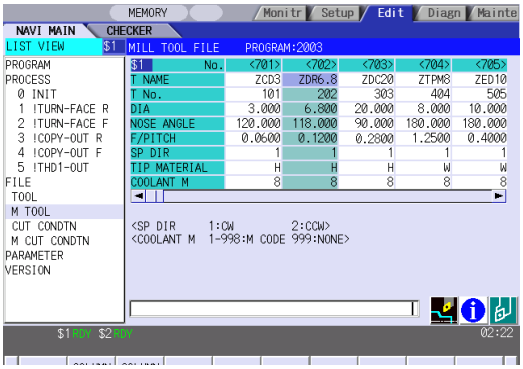
Operation method ([COPY], [PASTE])

(1) Display the tool file screen and move the cursor to the position where the column is to be copied.

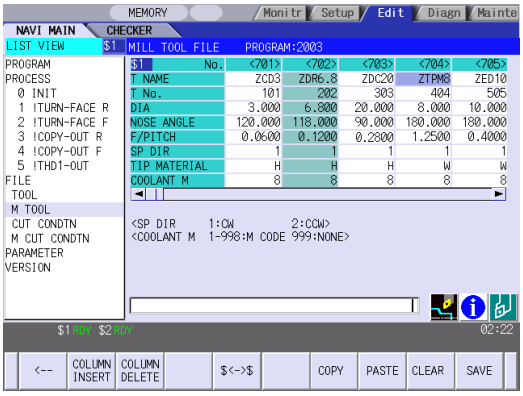


(2) Press the [COPY] menu key.

The background color of the copied column is highlighted.

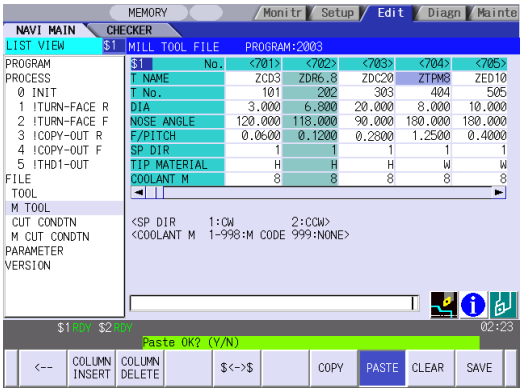


(3) Move the cursor to the position where the column is to be pasted.



(4) Press the [PASTE] menu key.

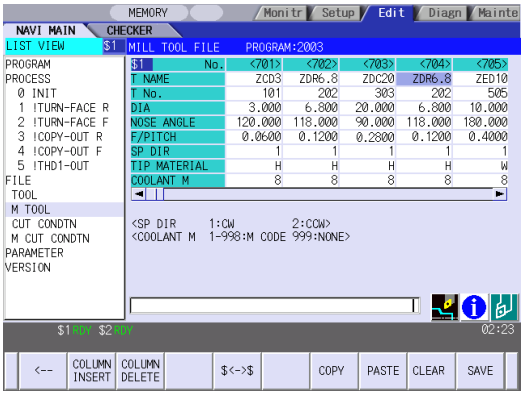
➡ The confirmation message appears.



(5) Press the [Y] key.

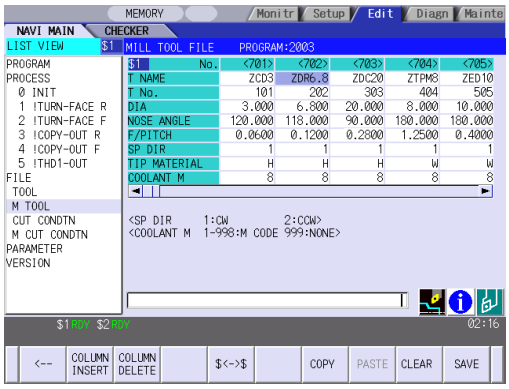
Press the [N] key in order not to paste the column.

➡ The data of the copied column is written to the cursor position.



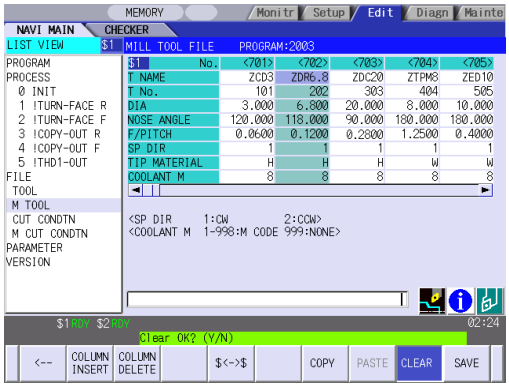
Operation method ([CLEAR])

(1) Display the tool file screen and move the cursor to the position where the column is to be cleared.



(2) Press the [CLEAR] menu key.

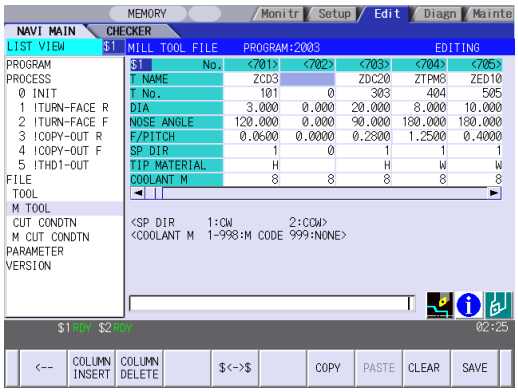
The column to be cleared is highlighted and the confirmation message appears.



(3) Press the [Y] key.

Press the [N] key in order not to clear.

The data of the cursor position column is cleared.

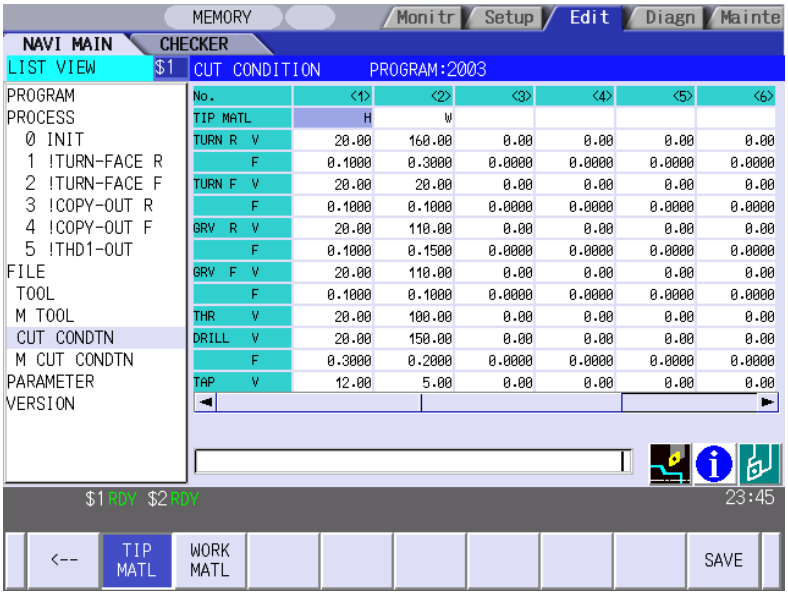


* After clearing, the data will not be aligned.

4.4.3 Cutting Condition File Screen for Turning

The cutting conditions (cutting speed, feedrate) of each process are registered, corresponding to each tip material type. Also, the cutting conditions (speed rate) of each process are registered, corresponding to each workpiece material type. When [CUT CONDTN] is selected in the LIST VIEW area, this screen is displayed.

Screen layout



(Note) Menu for the currently selected cutting condition is highlighted.

Screen display items

• Cutting condition file (Tip material)

No.	Display item		Details	Setting range
1	No.		Tip registration No.	1 to 8
2	TIP MATL		Input the name that represents the tip material.	Max. 4 alphanumeric characters
3	TURN R	V	Input the cutting speed for the rough turning machining.	Cutting speed: 1.00 to 9999.00m/min 1.00 to 9999.00feet/min Feedrate: 0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev
4		F	Input the feedrate for the rough turning machining.	
5	TURN F	V	Input the cutting speed for the finishing turning machining.	
6		F	Input the feedrate for the finishing turning machining.	
7	GRV R	V	Input the cutting speed for the rough grooving machining.	
8		F	Input the feedrate for the rough grooving machining.	
9	GRV F	V	Input the cutting speed for the finishing grooving machining.	
10		F	Input the feedrate for the finishing grooving machining.	
11	THR	V	Input the cutting speed for the threading machining.	
12	DRILL	V	Input the cutting speed for the drilling machining.	
13		F	Input the feedrate for the drilling machining.	
14	TAP	V	Input the cutting speed for the tapping machining.	

• Cutting condition file (Workpiece material)

No.	Display item		Details	Setting range
1	No.		Workpiece registration No.	1 to 8
2	WORK MATL		Input the name that represents the workpiece material.	Max. 5 alphanumeric characters
3	TURN R	V	Input the rate (%) of the workpiece material in respect to the cutting speed during rough turning machining.	1 to 200%
4		F	Input the rate (%) of the workpiece material in respect to the feedrate during rough turning machining.	
5	TURN F	V	Input the rate (%) of the workpiece material in respect to the cutting speed during finishing turning machining.	
6		F	Input the rate (%) of the workpiece material in respect to the feedrate during finishing turning machining.	
7	GRV R	V	Input the rate (%) of the workpiece material in respect to the cutting speed during rough grooving machining.	
8		F	Input the rate (%) of the workpiece material in respect to the feedrate during rough grooving machining.	
9	GRV F	V	Input the rate (%) of the workpiece material in respect to the cutting speed during finishing grooving machining.	
10		F	Input the rate (%) of the workpiece material in respect to the feedrate during finishing grooving machining.	
11	THR	V	Input the rate (%) of the workpiece material in respect to the cutting speed during threading machining.	
12	DRILL	V	Input the rate (%) of the workpiece material in respect to the cutting speed during drilling machining.	
13		F	Input the rate (%) of the workpiece material in respect to the feedrate during drilling machining.	
14	TAP	V	Input the rate (%) of the workpiece material in respect to the cutting speed during tapping machining.	

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	TIP MATL	Displays the cutting condition file (Tip material) screen.
3	WORK MATL	Displays the cutting condition file (Workpiece material) screen.
10	SAVE	Saves the changes in the cutting condition file.

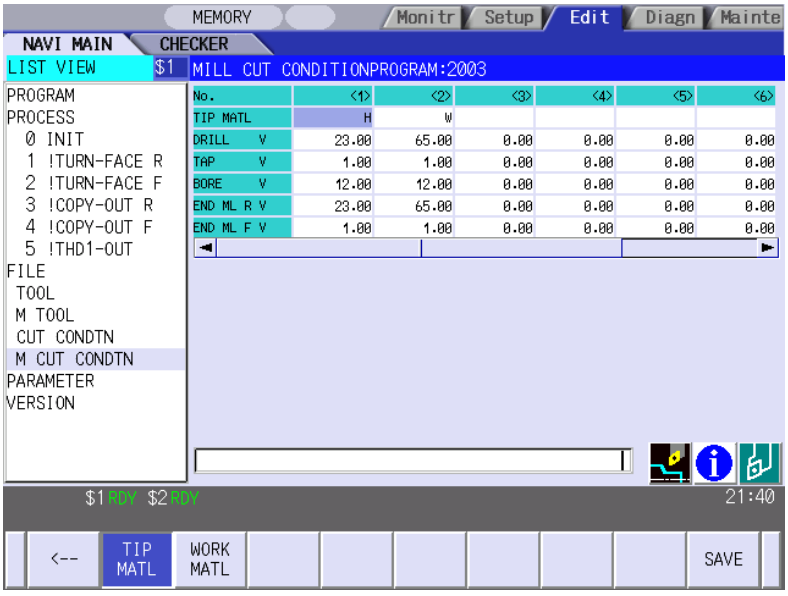
⚠ CAUTION

⚠ When either "TOOL REG No." or "CYCLE" is input in each machining process screen, the cutting speed and feedrate are automatically determined using the data in the tool file screen and the cutting condition file screen. Note that the cutting speed and feedrate of each process determined once will not be changed by changing the data in the tool file screen and the cutting condition file screen.

4.4.4 Cutting Condition File Screen for Milling

The cutting conditions (cutting speed, feedrate) of each process are registered, corresponding to each tip material type for milling. Also, the cutting conditions (speed rate) of each process are registered, corresponding to each workpiece material type. When [M CUT CONDTN] is selected in the LIST VIEW area, this screen is displayed.

Screen layout



(Note) Menu for the currently selected cutting condition is highlighted.

Screen display items

• Cutting condition file (Tip material)

No.	Display item	Details	Setting range
1	No.	Tip registration No. (1 to 8)	-
2	TIP MATL	Input the name that represents the tip material.	Max. 4 alphanumeric characters
3	DRILL V	Input the cutting speed for the drilling machining.	Cutting speed: 1.00 to 9999.00 m/min 1.00 to 9999.00 feet/min
4	TAP V	Input the cutting speed for the tapping machining.	
5	BORE V	Input the cutting speed for the boring machining.	
6	END ML R V	Input the cutting speed for the rough keyway/contour machining.	
7	END ML F V	Input the cutting speed for the finishing keyway/contour machining.	

• Cutting condition file (Workpiece material)

No.	Display item		Details	Setting range
1	No.		Workpiece registration No. (1～8)	-
2	WORK MATL		Input the name that represents the workpiece material. The workpiece material name input on the cutting condition file screen (for turning) is displayed.	-
3	DRILL	V	Input the rate (%) of the workpiece material in respect to the cutting speed during drilling machining.	1 to 200%
4		F	Input the rate (%) of the workpiece material in respect to the feedrate during drilling machining.	
5	TAP	V	Input the rate (%) of the workpiece material in respect to the cutting speed during tapping machining.	
6	BORE	V	Input the rate (%) of the workpiece material in respect to the cutting speed during boring machining.	
7		F	Input the rate (%) of the workpiece material in respect to the feedrate during boring machining.	
8	END ML R	V	Input the rate (%) of the workpiece material in respect to the cutting speed during rough keyway/contour machining.	
9		F	Input the rate (%) of the workpiece material in respect to the feedrate during rough keyway/contour machining.	
10	END ML F	V	Input the rate (%) of the workpiece material in respect to the cutting speed during finishing keyway/contour machining.	
11		F	Input the rate (%) of the workpiece material in respect to the feedrate during finishing keyway/contour machining.	

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	TIP MATL	Displays the cutting condition file (Tip material) screen for milling.
3	WORK MATL	Displays the cutting condition file (Workpiece material) screen for milling.
10	SAVE	Saves the changes in the cutting condition file.

⚠ CAUTION

⚠ When either "TOOL REG No." or "CYCLE" is input in each machining process screen, the cutting speed and feedrate are automatically determined using the data in the tool file screen and the cutting condition file screen. Note that the cutting speed and feedrate of each process determined once will not be changed by changing the data in the tool file screen and the cutting condition file screen.

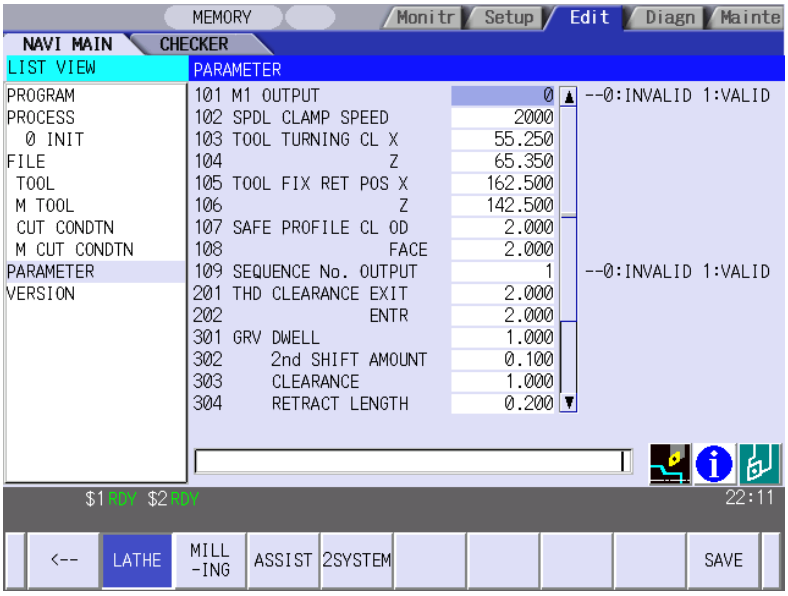
4.5 Screen Related to the Parameters

4.5.1 Parameter Screen

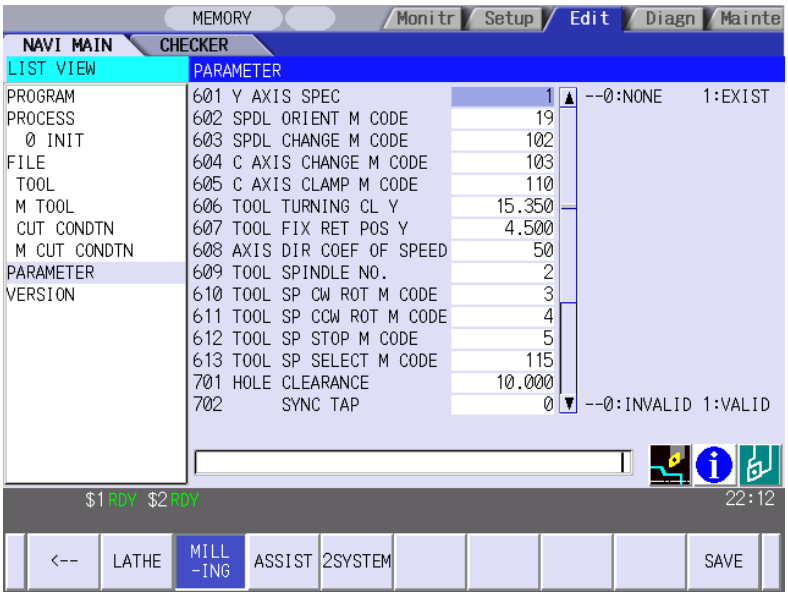
The parameter screen, on which the parameters for the machining program are entered, is provided for the turning and the milling machining.
When [PARAMETER] is selected in the LIST VIEW area, this screen is displayed.

Screen layout

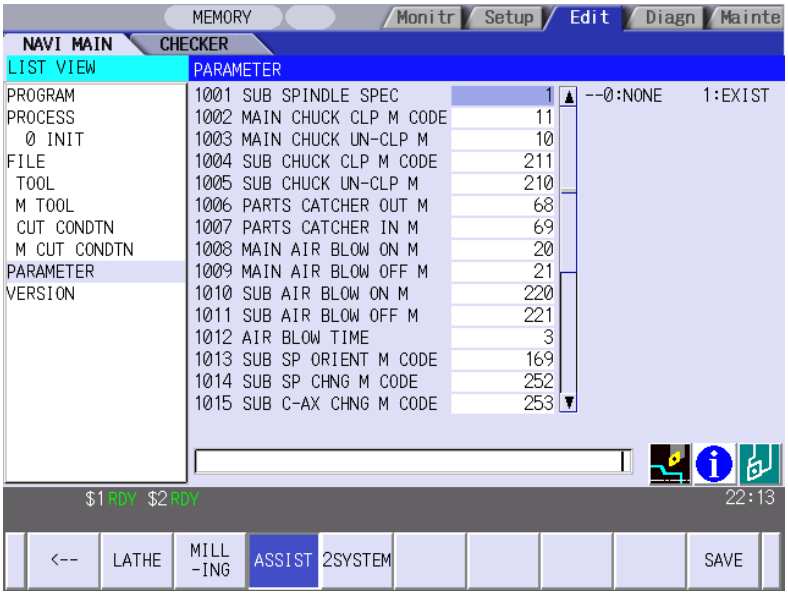
- Parameter for turning



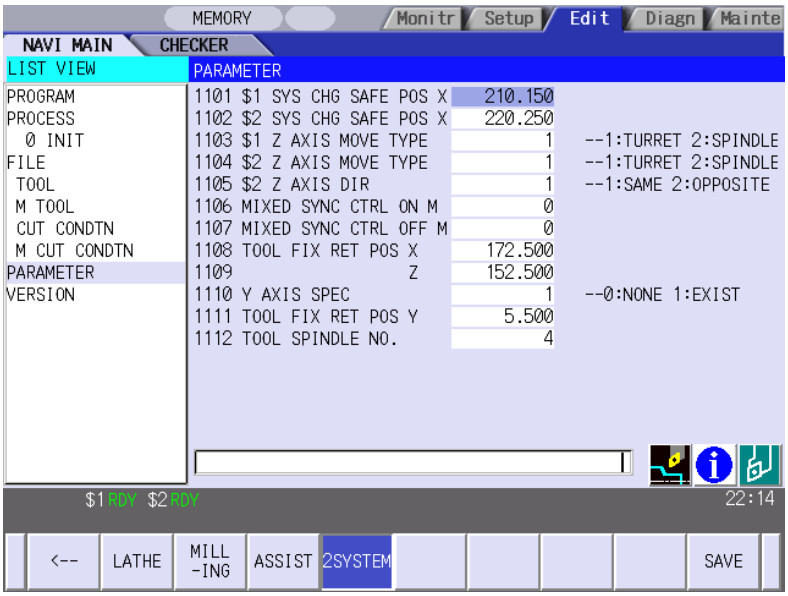
- Parameter for milling



- Parameter for ASSIST

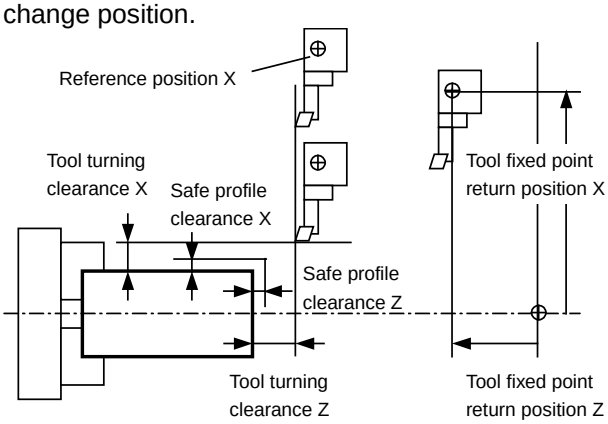


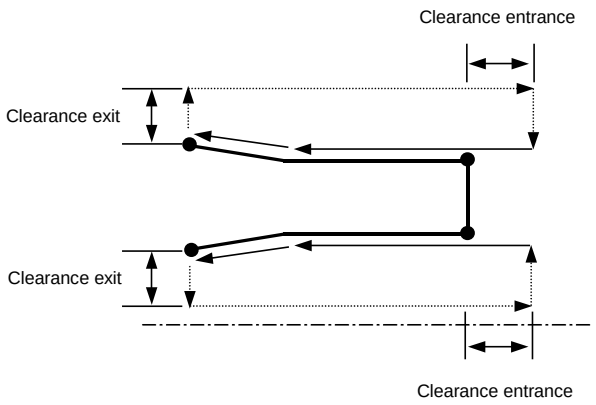
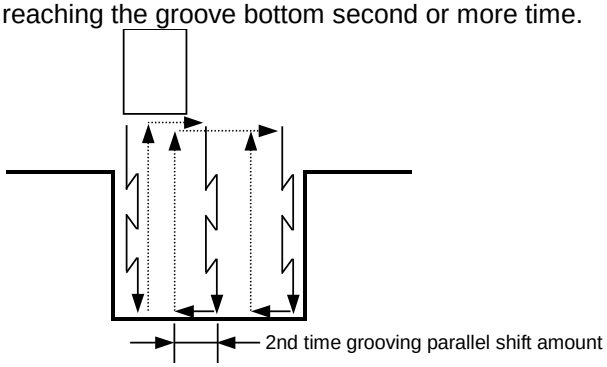
- Parameter for 2SYSTEM

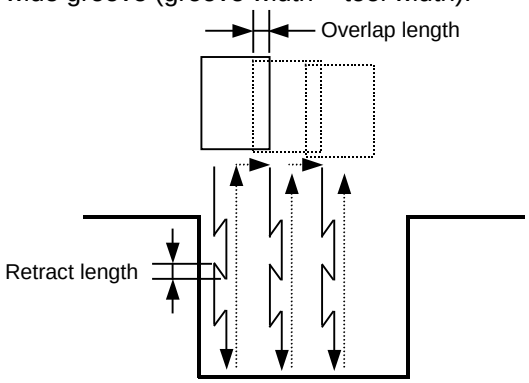
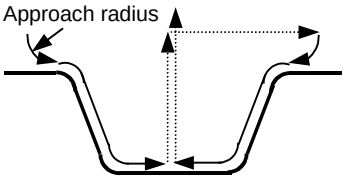


Screen display items

• Parameters for turning

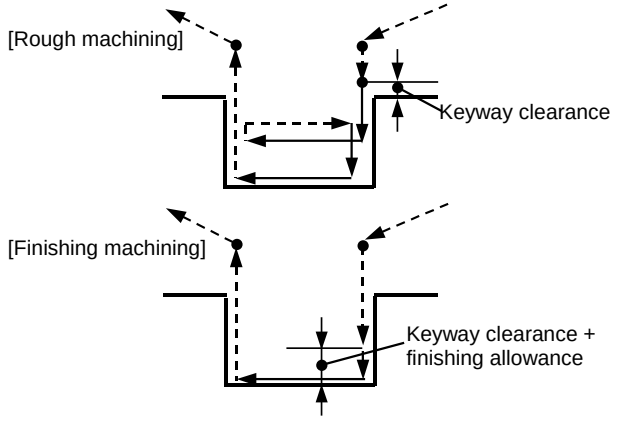
No.	Display item	Details	Setting range
101	M1 OUTPUT	Specify whether to output the M1 code before tool indexing command. 0: Not output 1: Output	0,1
102	SPDL CLAMP SPEED	Input the maximum spindle clamp speed of a machining program.	1 to 99999 rev/min
103	TOOL TURNING CL X	This is a constant to specify the turret positioning point when the tool is determined.	0.001 to 99999.999mm
104	TOOL TURNING CL Z		0.0001 to 9999.9999inch
105	TOOL FIX RET POS X	Input the tool change position in the machine coordinate system. This is valid when fixed point is selected for the tool change position.  	-99999.999 to 99999.999mm
106	TOOL FIX RET POS Z		-9999.9999 to 9999.9999inch
107	SAFE PROFILE CL OD	Input the clearance for the outer diameter area in radius value when the approaching/escaping path is used between processes. (Note) When approaching, two axes move together. But when escaping, the axes move one by one in the order of Z and X axes. Thus, set the safe profile clearance to avoid any interference with the tailstock, etc.	0.001 to 99999.999mm
108	SAFE PROFILE CL FACE	Input the clearance for the front area in radius value when the approaching/escaping path is used between processes. (Note) When approaching, two axes move together. But when escaping, the axes move one by one in the order of Z and X axes. Thus, set the safe profile clearance to avoid any interference with the tailstock, etc.	0.0001 to 9999.9999inch
109	SEQUENCE No. OUTPUT	Specify whether to output sequence No. in each process of the machining program. 0: Do not output 1: Output	0,1

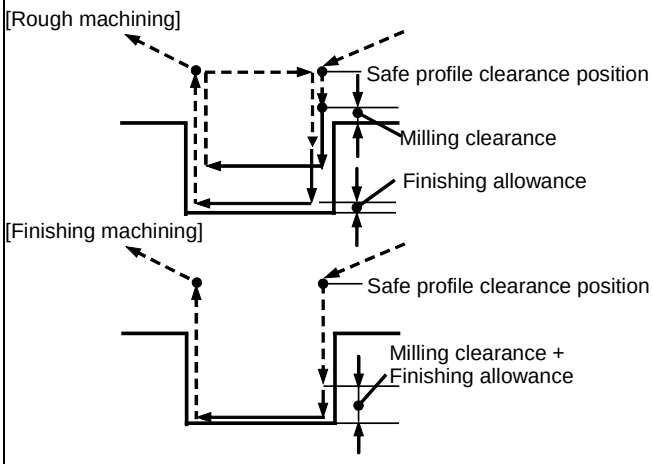
No.	Display item	Details	Setting range
201	THD CLEARANCE EXIT	Input the clearance between the highest part of the thread shape and the tool retract position in the radius value. 	0.001 to 99999.999mm 0.0001 to 9999.9999inch
202	THD CLEARANCE ENTR	Input the distance between the threading start point and machining start point.	0.000 to 99999.999mm 0.0000 to 9999.9999inch
301	GRV DWELL	Input the dwell value at the bottom of the groove.	0.000 to 99.999sec
302	GRV 2nd SHIFT AMOUNT	Input the amount of which the tool is shifted with cutting feed toward the machined area after reaching the groove bottom second or more time. 	0.001 to 99999.999mm 0.0001 to 9999.9999inch
303	GRV CLEARANCE	Input the distance from the point where cutting feedrate for grooving is started and the top surface position of the groove in radius value.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
304	GRV RETRACT LENGTH	Input the retract length of the tool used for the grooving machining in the radius value.	0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
305	GRV OVERLAP LENGTH	Input the tool overlap length when machining the wide groove (groove width > tool width).  	0.001 to 99999.999mm 0.0001 to 9999.9999inch
306	GRV FIN APPROACH R	Input the approach radius when approaching to the groove's entrance with smooth arc for the finishing machining of the trapezoidal groove.  	0.001 to 99999.999mm 0.0001 to 9999.9999inch
401	HOLE CLEARANCE	The distance from the R-point, where the cutting feed begins, to the hole top position is set in the radius value.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
402	HOLE SYNC TAP	Set valid or invalid of synchronous tapping for tapping cycle machining. 0: INVALID (ASYNC) 1: VALID (SYNC)	0 to 1

• Parameters for milling

No.	Display item	Details	Setting range
601	Y AXIS SPEC	Set whether Y-axis specifications are provided or not. 0: Not provided 1: Provided	0,1
602	SPDL ORIENT M CODE	Input the M command value for the spindle set position stop.	0 to 9999
603	SPDL CHANGE M CODE	Input the M command value to change the spindle to the normal one for the turning rotation.	0 to 9999
604	C AXIS CHANGE M CODE	Input the M command value to change the spindle to the one for milling (with C axis control).	0 to 9999
605	C AXIS CLAMP M CODE	Input the M command value for C axis clamp in the C axis control. M command for C axis unclamp is set by adding 1 to this value.	0 to 9999
606	TOOL TURNING CL Y	This is a constant to specify the turret positioning point when the tool is determined.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
607	TOOL FIX RET POS Y	Input the tool change position in the machine coordinate system. This is valid when fixed point is selected for the tool change position.	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch

No.	Display item	Details	Setting range
608	AXIS DIR COEF OF SPEED	The keyway/contour cutting feedrate in the diameter direction is automatically set. The cutting feedrates in the axis direction are determined by multiplying the value in the diameter direction by this coefficient. $F1 = F * \alpha$ F: Feedrate in the diameter direction F1: Feedrate in the axis direction α: Coefficient	1 to 200%
609	TOOL SPINDLE NO.	Input the tool spindle No. This No. is used to specify the spindle in the tapping cycle. (Note) Do not set a value larger than the value of "#1039 spinno".	1 to 4 (Default: 2)
701	HOLE CLEARANCE	The distance from the R-point, where the cutting feed begins, to the hole top position is set.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
702	HOLE SYNC TAP	Set “asynchronous tapping: 0” or “synchronous tapping: 1” for the tapping cycle (C=4) machining. 0: INVALID (ASYNC) 1: VALID (SYNC)	0,1
703	TAP ON M CODE	Input the M command value to turn ON the TAP mode for the tool spindle.	0 to 9999
704	TAP OFF M CODE	Input the M command value to turn OFF the TAP mode for the tool spindle.	0 to 9999
801	K-WAY CUT WIDTH PCT (%)	Set the overlap of the tool shift (“overlap percentage”) with “%” when the keyway width is larger than the diameter of the end mill. For example, if the overlap percentage is 70% when the machining is performed with the tool of ϕ 100, the machining is performed to the second line in the width of maximum 70mm. When this data is not input, 50% is applied.	1 to 100%
802	K-WAY CLEARANCE	Set the distance from the cutting start position of the keyway to the base plane position. In the second rough machining or later, the cutting start position approaches to the position at the distance of this clearance amount from the previous position.  	0.001 to 99999.999mm 0.0001 to 9999.9999 inch

