Changes for the Better

MITSUBISHI ELECTRIC

MITSUBISHI CNC

Simple Programming Function NAVI LATHE Instruction Manual

M700/M70 Series

IB-1500146(ENG)-C

Introduction

This manual is an instruction manual for NAVI LATHE for 700/70 (hereafter NAVI LATHE). This manual explains how to operate NAVI LATHE, so read this manual thoroughly before use. Be sure to study "Precautions for Safety" on the next page and use the system safely.

Details described in this manual

⚠ CAUTION

- ⚠ 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 to 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.

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

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	#150 to #197
MTB macro mode	#450 to #497

When NC program mode is user macro mode, do not use common variables (#150 to #197). 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 the power is turned 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.

(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.

Disposal

(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.
This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.
If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:
Hg: mercury (0,0005%), Cd: cadmium (0,002%), Pb: lead (0,004%)
In the European Union there are separate collection systems for used batteries and accumulators.
Please, dispose of batteries and accumulators correctly at your local community waste collection/ recycling centre.

Please, help us to conserve the environment we live in!

Contents

1. OUTLINE

1.1 System Outline

This manual is an instruction manual for NAVI LATHE for 700/70 (hereafter NAVI LATHE).
The part program for the turning center is created with the NAVI LATHE.
NAVI LATHE provides the turning function and the milling function.

(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

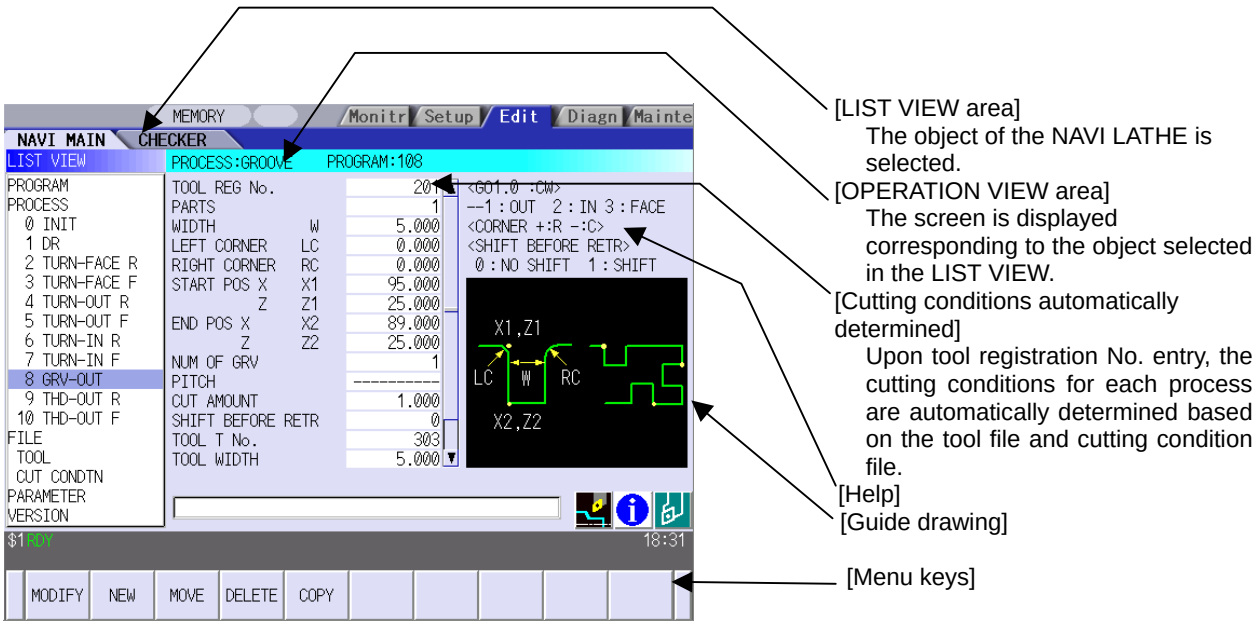
Milling Processes

- Milling hole drilling (Drilling, deep-hole drilling, boring, tapping)
- Keyway cutting (Front face, outer surface, side surface)
- Contour cutting (Front face, outer surface, side surface)

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

(2) The tool file and the cutting condition file 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.

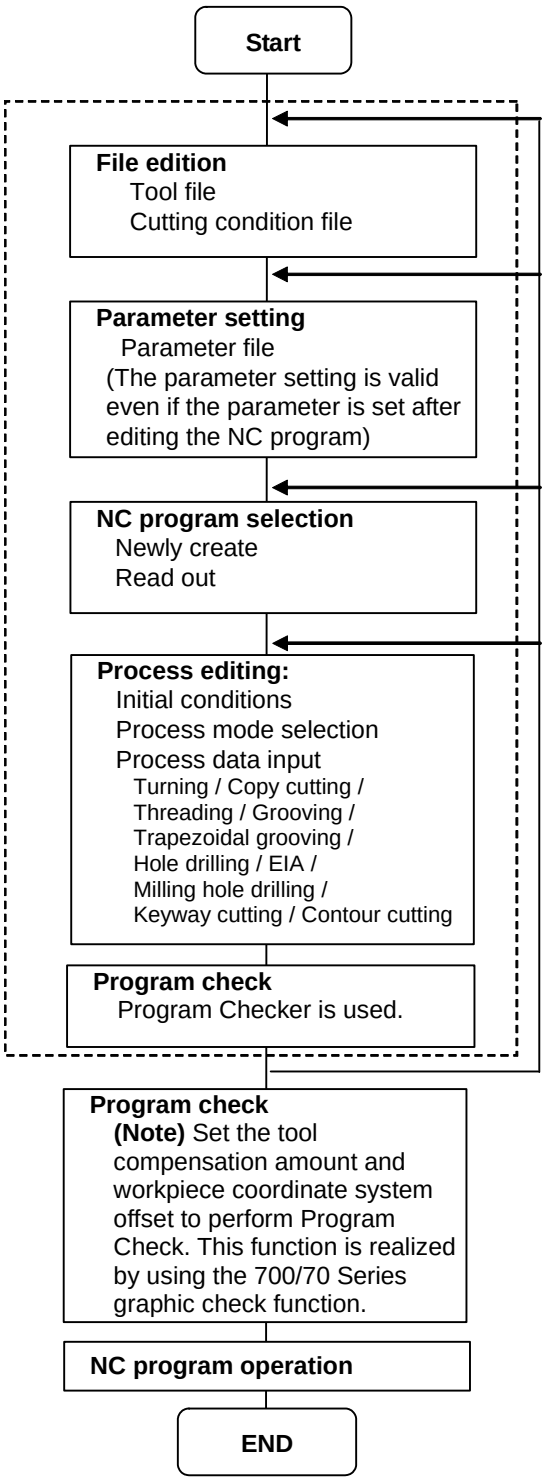
(6) Part program is a macro-program-based NC program. Commands can be added between processes from the edit screen of the standard MITSUBISHI CNC 700/70 Series.

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

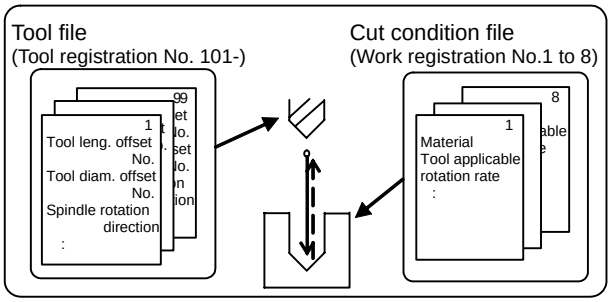
1.2 Input Procedures

The input procedure for the NAVI LATHE is shown below.