No.	Display item	Details	Setting range
901	E-ML CUT WIDTH PCT (%)	In the contour machining, when the machining is performed to the second step after the machining for the first step, the machining is performed with the tool overlapping the machining width of the first step. Set such overlap of the tool ("overlap percentage") with "%". For example, if the overlap percentage is 70% when the machining is performed with the tool of ϕ 100, the machining is performed to the second line in the width of maximum 70mm. When this data is not input, 50% is applied.	1 to 100%
902	E-ML CLEARANCE	Set the distance from the cutting start position of the contour shape to the base plane position. In the second rough machining or later, the cutting start position approaches to the position at the distance of this clearance amount from the previous position. 	0.001 to 99999.999mm 0.0001 to 9999.9999 inch
903	E-ML EMPTY D OFS NUM	Set the temporary offset No. to set the offset of the tool diameter in the contour machining.	1 to tool sets

• Parameters for ASSIST

No.	Display item	Details	Setting range
1001	SUB SPINDLE SPEC	Set whether to exist the sub spindle specification. 0: NONE 1: EXIST	0,1
1002	MAIN CHUCK CLP M CODE	Input the M command value to clamp the main spindle's chuck.	0 to 9999
1003	MAIN CHUCK UN-CLP M CODE	Input the M command value to unclamp the main spindle's chuck.	0 to 9999
1004	SUB CHUCK CLP M CODE	Input the M command value to clamp the sub spindle's chuck.	0 to 9999
1005	SUB CHUCK UN-CLP M CODE	Input the M command value to unclamp the sub spindle's chuck.	0 to 9999
1006	PARTS CATCHER OUT M CODE	Input the M command value to output the parts catcher.	0 to 9999
1007	PARTS CATCHER IN M CODE	Input the M command value to input the parts catcher.	0 to 9999
1008	MAIN AIR BLOW ON M CODE	Input the M command value to turn ON the main spindle's air blow.	0 to 9999

No.	Display item	Details	Setting range
1009	MAIN AIR BLOW OFF M CODE	Input the M command value to turn OFF the main spindle's air blow.	0 to 9999
1010	SUB AIR BLOW ON M CODE	Input the M command value to turn ON the sub spindle's air blow.	0 to 9999
1011	SUB AIR BLOW OFF M CODE	Input the M command value to turn OFF the sub spindle's air blow.	0 to 9999
1012	AIR BLOW TIME	Set the air blow time for the main and sub spindles.	0 to 99 sec
1013	SUB SP ORIENT M CODE	Input the M command value for the sub spindle set position stop.	0 to 9999
1014	SUB SP CHNG M CODE	Input the M command value to change the sub spindle to the normal one for the turning rotation.	0 to 9999
1015	SUB C-AX CHNG M CODE	Input the M command value to change the sub spindle to the one for milling (with C axis control).	0 to 9999
1016	SUB C-AX CLP M CODE	Input the M command value for C-axis clamp during C-axis control of the sub spindle. The M command value for the C-axis unclamp is the value for this parameter plus 1.	0 to 9999
1017	SUB SPINDLE No.	Input the spindle number for the sub spindle.	1 to 4 (Default: 2)
1018	SUB SP CW ROT M CODE	Input the M command value to turn ON a forward rotation of the sub spindle. When 0 is set, M3 will be output.	0 to 9999
1019	SUB SP CCW ROT M CODE	Input the M command value to turn ON a reverse rotation of the sub spindle. When 0 is set, M4 will be output.	0 to 9999
1020	SUB SP STOP M CODE	Input the M command value to stop the sub spindle. When 0 is set, M5 will be output.	0 to 9999
1021	SUB SP C AXIS NAME	Select the C-axis name while the sub spindle is under the C-axis control. 1: A 2: B (Note) Even if the parameter is changed, the already created machining programs or machining processes are not reflected. Create a machining program again after changing the parameter.	1, 2 (Default: 1)
1022	TRANSFER AXIS NAME	Set the name of the transfer axis. 1: A 2: B	1, 2 (Default: 2)
1023	OVER TRAVEL OF PUSH	Input the value of over travel of sub spindle when checking if the sub spindle is set properly during the transfer process.	0.000 to 99999.999 mm 0.0000 to 9999.9999 inch
1024	SUB SP ORIGIN	Set the zero point of sub spindle with machine coordinate system.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
1025	MAIN SP SELECT M CODE	Input the M command value to set the main spindle.	0 to 9999
1026	SUB SP SELECT M CODE	Input the M command value to set the sub spindle.	0 to 9999

• Parameters for 2SYSTEM

No.	Display item	Details	Setting range
1101	$1 SYS CHG SAFE POS X	Set the retract position of the X axis direction of the turret when switching the machining part system.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch
1102	$2 SYS CHG SAFE POS X		
1103	$1 Z AXIS MOVE TYPE	Designate either a turret or a spindle to be moved by the Z axis command. 1: TURRET 2: SPINDLE	1, 2 (Default: 1)
1104	$2 Z AXIS MOVE TYPE		
1105	$2 Z AXIS DIR	Designate the Z axis direction for $2. 1: SAME (same as $1's Z axis direction) 2: OPPOSITE (opposite to $1's Z axis direction)	1, 2 (Default: 1)
1106	MIXED SYNC CTRL ON M	Input the M command value to turn ON the mixed synchronous control. (Note) This is available for mixed synchronous control (cross axis control) II. For the mixed synchronous control (cross axis control) I, set "0".	0 to 9999 (Default: 112)
1107	MIXED SYNC CTRL OFF M	Input the M command value to turn OFF the mixed synchronous control. (Note) This is valid for mixed synchronous control (cross axis control) II. For the mixed synchronous control (cross axis control) I, set "0".	0 to 9999 (Default: 113)
1108	TOOL FIX RET POS X	Set the tool change position for $2 relative to the machine coordinate system. This is valid when the fixed point is selected for the tool change position. (Note) These are the parameters for $2 of the lathe turning process "105 TOOL FIX RTE POS X" and "106 TOOL FIX RTE POS Z".	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch (Default: 0.000)
1109	TOOL FIX RET POS Z		
1110	Y AXIS SPEC	Set whether to exist the Y axis specification for $2. 0: NONE 1: EXIST (Note) This is the parameter for $2 of the milling process "601 Y AXIS SPEC", and is available only when the milling specification is valid.	0, 1 (Default: 0)
1111	TOOL FIX RET POS Y	Set the tool change position for $2 relative to the machine coordinate system. This is valid when the fixed point is selected for the tool change position. (Note) This is the parameter for $2 of the milling process "607 TOOL FIX RTE POS Y", and is available only when the milling specification is valid.	-99999.999 to 99999.999 mm -9999.9999 to 9999.9999 inch (Default: 0.000)
1112	TOOL SPINDLE NO.	Input the tool spindle No. for $2 This is used for designating the spindle for tap cycle, etc. (Note) This is the parameter for $2 of the milling process "609 TOOL SPINDLE NO.", and is available only when the milling specification is valid.	1 to 4 (Default: 4)

(Note 1) The parameters 1110 to 1112 are able to set the cursor movement even if the milling specification is invalid. However, the parameter values are not used.

(Note 2) When changing the parameter, the change is not reflected in an existing machining program. Create the machining program again.

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	LATHE	Displays the parameter input screen for turning.
3	MILLING	Displays the parameter input screen for milling.
4	ASSIST	Displays the parameter input screen for assist process.
5	2SYSTEM	Displays the parameter input screen for 2-part system process. This menu is available only when the 2-part system specification is "1: EXIST".
10	SAVE	Saves the changes in the parameters.

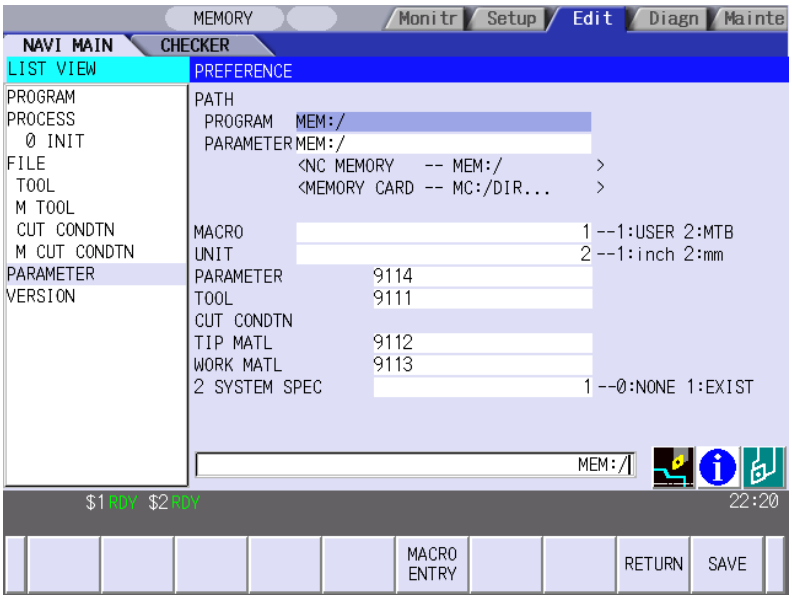
4.5.2 PREFERENCE Screen

Prior to the NAVI LATHE operation, system setups are done on this screen.
The followings are the items to be setup.

- Path to the folder in which NC program is saved
- Path to the folder in which tool file, cutting condition file and parameter file are saved
- Macro program mode (1: User Macro, 2: MTB Macro)
- Unit for data input (1: inch, 2: mm)
- Parameter file name
- Toll file name
- Cutting condition file (for tip material, workpiece material) name
- 2-part system specification (0: NONE, 1: EXIST)

This screen is displayed when [PREFERENCE] menu, which appears when 1 is input in the parameter "999 MAINTE", is pressed.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	PATH PROGRAM	Set the path to the folder in which NC program is saved.	(Drive name) : (Folder name)
2	PATH PARAMETER	Set the path to the folder in which tool file, cutting condition file and parameter file are saved.	
3	MACRO	Set the macro program mode. 1: User Macro 2: MTB Macro	1,2
4	UNIT	Set the unit for data input. 1: inch 2: mm	1,2
5	Parameter	Name of parameter file	
6	Tool file	Name of tool file	
7	Cutting condition file tip materials	Name of cutting condition file (tip material)	
8	Cutting condition file workpiece materials	Name of cutting condition file (workpiece material)	
9	2-part system specification	Whether 2-part specification is provided or not. 0: NONE (2-part system is not provided) 1: EXIST (2-part system is provided) (Note) When only 1-part system, this cannot be set to "1". Restart is required after changing the setting value.	0,1

(Note) If the following conditions are satisfied at the start of NAVI LATHE, the operation will be the same as when the 2-part system is set to "0: NONE", even when it is set to "1: EXIST".
- The number of part systems is less than 2.
- The multi-system program generation and operation (basic specification parameter #1285 ext21/bit2) is ON.

Menus

No.	Menu	Details
6	MACRO ENTRY	User macro program or MTB macro program is registered in the NC system. (Note 1) (Note 2)
9	RETURN	Returns to the parameter screen.
10	SAVE	Saves the changes in the preference setting data. (Note 3)

(Note 1) When changing the following parameters, make sure to do the macro entry.
Basic specification parameters
- #1037 Command type
- #1309 GType (Switch command format)

(Note 2) If "2" (MTB macro) is set at the PREFERENCE - MACRO even though there is no specification of the machine tool builder macro, an error message "E292 Program entry over" appears and the entry cannot be registered.
When there is no specification of the machine maker macro, set "1" (user macro) at the PREFERENCE - MACRO.

(Note 3) The PREFERENCE data is saved as the preference setting file (navi.ini) in the following folders.

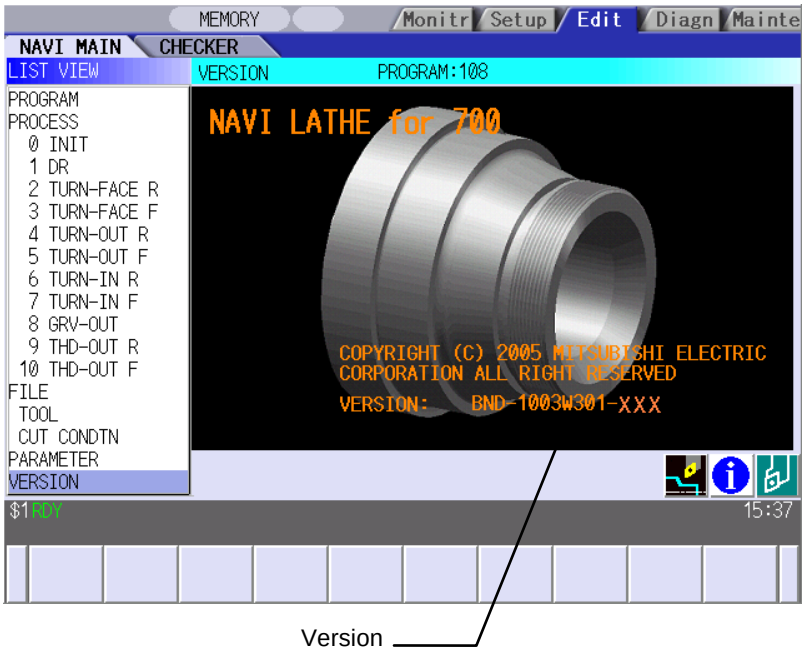
Model	MTB macro specification	Save folder
M700/M700VW series	EXIST	C:¥ncsys¥navilathe¥
	NONE	
M700VS series	EXIST	/PRG/MMACRO/
	NONE	/PRG/USER/
M70/M70V series	EXIST	/PRG/MMACRO/
E70	NONE	/PRG/USER/

4.6 Screen Related to the Version

4.6.1 Version Screen

The version data for the NAVI LATHE is displayed on this screen. When [VERSION] is selected in the LIST VIEW area, this screen is displayed.

Screen layout



4.7 Program Checker Screen

Machining shapes of a NC program are graphically displayed on this screen.
Program Checker screen appears when  or the tab is pressed while MAIN screen is displayed.
Program Checker screen also appear when the checker icon  is clicked.
On the checker screen, you can choose the following two check modes.
(1) Simple check: This is the mode where the NAVI LATHE analyzes and draws the machining shape of the machining program which was created by the NAVI LATHE.
(2) NC check: This is the mode that the graphic check function of NC analyzes and draws the tool path and the machining shape of the machining program which was created by the NAVI LATHE.

The checker screen starts with the simple check mode at the initial startup. After that, it starts according to the mode last used.
Switch the check mode using the mode change menu.

Main menus (Mode change)

While the simple check menu or the NC check menu is displayed, the following menus are displayed by pressing the menu change key.

No.	Menu	Details
1	EXIT	Terminates the Program Checker and then closes the screen.
2	SIMPLE CHECK	This menu is to change the check mode to the simple check. While the current check mode is the simple check, this menu is highlighted.
3	NC CHECK	This menu is to change the check mode to the NC check. While the current check mode is the NC check, this menu is highlighted.

The transition between each menu is as follows.

	Menu change key	[SIMPLE CHECK] menu	[NC CHECK] menu
Mode change menu	Move to the menu of the currently selected mode	Select the simple check mode Move to the simple check menu	Select the NC check mode Move to the NC check menu
Simple check menu	Move to the mode change menu	-	-
NC check menu	Move to the mode change menu	-	-

Screen layout

The screen layout may change according to the presence or absence of the following specifications.

No.	Name	Outline
1	Milling specification	When this specification exists, this enables two-plane graphics, and displays "VIEW". Also the "VIEW" menu is displayed.
2	Sub spindle specification	When this specification and the milling specification exist together, "FACE/BACK selection" of "DRAW STATUS" is displayed.
3	Two-part system specification	When this specification exists, "FACE/BACK selection" and "part system selection" of "DRAW STATUS" are displayed. Also the menu [$<->$] is displayed.

Display/non-display of the display items is determined by the combination of the specifications as follows.

○: Display × : Non-display

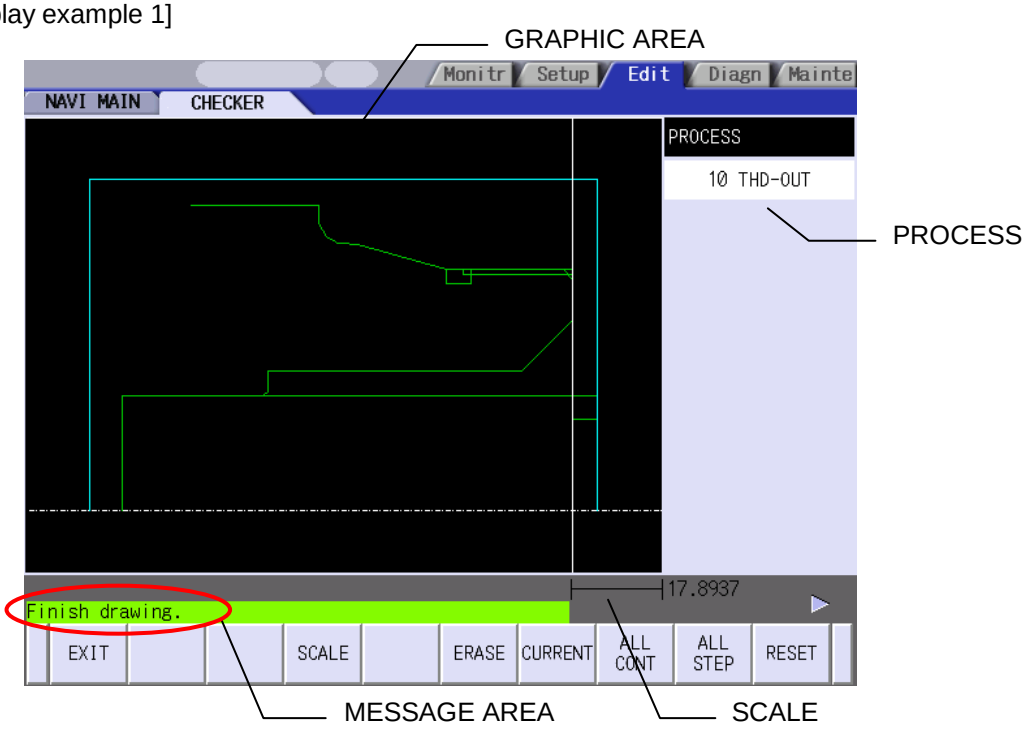
No.	Specification none/exist			Display item			Display example
	Milling spec.	Sub spindle spec.	Two-part system spec.	"VIEW"	"DRAW STATUS" FACE/BACK selection	"DRAW STATUS" part system selection	
1	None	None	None	×	×	×	1
2	None	None	Exist	×	×	○	2
3	None	Exist	None	×	×	×	1
4	None	Exist	Exist	×	×	○	2
5	Exist	None	None	○	×	×	3
6	Exist	None	Exist	○	×	○	5
7	Exist	Exist	None	○	○	×	4
8	Exist	Exist	Exist	○	○	○	6

4.7.1 Simple Check Mode

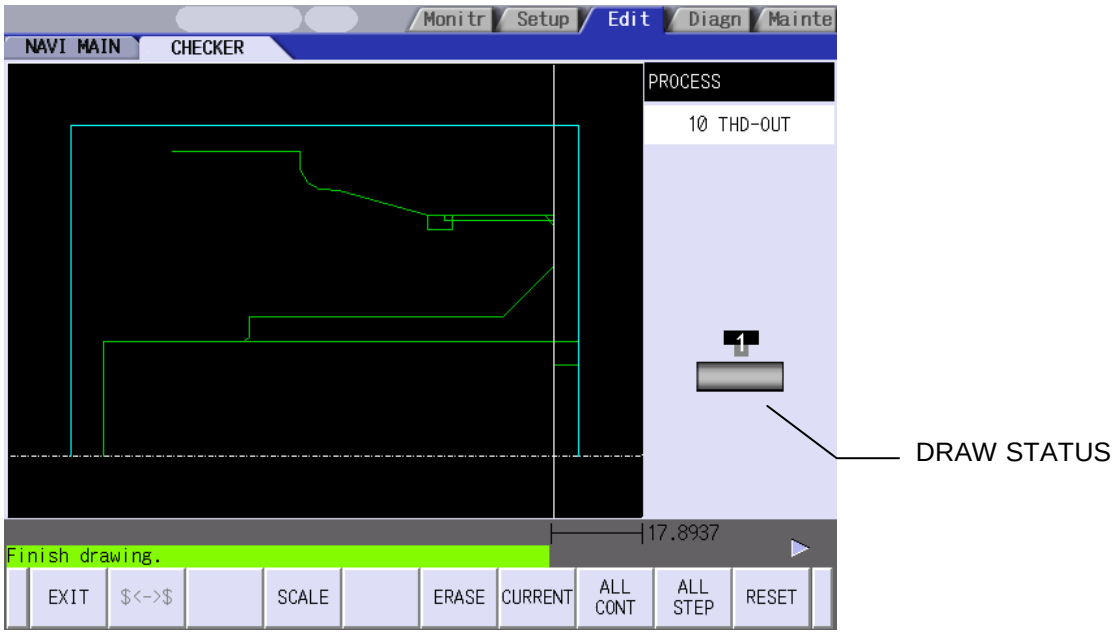
The NAVI LATHE analyzes and draws the machining shape of the machining program which was created by the NAVI LATHE.

Screen layout

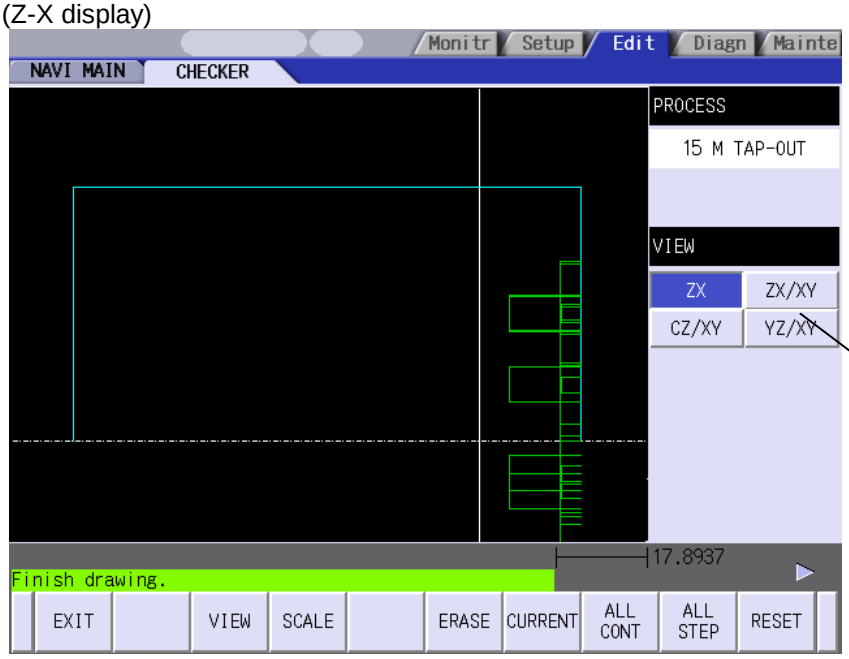
[Display example 1]



[Display example 2]

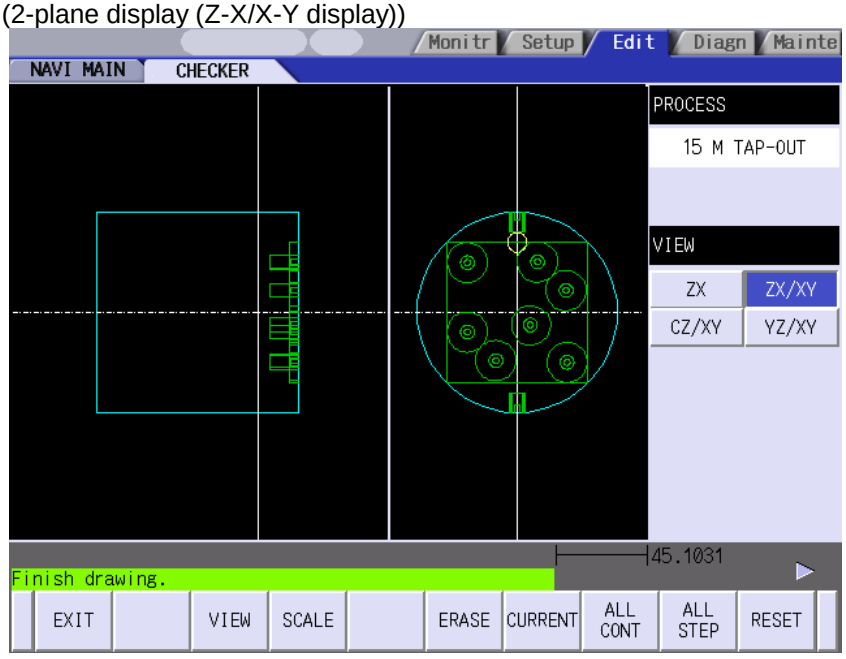


[Display example 3]
(Z-X display)

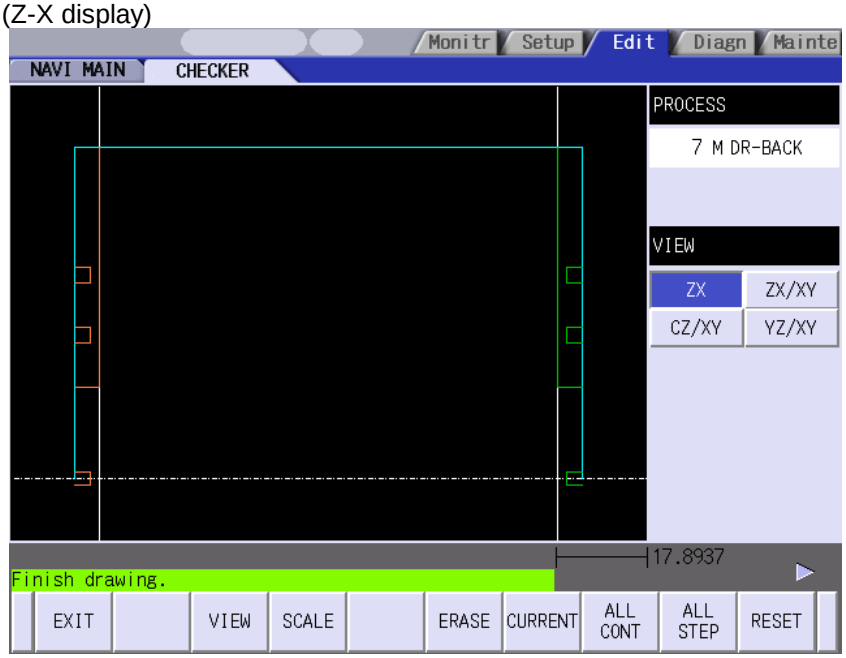


VIEW
(PLANE)

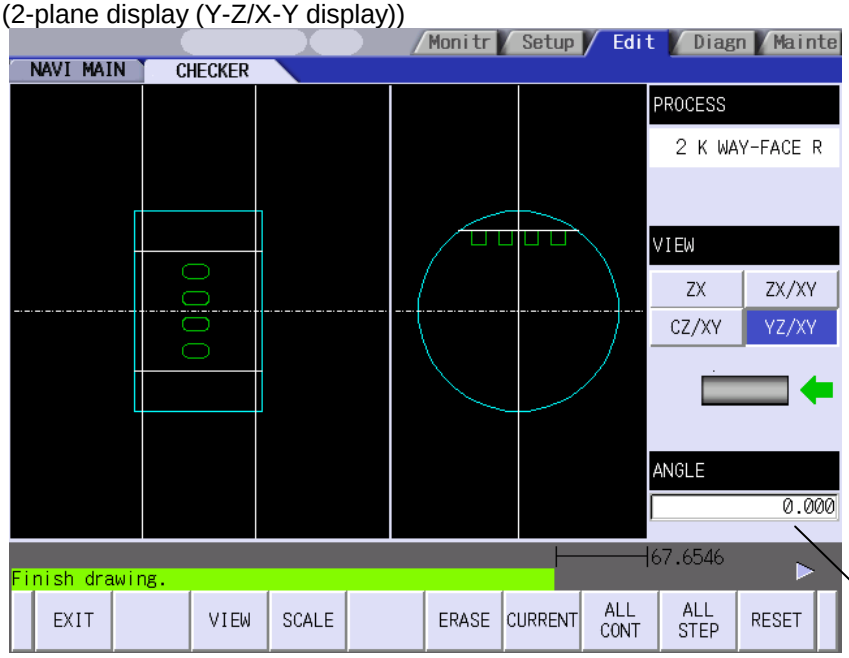
(2-plane display (Z-X/X-Y display))



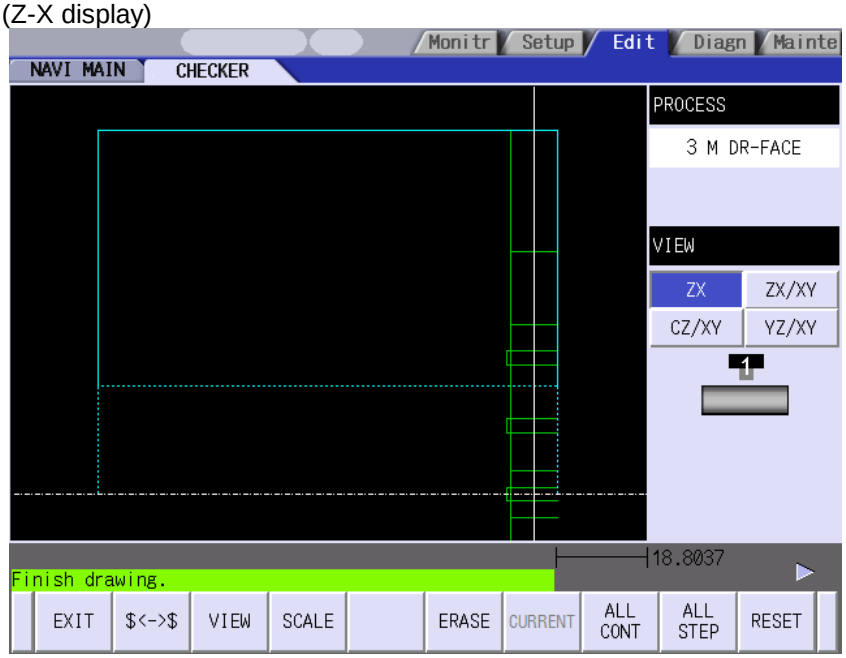
[Display example 4]
(Z-X display)



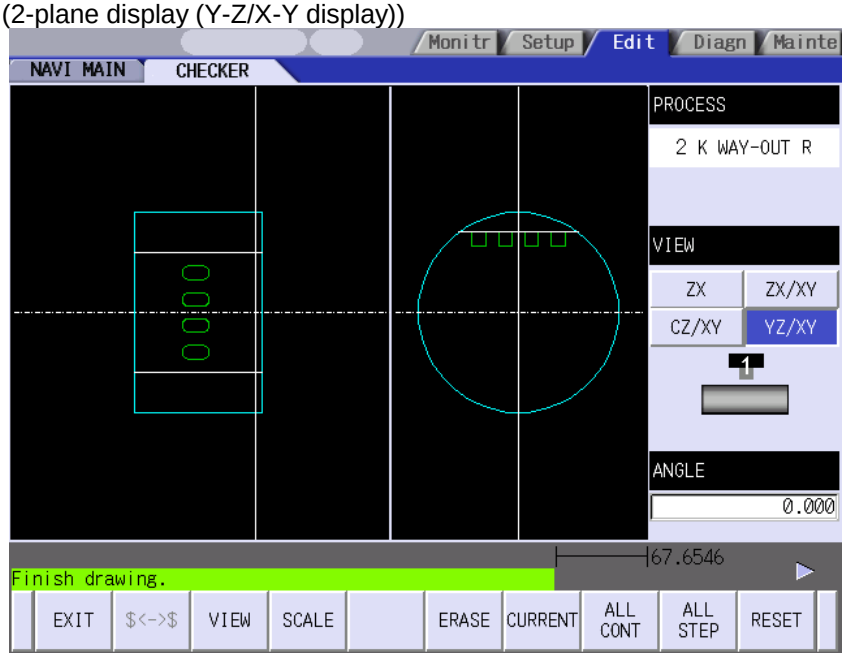
(2-plane display (Y-Z/X-Y display))



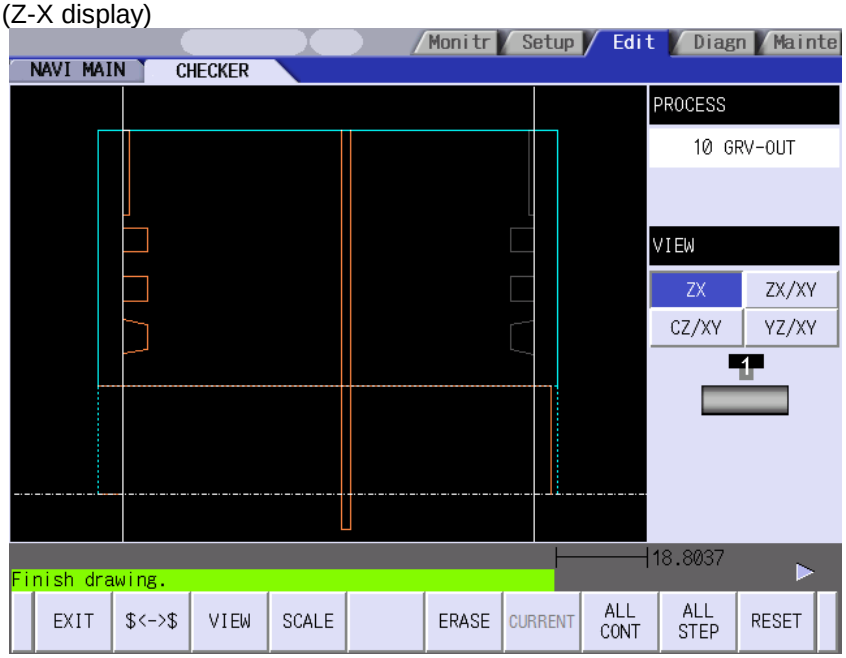
[Display example 5]
(Z-X display)



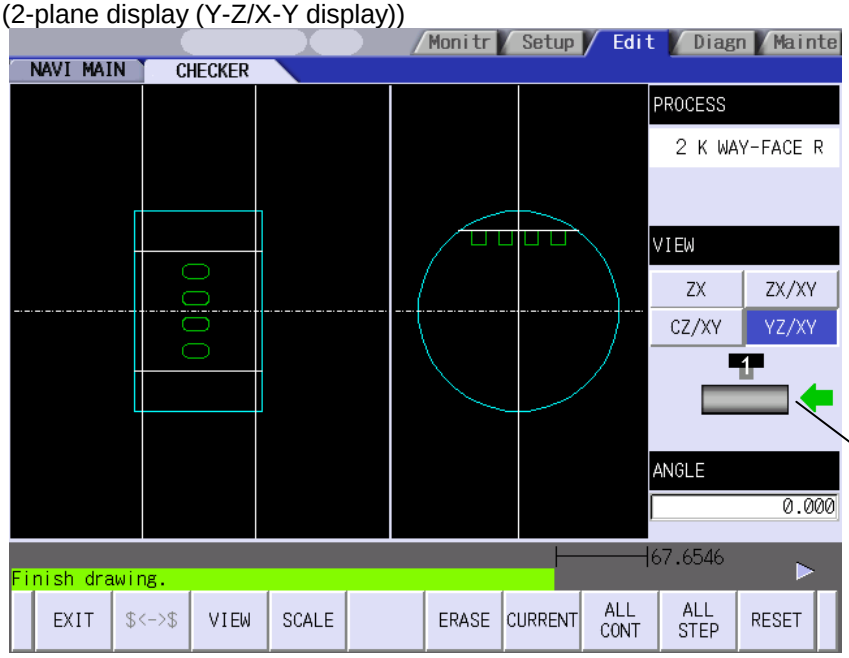
(2-plane display (Y-Z/X-Y display))



[Display example 6]
(Z-X display)



(2-plane display (Y-Z/X-Y display))

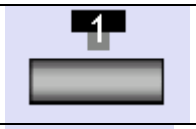
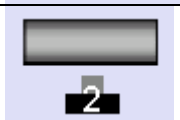
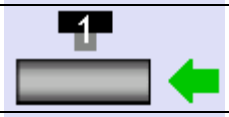
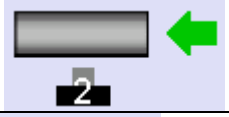
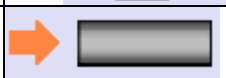
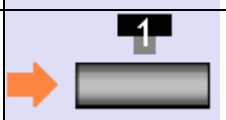
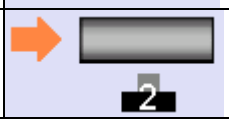


DRAW STATUS

Screen display items

No.	Menu	Details
1	GRAPHIC AREA	Graphically displays the workpiece shape and the machining shape. Items and their display colors on the screen are as follows: 1) Machining shape (main spindle) --- Green 2) Machining shape (sub spindle) --- Orange 3) Workpiece --- Light blue 4) Cutting plane on Y-Z view --- White 5) Radius display --- Yellow (Note) When the 2-part system specification is "1: EXIST", the machining shape of the part system not selected is drawn in gray.
2	PROCESS	Indicates the name of the process of which machining shape is currently displayed. (Note) When the 2-part system specification is "1: EXIST", the process names of the 2nd part system are also displayed.
3	VIEW	Displays the currently selected view. (Note) This is available when the milling interpolation specifications are provided. Not available unless the milling interpolation specifications are provided.
4	SCALE	Indicates the scale value of the graphic display area.
5	MESSAGE AREA	Messages on graphic display of the machining shape appear here.
6	RADIUS/ANGLE	Base radius and base angle of the graphic display area are input and indicated. Base radius is indicated when C-Z view is selected, while base angle is shown when Y-Z view is selected. This is not displayed unless C-Z view or Y-Z view is selected. When the [R/A] menu is selected in the VIEW change menu while ALL CONT or ALL STEP is performed, the cursor appears to set base radius and base angle.

<table>
<tr><th>No.</th><th>Menu</th><th>Details</th></tr>
<tr><td>7</td><td>DRAW STATUS</td><td>
Displays the following drawing modes.
1) FACE/BACK selection
There are the FACE selection and the BACK selection in this selection.
While the FACE is selected, a green arrow is displayed at the right side of the workpiece. In this mode, only the machining on the front face of workpiece is drawn. The drawing of the back surface machining is not performed.
While the BACK is selected, an orange arrow is displayed at the left side of the workpiece. In this mode, only the machining on the back surface of workpiece is drawn. The drawing of the front face machining is not performed.
FACE/BACK selection is switched by the [CHAGE FACE] menu.
2) Part system selection
There are $1 part system selection and the $2 part system selection in this selection.
While $1 is selected, the tool mark is displayed above the workpiece.
While $2 is selected, the tool mark is displayed below the workpiece.
Part system selection is switched with the [$<->$] menu key.
(Note) The DRAW STATUS differs according to the presence or absence of the sub spindle specification and the 2-part system specification. Refer to the chapter of screen layout for details.

Display combinations of the DRAW STATUS are as follows.
<table>
<tr><th>No.</th><th>FACE/BACK selection</th><th>Part system selection</th><th>Graphic icon</th></tr>
<tr><td>1</td><td>Invalid</td><td>Invalid</td><td>No icon</td></tr>
<tr><td>2</td><td>Invalid</td><td>$1</td><td>  </td></tr>
<tr><td>3</td><td>Invalid</td><td>$2</td><td>  </td></tr>
<tr><td>4</td><td>FACE</td><td>Invalid</td><td>  </td></tr>
<tr><td>5</td><td>FACE</td><td>$1</td><td>  </td></tr>
<tr><td>6</td><td>FACE</td><td>$2</td><td>  </td></tr>
<tr><td>7</td><td>BACK</td><td>Invalid</td><td>  </td></tr>
<tr><td>8</td><td>BACK</td><td>$1</td><td>  </td></tr>
<tr><td>9</td><td>BACK</td><td>$2</td><td>  </td></tr>
</table>
</td></tr>
</table>

Main menus

No.	Menu	Details
1	EXIT	Terminates the Program Checker and then closes the screen.
2	$<->$	Use this menu to change the part system in the DRAW STATUS. The part system will be changed from $1 to 2 or $2 to $1 by pressing this menu. (Note 1) During the drawing of the current process, this menu is not displayed because the tool part system of the currently edited process is selected and is unable to be changed. (Note 2) During the drawing of all the processes, $1 is set as the default and the part system can be changed by pressing this menu. (Note 3) When the 2-part system specification of NAVI LATHE is set to "0: NONE", this menu is not displayed. (Note 4) When the NC has no specification of the multi-part system program management, this menu is not displayed even if the 2-part system specification of NAVI LATHE is set to "1: EXIST".
3	VIEW	Use this menu to change view, base radius and base angle. Select a view from ZX, ZX/XY, CZ/XY or YZ/XY. The menu will be changed to the VIEW change menu by pressing this menu. (Note) Not available unless the milling interpolation specifications are provided.
4	SCALE	Use this menu when changing scale. Standard scale setting, scaling up/down, and graphic area shifting can be performed. The menu will be changed to SCALE change menu by pressing this menu. In the 2-plane display mode, scale frames are made on both of the planes.
6	ERASE	Deletes the drawing data.
7	CURRENT	Draws the machining shapes of the currently selected process. The shapes are drawn based on the view and scale set for CURRENT display.
8	ALL CONT	Draws the machining shapes of all the processes successively. The shapes are drawn based on the view and scale set for ALL CONT display.
9	ALL STEP	Draws the machining shapes of one process at a time. The shapes are drawn based on the view and scale set for ALL CONT display.
10	RESET	Resets the graphic display of the machining shapes.

(Note 1) Views and scales are arranged for CURRENT display and for ALL CONT display.
(Note 2) The views and scales selected in the CURRENT display are retained for the CURRENT display. When the CURRENT display is performed for any other process, the views and scales for the CURRENT display turn to the standard ones.
(Note 3) The scales, the views selected and the part system selected in the ALL CONT or ALL STEP display are retained for the ALL CONT display. These views, scales and part system are retained for the ALL CONT display until the NAVI LATHE is closed.
(Note 4) The selection of CURRENT or ALL CONT is retained even when the check mode is switched between Simple and NC check.

View change menu

This is the sub menu displayed by pressing the [VIEW] menu.

No.	Menu	Details
1	CANCEL	Returns to the main menu.
3	ZX	Converts the view into the Z-X view and returns to the main menu.
4	ZX/XY	Converts the view into the 2-plane display of Z-X and X-Y, and then returns to the main menu.
5	CZ/XY	Converts the view into the 2-plane display of C-Z and X-Y, and then returns to the main menu. In ALL CONT and ALL STEP display, C-Z view only displays the shapes made upon the fixed base radius for the machining process.
6	YZ/XY	While drawing the VIEW of the Y-Z and X-Y in the ALL CONT and ALL STEP, Y-Z VIEW displays only the shapes made upon the fixed base radius for the machining process.
8	R/A	Set the base radius and the base angle. These are selectable only when Y-Z or C-Z view is selected. When this menu is pressed, the cursor appears in the RADIUS/ANGLE display area. The [R/A] menu does not appear when Z-X or ZX/XY view is selected, or when the CURRENT display is performed.
9	CHANGE FACE	Switches the end surface for drawing in the XY plane. When the end surface is the FACE , the drawing is switched to the BACK, when the end surface is the BACK, the drawing is switched to the FACE. This menu does not appear while ZX VIEW is selected. When the parameter "1001 SUB SPINDLE SPEC" is "0: NONE", the [CHANGE FACE] menu is not displayed.

(Note 1) [VIEW] menu is not available while graphic display is performed. Press [RESET] menu and cancel the graphic display in advance.
(Note 2) The displayed shapes are deleted upon any change of the VIEW.
(Note 3) The views in the CURRENT display are set as follows, according to the machining process and the machining area.

Machining Process		View
Turning		ZX
Milling hole drilling	Front face	ZX/XY
	Outer diameter	CZ/XY
	Side surface	YZ/XY
	Back surface	ZX/XY
Keyway cutting	Front face	ZX/XY
	Outer diameter	CZ/XY
	Side surface	YZ/XY
	Back surface	ZX/XY
Contour cutting	Front face	ZX/XY
	Outer diameter	CZ/XY
	Side surface	YZ/XY
	Back surface	ZX/XY
Balance cut (turn)		ZX
Balance cut (copy)		ZX
Two-part system simultaneous thread cutting (identical screw)		ZX

(Note 4) Some views selected may not display the machining shapes. Refer to the examples of the graphic display for the machining shapes of the process displayed on each view.
(Note 5) When the checker runs while any object except PROCESS is selected in the LIST VIEW area, views for ALL CONT display are applied.

SCALE change menus

This is the sub menu of the [SCALE] menu.

No.	Display item	Details
1	CANCEL	Cancels the SCALE change and returns to the main menu.
2	STANDARD	Changes the scale to the standard setting and returns to the main menu. Scale value is automatically calculated based on the workpiece sizes. The center of workpiece displayed coincides with that of the screen.
3	ENLARGE	Enlarges the scale. The same function can be achieved by pressing – key.
4	REDUCE	Reduces the scale. The same function can be achieved by pressing + key. (Note) The solid scale frame will be drawn in dotted lines when its size exceeds 100%.
5	↑	Moves up the scale frame. The same function can also be achieved by pressing ↑ key. When ZX/XY view is selected, the two planes are simultaneously moved. When CZ/XY or YZ/XY view is selected, the scale frame in the selected area is moved.
6	↓	Moves down the scale frame. The same function can also be achieved by pressing ↓ key. When ZX/XY view is selected, the two planes are simultaneously moved. When CZ/XY or YZ/XY view is selected, the scale frame in the selected area is moved.
7	←	Moves the scale frame toward the left. The same function can also be achieved by pressing ← key. In the 2-plane display, the scale frame in the selected area is moved.
8	→	Moves the scale frame toward the right. The same function can also be achieved by pressing → key. In the 2-plane display, the scale frame in the selected area is moved.
9	SELECT	Select the area to adjust the scale. This is available in the 2-plane display.
10	SET	Determines the scale and returns to the main menu. The same result can also be achieved by pressing [INPUT] key.

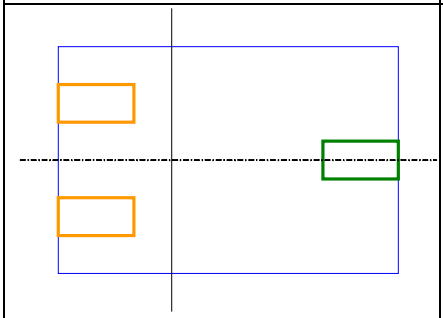
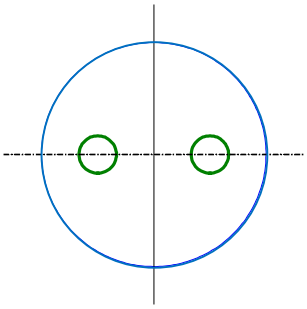
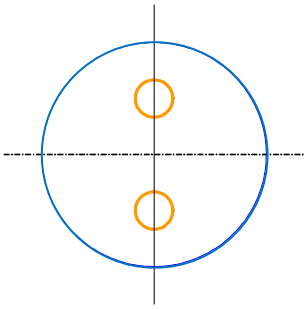
(Note 1) Display area is shown with a white frame.
(Note 2) The displayed machining shape will be deleted upon change of display scale or position.
(Note 3) The [SCALE] menu cannot be pressed during the drawing.
Press the [RESET] menu to stop the drawing, then press the [SCALE] menu.

Drawings of XY VIEW

The XY VIEW draws either the FACE or BACK end surface.
The end surface to be drawn differs according to the drawing mode or the machining spindle.

Drawing mode	Machining spindle	Drawing end surface
CURRENT (Drawing mode for the current process)	FACE	Draws only the machining shape of front side end surface in the XY VIEW.
	BACK	Draws only the machining shape of back side end surface in the XY VIEW.
ALL CONT and ALL STEP (Drawing mode for all processes and all steps)	FACE/BACK	Draws the machining shape of the selected side of end surface in the XY VIEW. FACE/BACK of the drawing end surface is switched with the [DRAW BACK] menu. When the [DRAW BACK] menu is OFF, the shape of front side end surface is drawn. When the [DRAW BACK] menu is ON, the shape of back side end surface is drawn.

Drawing images of FACE/BACK (when drilling both end surfaces)

ZX VIEW	XY VIEW selecting the front side of end surface	XY VIEW selecting the back side of end surface
		

Drawing of ZX/CZ/YZ VIEW

Whether the front or back side is selected, the center of the drawing area coincides with that of the workpiece.
The horizontal axis (Z axis) is drawn in the program zero position of the main and sub spindles respectively as listed below.

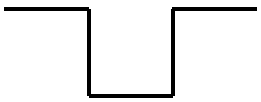
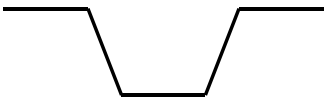
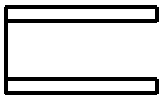
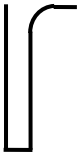
Spindle	Drawing of Z axis
Main spindle	The drawing is calculated by using the +Z and -Z values set in the initial setting screen. (Note) When +Z=100 and -Z=0 are set in the initial setting screen, the Z axis is drawn on the back surface of the workpiece.
Sub spindle	The drawing is calculated by the setting value "Z SUB SP" in the initial setting screen. (Note) When Z SUB SP=0 is set in the initial setting screen, the Z axis is drawn on the back surface of the workpiece.

Restrictions on the graphic display function

- Graphic display is not available for the EIA process.
- When there is an error in the specified shape data for the turning/copy cutting, the shape data is displayed up to the error point.

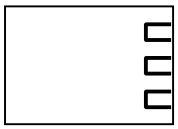
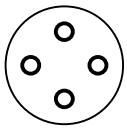
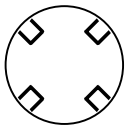
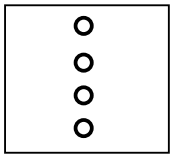
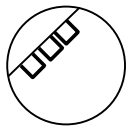
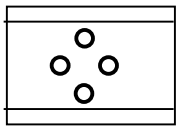
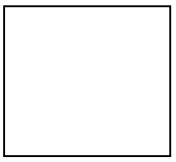
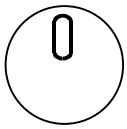
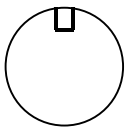
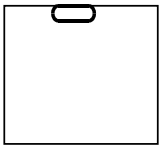
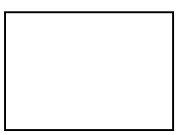
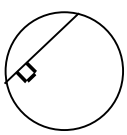
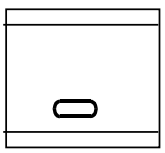
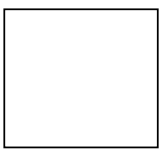
Examples of graphic drawings

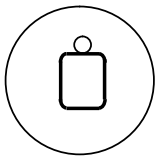
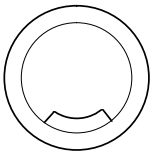
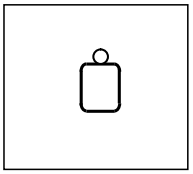
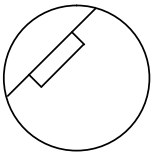
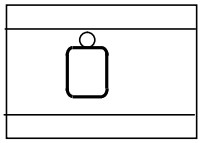
[Turning]
Only ZX view is displayed for turning.

[Turning / Copy cutting] 	[Threading] 
[Grooving] 	[Trapezoidal grooving] 
[Hole drilling --- Drilling ---] 	[Hole drilling --- Tapping ---] 
[Cut off] 	

(Note) When the 2-part system specification is "1: EXIST", the machining shape of the balance cut process will be the same as the machining shape of each turning.

[Milling]
For milling process, machining shapes are displayed on the views that correspond to each machining area.

Process	Area	Z-X view	X-Y view	Y-Z view	C-Z view
Milling hole drilling	FACE/ BACK				
		Machining shapes are not displayed on Y-Z or C-Z view.			
	OUT				
		Machining shapes are not displayed on Z-X or Y-Z view.			
	SIDE				
		Machining shapes are not displayed on Z-X or C-Z view.			
Keyway cutting	FACE/ BACK				
		Machining shapes are not displayed on Y-Z or C-Z view.			
	OUT				
		Machining shapes are not displayed on Z-X or Y-Z view.			
	SIDE				
		Machining shapes are not displayed on Z-X or C-Z view.			

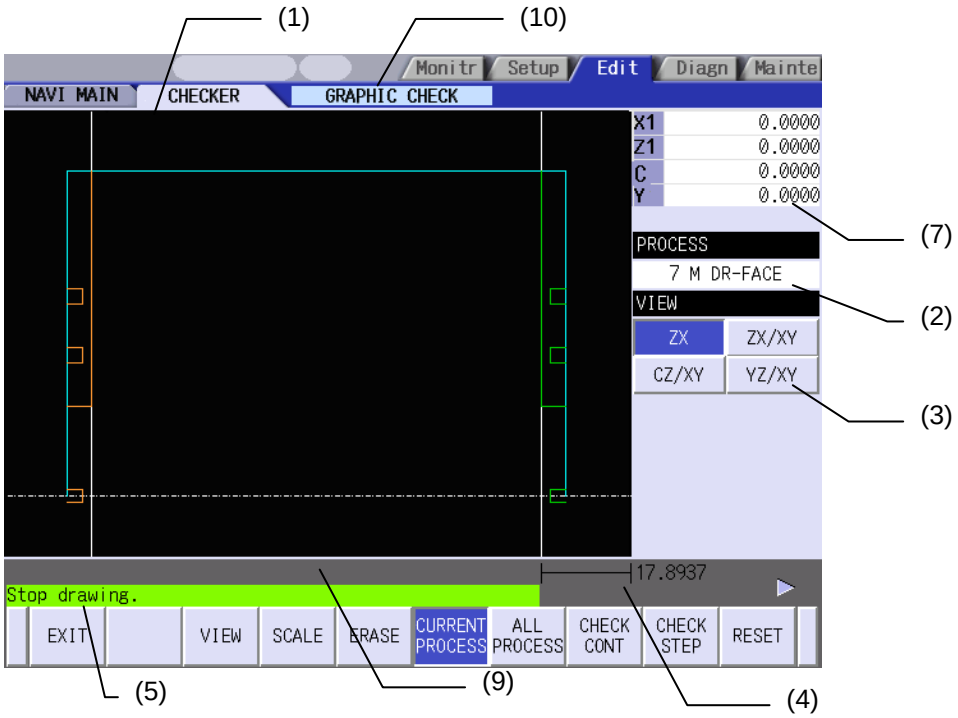
Process	Area	Z-X view	X-Y view	Y-Z view	C-Z view
Contour cutting	FACE				
		Machining shapes are not displayed on Y-Z or C-Z view.			
	OUT				
		Machining shapes are not displayed on Z-X or Y-Z view.			
	SIDE				
		Machining shapes are not displayed on Z-X or C-Z view.			

4.7.2 NC Check Mode

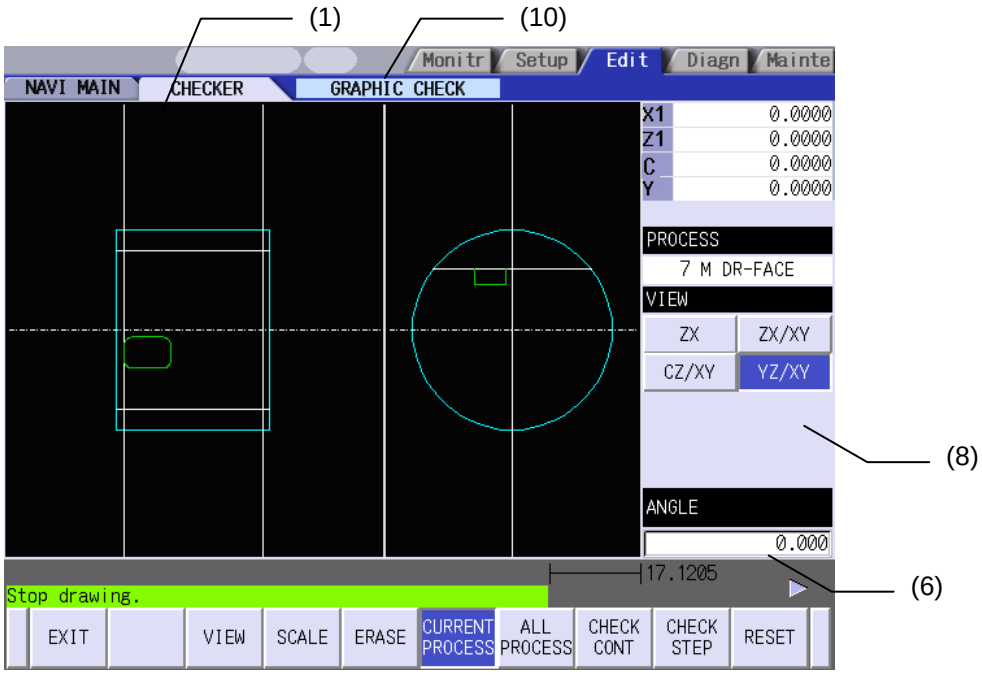
This mode draws the tool path by using the graphic check function of the NC and the machining shape of the program which was created with the NAVI LATHE.

Screen layout (NC check)

[Z-X display]



[2-plane display (example of Z-X/X-Y display))]

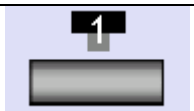
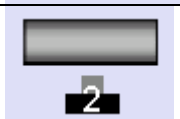
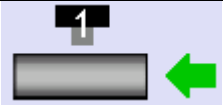
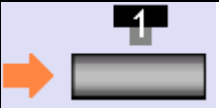
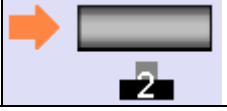


(Note 1) The 2-plane display is available only when the specification of milling interpolation exists.
(Note 2) Maximum 4 axes are displayed in the counter according to the presence or absence of the C and Y axes specification.

Screen display items

No.	Display item	Details
1	GRAPHIC AREA	Graphically displays the workpiece shape and the machining shape. Items and their display colors on the screen are as follows: 1) Machining shape (front face spindle) --- Green 2) Machining shape (back surface spindle) --- Orange 3) Workpiece --- Light blue 4) Cutting plane on Y-Z view --- White 5) Radius display --- Yellow 6) Tool path --- rapid traverse Blue, cutting feed Green (Note) When the 2-part system specification is "1: EXIST", the machining shape of the part system not selected is drawn in gray.
2	PROCESS	Indicates the name of the process of which machining shape is currently displayed. (Note) When the 2-part system specification is "1: EXIST", the process name of the 2nd part system is also displayed.
3	VIEW	Displays the currently selected view. (Note) This is available when the milling interpolation specifications are provided. Not available unless the milling interpolation specifications are provided.
4	SCALE	Indicates the scale value of the graphic display area.
5	MESSAGE AREA	Messages on graphic display of the machining shape appear here.
6	RADIUS/ANGLE	Base radius and base angle of the graphic display area are input and indicated. Base radius is indicated when C-Z view is selected, while base angle is shown when Y-Z view is selected. This is not displayed unless C-Z view or Y-Z view is selected. When the [R/A] menu is selected in the VIEW change menu while ALL CONT or ALL STEP is performed, the cursor appears to set base radius and base angle.
7	WORK COORDINATE POSITION	Displays the counter of workpiece coordinate position. The counter displays up to four axes, two of which are X and Z axes, according to the presence or absence of C axis and Y axis specifications. The axis name set in the axis parameter #1022 axname2 is displayed. For the 2-part system specification, the workpiece coordinate position counter of the part system being selected is displayed.

<table>
<tr><th>No.</th><th>Display item</th><th>Details</th></tr>
<tr><td>8</td><td>DRAW STATUS</td><td>
Displays the following drawing modes.
1) FACE/BACK selection
There are the FACE selection and the BACK selection in this selection. While the FACE is selected, a green arrow is displayed at the right side of the workpiece. In this mode, only the machining on the front face of workpiece is drawn. The drawing of the back surface machining is not performed.
While the BACK is selected, an orange arrow is displayed at the left side of the workpiece. In this mode, only the machining on the back surface of workpiece is drawn. The drawing of the front face machining is not performed.
FACE/BACK selection is switched by the [CHANGE FACE] menu.
2) Part system selection
There are $1 part system selection and the $2 part system selection in this selection.
While $1 is selected, the tool mark is displayed above the workpiece.
While $2 is selected, the tool mark is displayed below the workpiece.
Part system selection is switched with the [$<¥>$] menu key.
(Note) The DRAW STATUS differs according to the presence or absence of the sub spindle specification and the 2-part system specification. Refer to the chapter of screen layout for details.

Display combinations of the DRAW STATUS are as follows.
<table>
<tr><th>No.</th><th>FACE/BACK selection</th><th>Part system selection</th><th>Graphic icon</th></tr>
<tr><td>1</td><td>Invalid</td><td>Invalid</td><td>No icon</td></tr>
<tr><td>2</td><td>Invalid</td><td>$1</td><td>  </td></tr>
<tr><td>3</td><td>Invalid</td><td>$2</td><td>  </td></tr>
<tr><td>4</td><td>FACE</td><td>Invalid</td><td>  </td></tr>
<tr><td>5</td><td>FACE</td><td>$1</td><td>  </td></tr>
<tr><td>6</td><td>FACE</td><td>$2</td><td>  </td></tr>
<tr><td>7</td><td>BACK</td><td>Invalid</td><td>  </td></tr>
<tr><td>8</td><td>BACK</td><td>$1</td><td>  </td></tr>
<tr><td>9</td><td>BACK</td><td>$2</td><td>  </td></tr>
</table>
</td></tr>
<tr><td>9</td><td>NC MESSAGE AREA</td><td>Displays the alarm messages output from the NC.</td></tr>
<tr><td>10</td><td>CHECK MODE DISPLAY AREA</td><td>When the current check mode is the NC check, the letters "NC" appear to indicate the NC check is being selected.</td></tr>
</table>

Main menus

No.	Menu	Details
1	EXIT	Terminates the Program Checker and then closes the screen.
2	$<->$	Use this menu to change the part system in the DRAW STATUS. The part system will be changed from $1 to 2 or $2 to $1 by pressing this menu. (Note 1) During the drawing of the current process, this menu is not displayed because the tool part system of the currently edited process is selected and is unable to be changed. (Note 2) During the drawing of the all processes, $1 is selected as the default and the part system can be changed by pressing this menu. (Note 3) When the 2-part system specification of NAVI LATHE is set to "0: NONE", this menu is not displayed. (Note 4) When the NC has no specification of the multi-part system program management, this menu is not displayed even if the 2-part system specification of NAVI LATHE is set to "1: EXIST".
2	VIEW	Use this menu to change view, base radius and base angle. Select a view from ZX, ZX/XY, CZ/XY or YZ/XY. The menu will be changed to the VIEW change menu by pressing this menu. (Note) Not available unless the milling interpolation specifications are provided.
3	SCALE	Use this menu when changing scale. Standard scale setting, scaling up/down, and graphic area shifting can be performed. The menu will be changed to SCALE change menu by pressing this menu. In the 2-plane display mode, scale frames are made on both of the planes.
4	ERASE	Deletes the drawing data.
5	CURRENT PROCESS	Use this menu when drawing the tool path of the currently selected process. While this menu is selected, it is highlighted.
6	ALL PROCESS	Use this menu when drawing the tool path of the entire processes continuously. While this menu is selected, it is highlighted.
7	CHECK CONT	Draws the tool path continuously according to the mode in the current process or the entire processes.
8	CHECK STEP	Draws the tool path for every movement command block according to the mode of the current process or the entire processes.
9	RESET	Resets the graphic display of the machining shapes.

(Note 1) Views and scales are arranged for CURRENT display and for ALL CONT display.
(Note 2) The views and scales selected in the CURRENT display are retained for the CURRENT display. When the CURRENT display is performed for any other process, the views and scales for the CURRENT display turn to the standard ones.
(Note 3) The scales, the views selected and the part system selected in the ALL CONT or ALL STEP display are retained for the ALL CONT display. These views, scales and part system are retained for the ALL CONT display until the NAVI LATHE is closed.
(Note 4) After the CURENT PROCESS drawing or the ALL PROCESS drawing is completed, if the [CHECK CONT] or the [CHECK STEP] menu is pressed again, it starts drawing without deleting the displayed tool path.
(Note 5) The selection of CURRENT or ALL CONT is retained even when the check mode is switched between Simple and NC check.

View Change Menu

Refer to "4.7.1 Simple Check Mode VIEW change menus".

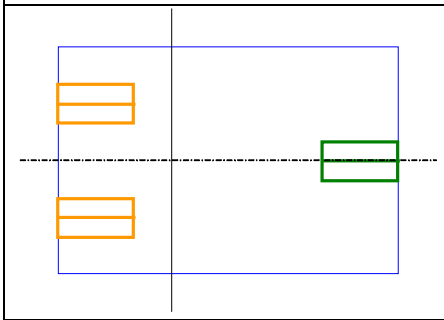
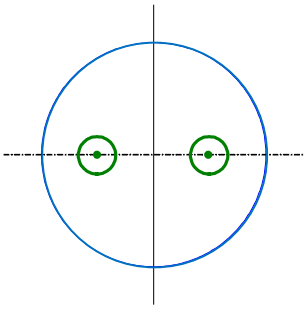
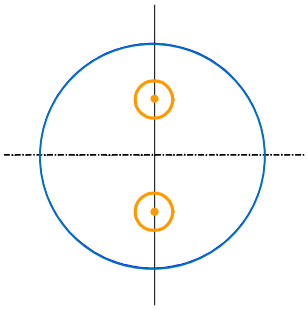
SCALE change menus

Refer to "4.7.1 Simple Check Mode SCALE change menus".

Drawings of XY VIEW

Refer to "4.7.1 Simple Check Mode Drawings of XY VIEW".

Drawing images of FACE/BACK (when drilling both end surfaces)

ZX VIEW	XY VIEW selecting the front side of end surface	XY VIEW selecting the back side of end surface
		

Drawings of ZX/CZ/YZ VIEW

Refer to "4.7.1 Simple Check Mode Drawing of ZX/CZ/YZ VEIEW".

Restrictions on the graphic display function

- Graphic display is also available for the EIA process.
- When there is an error in the specified shape data for the turning/copy cutting/contour cutting, the shape data is displayed up to the error point. In this case, the [CHECK CONT] or the [CHECK STEP] menu cannot be pressed.
- Whether the graphic mode is in the foreground (70V) or the background (M700VM/M700VS), the NC check is available even during the graphic check on the 700 HMI screen.
- When the NC check of the NAVI LATHE is executed during cycle start with the foreground (70V) graphic check mode, an error message "E294 Program running" appears. In this case, the program shape is drawn, but the [CHECK CONT] or [CHECK STEP] menu cannot be pressed.

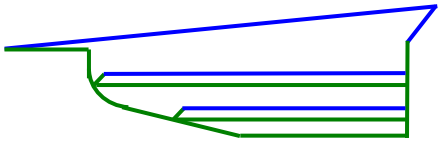
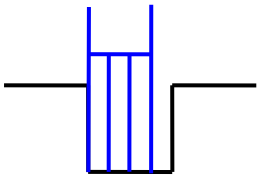
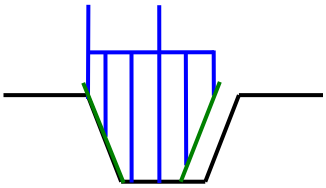
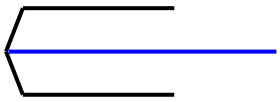
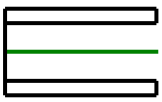
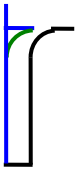
	M70V/E70	700VS/700VW	Remarks
Graphic check mode	Foreground	Background	
In automatic operation	NC check NG	NC check OK	
In emergency stop	NC check NG	NC check OK	
In the HMI graphic check	NC check OK	NC check OK	
Emergency stop during the NAVI2D check	NC check stop	No impact	
NC reset during the NAVI2D check	NC check stop	No impact	
Automatic operation during the NAVI2D check	Operation alarm	No impact	

- When the cycle start button is pressed during the NC check of the NAVI LATHE with the foreground (70V) graphic check mode, an operation alarm "M01 Program check mode" appears and the cycle start is disabled.

Examples of graphic drawings

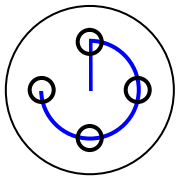
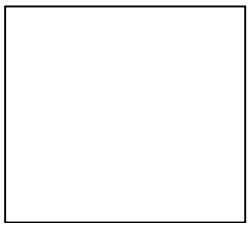
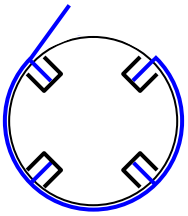
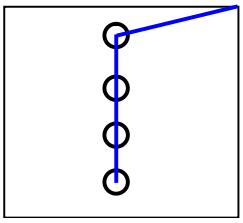
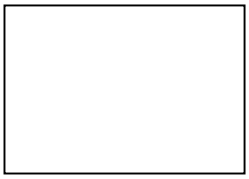
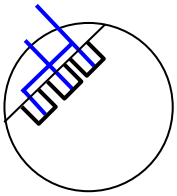
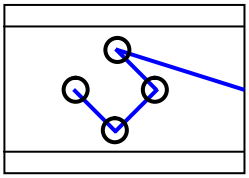
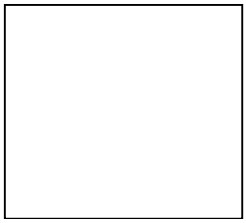
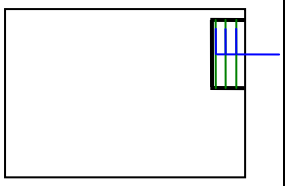
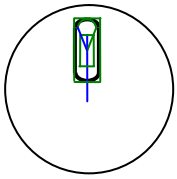
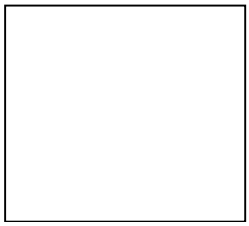
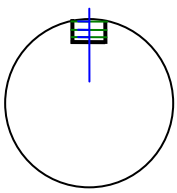
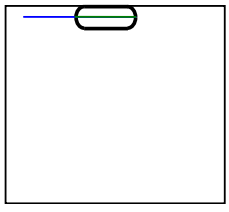
[Turning]

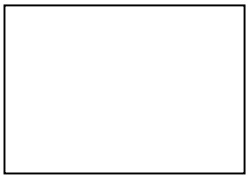
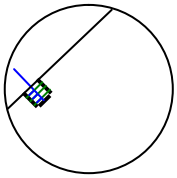
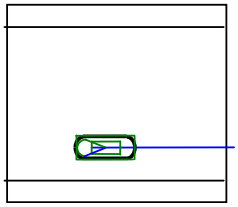
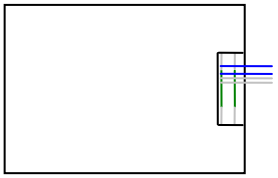
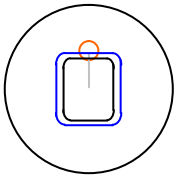
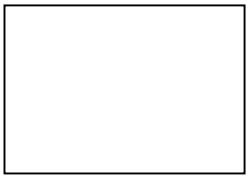
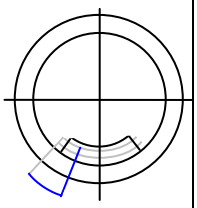
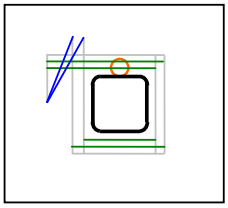
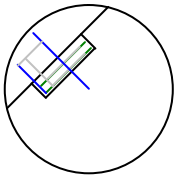
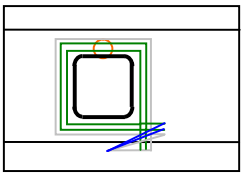
Only Z-X VIEW is displayed for turning.

[Turning] 	[Threading] 
[Grooving] 	[Trapezoidal grooving] 
[Hole drilling ---Drilling---] 	[Hole drilling ---Tapping---] 
[Cut off] 	

[Milling]

For milling process, machining shapes are displayed on the views that correspond to each machining area.

<table>
<tr><th></th><th></th><th>Z-X VIEW</th><th>X-Y VIEW</th><th>Y-Z VIEW</th><th>C-Z VIEW</th></tr>
<tr><td rowspan="6">Milling hole drilling</td><td>FACE/
BACK</td><td>  </td><td>  </td><td>  </td><td>  </td></tr>
<tr><td colspan="5">Machining shapes are not displayed on Y-Z or C-Z VIEW.</td></tr>
<tr><td>OUT</td><td>  </td><td>  </td><td>  </td><td>  </td></tr>
<tr><td colspan="5">Machining shapes are not displayed on Z-X or Y-Z VIEW.</td></tr>
<tr><td>SIDE</td><td>  </td><td>  </td><td>  </td><td>  </td></tr>
<tr><td colspan="5">Machining shapes are not displayed on Z-X or C-Z VIEW.</td></tr>
<tr><td rowspan="6">Keyway cutting</td><td>FACE/
BACK</td><td>  </td><td>  </td><td>  </td><td>  </td></tr>
<tr><td colspan="5">Machining shapes are not displayed on Y-Z or C-Z VIEW.</td></tr>
<tr><td>OUT</td><td>  </td><td>  </td><td>  </td><td>  </td></tr>
<tr><td colspan="5">Machining shapes are not displayed on Z-X or Y-Z VIEW.</td></tr>
</table>

		Z-X VIEW	X-Y VIEW	Y-Z VIEW	C-Z VIEW
Keyway cutting	SIDE				
	Machining shapes are not displayed on Z-X or C-Z VIEW.				
Contour cutting (Note 1)	FACE/ BACK				
	Machining shapes are not displayed on Y-Z or C-Z VIEW.				
	OUT				
	Machining shapes are not displayed on Z-X or Y-Z VIEW.				
	SIDE				
	Machining shapes are not displayed on Z-X or C-Z VIEW.				

(Note) If the following option/parameter are not set as shown below, the FACE/BACK/OUT/SIDE drawings in contour cutting are not traced properly.

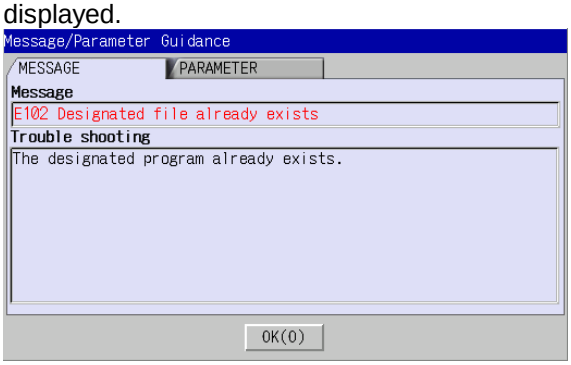
Option	Setting value
Graphic check	ON

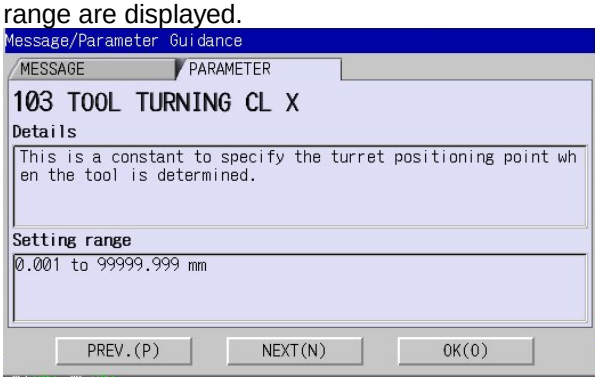
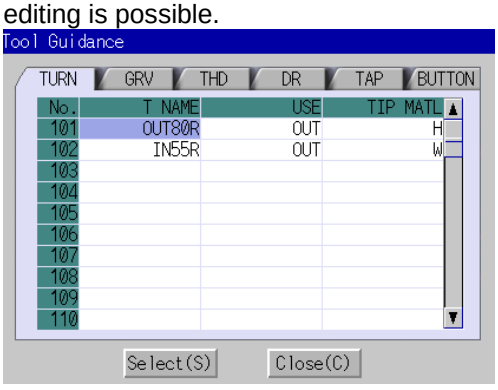
Parameter	Setting value
#19405 Rotary ax drawing	C

4.8 Guidance Function

Guidance Function helps an operator perform data inputting.
Guidance Function includes Message Guidance and Tool Guidance. Message Guidance screen will be appeared by pressing [/] key or by clicking the icon , and Tool Guidance screen will be appeared by pressing [LIST] key or by clicking the icon . The message/parameter guidance screen will be closed by clicking [OK] , and the tool guidance screen will be closed by clicking [Close] button.

<table>
<tr><th rowspan="2">Guidance Type</th><th colspan="2">Starting method</th><th rowspan="2">Details</th></tr>
<tr><th>Key-board</th><th>Icon</th></tr>
<tr><td>Message Guidance</td><td>[?]</td><td></td><td>When selecting the parameter screen, the parameter guidance is displayed.
When not selecting the parameter screen, the message guidance is displayed.

<Message guidance>
Details or countermeasures related to the current error and message are displayed.
 

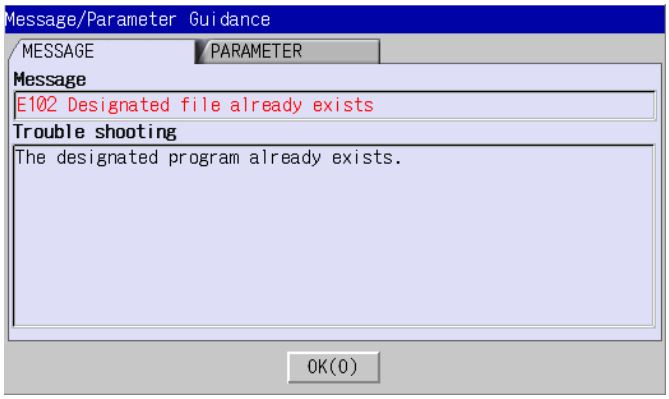
<Parameter guidance>
For the parameter selected with the cursor, the detail and the setting range are displayed.
 
</td></tr>
<tr><td>Tool Guidance</td><td>[LIST]</td><td></td><td>A segment of tool data registered in the tool file is displayed. Note that no editing is possible.
 
</td></tr>
</table>

4.8.1 Message/Parameter Guidance Screen

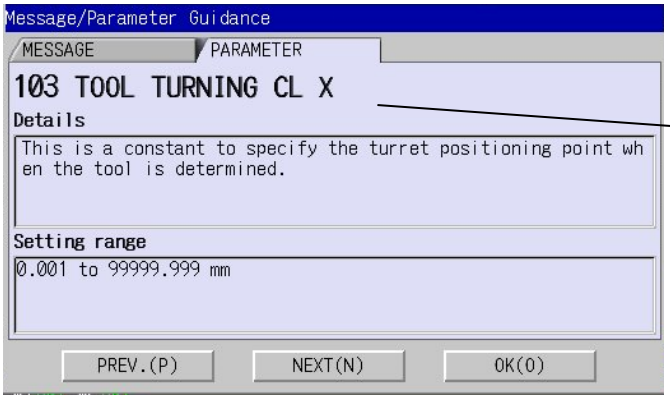
The message guidance or the parameter guidance is displayed in this function.
When the parameter screen is selected, the parameter guidance is displayed, and when the other screen other than the parameter is selected, the message guidance is displayed. The message guidance or the parameter guidance is switched by the tab.

Screen layout

<Alarm guidance>



<Parameter guidance>



Screen display items

No.	Display item	Details
1	Message	Displays the message of the alarm occurring currently.
2	Trouble shooting	Displays the indication for the alarm currently occurring.
3	Parameter name	Displays the parameter name. At the time of the parameter guidance start, the selected parameter is displayed.
4	Details	Indicates the explanation of the displaying parameter. When the message exceeds five lines, the scroll bar will appear.
5	Setting range	Indicates the setting range of the displaying parameter.

Buttons

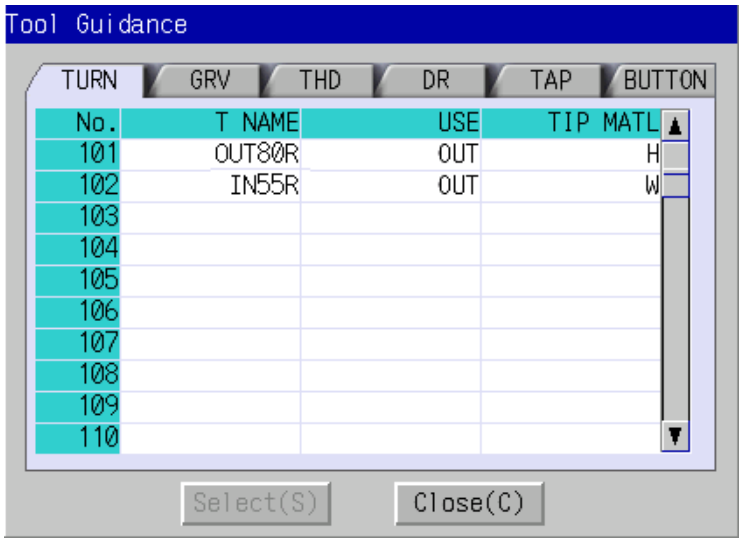
No.	Button	Details
1	OK(O)	Closes the guidance window.
2	PREV.(P)	Goes back to a previous parameter. When the first parameter is displayed, the letters are grayed and the button is disabled.
3	NEXT(N)	Goes to a next parameter. When the last parameter is displayed, the letters are grayed and the button is disabled. (Note) The last parameter differs according to the presence or the absence of the 2-part system specification. When 2-part system specification is “1: EXIST”: “1112 TOOL SPINDLE NO” When 2-part system specification is “0: NONE: “1026 SUB SP SELECT M CODE”

4.8.2 Tool Guidance Screen

The primary data of tool data registered with the tool file of the part system selecting are displayed.

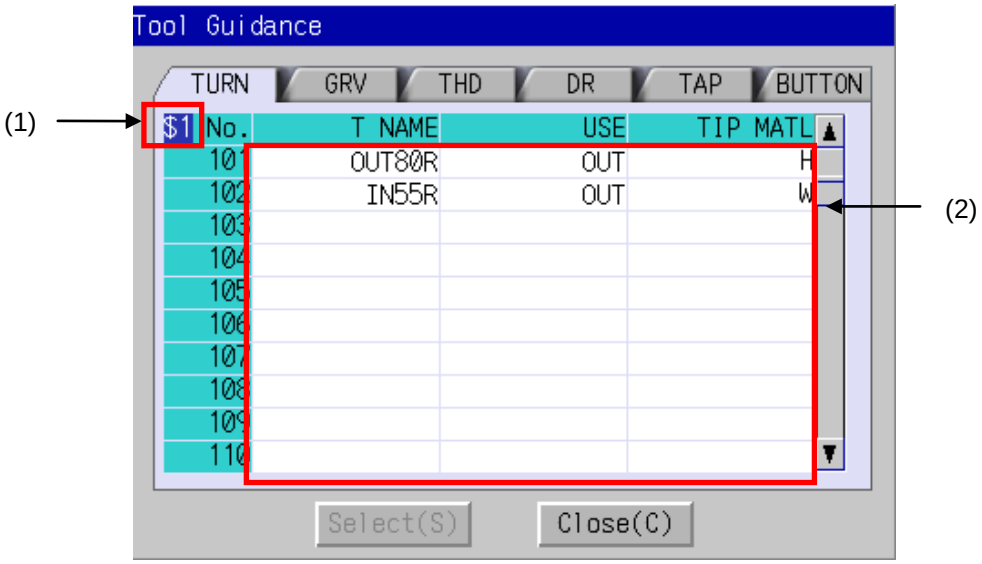
Screen layout

When the 2-part system specification is “0: NONE”
- Displaying the tools for turning

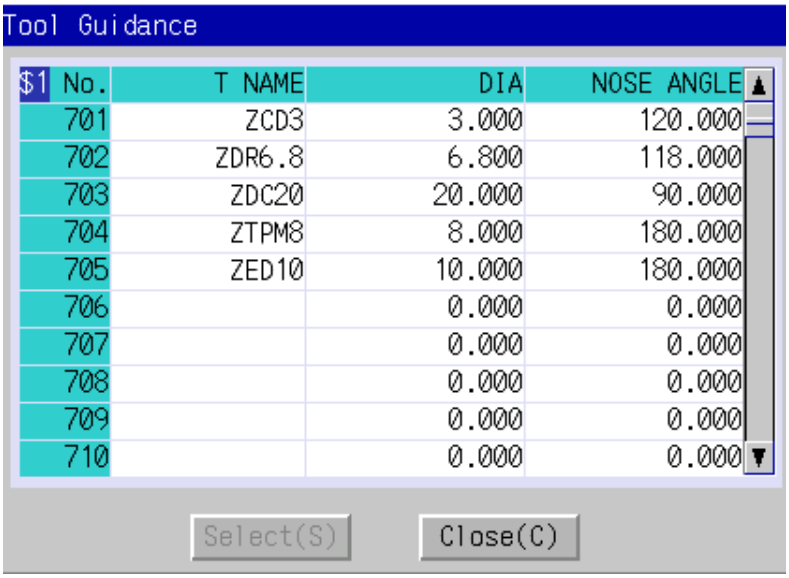


- Displaying the tools for milling

When the 2-part system specification is “1: EXIST”
- Displaying the tools for turning



- Displaying the tools for milling



Screen display items

No.	Display item	Details
1	Part system of tool data displayed	Indicates the part system number of the tool data displayed. (Note) This is available only when the 2-part system specification is "1: EXIST".
2	Tool file list	Indicates the tool data of the part system displayed (Note 1) on the screen. The tool No. of the 2nd part system is represented with the tool No. of the 1st part system plus 1000. Tool No. of 1st part system: 101 to 650 Tool No. of 2nd part system: 1101 to 1650 (Note) The tool data of the 2nd part system is available only when the 2-part system specification is "1: EXIST".

(Note 1) The part system displayed differs according to the conditions as follows.

Condition	Judgment point of part system	Selecting part system	Part system of tool guidance to be displayed.
When 2-part system is "1: EXIST" and multi-part system program management is ON.	Selected part system in the LIST VIEW	$1	$1
		$2	$2
When 2-part system is "1: EXIST" and multi-part system program management is OFF.	Selected part system in the initial setting screen.	$1	$1
		$2	$2

However, this displays the tool data of $1 when the cursor locates at the “TOOL REG No. $1”, and displays the tool data of $2 when the cursor locates at the “TOOL REG No. $2” in the balance cut process.

Buttons

No.	Button	Details
1	SELECT(S)	The tool registration No. at the cursor position is set to "TOOL REG No." in the process editing screen. This button is valid only when the tool guidance screen is opened while the cursor is at the "tool registration No." in the process edit screen. (Note 2)
2	CLOSE(C)	Closes the tool guidance window.

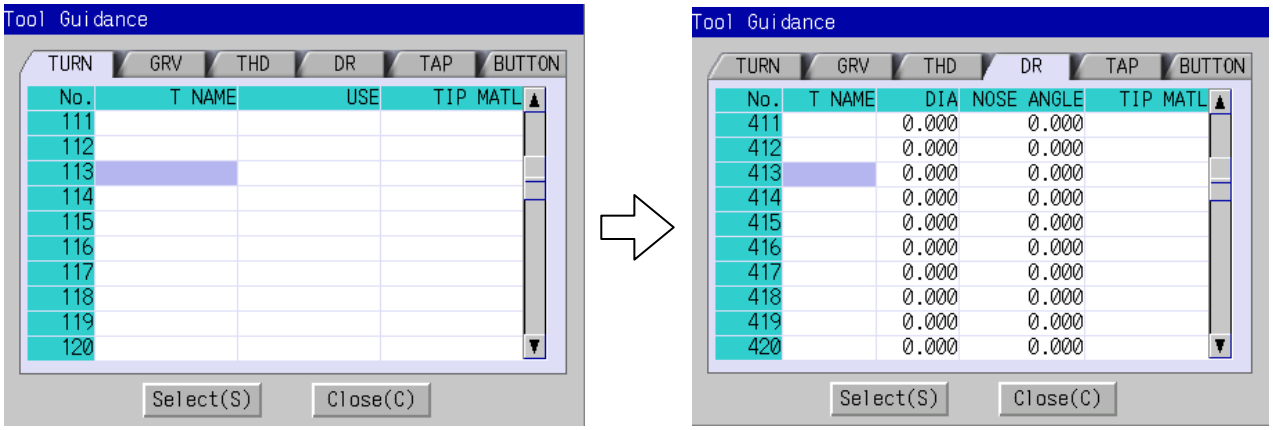
(Note 2) The initial display of tool guidance differs according to the active screen before opening the guidance as follows.

No.	Active screen before opening		Initial display of the tool guidance	
			Guidance type	Displaying tool
1	LIST VIEW	Program	For turning	Turning tool
2		PROCESS		
3		Initial setting		
4		Turning		Turning tool or button tool
5		Copy cutting		
6		Grooving		Grooving tool
7		Trapezoidal grooving		

No.	Active screen before opening		Initial display of the tool guidance	
			Guidance type	Displaying tool
8	LIST VIEW	Threading		Threading tool
9		Hole drilling		Drilling tool or tapping tool
10		EIA		Turning tool
11		Cutting off		Grooving tool
12		Milling hole	For milling	—
13		Keyway cutting		
14		Contour cutting		
15		Transfer	For turning	Turning tool
16		Balance cut (Turn)		
17		Balance cut (Copy)		
18		Two-part system simultaneous thread cutting		Threading tool
19		A process which indicates the other part system is machining		Turning tool
20		Tool file		
21		Milling tool file	For milling	—
22		Cutting condition file	For turning	Turning tool
23		Milling cutting condition file	For milling	—
24		Parameter	For turning	Turning tool
25		Version		
26	OPERATION VIEW	Program editing screen	For turning	Turning tool
27		Process list screen		
28		System synchro screen		
29		Initial condition setting screen		
30		Turning screen		Turning tool or button tool
31		Copy cutting screen		
32		Grooving screen		Grooving tool
33		Trapezoidal grooving screen		
34		Threading screen		Threading tool
35		Hole drilling screen		Drilling tool or tapping tool
36		EIA screen		Turning tool
37		Cutting off screen		Grooving tool
38		Milling hole drilling screen	For milling	—
39		Keyway cutting screen		
40		Contour cutting screen		
41		Transfer process screen	For turning	Turning tool
42		Balance cut (Turn) process screen		
43		Balance cut (Copy) process screen		

No.	Active screen before opening		Initial display of the tool guidance	
			Guidance type	Displaying tool
44	OPERATION VIEW	Two-part system simultaneous thread cutting screen		Threading tool
45		Brank process screen		Turning tool
46		Tool file screen		
47		Milling tool file screen	For milling	—
48		Cutting condition file screen	For turning	Turning tool
49		Milling cutting condition file screen	For milling	—
50		Parameter screen	For turning	Turning tool

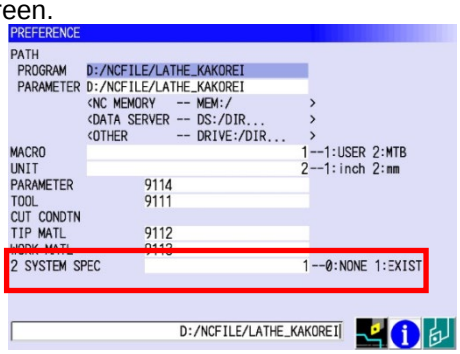
(Note 3) The cursor position does not move after changing the tool type.



The cursor and scroll bar stay the same position

5. 2-part System Function

The 2-part system function can be used when setting "2 SYSTEM SPEC" to exist on the Preference screen.



(Note) "2 SYSTEM SPEC" is enabled only when the number of part systems is 2 or more.

The 2-part system function allows the following machining.

(1) Independent machining in each part system
Machining is carried out only in the 1st part system or 2nd part system.
Available for all the conventional turning processes and milling processes.

(2) Timing synchronization machining between 2-part systems
2-part systems (turrets) are synchronized by each machining process and machined alternately.
Available for all the conventional turning processes and milling processes.
(Note) The timing synchronization machining between 2-part systems is only available for the following cases.
- When the multi-part system program management is enabled ("#1285 ext21/bit0" is ON)
- When the multi-part system program generation and operation is disabled ("#1285 ext21/bit2" is OFF)

(3) Balance cut machining
Machining is simultaneously carried out in 2-part systems.
Available only for the balance cut dedicated process (turning balance cut, copying balance cut, and two-part system simultaneous thread cutting).
(Note) Balance cut machining is only available for the following cases.
- When the multi-part system program management is enabled ("#1285 ext21/bit0" is ON)
- When the multi-part system program generation and operation is disabled ("#1285 ext21/bit2" is OFF)

The machining process corresponding to each machining is as follows.

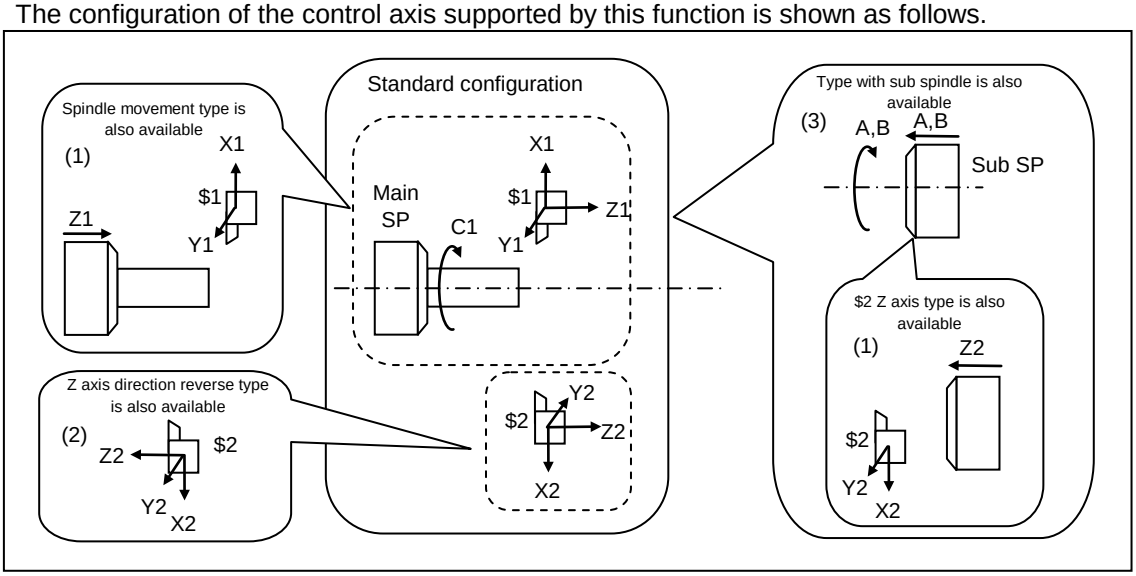
No.	Machining process	Independent machining	Synchronization machining	Balance cut	Remarks
1	Turning Processes				
-1	Turning	○	○	○	
-2	Copy cutting	○	○	○	
-3	Grooving	○	○		
-4	Trapezoidal grooving	○	○		
-5	Threading	○	○	○	There are following restrictions for balance cut. - Machining area: Only the outer dia. and inner dia. - Cut method: Only "Constant area-normal"
-6	Hole drilling	○	○		
-7	Cutting off	○	○		
2	Milling				
-1	Milling hole	○	○		
-2	Keyway cutting	○	○		
-3	Contour cutting	○	○		
3	Others				
-1	Workpiece transfer	○	○		

(Caution)

- The machining program created with NAVI LATHE is for a part system when the multi-part system program management is disabled.
- NAVI macro program is registered only for the 1st part system because of the NC memory capacity. Accordingly, when the multi-part system program generation and operation are enabled ("#1285 ext21/bit2" is ON), a program error occurs because the 2nd part system and the following cannot call an NAVI LATHE macro program.
  Therefore, it is required to disable the multi-part system program generation and operation to operate the program for the 2nd part system.

5.1 Control Axis Configuration

The configuration of the control axis supported by this function is shown as follows.



The following axis configuration is the standard for this function.

- Workpiece spindle: 1 axis (main spindle)
- Tool spindle: Up to 2 axes (1 axis for each system)
- 1st part system control axis: Up to 4 axes (X, Z, Y, and C) *Y and C are used when milling function is enabled. The turret is moved by the Z axis command.
- 2nd part system control axis: Up to 4 axes (X, Z, Y, and C) *Y and C are used when milling function is enabled. The turret is moved by the Z axis command.
  Z axis direction is the same as the 1st part system.
- Phase difference between 1st part system and 2nd part system: 180°

The following changes to the above are available.

(1) Change to spindle movement type

When the spindle moves according to the Z axis command, change the following NAVI dedicated parameter.

- #1103 $1 Z axis movement type, #1104 $2 Z axis movement type (Setting value: 1 = Turret movement type, 2 = Spindle movement type)
  * Default setting is 1.

Some spindle movement types using the mixed synchronization control or control axis superimposition of NC are required to set the NC parameter separately. Refer to "7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS" for the setting of NC parameter.

(2) Change to Z axis direction reverse type

When the Z axis direction of the 2nd part system is reverse to the 1st part system, change the following parameter.

- #1105 $2 Z axis direction (Setting value: 1 = Same direction with the 1st part system, 2 = Reverse direction to the 1st part system) * Default setting is 1.
  (Note) Z axis direction of the 1st part system is fixed.

(3) Change to the type with sub spindle

For the type with sub spindle, change the following parameter.

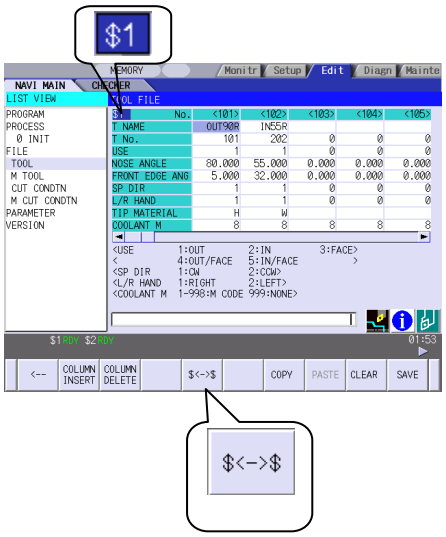
- #1001 Sub spindle specification (Setting value: 0 = NONE 1 = EXIST) * Default setting is 0
  (Note) Separately setting the parameter #1002 or later is also required.

The machining on the sub spindle side is available for the type with sub spindle.
Note that the simultaneous machining of the both spindles, which the workpieces are set on both of the main spindle and sub spindle, is not available.

5.2 Editing Tool Data

When the 2-part system specification is enabled, the part system changeover of the tool data is available on the tool file screen and mill tool file screen.
Select the part system to input the tool data with the [$<->$] menu, and input the tool data.
The selected part system is displayed on the upper left of the data input table.

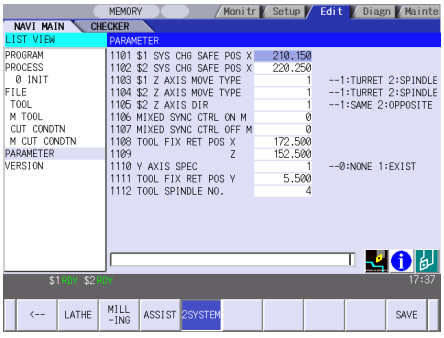
Example of tool file screen



Refer to "4.4.1 Tool File Screen for Turning" and "4.4.2 Tool File Screen for Milling" for details.

5.3 Editing Parameter

When the 2-part system specification is enabled, the [2 SYSTEM] menu on the parameter screen is enabled and parameters for 2-part systems can be set.



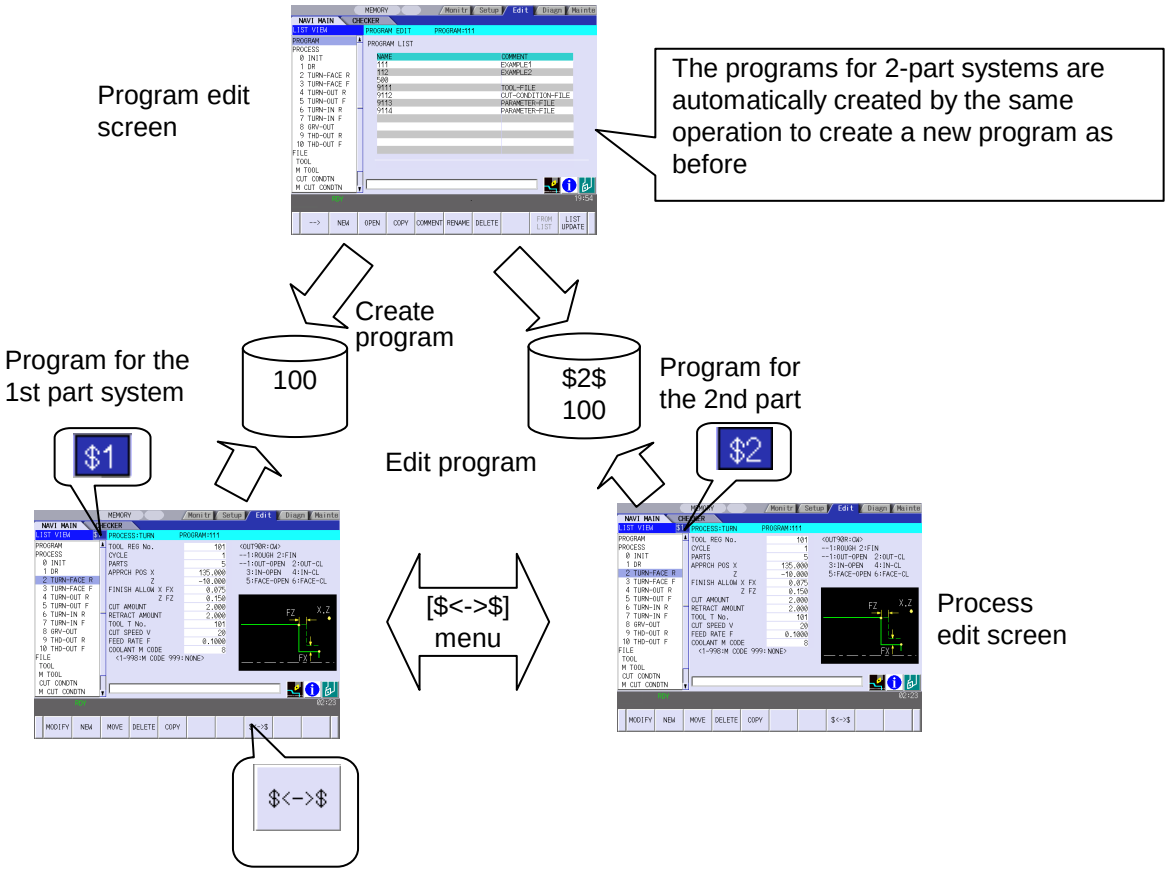
Refer to "4.5.1 Parameter Screen" for details.

5.4 Editing 2-part System Program

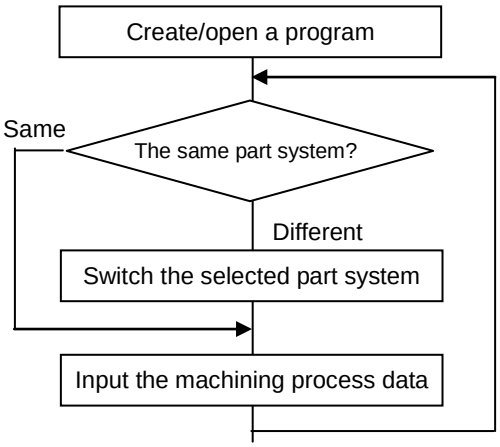
How to edit a program changes depending on whether the multi-part system program management is enabled or disabled.

When the multi-part system program management is enabled

When creating a new program on the program edit screen, programs for 2-part systems are created.
When opening an existing program, programs for 2-part systems are opened.
Select the part system to be edited with the [$<->$] menu, and edit the machining process.



Program with the following procedure when the multi-part system program management is enabled.



(1) Creating or opening a program

The operation method for the program edit screen is the same as when the 2-part system specification is disabled.

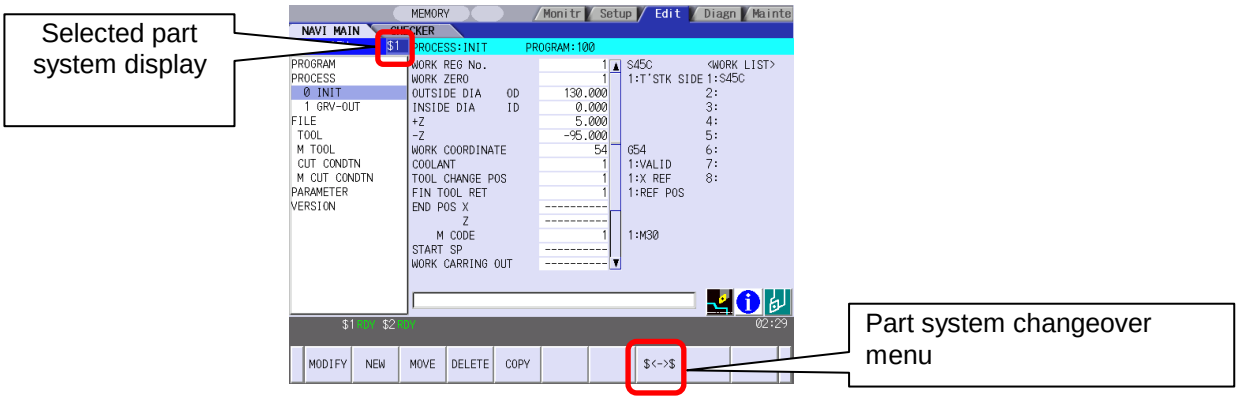
When creating a program with the [NEW] menu, a program for the 1st part system (the program with the specified name) and a program for the 2nd part system (the program with the name added "2" to the beginning of the specified one) are created.

If opening a program which only either part system exists on the [OPEN] menu, the program for not existing part system is automatically created.

The selected part system is switched to the 1st part system when creating or opening a program.

(2) Selecting the part system to edit

The selected part system is displayed at the top of the screen. The selected part system is switched by pressing the part system changeover menu ([$<->$]) which is displayed when activating LIST VIEW. (The selected part system switches from $1->$2->$1 … each time this menu key is pressed.)



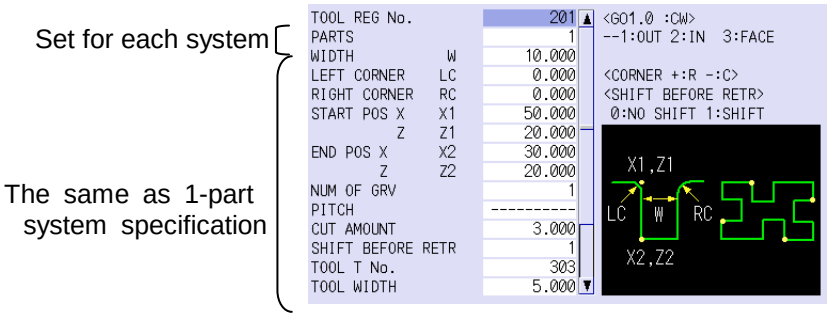
(3) Adding a machining process

A process is added to the program of selectd part system by pressing the [NEW] menu with activating LIST VIEW. When the balance cut process is added, it is also added to the program of non-selected part system. When a process other than the balance cut process is added, the process indicating that the other part system is in machining for timing synchronization is automatically inserted to keep the synchronization between the systems.

(4) Editing data of the machining process

- Turning process and milling process

Select the tool corresponding to the selected part system, and other editing method is the same as when the 2-part system specification is disabled.



- Assist process (Transfer process)

The operation method is the same as when the 2-part system specification is disabled.

- Balance cut process

Data other than tool registration No. and tool No. are common to both systems for the balance cut process. Therefore, the edited contents are saved to programs of both part systems regardless of the selected part system.

(5) Displaying LIST VIEW

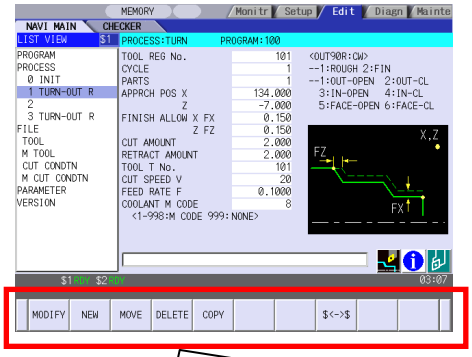
The contents of the selected part system are displayed on LIST VIEW.
The process indicating that the other part system is in machining is displayed as a blank based on the synchronization between the systems.

$1 LIST VIEW	**$2** LIST VIEW	**Remarks**
0 INIT	0 INIT	
1 DR	1 (Blank)	$1 operation only; $2 waiting
2 TURN-FACE R	2 (Blank)	
3 TURN-FACE F	3 (Blank)	
4 ! TURN-OUT R	4 ! TURN-OUT R	Balance cut ($1,$2 simultaneously)
5 ! TURN-OUT F	5 ! TURN-OUT F	
6 (Blank)	6 GRV-OUT	$2 operation only; $1 waiting
7 THD-OUT R	7 (Blank)	$1 operation only; $2 waiting
8 THD-OUT F	8 (Blank)	

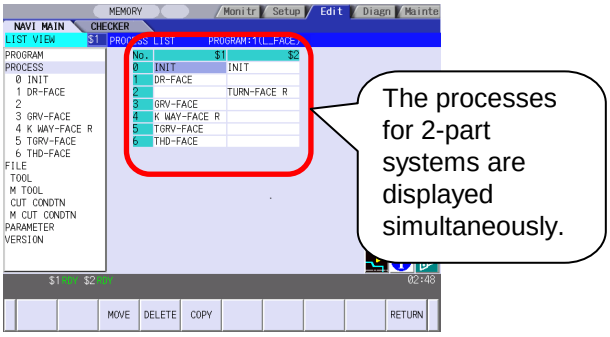
(6) Editing machining order

The machining order and the part system to machine can be changed after creating a program.
This operation is carried out on LIST VIEW menu or System synchro screen.

LIST VIEW menu



System synchro screen



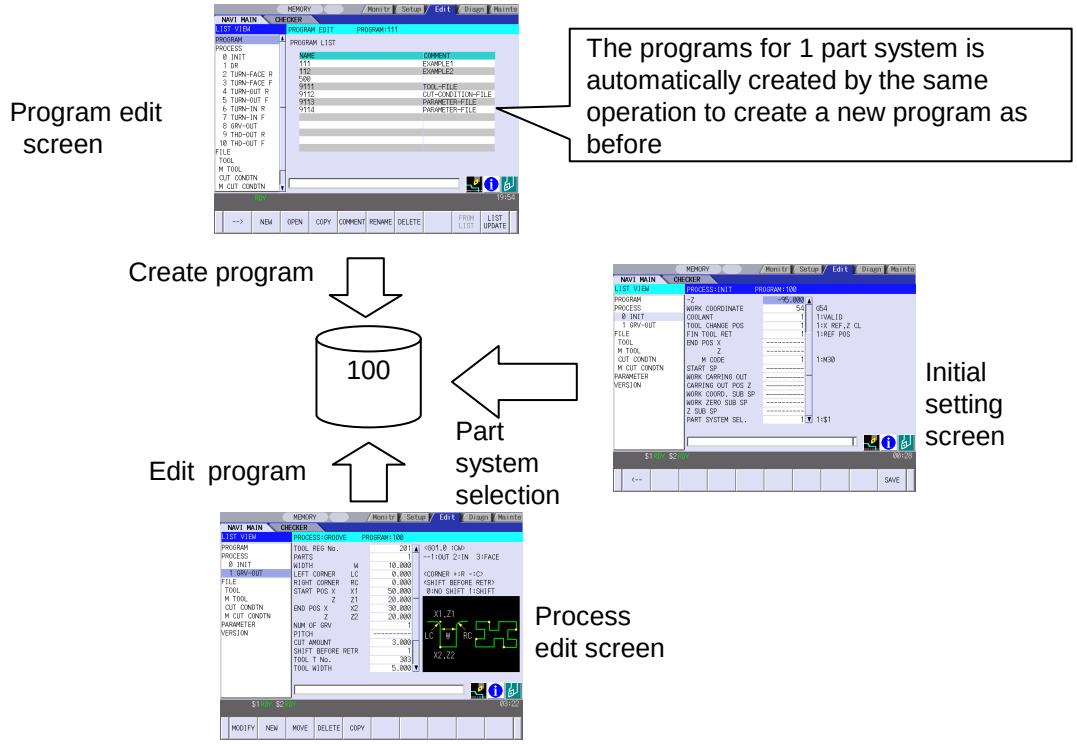
The processes for 2-part systems are displayed simultaneously.

This allows operating in the same way as when the 2-part system specification is disabled.

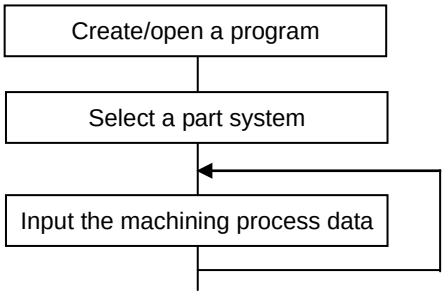
Refer to "Operating Process" and "System Synchro Screen" for details.

When the multi-part system program management is disabled

When creating a new program on the program edit screen, a program for one part systems is created.
The machining processes are edited by the selected machining part system on the initial setting screen.



Program with the following procedure when the multi-part system program management is disabled.



(1) Creating or opening a program

The operation method for the program edit screen is the same as when the 2-part system specification is disabled.

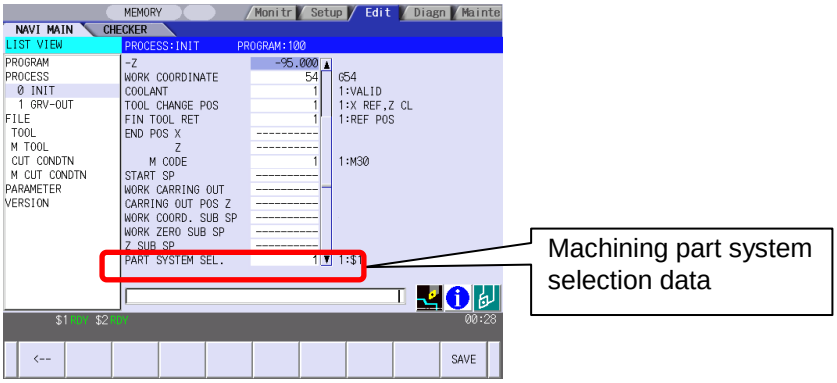
When creating a program with the [NEW] menu, a program for one part system is created.

When creating a program with the [NEW] menu, the program is created in the condition of the first part system selected.

(2) Selecting the part system to edit

Select the part system to edit at the setting item "PART SYSTEM SEL." on the initial setting screen. (1: $1 (1st part system), 2: $2 (2nd part system))

(Note) The selected part system is not displayed in the upper screen.



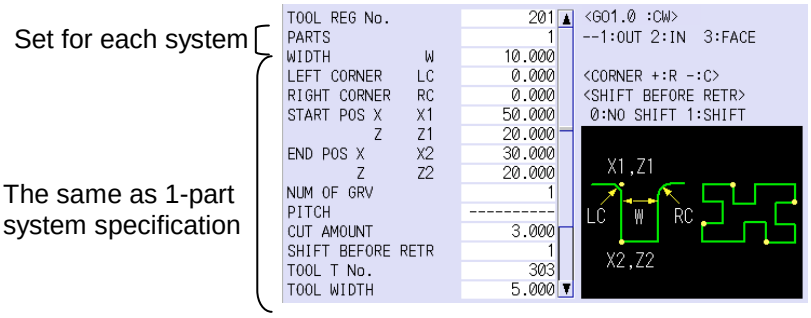
(3) Adding a machining process

The operation method is the same as when the 2-part system specification is disabled.

(4) Editing a machining process

・Turning process and milling process

Select the tool corresponding to the selected part system, and other editing method is the same as when the 2-part system specification is disabled.



・Assist process (Transfer process)

The operation method is the same as when the 2-part system specification is disabled.

(Note) The balance cut process cannot create when the multi-part system program management is disabled.

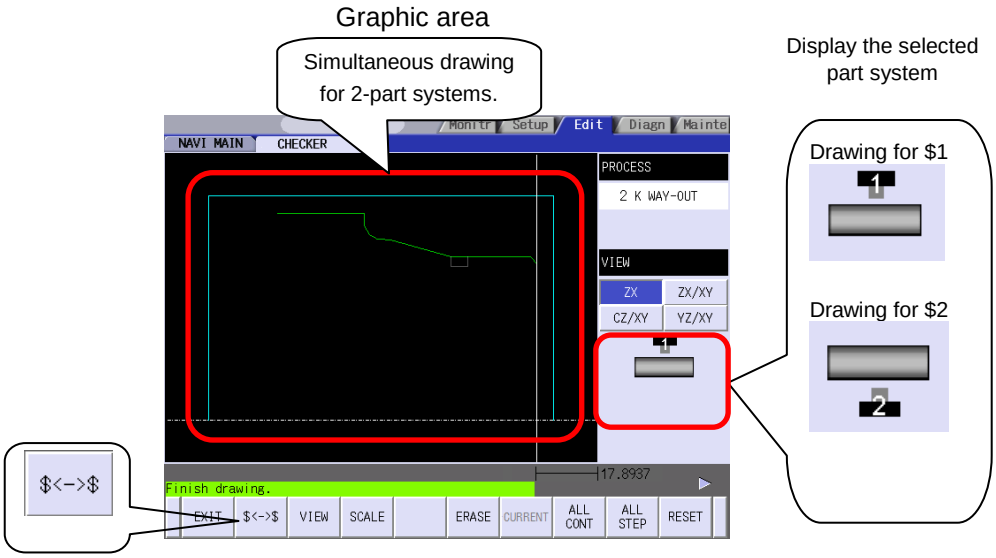
(5) Displaying LIST VIEW

The displaying method is the same as when the 2-part system specification is disabled.

5.5 Check for 2-part System Programs (Checker Function)

When 2-part system specification is enabled, the drawing on the checker screen is performed for the 2-part systems.

When the 2-part system specification is enabled, the machining shapes are drawn for 2-part systems on the checker screen.
The drawing of each part system is identifiable with a color.
・Selected part system: drawn in green
・Non-selected part system: drawn in gray
The selected part system is indicated on the right side of the screen with the icon, and it switches at the [$<->$] menu.

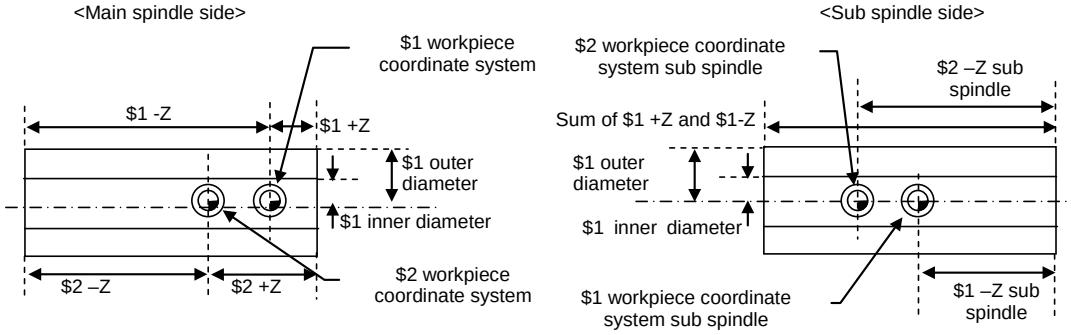


Refer to "4.7 Program Checker Screen" for details.

5.6 Machining Motion

5.6.1 About the Setting of the Work Coordinate System

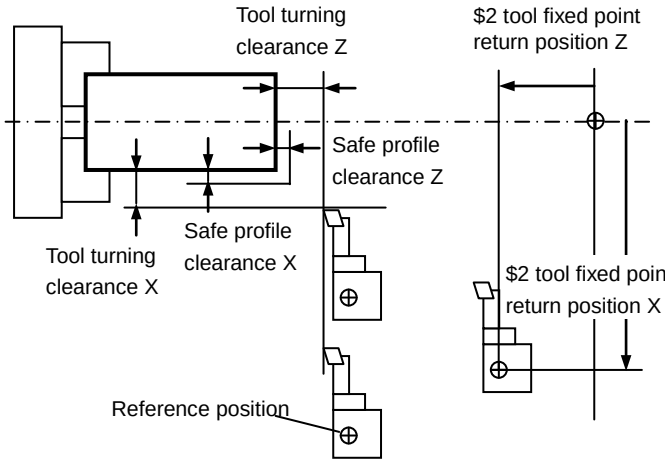
The workpiece coordinate zero point is set for every part system.
It is possible to set a different zero point position for each part system. But when machining the balance cut process, the workpiece coordinate zero point of both part systems need to be the same.



5.6.2 Independent Machining at Each Part System

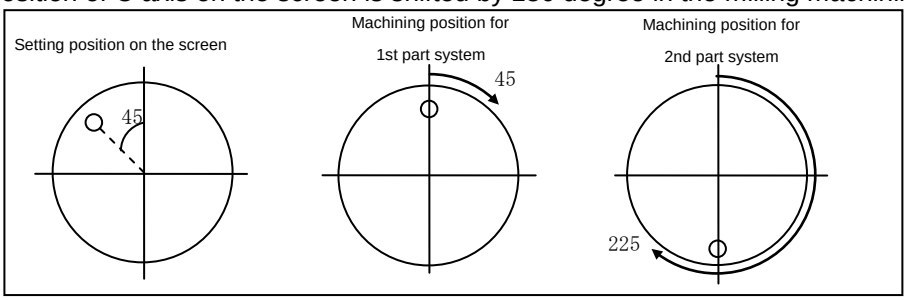
(1) Machining motion for the 1st part system
The machining motion for the 1st part system is the same as when the 2-part system specification is disabled.

(2) Machining motion for the 2nd part system
(a) Approach and escape motions
The clearances such as tool turning clearance, safe profile clearance, etc. use the same value as the 1st part system, and the tool fixed point return position uses another value. The movement order of the axis for the approach and escape is the same specification as the 1st part system.



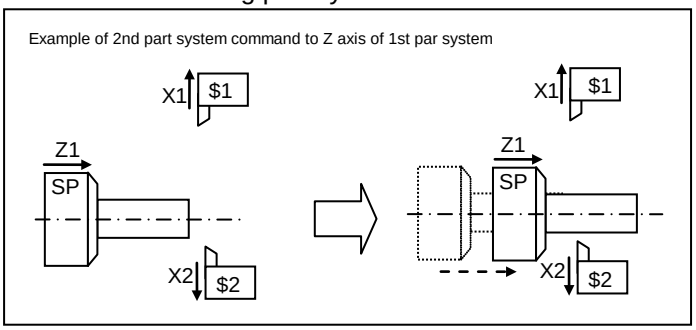
(b) Cutting motion

Cutting motion (motion after the approach to before the escape) is the same specification as the 1st part system. However, because of the phase difference with the 1st part system, the setting position of C axis on the screen is shifted by 180 degree in the milling machining.



The following case uses the mixed synchronous control. In that case, parameter setting of the NC is necessary.

・When the machining part system commands to Z axis of other part system



・When the machining part system commands to C axis of other part system (Milling machining)

The cases below also run automatic start to the non-machining part system in the program operation. In the result, each axis of non-machining part system returns the zero point, and moves to the part system change safe position.

・The multi-part system program management is enabled. (“#1285 ext21/bit0” is ON)

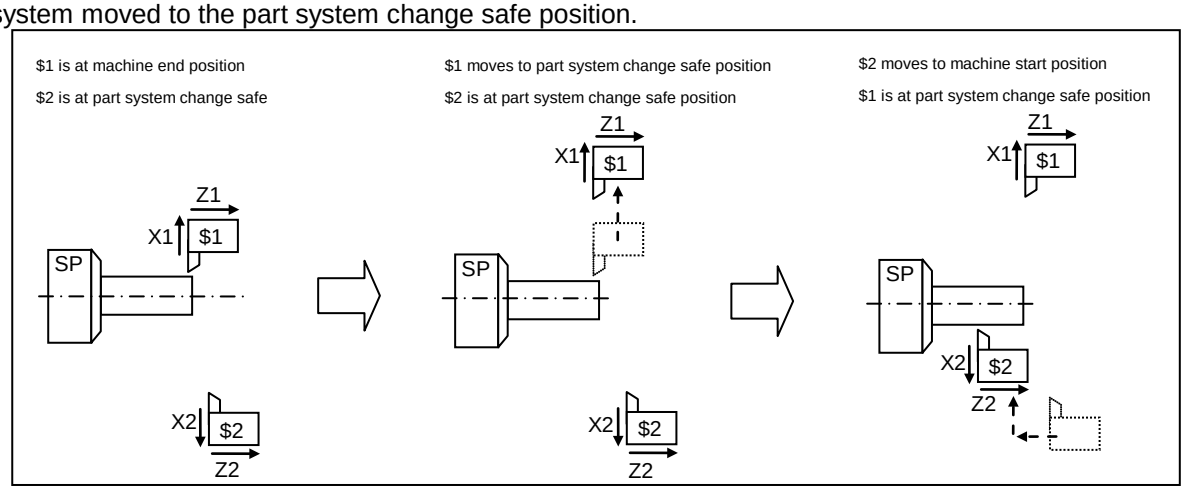
・The multi-part system program generation and operation is disabled. (“#1285 ext21/bit2” is OFF)

5.6.3 Timing Synchronization Process between 2-part Systems

2-part systems (turrets) are synchronized by each machining process and machined alternately.

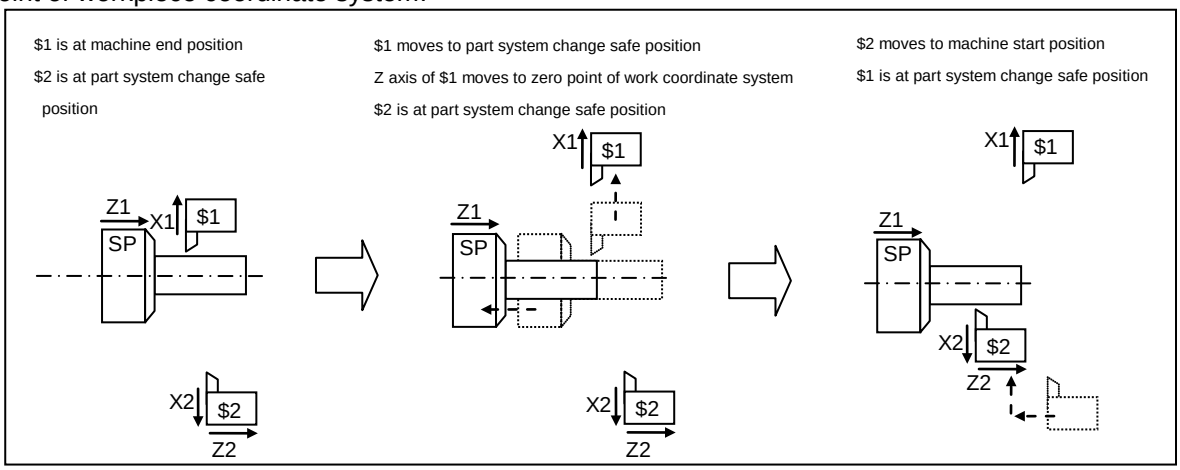
(1) Changing operation for machining part system

When changing the machining part system, the waiting part system moves to the part system change safe position. The machining part system moves to the machine start position after the waiting part system moved to the part system change safe position.



(Note) The part system change safe position is designated in the parameter (#1101 $1 SYS CHG SAFE POS X, #1102 $2 SYS CHG SAFE POS X).

For the spindle movement type, when changing the machining part system, Z axis moves to the zero point of workpiece coordinate system.



(2) Approach and escape motions

Same as the independent machining.

(3) Cutting motion

The cutting motion (after the approach to before the escape) is the same specification as the independent machining.

5.6.4 Balance Cut

Machining is simultaneously carried out in 2-part systems.
Available only for the balance cut dedicated process (turning balance cut, copying balance cut, and two-part system simultaneous thread cutting).
In Comparison with the each part system independent machining, this is able to set the cutting speed faster.

(1) Balance cut (turn)
The turning process is simultaneously carried out in 2-part systems.
The machining motions of each part system are the same specification as the independent machining.

(2) Balance cut (copy)
The copy cutting process is simultaneously carried out in 2-part systems.
The machining motions of each part system are the same specification as the independent machining.

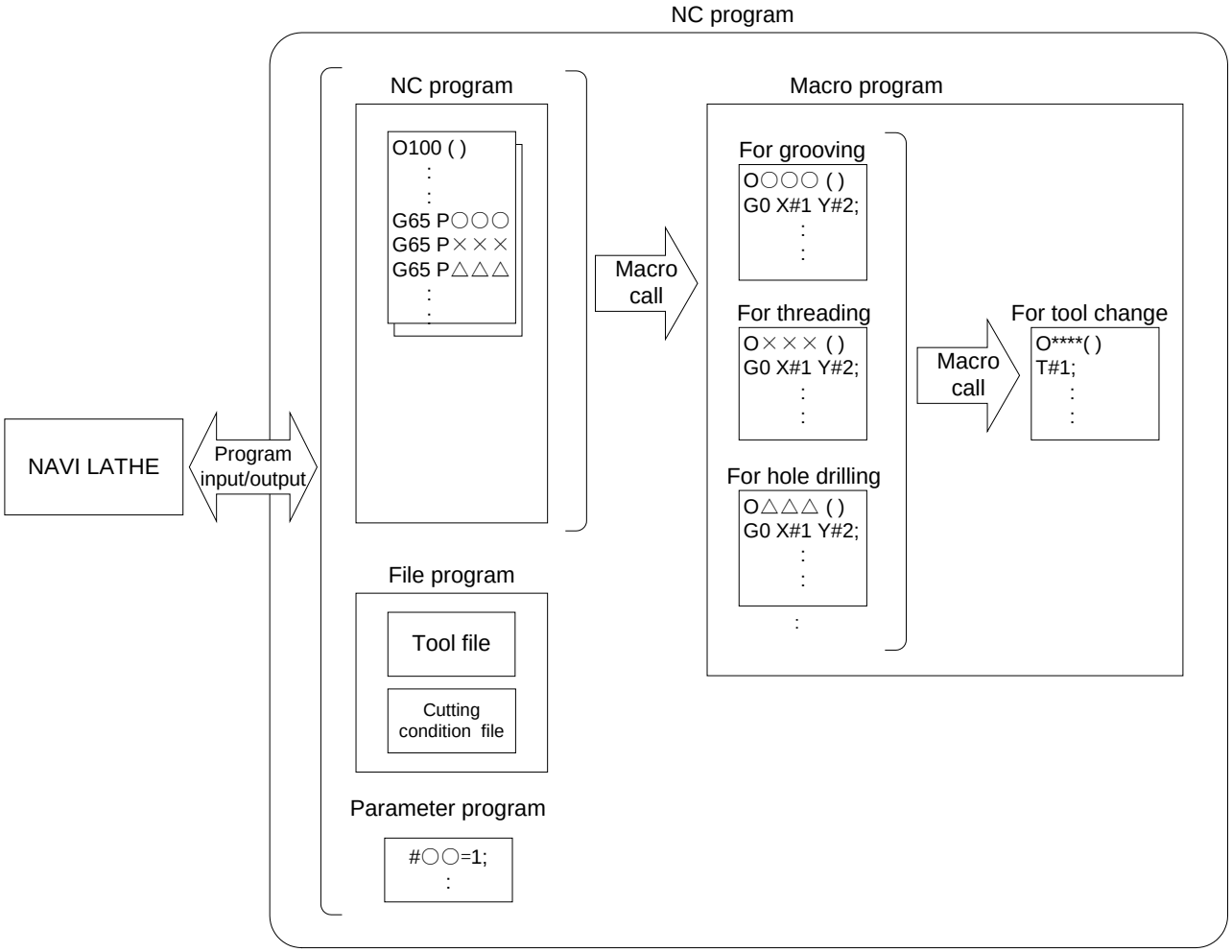
(3) Two-part system simultaneous thread cutting
The threading process is simultaneously carried out in 2-part systems.
Two-part system simultaneous thread cutting cycle II (G76.2) of NC is used.
Refer to “MITSUBISHI CNC M700V/M70V Series Programing Manual (Lathe System) IB-1500924” for the machine operations.

6. PROGRAM SPECIFICATIONS

The configuration of the program related to the NAVI LATHE is as shown below.

(1) NC program
(2) File program
(3) Miscellaneous parameter program
(4) Macro program

(Note) Macro program is registered in the NC memory of 700/70 series in which NAVI LATHE is installed.



6.1 NC Program

NAVI LATHE outputs the NC programs. The NC program No. ranges from 1 to 7999 or from 10000 to 99999999.

6.1.1 Output Method for NC Program

In the NAVI LATHE, the NC program is output in the process unit.
The output method for the NC program is as follows.

Process	Machining program	
Hole drilling (Drill Line)	(NAVI-HOLE-PECK); • • • (/NAVI);	Machining start comment Process data Process end comment
Turning (Outer diameter)	(NAVI-TURN-OUT); • • • (/NAVI);	
Turning (Face)	(NAVI-TURN-FACE); • • • (/NAVI);	
Grooving (Outer diameter)	(NAVI-GRV-OUT); • • • (/NAVI);	
Threading (Outer diameter)	(NAVI-THD-OUT); • • • (/NAVI);	
Milling hole drilling (Drill Front face)	• • • (NAVI-M HOLE-FACE-DRILL); • • • (/NAVI);	
Keyway cutting (Outer surface)	(NAVI-M KWAY-OUT); • • • (/NAVI);	
Contour cutting (Side surface)	(NAVI-M CONT-SIDE); • • • (/NAVI); • • •	

Process start comment

Process		Comment	Remarks
Initial setting		(NAVI-INIT);	The symbol which indicates the machining area is set in the **** part. OUT: Outer diameter IN: Inner diameter FACE: Front face BACK: Back surface
Turning		(NAVI-TURN-****)	
Copy cutting		(NAVI-COPY-****)	
Threading		(NAVI-THD-****)	
Trapezoidal grooving		(NAVI-TGRV-****)	
Hole drilling	Drilling	(NAVI-HOLE-DRILL);	
	Pecking	(NAVI-HOLE-PECK);	
	Boring	(NAVI-HOLE-BORE);	
	Tapping	(NAVI-HOLE-TAP);	
EIA		(NAVI-EIA);	
Milling hole drilling	Drilling	(NAVI-M HOLE-****-DRILL);	The symbol which indicates the machining area is set in the **** part. OUT: Outer surface SIDE: Side surface FACE: Front face BACK: Back surface
	Deep hole drilling	(NAVI-M HOLE-****-PECK);	
	Boring	(NAVI-M HOLE-****-BORE);	
	Tapping	(NAVI-M HOLE-****-TAP);	
Keyway cutting		(NAVI-M KWAY-****);	
Contour cutting		(NAVI-M CONT-****);	
Transfer		(NAVI-TRANS-TO-****);	The symbol which indicates the transfer direction is set in the **** part. MAIN: Transfer to the main spindle SUB: Transfer to the sub spindle SYNC: Spindle synchronization
Balance cut (turn)		(NAVI-!TURN-****);	The symbol which indicates the machining area is set in the **** part. OUT: Outer diameter IN: Inner diameter FACE: Front face BACK: Back surface
Balance cut (copy)		(NAVI-!COPY-****);	
Two-part system simultaneous thread cutting (identical screw)		(NAVI-!THD1-****);	
Waiting process		(NAVI-WAIT);	
End process		(NAVI-FIN);	

Process data

Process		Program block	Remarks
Initial setting		G65 P9110 A B C D E F・・・ Z;	Zero point return, spindle clamp
Turning	ROUGH	G65 P9120 C F I ・・・Z; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G71(72) U(W)_ R_ H_; G71(72) P_ Q_ U_ W_; N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; M5;	Movement to the tool change position, T command Workpiece coordinate system setting Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel.
	FIN	G65 P9120 C F I ・・・Z; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; M5;	Movement to the tool change position, T command Workpiece coordinate system setting Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel.

Process		Program block	Remarks
Copy cutting	ROUGH	G65 P9130 C F I ・・・Z; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G73 U_ W_ R_; G73 P_ Q_ U_ W_; N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; M5;	Movement to the tool change position, T command Workpiece coordinate system setting Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel
	FIN	G65 P9130 C F I ・・・Z; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; M5;	Movement to the tool change position, T command Workpiece coordinate system setting Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel
Threading		G65 P9140 A B C・・・Z;	
Grooving		G65 P9150 B C D・・・Z;	
Trapezoidal grooving		G65 P9160 A B C・・・Z;	
Hole drilling	DRILL PECK BORING TAP	G65 P9170 C D E・・・Z;	Common in drilling, pecking, boring and tapping.
EIA process		・・・;	
Cut off		G65 P9107 C D F・・・Z;	
Milling hole drilling	DRILL PECK BORING TAP	G65 P9171 C D F・・・Z;	Common in drilling, deep hole drilling, boring and tapping.
Keyway cutting		G65 P9155 A C D・・・Z;	

Process		Program block	Remarks
Contour cutting		・・・ G65 P9180 C D E・・・Z; G41(42); G0 X_ Y_; G0 Z_; G0 Z_ F_; F_; G1 X_ Y_; ・・・ G1 X_ Y_; G65 P9105 C; G40; G65 P9105 C; ・・・ G13.1; M5;	Polar coordinate interpolation mode ON Nose R compensation mode ON Movement to the approach point Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Z Nose R compen. mode cancel. Move. to the safe profile clearance position X Polar coordinate interpolation mode cancel
Transfer		G65 P9107 C D F・・・Z;	
Balance cut (turn)	ROUGH	G65 P9101 A B C E; G65 P9121 C F M ・・・Z; G65 P9105 C S M T; G15; G0 X_ Z_ F_; G41(42); G71(72) U(W)_ R_ H_; G71(72) P_ Q_ U_ W_; N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; G14; G65 P9105 C D T M; G65 P9105 C T;	Movement to the tool change position, T command Workpiece coordinate system setting Balance cut command ON Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel. Balance cut command OFF Superimposition control cancel Spindle stop M code output
	FIN	G65 P9101 A B C E; G65 P9121 C F M ・・・Z; G65 P9105 C S M T; G15; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; G14; G65 P9105 C D T M; G65 P9105 C T;	Movement to the tool change position, T command Workpiece coordinate system setting Balance cut command ON Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel. Balance cut command OFF Superimposition control cancel Spindle stop M code output

Process		Program block	Remarks
Balance cut (copy)	ROUGH	G65 P9101 A B C E; G65 P9131 C F I ・・・Z; G65 P9105 C S M T; G15; G0 X_ Z_ F_; G41(42); G73 U_ W_ R_; G73 P_ Q_ U_ W_; N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; G14; G65 P9105 C D T M; G65 P9105 C T;	Movement to the tool change position, T command Workpiece coordinate system setting Balance cut command ON Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel. Balance cut command OFF Superimposition control cancel Spindle stop M code output
	FIN	G65 P9101 A B C E; G65 P9131 C F I ・・・Z; G65 P9105 C S M T; G15; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G0 X_ Z_; ・・・ N_ G1 X_ Z_; N_ G65 P9105 C; G40; G14; G65 P9105 C D T M; G65 P9105 C T;	Movement to the tool change position, T command Workpiece coordinate system setting Balance cut command ON Movement to the approach point Nose R compensation mode ON Start point of the cutting shape End point of the cutting shape Move. to the safe profile clearance position Nose R compen. mode cancel. Balance cut command OFF Superimposition control cancel Spindle stop M code output
Two-part system simultaneous thread cutting (identical screw)		G65 P9101 A B C E; G65 P9146 A B C ・・・Z;	
End process		G65 P9190; M#156;	

(Note 1) Macro program No. (P***) in the table is used when user macro is selected. For the macro program No. used when MTB macro is selected, refer to the section "6.4 Macro Program".

(Note 2) The data that follows each address in the table is output at μm level.

(Note 3) The programs in the above table are some examples. The output program may change by the presence or absence of the milling interpolation, the NAVI parameter "1001 SUB SPINDLE SPEC" and the 2-part system specifications.

Process end comment

Process	Program block	Remarks
All processes are common.	(/NAVI);	

6.1.2 Restrictions

The NC program output from the NAVI LATHE can be edited with various commercially available editor tools. Note that there are the following restrictions.

(1) Deleting block

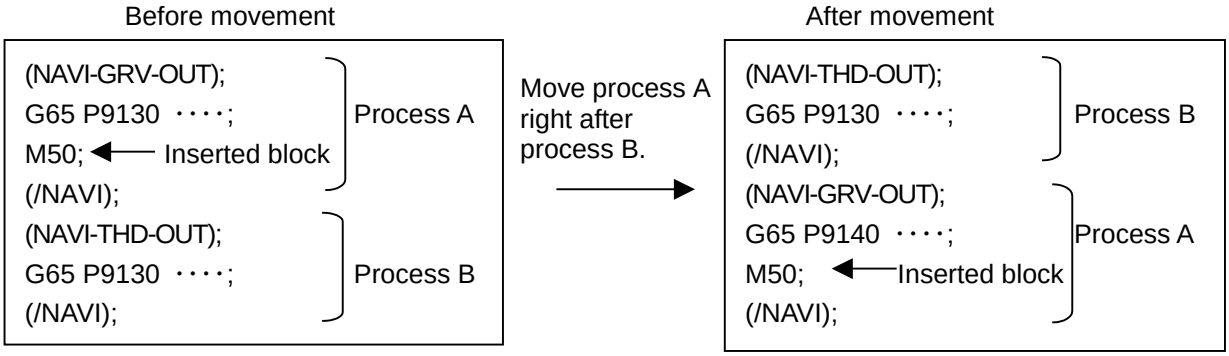
Deleting a block in the NC program process unit (process start comment to end comment) is no problem. However, if either block of process start comment, process data or process end comment is deleted, NAVI LATHE may not be able to edit the program. Do not delete any block of process start comment, process data or process end comment.

(2) Inserting block

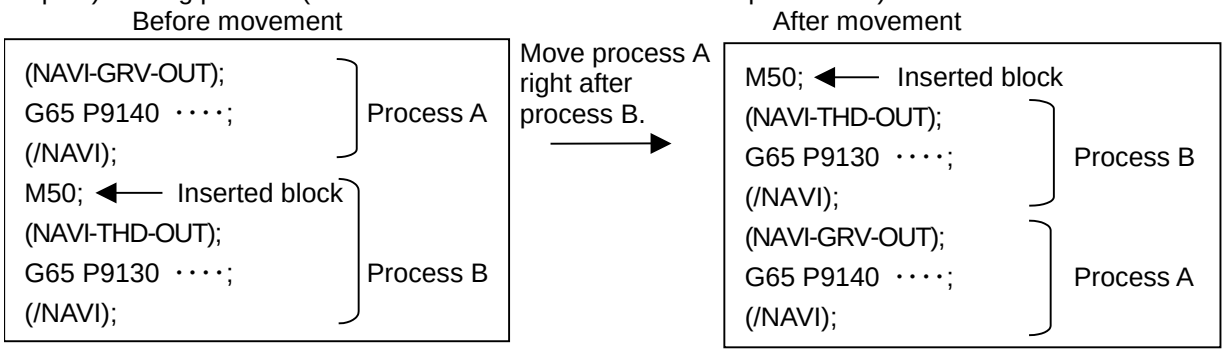
Inserting a block between the processes of the NC program (between the process end comment and next process start comment) is no problem.
If a block is inserted into the process of the NC program (between the process start comment and process end comment), the inserted block will not be recognized in most cases while NAVI LATHE is editing the process. Note that if NAVI MILL edits the process which a block is inserted into, the block may be lost.
In response to the operating process (moving process, deleting process, copying process) with NAVI LATHE, an inserted block is operated as follows.

Process operation	Inserted block in the process	Inserted block between the processes
Moving process	Moved with the process.	The inserted block is not moved.
Deleting process	Deleted with the process.	The inserted block is not deleted.
Copying process	Copied with the process.	The inserted block is not copied.

(Example1) Moving process (An inserted block exists in the process.)



(Example2) Moving process (An inserted block exists between the processes.)



(3) Changing process data

If the contents of the macro program call block in the process data is changed, editing the program with the NAVI LATHE may be disabled. Therefore, do not change the contents of the macro program call block in the process data.

6.2 File Program

This program is used to store the contents of each NAVI LATHE file.

<Program No., Comment>

No.	Name	User macro No.	MTB macro No.	Program comment
1	Tool file	9111	100019111	TOOL FILE
2	Cutting condition file (Tip material)	9112	100019112	CUT CONDITION FILE TIP
3	Cutting condition file (Workpiece material)	9113	100019113	CUT CONDITION FILE TIP WORK

(Note 1) Tool files and cutting condition files are saved via "parameter path" specified in the PREFERENCE screen.
(Note 2) Tool files and cutting condition files are saved under the file name specified in the PREFERENCE screen.

6.3 Parameter Program

This program is used to store the contents of the NAVI LATHE's parameters.

<Program No., Comment>

No.	Name	User macro No.	MTB macro No.	Program comment
1	Parameter	9114	100019114	PARAMETER

(Note 1) Parameters are saved via "parameter path" specified in the PREFERENCE screen.
(Note 2) Parameters are saved under the file name specified in the PREFERENCE screen.

6.4 Macro Program

This program is called from the NC program.
(Macro program will be registered in the NC memory of 700/70 Series in which NAVI LATHE is installed.)

<Program No., Comment>

No.	Name	User macro No.	MTB macro No.	Program comment
1	Macro program for waiting process	9101	100019101	WAIT-MACRO
2	Macro program for tool change	9102	100019102	TOOL-CHANGE-MACRO
3	Macro program for parameter setting	- 9108	100019104	PARAM-SET-MACRO
4	Macro program for variable control	9105	100019108	VARIABLE-CTRL-MACRO
5	Macro program for transfer process	9106	100019105	WORK-TRANS-MACRO
6	Macro program for cutting off process	9107	100019106	CUTOFF-MACRO
7	Macro program for workpiece coordinate system setting	9109	100019107	WORK-COORD-SET-MACRO
8	Macro program for INIT process	9110	100019109	INIT-MACRO
9	Macro program for turning process	9120	100019110	TURN-MACRO
10	Macro program for balance cut (turn) process	9121	100019120	TURN-BALANCE-MACRO
11	Macro program for copy-cutting process	9130	100019121	COPY-MACRO
12	Macro program for balance cut (copy) process	9131	100019130	COPY-BALANCE-MACRO
13	Macro program for threading process	9140 to 9145	100019131	THREAD-MACRO
14	Macro program for two-part system simultaneous thread cutting (identical screw) process	9146,9148	100019140 to 100019145	2SYS_THREAD
15	Macro program for grooving process	9150 to 9154	100019146,9148	GROOVE-MACRO
16	Macro program for keyway cutting process	9155 to 9158	100019150 to 100019154	KEYWAY-MACRO
17	Macro program for trapezoidal grooving process	9160 to 9166	100019155 to 10019158	TGROOVE-MACRO
18	Macro program for hole drilling process	9170	100019160 to 100019166	HOLE-MACRO
19	Macro program for milling hole drilling process	9171 to 9177	100019170	M-HOLE-MACRO
20	Macro program for contour cutting process	9180	100019171 to 10019177	CONT-MACRO
21	Macro program for cross control	9189	100019180	CROSS-MODE_ON_OFF
22	Macro program for end process	9190	100019189	END-MACRO

(Note 1) Modal initialization:

The following commands are output at the head of each macro program.

(a) Hole drilling fixed cycle cancel (G80)
(b) Tool nose R compensation cancel (G40)
(c) Plane selection Z-X(G18)
(d) Absolute value command (G90)
(d) is commanded only when G code system 3 or 5 is selected.

(Note 2) T command:

If "0" is specified for the tool No. when using NAVI LATHE, tool change (T command) will not be carried out. The number of digits for the tool length compensation No. is determined according to the settings of "#1098 Tlno.".

(Note 3) Reading a program again when the date of the program is updated

When the date of the following programs is updated while NAVI LATHE is running, the updated program can be read.

(1) NC program
(2) File program (tool file program and cutting condition file program)
(3) Parameter program

When the date of the program is updated, the message to confirm the reading of the updated program is displayed. Press the [Y] key to read the program, and press the [N] key not to read the program.

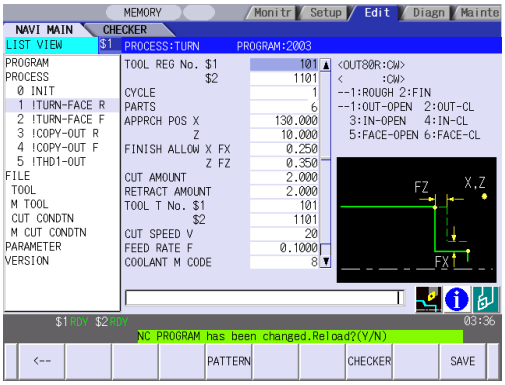
The screen display is updated as follows by reading the program.

- Reading the NC program
  The display is in the state immediately after opening the program with the [OPEN] menu.
- Reading the file program
  The head of the data is displayed when the file screen corresponding to the program read is displayed.
- Reading the parameter program
  The head of the data is displayed when the parameter screen is displayed.

< Example of when the NC program is updated >

(1) Update the opening NC program with NAVI LATHE from other than NAVI LATHE.

➡ The message to confirm the reading of the updated program appears.

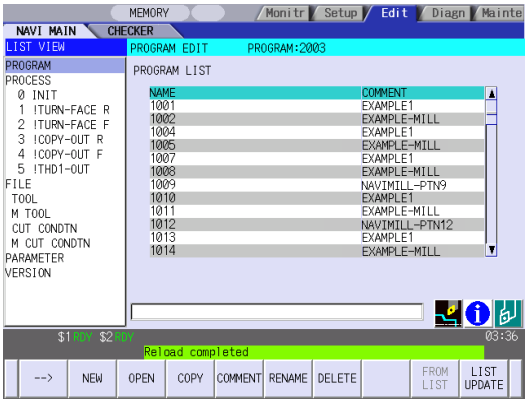


(2) Press the [Y] key.

Press the [N] key in order not to read the program.

➡ The program is read and the screen display is updated.

(Note) The display is the same as the state after opening the NC program with the [OPEN] menu.



If the program is updated when NAVI LATHE is not activated, the confirmation of reading the updated program will be performed in the next NAVI LATHE activation.

7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

NAVI LATHE operations and the creations of machining programs with NAVI LATHE require the following specifications for 700/70 Series CNC functions.

Required specifications

Division	Specifications	Remarks
Additional specifications	Synchronous tapping cycle	
	Constant surface speed control	
	Tool offset 80 sets	This is necessary when 21 or higher value is set for the offset No.
	Expansion workpiece coordinate system selection (48 sets)	This is necessary when specifying G54.1Pn (n=1 to 48) in the workpiece coordinate system.
	User macro	
	MTB macro	This is necessary when the macro program mode is MTB macro. 128KB of free space is required. (Note) MTB macro is not available for E70.
	Compound type fixed cycle for the turning	
	Compound type fixed cycle for turning (Type II)	
	Variable command 200 sets or more	
	Conner chamfering / Corner R	
	Milling interpolation / Polar coordinate interpolation	These are necessary for milling. The cylindrical interpolation is necessary for the G code system 6 or 7.
	Cylindrical interpolation	
	Multiple-spindle control II	
	Spindle position control (spindle/C axis control)	
	Balance cut	To enable the balance cut function in the G code system 6 or 7, disable the option of mirror image for facing tool posts. Enable this function only when using the 2-part system specification. (Note) This function is invalid for the M70 type B.
	Two-part system simultaneous thread cutting	Enable this function only when using the 2-part system specification. (Note) This function is invalid for the M70 type B.
	Control axis superimposition	
	Mixed synchronous control	

Division	Parameter name	Setting details	Remarks
Parameter specifications	#1013 axname	1:X 2:Z	Address of each axis name is specified.
	#1014 incax	1:U 2:W	Specify the incremental command axis name address for each axis.
	#1017 rot	3:1	Specify the 3rd axis as the rotary axis for the milling machining.
	#1019 dia	1(X axis):1	The diameter specification axis is selected by the X axis. The radius specification axis is selected by the other axes.
	#1026 base-I	X	Address of the axes configuring a plane is specified.
	#1027 base-J	Y	
	#1028 base-K	Z	
	#1029 aux-I	X	If there is an axis parallel to #1026 base_I, specify that axis address.
	#1030 aux-J	Y,C	If there is an axis parallel to #1027 base_I, specify that axis address. Set Y when G code system is any of 2 to 5. Set C when G code system is 6 or 7.
	#1037 cmdtyp	3 to 6	Specify the G code system of a program. When the G code system has been changed, the macro has to be registered again.
	#1076 AbsInc	1	Absolute command and incremental command are switched by the address code.
	#1098 Tlno.	0	The high-order 2 digits or 3 digits are designated as tool NO. The low-order 2 digits or 1 digit are designated as tool length and wear offset number.
	#1128 RstVCl	0	Specify how to handle the common variables when resetting. Common variables are not cleared after resetting. Set "0" when user macro mode is applied to the macro program. MTB macro mode does not require the setting "0".
	#1129 PwrVCl	0	Specify how to handle the common variables when the power is turned ON. Common variables are not cleared after the power is turned ON. Set "0" when user macro mode is applied for the macro program. MTB macro mode does not require the setting "0".

Division	Parameter name	Setting details	Remarks
Parameter specifications	#1181 G96_ax	1	Specify the 1st axis for the axis to be targeted for constant surface speed control.
	#1183 clmp_M	–	Set the M code for C axis clamp. Input the same value as set in “605 C AXIS CLAMP M CODE” which is the parameter for milling.
	#1146 Sclamp	1	Specify how to handle the spindle speed clamp function with G92S command. If S command and G92 command are in the same block, S command is always handled as a clamp command.
	#1227 aux11 (bit5)	0	Clamp the rotation regardless of the constant surface speed mode when the spindle rotation speed clamp command is issued.
	#1228 aux12 (bit5)	0	Select the workpiece coordinate for the coordinates during constant surface speed.
	#1229 set01 (bit2)	0	When the start-up and cancel commands are operated during nose R and radius compensation, their blocks are not handled by intersection operation processing; they are handled as offset vectors in the direction vertical to that of the commands.
	#1265 ext01	bit0: 0 bit2: 0	Select the conventional format for the following command format. • Compound type fixed cycle for turning. • Hole drilling fixed cycle MITSUBISHI CNC special format cannot be used.
	#1273 ext09	bit2: 1	When the shape specified at the turning (ROUGH) G71/G72, or copy cutting (ROUGH) G73 is not a monotone increasing or decreasing shape, set "1".
	#1280 ext16 (bit4)	0/1	Select how to command mixed control. 0: Use PLC interface signal for mixed control 1: Use G command for mixed control

Division	Parameter name	Setting details	Remarks
Parameter specifications	#1285 ext21 (bit2)	0	Select whether to perform the following processes for all the part systems or for each part system separately in multi-part system program management: newly create, delete or rename the machining programs in NC memory or transfer, compare, merge the programs between NC memory and other device. Set "0". (Perform these processes for the programs in all the part systems.)
	#1316 CrossCom	1	When using the common variables from #100100 to #800199, set this parameter to "1".
	#1516 mill_ax	C	Select C for the name of the rotary axis used in milling interporation.
	#1517 mill_C	0, 1	Specify Y axis as the hypothetical axis for milling interpolation. Set “0” (Y axis) when G code system is any of 2 to 5. Set “1” (C axis) when G code system is 6 or 7.
	#1537 crsax[1]	-	Set the axis to be interchanged during cross machining control. Using two digits, set the name of the axis interchanged with that where the mixed synchronous control (cross axis control) request signal is input, or that moves to the position where the signal is input. For lathe system process: Designate Z2 to crsax[1] of the 1st part system. Designate Z1 to crsax[1] of the 2nd part system. For milling system process: Designate C1 to the crsax[1] of 2nd part system.
	#2143 polar Z1	1	Set "1 (negative)" for the reference axis Z1.
	#2143 polar Z2	0/1	When the superimposed axis Z2 is in the same direction as the reference axis Z1, set "0 (positive)" to the superimposed axis Z2. (1105 $2 Z axis moving direction: 2) When the superimposed axis Z2 is in the opposite direction to the reference axis Z1, set "1 (negative)" to the superimposed axis Z2. (1105 $2 Z axis moving direction: 1)
	#8102 COLL. ALM OFF	1	This is validated when executing the machining program created with NAVI LATHE.

Division	Parameter name	Setting details	Remarks
Parameter specifications	#8111 Milling Radius	0	Select all axes radius command to set the linear axis for milling interpolation.
	#8112 G04P DECIMAL PNT-P	1	The decimal point command for G04 address P is validated.
	#8117 OFS Diam DESIGN	0	The tool radius compensation amount is designated with tool radius.

- When the multi-system program generation and operation is enabled (#1285 ext21(bit2) is ON), the program of the second part system cannot call the NAVI macro program and it causes the error. Thus, when operating the second part system, the multi-part system program generation and operation are required to turn OFF.
- In the G code system 6 or 7, the control axis superimposition or mixed synchronous control function cannot be used.
- When executing the balance cut, the coordinate zero point of both part systems must be the same. If the coordinate zero point of both part systems are set at different points, the balance cut cannot be processed properly.
- "!" code is used for the waiting. The waiting by use of M code is not supported.
- The control axis superimposition cannot be performed on the sub spindle side. Thus, if the sub spindle side is the spindle moving type, attempting balance cut causes a program error.
- When the $2 Z axis move type is the spindle moving type, machining cannot be performed on the sub spindle side.

7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

Listed below are the other cases which disable the machining .

○ : machinable
× : non-machinable

When selecting the main spindle:

No.	Machine configuration		Machining content		
	1003 $1 Z AXIS MOVE TYPE	1004 $2 Z AXIS MOVE TYPE	$1 independent machining	$2 independent machining	Balance cut machining
1	Turret moving type	Turret moving type	○	○	○
2	Spindle moving type	Turret moving type	○	○	○
3	Turret moving type	Spindle moving type	○	×	×
4	Spindle moving type	Spindle moving type	○	○	×

○ : machinable
× : non-machinable

When selecting the sub spindle:

No.	Machine configuration		Machining content		
	1003 $1 Z AXIS MOVE TYPE	1004 $2 Z AXIS MOVE TYPE	$1 independent machining	$2 independent machining	Balance cut machining
1	Turret moving type	Turret moving type	○	○	○
2	Spindle moving type	Turret moving type	×	○	×
3	Turret moving type	Spindle moving type	○	×	×
4	Spindle moving type	Spindle moving type	×	×	×

(Note 1) When either the mixed control (cross axis control) II or $2 Z axis move type is the spindle moving type, the 2nd part system is unable to perform milling. Milling is enabled when $1 Z axis move type is the turret moving type, and $2 Z axis move type is the turret moving type. When performing milling, select the mixed control (cross axis control) I.

(Note 2) When the mixed control (cross axis control) II is selected, the transfer process cannot perform at the 2nd part system. When processing the transfer, select the mixed control (cross axis control) I.

Recommended specifications

Division	Specifications	Remarks
Additional specifications	Graphic check	
	Graphic trace	

8. ALARM MESSAGE

8.1 Error Message

Division	Message	Details
Common	E001 No Data setting	The data with no setting exists.
	E002 Data range over	The data exceeded a set range was input.
	E003 Setting data error	The setting data is illegal.
	E004 System error	An unexpected error exists.
	E005 No data setting on pattern screen	Incomplete data exists on the pattern screen.
	E007 Data range over on pattern screen	The data exceeded a set range was input on the pattern screen.
Program editing	E101 Designated file does not exist	The designated program does not exist.
	E102 Designated file already exists	The designated program already exists.
	E103 Program running	The program is running.
	E104 Program entry over	The number of program registrations was exceeded.
	E105 Memory over	The number of program memory characters was exceeded.
	E106 Data protect	Saving of the parameters is prohibited because the data protect key is validated. Reconsider the data protect key setting and save the parameters on Parameter Screen.
	E107 TOOL file read error	Reading of the tool file was failed. Check the path (drive/folder) of the file.
	E108 TOOL file write error	Writing to the tool file was failed. Check the path (drive/folder) of the file.
	E109 CUT CONDITION file read error	Reading of the cutting condition file was failed. Check the path (drive/folder) of the file.
	E110 CUT CONDITION file write error	Writing to the cutting condition file was failed. Check the path (drive/folder) of the file.
	E111 PARAMETER file read error	Reading of the parameter file was failed. Check the path (drive/folder) of the file.
	E112 PARAMETER file write error	Writing to the parameter file was failed. Check the path (drive/folder) of the file.
	E113 PREFERENCE data read error	Reading of the PREFERENCE data was failed.
	E114 PREFERENCE data write error	Writing to the PREFERENCE data was failed.
	E115 PROGRAM file read error	Reading of the NC program file was failed. Check the path (drive/folder) of the file.
	E116 PROGRAM file write error	Writing to the NC program file was failed. Check the path (drive/folder) of the file.
	E117 Program name illegal	The designated program name is illegal.
	E198 Program format error	Program format is illegal.
	E199 File system error	An error occurred during file input or output.
List view	E201 Process number over	The number of processes exceeded 100.

Division	Message	Details
Turning, copy cutting	E211 Geometry record number entry over	Exceeded the number of records currently registered.
	E212 Geometry maximum record number over	The maximum number of records (35) is exceeded.
	E213 Geometry record number entry over	The record No. is illegal.
	E214 I,K disagreement with A (angle) E214 I,J disagreement with A (angle) E214 J,K disagreement with A (angle)	Linear I,K and angle are contradictory. Linear I,J and angle are contradictory. Linear J,K and angle are contradictory.
	E215 No end point on surface (line number)	The end point does not exist on the surface.
	E216 No continuity with previous line (line number)	There is no continuity with the previous line.
	E217 No circle (line number)	Circle cannot be determined from set data.
	E218 Corner C error (line number)	Corner C cannot be determined.
	E219 Corner R error (line number)	Corner R cannot be determined.
	E220 shape input error (line number)	Shape input error
	E221 Last line has corner R/C (line number)	Corner R/C was set in the last line.
	E222 Start point error (line number)	Start point error
	E223 Corner no move	The block following corner R or corner C is not a movement command.
	E224 Corner short	When issuing corner C or corner R command, the movement distance in the next block is smaller than corner C or corner R.
	E225 Cutting shape reversed	The cutting shape is not incremented or decremented monotonously.
	E226 Depth of cutting shape <= CUT AMOUNT	“Depth of cutting shape <= cutting amount” is applied.
	E227 Starting shape not linear	Starting shape is circular. When OPEN type is selected in PARTS, circular cannot be specified for the starting shape.
	E228 APPRCH POS illegal	Approach point is illegal for the cutting shape.
	E229 Halfway position of cutting shape illegal	Halfway position of the cutting shape is beyond the end position.
Threading	E231 H < FIN ALLOW	"Thread height < finishing allowance" is applied.
	E232 H < CUT AMOUNT	"Thread height < cutting amount" is applied.
	E233 THREAD angle > 45 deg.	"Thread angle > 45°" is applied for taper thread.
	E234 THREAD length = 0	"Thread length = 0" is applied.
	E235 PITCH isn't set	Thread height cannot be calculated because the pitch is not set. Set the pitch.
Grooving	E241 W < TOOL WIDTH	"Groove width < tool width" is applied.
	E242 GRV Height < CUT AMOUNT	"Groove height < cutting amount" is applied.
	E243 GRV Height < |Corner Size|	"Groove height < corner size" is applied.
	E244 Corner R/C input error	Corner R/C is specified for the taper grooving.
	E245 GRV angle > 45 deg.	"Groove angle > 45°" is applied for taper groove.

Division	Message	Details
Trapezoidal grooving	E251 W < TOOL WIDTH	"Groove width < tool width" is applied.
	E252 H< CUT AMOUNT	"Groove height < cutting amount" is applied.
	E253 H< FIN ALLOW	"Groove height < finishing allowance" is applied.
	E254 H/2 < |Corner Size|	"Groove height/2 < corner size" is applied.
	E255 W/2 < |Corner Size|	"Groove width/2 < corner size" is applied.
	E256 Can't insert tool	The width of groove is small or tool diameter is large.
	E257 GRV ANG illegal	"GRV ANG1 + GRV ANG3 >= 90" or "GRV ANG2 + GRV ANG4 >= 90" is applied.
Hole drilling	E261 B < H	"Tool nose depth < hole depth" is applied.
	E262 D > Tool diameter	"Spot radius > tool diameter" is applied.
	E263 CUT AMOUNT illegal	Cutting amount is illegal.
	E264 Feedrate over	The feedrate (mm/min, inch/min) exceeded the commanded range. Check the cutting speed and feedrate again.
EIA	E271 Block number over	The number of EIA blocks was exceeded.
Cutting off	E701 X1 <= X2	Set the values to meet the relation: Start position > End position.
INIT	E281 ID >= OD	Workpiece's inner diameter is larger than the outer diameter.
	E282 - Z >= +Z	The position of -Z is greater than that of +Z.
	E283 Work coordinate setting error	Any of P1 to P48 (extended workpiece coordinate system) is set while WORK COORDINATE and WORK COORD. SUB SP are set to the same value. If the main and sub spindles are in the same coordinate system, use G54 to G59.
Milling hole drilling	E601 B < H	"Tool nose depth < hole depth" is applied.
	E602 D > Tool diameter	"Spot diameter > tool diameter" is applied.
	E603 CUT AMOUNT illegal	Cutting amount is illegal.
	E604 Omit number illegal	Omit No. is illegal.
	E605 Maximum hole number over	The number of holes exceeded the maximum hole number (35 points).
	E264 Feedrate over	The feedrate (mm/min, inch/min) exceeded the commanded range. Check the cutting speed and feedrate again.
Keyway cutting	E611 W < TOOL WIDTH	"Groove width < tool width" is applied.
	E612 GRV Height < CUT AMOUNT	"Groove height < cutting amount" is applied.
	E264 Feedrate over	The feedrate (mm/min, inch/min) exceeded the commanded range. Check the cutting speed and feedrate again.

Division	Message	Details
Contour cutting	E621 FH > WIDTH	“Finishing allowance FH > cutting width” is applied.
	E622 FV > DEPTH	“Finishing allowance FV > cutting depth” is applied.
	E623 WIDTH < TOOL WIDTH	“Cutting width < tool width” is applied.
	E624 DEPTH < CUT AMOUNT	“Cutting depth < cutting amount” is applied.
	E211 Geometry record number entry over	Exceeded the number of records currently registered.
	E212 Geometry maximum record number over	The maximum number of records (35) is exceeded.
	E213 Geometry record number entry over	The record No. is illegal.
	E214 I,K disagreement with A (angle) E214 I,J disagreement with A (angle) E214 J,K disagreement with A (angle)	Linear I,K and angle are contradictory. Linear I,J and angle are contradictory. Linear J,K and angle are contradictory.
	E215 No end point on surface (line number)	The end point does not exist on the surface.
	E216 No continuity with previous line (line number)	There is no continuity with the previous line.
	E217 No circle (line number)	Circle cannot be determined from set data.
	E218 Corner C error (line number)	Corner C cannot be determined.
	E219 Corner R error (line number)	Corner R cannot be determined.
	E220 shape input error (line number)	Shape input error
	E221 Last line has corner R/C (line number)	Corner R/C was set in the last line.
	E222 Start point error (line number)	Start point error
	E223 Corner no move	The block following corner R or corner C is not a movement command.
	E224 Corner short	When issuing corner C or corner R command, the movement distance in the next block is smaller than corner C or corner R.
	E264 Feedrate over	The feedrate (mm/min, inch/min) exceeded the commanded range. Check the cutting speed and feedrate again.
OTHERS	E291 Memory over	The number of program memory characters was exceeded during macro transfer.
	E292 Program entry over	The number of program registrations was exceeded during macro transfer.
	E293 Macro transporting error	An error occurred during macro transfer.
	E294 Program running	The program is running.
	E295 Tool register number over	The number of tools registered has exceeded 100.

8.2 Operation Message

Division	Message	Details
Common	OK? (Y/N)	Message to confirm the operation. Y: Execute the operation. N: Do not execute the operation.
	Save data?(Y/N)	Message to confirm saving data Y: Save data. N: Do not save data.
	Clear the pattern data? (Y/N)	Message to confirm clearing the pattern data Y: Clear the pattern data N: Cancel the pattern change.
	Delete OK? (Y/N)	Message to confirm deleting the program or process data Y: Delete the program or process data. N: Do not delete the program or process data.
	Select the position, please	During process movement mode.
	Loading program	The program is being loaded.
	No init process. Create OK?(Y/N)	INIT process creation confirmation Edited the program that was not created with NAVI LATHE. Y: Create the INIT process. N: Cancel opening the program.
	The data was changed. Save the changes?(Y/N)	Save confirmation for unsaved data Y: Save data. N: Not save data.
	The page cannot be changed during edit.	Editing the process data. Switch the screen after the saving operation.
	Macro transporting complete	Macro transporting complete
	Data protect	Saving of the program, file, parameters is prohibited because the data protect key is validated. Reconsider the data protect key setting.
File reload	NC program has been changed. Reloaded? (Y/N)	A message to confirm whether to update the program opened in NAVI LATHE. The opened program in the NAVI LATHE was updated by an external device. Y: Open the updated program. N: Not open the updated program.
	TOOL file was changed. Reloaded? (Y/N)	A message to confirm whether to update the data of TOOL file. The TOOL file was updated by an external device. Y: Open the updated TOOL file. N: Not open the updated TOOL file.
	CUT CONDITION file was changed. Reloaded? (Y/N)	A message to confirm whether to update the contents of CUT CONDITION file (tip materials or workpiece materials). The CUT CONDITION file was updated by an external device. Y: Open the updated CUT CONDITION file. N: Not open the updated CUT CONDITION file.

Division	Message	Details
File reload	PARAMETER file was changed. Reloaded? (Y/N)	A message to confirm whether to update the PARAMETER contents. The PARAMETER file was updated by an external device. Y: Open the updated PARAMETER file. N: Not open the updated PARAMETER file.
	NC program has been deleted. Initialize? (Y/N)	A message to confirm whether to cancel the program editing. The opened program in the NAVI LATHE has been deleted by an external device. Y: Cancel the program editing. N: Not cancel the program editing.
	TOOL file was deleted. Initialize? (Y/N)	A message to confirm whether to restore the contents of TOOL file to the default. The TOOL file was deleted by an external device. Y: Restore the contents of TOOL file to the default value. N: Not restore the contents of TOOL file to the default value.
	CUT CONDITION file was deleted. Initialize? (Y/N)	A message to confirm whether to restore the contents of CUT CONDITION file (tip materials or workpiece materials) to the default. The CUT CONDTION file was deleted by an external device. Y: Restore the contents of CUT CONDITION file to the default value. N: Not restore the contents of CUT CONDITION file to the default value.
	PARAMETER file was deleted. Initialize? (Y/N)	A message to confirm whether to restore the contents of PARAMETER file to the default. The PARAMETER file was deleted by an external device. Y: Restore the contents of PARAMETER file to the default value. N: Not restore the contents of PARAMETER file to the default value.
	Reload completed	Reload completed
	Initialize completed	Initialize completed
Tool file	Delete OK? (Y/N)	Message to confirm deleting the tool data column. Y: Delete. N: Do not delete.
	Clear OK? (Y/N)	Message to confirm clearing the tool data. Y: Clear. N: Do not clear.
	Paste OK? (Y/N)	Message to confirm pasting the tool data. Y: Paste. N: Do not paste.
Checker	Stop drawing	The shape drawing has been stopped.
	On drawing	The shape drawing is being carried out.
	Finish drawing	The shape drawing has been completed.

APPENDIX 1. VARIABLES USED IN NAVI LATHE

NAVI LATHE uses the following variables to run an NC program.
The data meanings of the variables change according to the process in operation and the setting of the parameter "1001 SUB SPINDLE SPEC".

(1) Operation variables during program operation

- Parameter “1001 SUB SPINDLE SPEC” NONE

Common variable No.		Data name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode				
#150	#450	WORK COORDINATE	54 to 59, 101 to 148	54	Variable for operation
#151	#451	COOLANT	0 to 1	1	Variable for operation
#152	#452	TOOL CHANGE POS	1 to 3	1	Variable for operation
#153	#453	FIN TOOL RET	1 to 3	1	Variable for operation
#154	#454	END POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Variable for operation
#155	#455	END POS Z		0	Variable for operation
#156	#456	END M CODE	1 to 3	1	Variable for operation
#157	#457	OUTSIDE DIA	0.001 to 99999.999mm 0.0001 to 9999.9999inch	100	Variable for operation
#158	#458	+Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	100	Variable for operation
#159	#459	Milling interpolation specification	0: NONE, 1: EXIST	0	Variable for operation

- Parameter “1001 SUB SPINDLE SPEC” EXIST

Common variable No.		Data name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode				
#150	#450	WORK COORDINATE	54 to 59, 101 to 148	54	Variable for operation
#151	#451	COOLANT	0 to 1	1	Variable for operation
#152	#452	TOOL CHANGE POS	1 to 3	1	Variable for operation
#153 (Integer part)	#453 (Integer part)	FIN TOOL RET	1 to 3	1	End process
		START SP	1: MAIN SP, 2: SUB SP	1	Excluding end process
#153 (decimal part)	#453 (decimal part)	WORK CARRING OUT	0: NONE, 1: EXIST	0	

Common variable No.		Data name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode				
#154	#454	END POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	End process
		WORK COORDINATE SUB SPIN	54 to 59, 101 to 148	54	Excluding end process
#155	#455	END POS Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Variable for operation
#156	#456	END M CODE	1 to 3	1	Variable for operation
#157	#457	OUTSIDE DIA	0.001 to 99999.999mm 0.0001 to 9999.9999inch	100	Variable for operation
#158	#458	+Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	100	Variable for operation
#159	#459	Milling interpolation specification	0: NONE, 1: EXIST	0	Variable for operation

(2) Parameter variables during program operation

- Parameter “1001 SUB SPINDLE SPEC” NONE

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#160	#460	101	M1 OUTPUT	0: INVALID, 1: VALID	0	Common
#161	#461	102	SPDL CLAMP SPEED	1 to 99999 rev/min	2000 rev/min	
#162	#462	103	TOOL TURNING CL X	0.001 to 99999.999mm	50.000mm	Common
#163	#463	104	TOOL TURNING CL Z	0.0001 to 9999.9999inch	1.9685inch	Common
#164	#464	105	TOOL FIX RET POS X	-99999.999 to 99999.999mm	0	Common
#165	#465	106	TOOL FIX RET POS Z	-9999.9999 to 9999.9999inch	0	Common
#166	#466	107	SAFE PROFILE CL OD	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	Common
#167	#467	108	SAFE PROFILE CL FACE			Common
#168	#468		PART SYSTEM SEL.	1: $1, 2: $2	1	Common
#169	#469		$2 TOOL CHANGE POS	1: X REF,Z CL, 2: XZ CL, 3: XZ FIX POS	1	Common
#170	#470	1101	$1 SYS CHG SAFE POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	INIT
		301	GRV DWELL	0.001 to 99.999sec	1.000sec	GRV, TGRV, CUTOFF
		105	$1 TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process

APPENDIX 1. VARIABLES USED IN NAVI LATHE

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#171	#471	1102	$2 SYS CHG SAFE POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	INIT
		302	GRV 2nd SHIFT AMOUNT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV
		106	$1 TOOL FIX RET POS Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#172	#472	303	GRV CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	1.000mm 0.0394inch	GRV, TGRV, CUTOFF
		1108	$2 TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#173	#473	304	GRV RETRACT LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.2mm 0.0079inch	GRV, TGRV, CUTOFF
		1109	$2 TOOL FIX RET POS Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#174	#474	201	THD CLEARANCE EXIT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.001mm 0.0001inch	THD
		305	GRV OVERLAP LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV, TGRV
#175	#475	202	THD CLEARANCE ENTR	0.000 to 99999.999mm 0.0000 to 9999.9999inch	0	THD
		306	GRV FIN APPROACH R	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.5mm 0.0197inch	GRV
#176	#476	401	HOLE CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	HOLE
#177	#477	402	HOLE SYNC TAP	0: INVALID, 1: VALID	0	HOLE
#180	#480	601	Y AXIS SPEC	0: NONE, 1: EXIST	0	Common
#181	#481	602	SPDL ORIENT M CODE	0 to 9999	19	Common
#182	#482	603	SPDL CHANGE M CODE	0 to 9999	102	Common
#183	#483	1110	$2 Y AXIS SPEC	0: NONE, 1: EXIST	0	INIT
		604	C AXIS CHANGE M CODE	0 to 9999	103	K WAY, M HOLE, CONT
#184	#484	605	C AXIS CLAMP M CODE	0 to 9999	110	Common
#185	#485	609	TOOL SPINDLE NO.	1 to 4	2	Common
#186	#486	606	TOOL TURNING CL Y	0.000 to 99999.999mm 0.0000 to 9999.9999inch	50.000mm 1.9685inch	Common

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#187	#487	607	TOOL FIX RET POS Y	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Common
#188	#488	608	AXIS DIR COEF OF SPEED	1 to 200%	50	Common
#189	#489	701	HOLE CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	M HOLE
#190	#490	702	HOLE SYNC TAP	0: INVALID, 1: VALID	0	M HOLE
#191	#491	801	K-WAY CUT WIDTH PCT(%)	1 to 100%	50	K WAY
#192	#492	802	K-WAY CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	K WAY
		903	E-ML EMPTY D OFS NUM	1 to tool sets	0	CONT
#193	#493	703	HOLE TAP ON M CODE	0 to 9999	0	M HOLE
		901	E-ML CUT WIDTH PCT(%)	1 to 100%	50	CONT
#194	#494	704	HOLE TAP OFF M CODE	0 to 9999	0	M HOLE
		902	E-ML CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	CONT
#195	#495	1105	$2 Z AXIS DIR	1: SAME, 2: OPPOSITE	1	Common
#196 (integer part)	#496 (integer part)	1103	$1 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	Common
#196 (decimal part)	#496 (decimal part)	1104	$2 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	Common
#197 (integer part)	#497 (integer part)	1106	MIXED SYNC CTRL ON M	0 to 9999	112	Common
#197 (decimal part)	#497 (decimal part)	1107	MIXED SYNC CTRL OFF M	0 to 9999	113	Common
#198 (integer part)	#498 (integer part)	610	TOOL SP CW ROT M CODE	0 to 9999	3	Common
#198 (decimal part)	#498 (decimal part)	611	TOOL SP CCW ROT M CODE	0 to 9999	4	Common
#199 (integer part)	#499 (integer part)	612	TOOL SP STOP M CODE	0 to 9999	5	Common
#199 (decimal part)	#499 (decimal part)	613	TOOL SP SELECT M CODE	0 to 9999	0	Common

- Parameter “1001 SUB SPINDLE SPEC” EXIST

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#160	#460	101	M1 OUTPUT	0: INVALID, 1: VALID	0	Common
#161	#461	102	SPDL CLAMP SPEED	1 to 99999 rev/min	2000 rev/min	INIT
			START SP	1: MAIN SP, 2: SUB SP	1	Each process
#162	#462	103	TOOL TURNING CL X	0.001 to 99999.999mm 0.0001 to 9999.9999 inch	50.000mm 1.9685inch	Each process
		1006	PARTS CATCHER OUT M	0 to 9999	68	End process
#163	#463	104	TOOL TURNING CL Z	0.001 to 99999.999 mm 0.0001 to 9999.9999 inch	50.000mm 1.9685inch	Common
		1007	PARTS CATCHER IN M	0 to 9999	69	End process
#164	#464	105	TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0 0	Common
#165	#465	106	TOOL FIX RET POS Z			Common
#166	#466	107	SAFE PROFILE CL OD	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	Common
#167	#467	108	SAFE PROFILE CL FACE			Common
#168	#468		PART SYSTEM SEL.	1: $1, 2: $2	1	Each process
		1025	MAIN SP SELECT M CODE	0 to 9999	297	TRS
#169	#469		$2 TOOL CHANGE POS	1: X REF,Z CL, 2: XZ CL, 3: XZ FIX POS	1	Common
#170	#470	1101	$1 SYS CHG SAFE POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	INIT
		301	GRV DWELL	0.001 to 99.999sec	1.000sec	GRV, TGRV, CUTOFF
		1019	SUB SP CCW ROT M CODE	0 to 9999	4	TRS
		105	$1 TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#171	#471	1102	$2 SYS CHG SAFE POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	INIT
		302	GRV 2nd SHIFT AMOUNT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV
		1026	SUB SP SELECT M CODE	0 to 9999	296	TRS
		106	$1 TOOL FIX RET POS Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#172 (integer part)	#472 (integer part)	303	GRV CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	1.000mm 0.0394inch	GRV, TGRV, CUTOFF
		1016	SUB C-AX CLP M CODE	0 to 9999	260	End process, K WAY, M HOLE, CONT
		1106	MIXED SYNC CTRL ON M	0 to 9999	112	TRS
		1108	$2 TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#172 (decimal part)	#472 (decimal part)	1107	MIXED SYNC CTRL OFF M	0 to 9999	113	TRS
#173	#473	304	GRV RETRACT LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.2mm 0.0079inch	GRV, TGRV, CUTOFF
		1021	SUB SP C AXIS NAME	1: A, 2: B	1	K WAY, M HOLE, CONT,TRS
		1109	$2 TOOL FIX RET POS Z	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Balance cut process
#174	#474	201	THD CLEARANCE EXIT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.001mm 0.0001inch	THD
		305	GRV OVERLAP LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV, TGRV
		1025	MAIN SP SELECT M CODE	0 to 9999	297	End process, K WAY, M HOLE, CONT
		1105	$2 Z AXIS DIR	1: SAME, 2: OPPOSITE	1	TRS

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#175	#475	202	THD CLEARANCE ENTR	0.000 to 99999.999mm 0.0000 to 9999.9999inch	0	THD
		306	GRV FIN APPROACH R	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.5mm 0.0197inch	GRV
		1026	SUB SP SELECT M CODE	0 to 9999	296	K WAY, M HOLE, CONT
		1023	OVER TRAVEL OF PUSH	0.000 to 99999.999mm 0.0000 to 9999.9999inch	0	TRS
#176	#476	401	HOLE CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	HOLE
		1014	SUB SP CHNG M CODE	0 to 9999	252	End process, K WAY, M HOLE, CONT,TRS
#177	#477	402	HOLE SYNC TAP	0: INVALID, 1: VALID	0	HOLE
		1015	SUB C-AX CHNG M CODE	0 to 9999	253	CUTOFF, K WAY, M HOLE, CONT,TRS
#178	#478		WORK OFFSET SHIFT AMOUNT	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Common
#179	#479		Z SUB SP	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Common
#180	#480	601	Y AXIS SPEC	0: NONE, 1: EXIST	0	Common
#181	#481	1014	SUB SP CHNG M CODE	0 to 9999	252	TRUN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		1024	SUB SP ORIGIN	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	TRS
#182	#482	603	SPDL CHANGE M CODE	0 to 9999	102	Common
#183	#483	1110	$2 Y AXIS SPEC	0: NONE, 1: EXIST	0	INIT
		1024	SUB SP ORIGIN	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	TRS
		604	C AXIS CHANGE M CODE	0 to 9999	103	K WAY, M HOLE, CONT,TRS

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#184	#484	1001	SUB SPINDLE SPEC	0: NONE, 1: EXIST	0	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		605	C AXIS CLAMP M CODE	0 to 9999	110	End process
		1017	SUB SPINDLE NO.	1 to 4	2	TRS
#185	#485	609	TOOL SPINDLE NO.	1 to 4	2	Common
#186	#486	606	TOOL TURNING CL Y	0.000 to 99999.999mm 0.0000 to 9999.9999inch	50.000mm 1.9685inch	Common
#187	#487	607	TOOL FIX RET POS Y	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	0	Common
#188	#488	608	AXIS DIR COEF OF SPEED	1 to 200%	50	Common
#189	#489	1022	TRANSFER AXIS NAME	1: A, 2: B	2	INIT, TRS
		1018	SUB SP CW ROT M CODE	0 to 9999	3	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		701	HOLE CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	M HOLE
#190	#490	1019	SUB SP CCW ROT M CODE	0 to 9999	4	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		702	HOLE SYNC TAP	0: INVALID, 1: VALID	0	M HOLE
		1002	MAIN CHUCK CLP M CODE	0 to 9999	11	TRS

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#191	#491	1020	SUB SP STOP M CODE	0 to 9999	5	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		801	K-WAY CUT WIDTH PCT(%)	1 to 100%	50	K WAY
		1003	MAIN CHUCK UN-CLP M	0 to 9999	10	TRS
#192	#492	1025	MAIN SP SELECT M CODE	0 to 9999	297	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		802	K-WAY CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	K WAY
		903	E-ML EMPTY D OFS NUM	1 to tool sets	0	CONT
		1004	SUB CHUCK CLP M CODE	0 to 9999	211	TRS
#193	#493	1026	SUB SP SELECT M CODE	0 to 9999	296	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		703	HOLE TAP ON M CODE	0 to 9999	0	M HOLE
		901	E-ML CUT WIDTH PCT(%)	1 to 100%	50	CONT
		1005	SUB CHUCK UN-CLP M	0 to 9999	210	TRS

Common variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#194	#494	1017	SUB SPINDLE NO.	1 to 4	2	TURN, COPY, THD, GRV, TGRV, HOLE, CUTOFF, balance cut process
		704	HOLE TAP OFF M CODE	0 to 9999	0	M HOLE
		902	E-ML CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	CONT
		1008	MAIN AIR BLOW ON M	0 to 9999	20	TRS
#195	#495	1105	$2 Z AXIS DIR	1: SAME, 2: OPPOSITE	1	Common
		1009	MAIN AIR BLOW OFF M	0 to 9999	21	TRS
#196 (integer part)	#496 (integer part)	1103	$1 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	Common
		1010	SUB AIR BLOW ON M	0 to 9999		TRS
#196 (decimal part)	#496 (decimal part)	1104	$2 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	Common
#197 (integer part)	#497 (integer part)	1106	MIXED SYNC CTRL ON M	0 to 9999	112	Common
		1011	SUB AIR BLOW OFF M	0 to 9999	221	TRS
#197 (decimal part)	#497 (decimal part)	1107	MIXED SYNC CTRL OFF M	0 to 9999	113	Common
#198 (integer part)	#498 (integer part)	610	TOOL SP CW ROT M CODE	0 to 9999	3	K WAY, M HOLE, CONT
		1012	AIR BLOW TIME	0 to 99 sec	3	TRS
#198 (decimal part)	#498 (decimal part)	611	TOOL SP CCW ROT M CODE	0 to 9999	4	K WAY, M HOLE, CONT
#199 (integer part)	#499 (integer part)	612	TOOL SP STOP M CODE	0 to 9999	5	K WAY, M HOLE, CONT
		1103	$1 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	TRS
#199 (decimal part)	#499 (decimal part)	613	TOOL SP SELECT M CODE	0 to 9999	0	CUTOFF, K WAY, M HOLE, CONT
		1104	$2 Z AXIS MOVE TYPE	1: TURRET, 2: SPINDLE	1	TRS

⚠ CAUTION

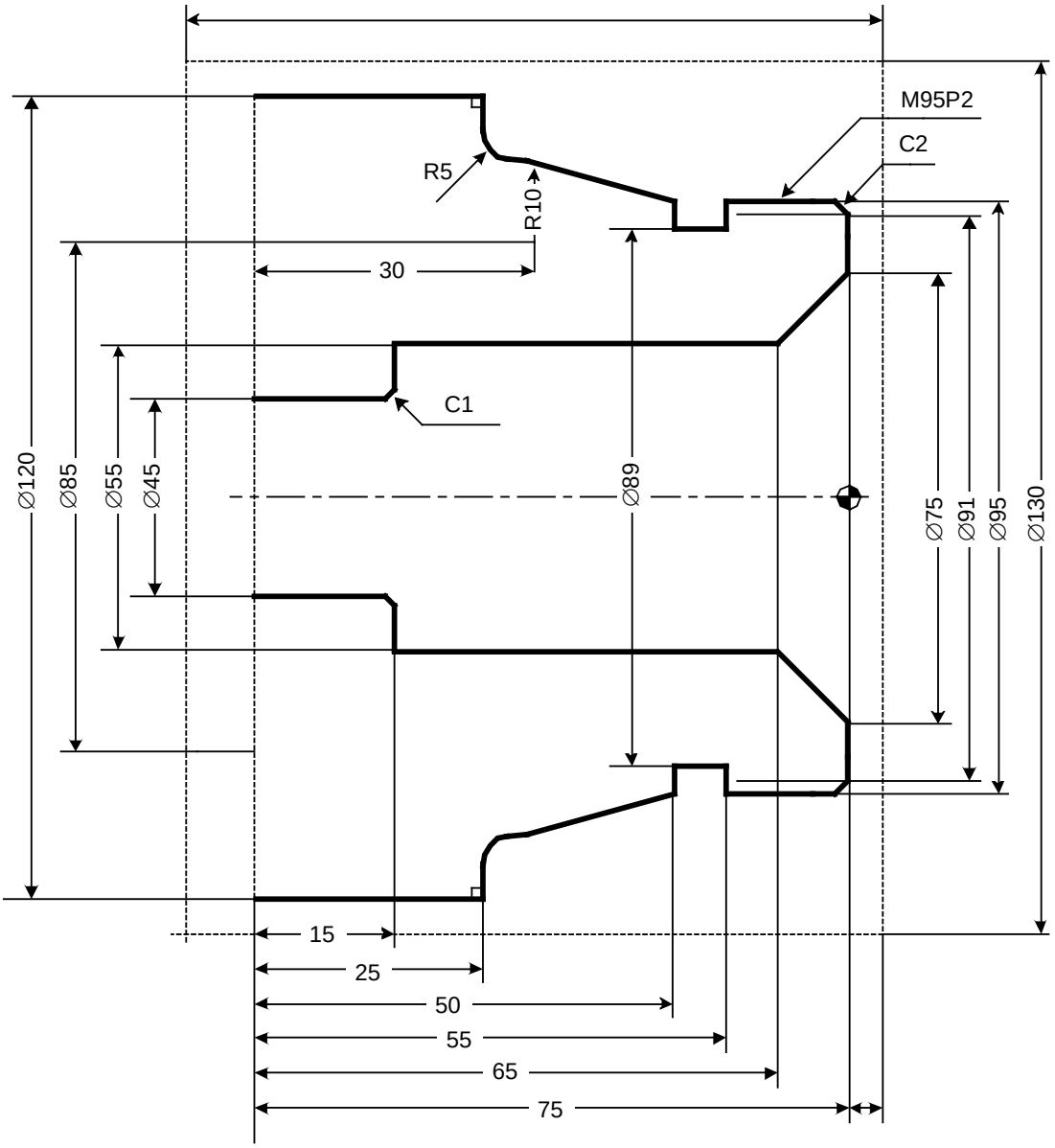
⚠ NAVI LATHE uses the following variables in order to operate the NC program.

NC program mode	Variables used by NAVI LATHE
User macro mode	#100 to #199
MTB macro mode	#450 to #499

When NC program mode is user macro mode, do not use common variables (#100 to #199). If those variables are written over, malfunction will be resulted. If mistakenly written them over, turn the NC power OFF after securing your safety. When starting NAVI LATHE by turning the NC power ON again,the system recovers the data. NC program mode is specified on the Preferences screen.

APPENDIX 2. PROGRAMMING EXAMPLE 1 (TURNING)

Appendix 2.1 Machining Drawing



Appendix 2.2 Process Table

Processes are shown below.

Process	Machining	Tool
1	Drilling machining	DR
2	Turning rough machining for front face	OUTR
	Turning finishing machining for front face	OUTR
3	Turning rough machining for outer diameter	OUTR
	Turning finishing machining for outer diameter	OUTR
4	Turning rough machining for inner diameter	INR
	Turning finishing machining for inner diameter	INR
5	Grooving for outer diameter	GO
6	Threading rough machining for outer diameter	TOMR
	Threading finishing machining for outer diameter	TOMR

Appendix 2.3 Condition Setting

Set the tool and cutting conditions before programming.

(1) Tool file screen

Register the tool data. Input the following values on the tool file screen.

No.	101	102	201	301	401
T NAME	OUT80R	IN55R	GO1.0	TOMR	DR45
T No.	101	202	303	404	505
USE	1	1	1	1	-
NOSE ANGLE	80.000	55.000	-	60.000	118.000
FRONT EDGE ANG	5.000	32.000	-	-	-
TOOL WIDTH	-	-	5.000	-	-
DIA	-	-	-	-	45.000
SP DIR	1	1	1	1	1
L/R HAND	1	1	1	1	-
TIP MATERIAL	H	W	W	W	W

(2) Cutting condition file screen

Register the cutting conditions for tip material and workpiece material. Input the following values on the cutting condition screen.

Item	1	2
TIP MATL	H	W
TURN R V	20.00	160.00
F	0.1000	0.3000
TURN F V	20.00	20.00
F	0.1000	0.1000
GRV R V	20.00	110.00
F	0.1000	0.1500
GRV F V	20.00	110.00
F	0.1000	0.1000
THR V	20.00	100.00
DRILL V	20.00	150.00
F	0.3000	0.2000
TAP V	12.00	5.00

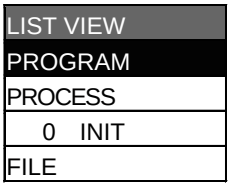
Item	1
WORK MATL	S45C
TURN R V	100
F	100
TURN F V	100
F	100
GRV R V	100
F	100
GRV F V	100
F	100
THR V	100
DRILL V	100
F	100
TAP V	100

Appendix 2.4 Creating Program

1. Open the program edit screen.

2. Press the [NEW] menu and create a new NC program.

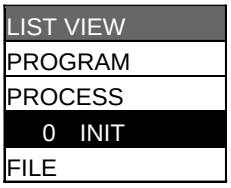
3. Move the cursor to "0 INIT" and press the [MODIFY] menu.



4. Input the following values.

Item	Setting value	Details
WORK REG No.	1	S45C
WORK ZERO	1	T'STK SIDE
OUTSIDE DIA OD	130.000	
INSIDE DIA ID	0.000	
+Z	5.000	
-Z	-95.000	
WORK COORDINATE	54	G54
COOLANT	1	VALID
TOOL CHANGE POS	1	X REF
FIN TOOL RET	1	REF
END POS X	-	
Z	-	
M CODE	1	M30

4.1 Save the initial conditions by pressing the [SAVE] menu.
4.2 Turn the LIST VIEW area active by pressing the [←] key.

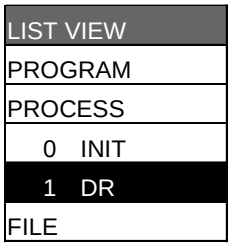


5. Process 1 Drilling machining (DR)

5.1 Open the process mode selection screen by pressing the [NEW] menu.
5.2 Open the hole drilling screen and set the following items.

Item	Setting value	Details
TOOL REG No.	401	DR45
HOLE CYCLE	1	DRILL
SURFACE Z ZF	-5.000	
DEPTH H	80.000	
NOSE DEPTH B	93.519	
SPOT DIAMETER D	45.000	
CUT AMOUNT	-	
DWELL	1.000	
TOOL T No.	505	
TOOL DIA	45.000	
CUT SPEED V	150	
FEED RATE F	0.2000	

5.3 Save the data of the drilling machining by pressing the [SAVE] menu.
5.4 Turn the LIST VIEW area active by pressing the [←] key.



6. Process 2 Turning rough machining for front face (OUTR)

6.1 Open the process mode selection screen by pressing the [NEW] menu.

6.2 Open the turning screen and set the following items.

<Turning screen>

Item		Setting value	Details
TOOL REG No.		101	OUT80R
CYCLE		1	ROUGH
PARTS		5	FACE-OPEN
APPRCH POS	X	134.000	
	Z	-7.000	
FINISH ALLOW	X FX	0.150	
	Z FZ	0.150	
CUT AMOUNT		2.000	
RETRACT AMOUNT		2.000	
TOOL T No.		101	
CUT SPEED	V	20	
FEED RATE	F	0.1000	

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
FILE

6.3 Press the [PATTERN] menu and set the following items.

<Turning pattern screen>

No.	M	X	Z	R/A
1		130.000	0.000	
2	1	36.000	0.000	(270.000)
3	1	36.000	-5.000	(180.000)

(Note) The value in the parentheses is calculated automatically.

6.4 After returning the screen to the turning screen by pressing the [RETURN] menu, save the data of the turning face rough machining by pressing the [SAVE] menu.

6.5 Turn the LIST VIEW area active by pressing the [←] key.

7. Process 2 Turning finishing machining for front face (OUTR)

7.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

7.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	FIN

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
FILE

7.3 Save the data of the turning face finishing machining by pressing the [SAVE] menu.

7.4 Turn the LIST VIEW area active by pressing the [←] key.

8. Process 3 Turning rough machining for outer diameter (OUTR)

8.1 Open the process mode selection screen by pressing the [NEW] menu.

8.2 Open the turning screen and set the following items.

<Turning screen>

Item		Setting value	Details
TOOL REG No.		101	OUT80R
CYCLE		1	ROUGH
PARTS		1	OUT-OPEN
APPRCH POS	X	134.000	
	Z	-7.000	
FINISH ALLOW X FX		0.150	
Z FZ		0.150	
CUT AMOUNT		4.875	
RETRACT AMOUNT		2.000	
TOOL T No.		101	
CUT SPEED	V	20	
FEED RATE	F	0.1000	

8.3 Press the [PATTERN] menu and set the following items.

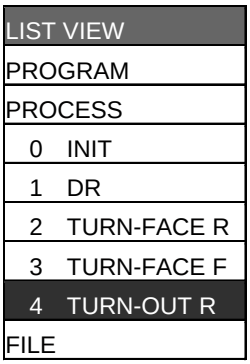
<Turning pattern screen>

No.	M	X	Z	R/A	I	K
1		91.000	0.000			
2	1	95.000	2.000	(45.000)		
3	1	95.000	25.000	(0.000)		
4	1	(104.320)	(42.415)	(14.981)		
5	3	(105.000)	(45.000)	10.000	85.000	45.000
6	2	(115.000)	(50.000)	5.000	(115.000)	(45.000)
7	1	120.000	50.000	90.000		
8	1	120.000	75.000	(0.000)		

(Note) The value in the parentheses is calculated automatically.

8.4 After returning the screen to the turning screen by pressing the [RETURN] menu, save the data of the turning outer diameter rough machining by pressing the [SAVE] menu.

8.5 Turn the LIST VIEW area active by pressing the [←] key.



9. Process 3 Turning finishing machining for outer diameter (OUTR)

9.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

9.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	FIN

9.3 Save the data of the turning outer diameter finishing machining by pressing the [SAVE] menu.

9.4 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
FILE

10. Process 4 Turning rough machining for inner diameter (INR)

10.1 Open the process mode selection screen by pressing the [NEW] menu.

10.2 Open the turning screen and set the following items.

<Turning screen>

Item		Setting value	Details
TOOL REG No.		102	IN55R
CYCLE		1	ROUGH
PARTS		3	IN-OPEN
APPRCH POS	X	45.000	
	Z	-10.000	
FINISH ALLOW X	FX	0.150	
	Z FZ	0.150	
CUT AMOUNT		3.500	
RETRACT AMOUNT		2.000	
TOOL T No.		202	
CUT SPEED	V	160	
FEED RATE	F	0.3000	

10.3 Press the [PATTERN] menu and set the following items.

<Turning pattern screen>

No.	M	X	Z	R/A
1		75.000	0.000	
2	1	55.000	10.000	(315.000)
3	1	55.000	60.000	(0.000)
4	1	47.000	60.000	(270.000)
5	1	45.000	61.000	(315.000)

(Note) The value in the parentheses is calculated automatically.

10.4 After returning the screen to the turning screen by pressing the [RETURN] menu, save the data of the turning inner diameter rough machining by pressing the [SAVE] menu.

10.5 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
6 TURN-IN R
FILE

11. Process 4 Turning finishing machining for inner diameter (INR)

11.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

11.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	FIN

11.3 Save the data of the turning inner diameter finishing machining by pressing the [SAVE] menu.

11.4 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
6 TURN-IN R
7 TURN-IN F
FILE

12. Process 5 Grooving for outer diameter (GO)

12.1 Open the process mode selection screen by pressing the [NEW] menu.

12.2 Open the grooving screen and set the following items.

Item			Setting value	Details
TOOL REG No.			201	GO1.0
PARTS			1	OUT
WIDTH		W	5.000	
LEFT CORNER		LC	0.000	
RIGHT CORNER		RC	0.000	
START POS	X	X1	95.000	
	Z	Z1	25.000	
END POS	X	X2	89.000	
	Z	Z2	25.000	
NUM OF GRV			1	
PITCH			0	
CUT AMOUNT			1.000	
SHIFT BEFORE RETR			0	
TOOL T No.			303	
TOOL WIDTH			5.000	
CUT SPEED		V	110	
FEED RATE		F	0.1500	

12.3 Save the data of the grooving outer diameter machining by pressing the [SAVE] menu.

12.4 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
6 TURN-IN R
7 TURN-IN F
8 GRV-OUT
FILE

13. Process 6 Threading rough machining for outer diameter (TOMR)

13.1 Open the process mode selection screen by pressing the [NEW] menu.

13.2 Open the threading screen and set the following items.

<Threading screen>

Item		Setting value	Details
TOOL REG No.		301	TOMR
CYCLE		1	ROUGH
PARTS		1	OUT
CUT METHOD		2	AR ZIG
ANG OF CUT	A	30.000	
PITCH	P	2.0000	
HEIGHT	H	1.227	
START POS X	X1	95.000	
Z	Z1	0.000	
END POS X	X2	95.000	
Z	Z2	21.499	
CHM. ANGLE		0	NONE
CHM. AMOUNT		1.000	
FIN ALLOW		0.200	
CUT AMOUNT		0.450	
TOOL T No.		404	
CUT SPEED	V	100	

13.3 Save the data of the rough threading outer diameter machining by pressing the [SAVE] menu.

13.4 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
6 TURN-IN R
7 TURN-IN F
8 GRV-OUT
9 THD-OUT R
FILE

14. Process 6 Threading finishing machining for outer diameter (TOMR)

14.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

14.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	FIN

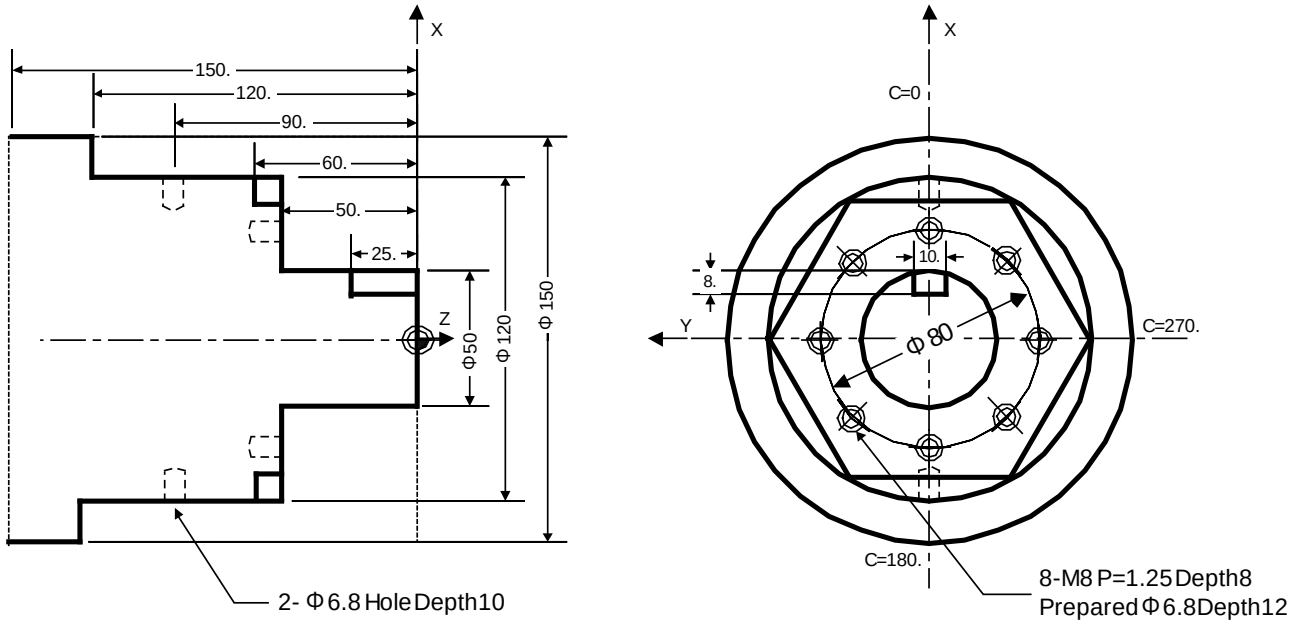
14.3 Save the data of the threading outer diameter finishing machining by pressing the [SAVE] menu.

14.4 Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 DR
2 TURN-FACE R
3 TURN-FACE F
4 TURN-OUT R
5 TURN-OUT F
6 TURN-IN R
7 TURN-IN F
8 GRV-OUT
9 THD-OUT R
10 THD-OUT F
FILE

APPENDIX 3. PROGRAMMING EXAMPLE 2 (MILLING)

Appendix 3.1 Machining Drawing



Appendix 3.2 Process Table

Processes are shown below.

Process	Machining	Tool
1 8-M8	Milling hole drilling for front face	ZCD3 (ϕ3 Center Drill)
	Milling hole drilling for front face	ZDR6.8 (ϕ6.8 Drill)
	Milling hole drilling for front face	ZDC20 (ϕ20 Countersink)
	Milling tap machining for front face	ZTPM8 (M8 P=1.25 Tap)
2	Contour rough cutting for front face	ZED10 (ϕ10 End Mill)
	Contour finishing cutting for front face	ZED10 (ϕ10 End Mill)
3 2-ϕ6.8	Milling hole drilling for outer surface	XCD3 (ϕ3 Center Drill)
	Milling hole drilling for outer surface	XDR6.8 (ϕ6.8 Drill)
	Milling hole drilling for outer surface	XDC20 (ϕ20 Countersink)
4	Keyway rough cutting for outer surface	XED10 (ϕ10 End Mill)
	Keyway finishing cutting for outer surface	XED10 (ϕ10 End Mill)

Appendix 3.3 Condition Setting

Set the tool and cutting conditions before programming.

(1) Tool file screen for milling

Register the tool data. Input the following values on the tool file screen for milling.

No.	701	702	703	704	705
T NAME	ZCD3	ZDR6.8	ZDC20	ZTPM8	ZED10
T No.	101	202	303	404	505
DIA	3.	6.8	20.	8.	10.
NOSE ANGLE	120.	118	90	180	180
F/PITCH	0.06	0.12	0.28	1.25	0.4
SP DIR	1	1	1	1	1
TIP MATERIAL	H	H	H	W	W

No.	711	712	713	714	715
T NAME	XCD3	XDR6.8	XDC20	XTPM8	XED10
T No.	1111	1212	1313	1414	1515
DIA	3	6.8	20.	8.	10.
NOSE ANGLE	120	118	90	180	180
F/PITCH	0.06	0.12	0.28	1.25	0.4
SP DIR	1	1	1	1	1
TIP MATERIAL	H	H	H	W	W

(2) Cutting condition file screen for milling

Set the cutting speed for the tip material, as well as coefficients of cutting speed rate and feedrate for the workpiece material. Input as follows on the cutting condition file screen for milling machining.

[Cutting condition file (tip material)]

Item	1	2
TIP MATL	H	W
DRILL V	23.	65.
TAP V	12.	12.
BORE V	23.	95.
END ML R V	22.	40.
END ML F V	25.	55.

[Cutting condition file (workpiece material)]

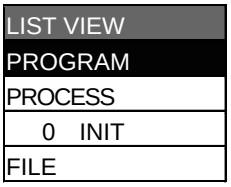
Item	1
WORK MATL	S45C
DRILL V	100
F	100
TAP V	100
BORE V	100
F	100
END ML R V	100
F	100
END ML F V	100
F	100

Appendix 3.4 Creating Program

1. Open the program edit screen.

2. Press the [NEW] menu and create a new NC program.

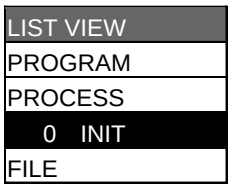
3. Move the cursor to "0 INIT" and press the [MODIFY] menu.



4. Input the following values.

Item	Setting value	Details
WORK REG No.	1	S45C
WORK ZERO	1	T'STK SIDE
OUTSIDE DIA OD	150.000	
INSIDE DIA ID	0.000	
+Z	0.000	
-Z	-150.000	
WORK COORDINATE	54	G54
COOLANT	1	VALID
TOOL CHANGE POS	1	X REF
FIN TOOL RET	1	REF
END POS X	-	
Z	-	
M CODE	1	M30

4.1 Save the initial conditions by pressing the [SAVE] menu.
4.2 Turn the LIST VIEW area active by pressing the [←] key.



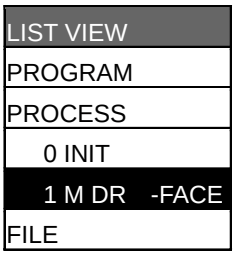
5. Process 1 Milling hole drilling for front face (ϕ3 Center Drill)

5.1 Open the process mode selection screen by pressing the [NEW] menu.
5.2 Open the milling hole drilling screen and set the following items.

Item	Setting value	Details
TOOL REG No.	701	ZCD3
PARTS	1	FACE
HOLE CYCLE	1	DRILL
BASE PLANE BZ	50.000	
DEPTH H	3.000	
NOSE DEPTH B	3.866	
SPOT DIAMETER D	3.000	
CUT AMOUNT	-	
DWELL	0.000	
RETURN POINT	2	R point
C-AXIS CLAMP	1	VALID
TOOL T No.	101	
TOOL DIAMETER	3.000	
CUT SPEED V	23	
FEED RATE F	0.06	

5.3 Press the [PATTERN] menu and set the following items.
<Hole drilling machining pattern screen (circle)>

Item	Setting value	Details
PATTERN	4	CIRCLE
BASE POS X	0.	
BASE POS Y	0.	
DIAMETER D	80	
START ANGLE A	0	
NUM OF HOLES	8	
OMIT 1	0	
2	0	
3	0	
4	0	



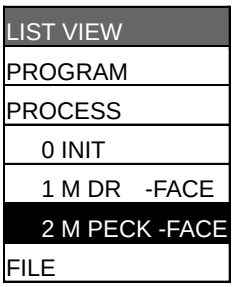
5.4 Press the [RETURN] menu to change to the milling hole drilling screen before pressing the [SAVE] menu.
5.5 Turn the LIST VIEW area active by pressing the [←] key.

6. Process 2 Milling hole drilling for front face (ϕ6.8 Drill)

6.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

6.2 Press the [MODIFY] menu and set the following items.

Item	Setting value	Details
TOOL REG No.	702	
HOLE CYCLE	2	PECK
DEPTH H	12	
CUT AMOUNT	2	



6.3 Press the [SAVE] menu.

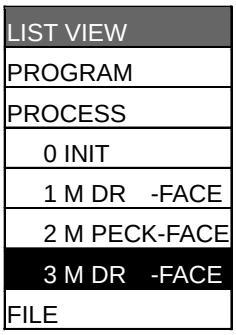
6.4 Turn the LIST VIEW area active by pressing the [←] key.

7. Process 3 Milling hole drilling for front face (ϕ20 Countersink)

7.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

7.2 Press the [MODIFY] menu and set the following items.

Item	Setting value	Details
TOOL REG No.	703	
HOLE CYCLE	1	DRILL
SPOT DIAMETER D	10	



7.3 Press the [SAVE] menu.

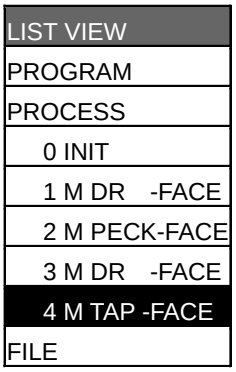
7.4 Turn the LIST VIEW area active by pressing the [←] key.

8. Process 4 Milling tap machining for front face (M8 P=1.25 Tap)

8.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.

8.2 Press the [MODIFY] menu and set the following items.

Item	Setting value	Details
TOOL REG No.	704	
HOLE CYCLE	4	TAP
DEPTH H	8	



8.3 Press the [SAVE] menu.

8.4 Turn the LIST VIEW area active by pressing the [←] key.

9. Process 5 Contour rough cutting for front face (ϕ10 End Mill)

9.1 Open the process mode selection screen by pressing the [NEW] menu.
9.2 Open the contour cutting screen and set the following items.

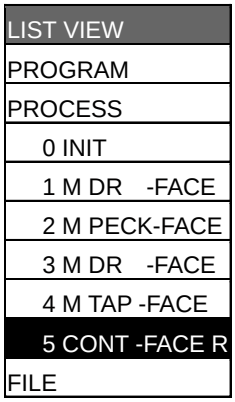
Item		Setting value	Details
TOOL REG No.		705	ZED10
CYCLE		1	Rough
PARTS		1	FACE
BASE PLANE	BZ	50.	
TOOL PATH		2	RIGHT
WIDTH W		18.	
DEPTH D		10.	
FIN ALLOW	FH	2.	
	FV	1.	
CUT AMOUNT		5.	
APPROACH IN AXIS DIR		1	RAPID (G00)
TOOL T No.		505	
DIA		10.	
CUT SPEED	V	40	
FEED RATE	F1	0.4	
	F2	0.2	

9.3 Press the [PATTERN] menu and set the following items.
<Contour cutting pattern screen>

No.	M	X	Y	R/A	I	J
1		70.	19.586			
2	1	0	60.	(150.)		
3	1	-51.962	30.	(210.)		
4	1	-51.962	-30.	(270.)		
5		0	-60.	(330.)		
6		51.962	-30.	(30.)		
7		51.962	47.	(90.)		

(Note) The value in the parentheses is calculated automatically.

9.4 Press the [RETURN] menu to change to the contour cutting screen before pressing the [SAVE] menu.
9.5 Turn the LIST VIEW area active by pressing the [←] key.

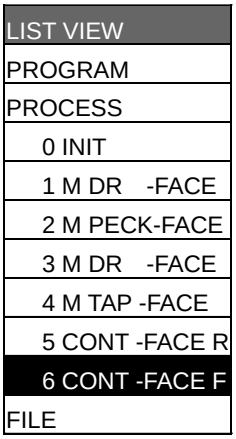


10. Process 6 Contour finishing cutting for front face (ϕ10 End Mill)

10.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.
10.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	Finishing

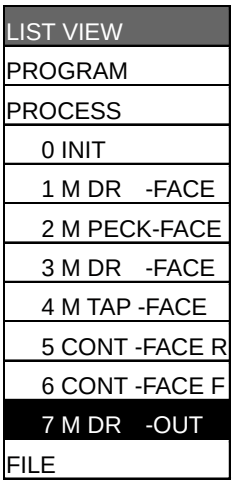
10.3 Press the [SAVE] menu.
10.4 Turn the LIST VIEW area active by pressing the [←] key.



11. Process 7 Milling hole drilling for outer surface (ϕ3 Center Drill)

11.1 Open the process mode selection screen by pressing the [NEW] menu.
11.2 Open the milling hole drilling screen and set the following items.

Item	Setting value	Details
TOOL REG No.	711	ZCD3
PARTS	2	OUT
HOLE CYCLE	1	DRILL
BASE PLANE BR	60.000	
DEPTH H	3.000	
NOSE DEPTH B	3.866	
SPOT DIAMETER D	3.000	
CUT AMOUNT	-	
DWELL	0.000	
RETURN POINT	2	R point
C-AXIS CLAMP	1	VALID
TOOL T No.	1111	
TOOL DIAMETER	3.000	
CUT SPEED V	23	
FEED RATE F	0.06	



11.3 Press the [PATTERN] menu and set the following items.
<Hole drilling machining pattern screen (RANDOM)>

No.	C	Z
1	0.	90.
2	180.	90.

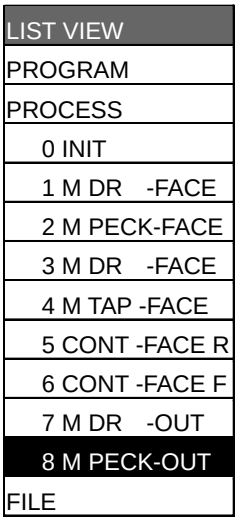
11.4 Press the [RETURN] menu to change to the milling hole drilling screen before pressing the [SAVE] menu.
11.5 Turn the LIST VIEW area active by pressing the [←] key.

12. Process 8 Milling hole drilling for outer surface (ϕ6.8 Drill)

12.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.
12.2 Press the [MODIFY] menu and set the following items.

Item	Setting value	Details
TOOL REG No.	712	
HOLE CYCLE	2	PECK
DEPTH H	12	
CUT AMOUNT	2	

12.3 Press the [SAVE] menu.
12.4 Turn the LIST VIEW area active by pressing the [←] key.



13. Process 9 Milling hole drilling for outer surface (ϕ20 Countersink)

13.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.
13.2 Press the [MODIFY] menu and set the following items.

Item	Setting value	Details
TOOL REG No.	713	
HOLE CYCLE	1	DRILL
SPOT DIAMETER D	10	

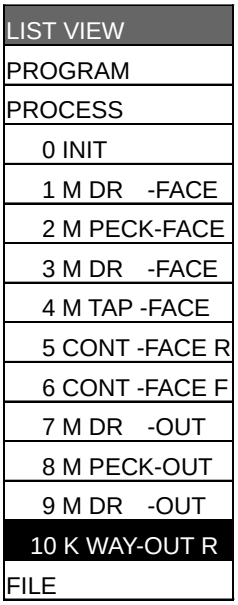
13.3 Press the [SAVE] menu.
13.4 Turn the LIST VIEW area active by pressing the [←] key.

14. Process 10 Keyway rough cutting for outer surface (ϕ10 End Mill)

14.1 Open the process mode selection screen by pressing the [NEW] menu.
14.2 Open the keyway cutting screen and set the following items.

Item		Setting value	Details
TOOL REG No.		715	XED10
CYCLE		1	Rough
PARTS		2	OUT
BASE PLANE	BR	25.	
WIDTH	W	10.	
DEPTH	H	8.	
FIN ALLOW		1.	
CUT AMOUNT		4.	
START ANGLE	SA	0.	
START POS	SZ	-7.	
END POS	EZ	20.	
NUM OF KEYWAY		1	
RETURN POINT		1	Initial point
C-AXIS CLAMP		1	VALID
APPROACH IN AXIS DIR		1	RAPID (G00)
TOOL T No.		1515	
DIA		10.	
CUT SPEED	V	40	
FEED RATE	F1	0.4	
	F2	0.2	

14.3 Press the [SAVE] menu.
14.4 Turn the LIST VIEW area active by pressing the [←] key.

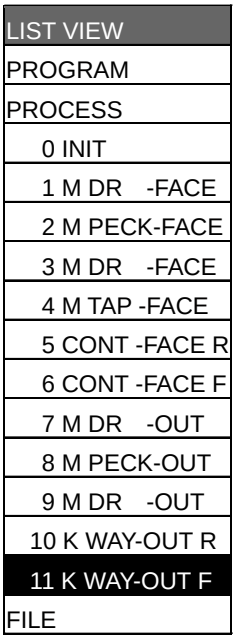


15. Process 11 Keyway finishing cutting for outer surface (ϕ10 End Mill)

15.1 Press the [COPY] menu and move down the cursor in the LIST VIEW area.
15.2 Press the [MODIFY] menu and set the following item.

Item	Setting value	Details
CYCLE	2	Finishing

15.3 Press the [SAVE] menu.
15.4 Turn the LIST VIEW area active by pressing the [←] key.



Revision History

Date of revision	Manual No.	Revision details
Nov. 2005	IB(NA)1500146-A	First edition created.
Mar.2007	IB(NA)1500146-B	・Milling function was added. ・Explanations for 70 Series were added. ・Mistakes were corrected.
Apr. 2010	IB(NA)1500146-C	・Reviewed "Precautions for Safety". ・Corrected the mistakes.
Oct. 2015	IB(NA)1500146-D	The descriptions were revised corresponding to S/W version FH of MITSUBISHI CNC M700/M70 Series. The descriptions were revised corresponding to S/W version L0 of MITSUBISHI CNC M700V/M70V/E70 Series. The assist and balance cut processes were added to the machining process edit. The description of 2-part system function was added. The description of NC check function was added. Restrictions for E70 series were added. Other contents were reviewed/corrected.
Apr. 2016	IB(NA)1500146-E	Precautions were added to "Precautions for Safety", "7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS" and "APPENDIX 1. VARIABLES USED IN NAVI LATHE".

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Michigan Service Satellite
ALLEGAN, MICHIGAN 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Ohio Service Satellite
LIMA, OHIO 45801, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
CINCINATTI, OHIO 45201, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
ROGERS, MINNESOTA 55374, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
16900 VALLEY VIEW AVE., LAMIRADA, CALIFORNIA 90638, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Northern CA Satellite
SARATOGA, CALIFORNIA 95070, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650

Pennsylvania Service Satellite
PITTSBURG, PENNSYLVANIA 15644, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Connecticut Service Satellite
TORRINGTON, CONNECTICUT 06790, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

South Region Service Center
1845 SATTELITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL +1-678-258-4529 / FAX +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519
HOUSTON, TEXAS 77001, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Tennessee Service Satellite
Nashville, Tennessee, 37201, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Florida Service Satellite
WEST MELBOURNE, FLORIDA 32904, U.S.A.
TEL: +1-678-258-4529 / FAX: +1-678-258-4519

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Canada Service Satellite
EDMONTON, ALBERTA T5A 0A1, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935

Mexico Region Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-3067-7500 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171

BRAZIL

MELCO CNC do Brasil Comércio e Serviços S.A

Brazil Region Service Center
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3363-9900
JOVIMAQ – Joinville, SC Satellite office
MAQSERVICE – Canoas, RS Satellite office

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.
Mitsubishi-Electric-Platz 1, 40882 RATINGEN, GERMANY
TEL: +49-2102-486-1850 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center DEPARTEMENT CONTROLE NUMERIQUE
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite DEPARTEMENT CONTROLE NUMERIQUE
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI, 7 - CENTRO DIREZIONALE COLLEONI PALAZZO SIRIO INGRESSO 1
20864 AGRATE BRIANZA (MB), ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Branch
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08173 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-630-4700 / FAX: +48-12-630-4701

Mitsubishi Electric Turkey A.Ş Ümraniye Şubesi
Turkey Service Center
ŞERIFALI MAH. NUTUK SOK. NO.5 34775
ÜMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
KAFKOVA 1853/3, 702 00 OSTRAVA 2, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

MITSUBISHI ELECTRIC EUROPE B.V. (SCANDINAVIA)
Sweden Service Center
HAMMARBACKEN 14 191 49 SOLLENTUNA, SWEDEN
TEL: +46-8-6251000 / FAX: +46-8-966877

Bulgaria Service Center
4 A.LYAPCHEV BOUL., POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY LANE 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +380-57-732-7774 / FAX: +380-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +380-44-494-3355 / FAX: +380-44-494-3366

Belarus Service Center
OFFICE 9, NEZAVISIMOSTI PR.177, 220125 MINSK, BELARUS
TEL: +375-17-393-1177 / FAX: +375-17-393-0081

South Africa Service Center
5 ALBATROSS STREET, RHODESFIELD, KEMPTON PARK 1619, GAUTENG, SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Malaysia (KL) Service Center
60, JALAN USJ 10 /1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
17 & 17A, JALAN IMPIAN EMAS 5/5, TAMAN IMPIAN EMAS, 81300 SKUDAI, JOHOR MALAYSIA.
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Philippines Service Center
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO.,LTD

Vietnam (Ho Chi Minh) Service Center
UNIT 01-04, 10TH FLOOR, VINCOM CENTER 72 LE THANH TON STREET, DISTRICT 1,
HO CHI MINH CITY, VIETNAM
TEL: +84-8-3910 5945 / FAX: +84-8-3910 5946

Vietnam (Hanoi) Service Satellite
6th Floor, Detech Tower, 8 Ton That Thuyet Street, My Dinh 2 Ward, Nam Tu Liem District, Hanoi,Vietnam
TEL: +84-4-3937-8075 / FAX: +84-4-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA

Indonesia Service Center (Cikarang Office)
JL.Kenari Raya Blok G2-07A Delta Silicon 5, Lippo Cikarang-Bekasi 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO.,LTD

Thailand Service Center
12TH FLOOR, SV.CITY BUILDING, OFFICE TOWER 1, NO. 896/19 AND 20 RAMA 3 ROAD,
KWAENG BANGPONGPANG, KHET YANNAWA, BANGKOK 10120,THAILAND
TEL: +66-2-682-6522-31 / FAX: +66-2-682-6020

INDIA

MITSUBISHI ELECTRIC INDIA PVT. LTD.

India Service Center
2nd FLOOR, TOWER A & B, DLF CYBER GREENS, DLF CYBER CITY,
DLF PHASE-III, GURGAON 122 002, HARYANA, INDIA
TEL: +91-124-4630 300 / FAX: +91-124-4630 399
Ludhiana satellite office
Jamshedpur satellite office

India (Pune) Service Center
EMERALD HOUSE, EL-3, J-BLOCK, MIDC BHOSARI. PUNE – 411 026, MAHARASHTRA, INDIA
TEL: +91-20-2710 2000 / FAX: +91-20-2710 2100
Baroda satellite office
Mumbai satellite office

India (Bangalore) Service Center
PRESTIGE EMERALD, 6TH FLOOR, MUNICIPAL NO. 2,
LAVELLE ROAD, BANGALORE - 560 043, KAMATAKA, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699
Chennai satellite office
Coimbatore satellite office

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Australia Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)

China (Shanghai) Service Center
1-3,5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2308-3000
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer
China (Hangzhou) Service Dealer
China (Wuhan) Service Satellite

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030
China (Beijing) Service Dealer

China (Tianjin) Service Center
UNIT 2003, TIANJIN CITY TOWER, NO 35 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite
China (Changchun) Service Satellite

China (Chengdu) Service Center
ROOM 407-408, OFFICE TOWER AT SHANGRI-LA CENTER, NO. 9 BINJIANG DONG ROAD,
JINJIANG DISTRICT, CHENGDU, SICHUAN 610021, CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China (Shenzhen) Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776
China (Xiamen) Service Dealer
China (Dongguan) Service Dealer

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)

Korea Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200 KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Daegu Service Satellite
4F KT BUILDING, 1630 SANGYEOK-DONG, BUK-KU, DAEGU 702-835, KOREA
TEL: +82-53-382-7400 / FAX: +82-53-382-7411

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)

Taiwan (Taichung) Service Center (Central Area)
NO.8-1, INDUSTRIAL 16TH RD., TAICHUNG INDUSTRIAL PARK, SITUN DIST.,
TAICHUNG CITY 40768, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center (North Area)
10F, NO.88, SEC.6, CHUNG-SHAN N. RD., SHI LIN DIST., TAIPEI CITY 11155, TAIWAN R.O.C.
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan (Tainan) Service Center (South Area)
11F-1., NO.30, ZHONGZHENG S. ROAD, YONGKANG DISTRICT, TAINAN CITY 71067, TAIWAN, R.O.C.
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

© 2005-2016 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED.

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	Simple Programming Function
MODEL CODE	008-366
Manual No.	IB-1500146

IB(NA)1500146(ENG)-E(1604)MEE
Made in JAPAN

Specifications are subject to change without notice.