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

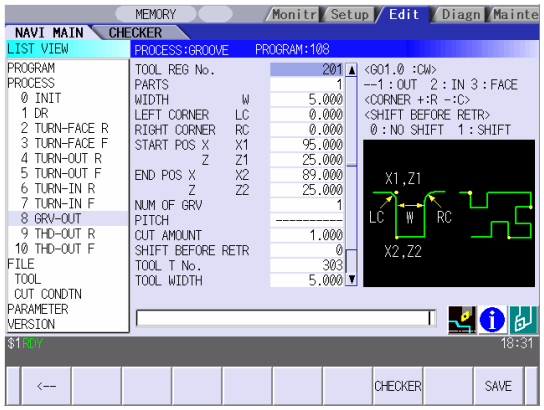


Supplements



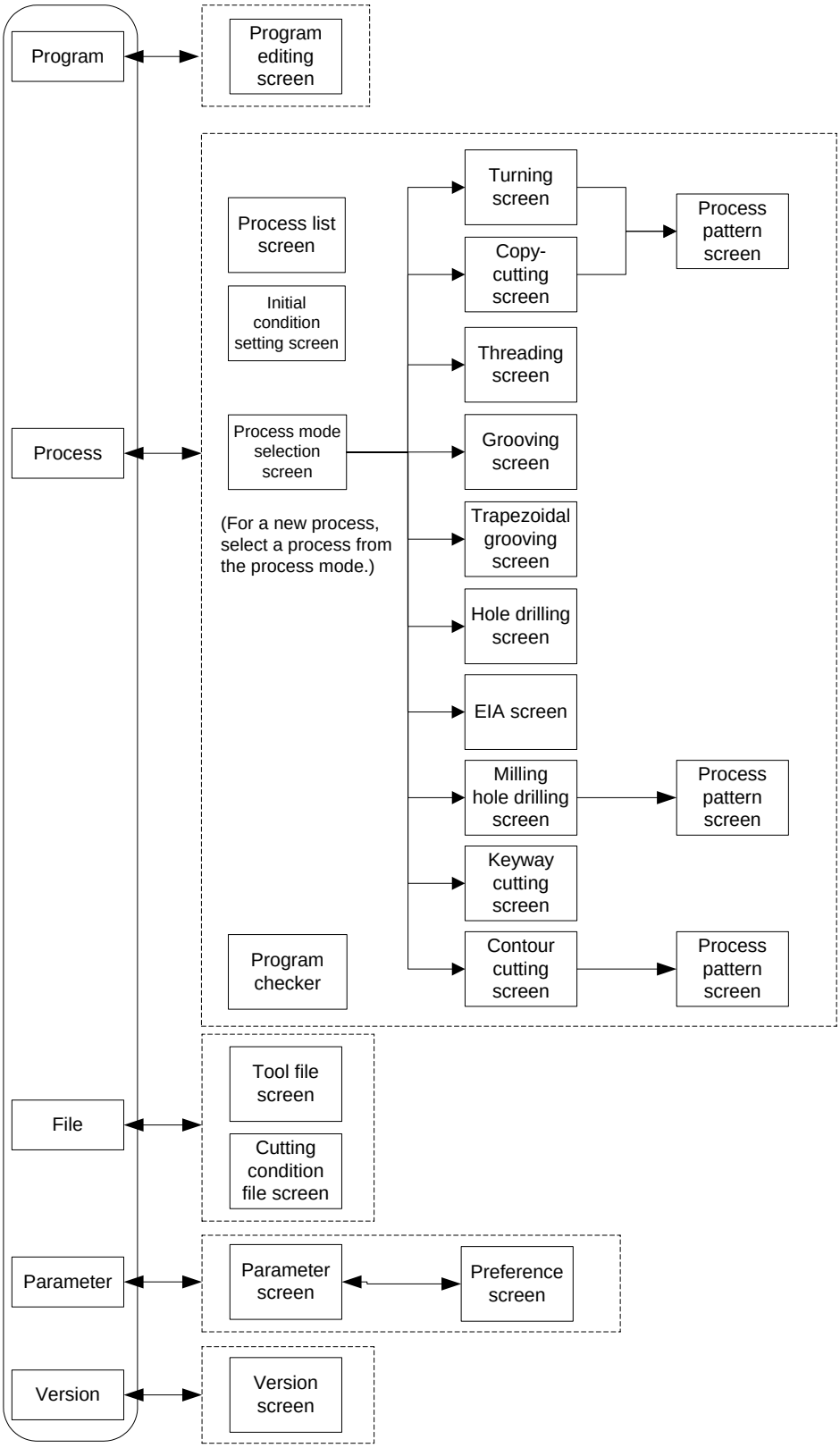
Parameter setting
- M0 output
- Maximum number of spindle rotations
- Clearance
- Tool return position
- Common parameters for threading process
- Common parameters for grooving process
- Common parameters for hole drilling process

Process editing



1.3 Screen Configuration

The screen configuration for the NAVI LATHE is shown below.



Screen name	Details
Program editing 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.
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.
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.
Tool file screen	The tool data by each tool 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.
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

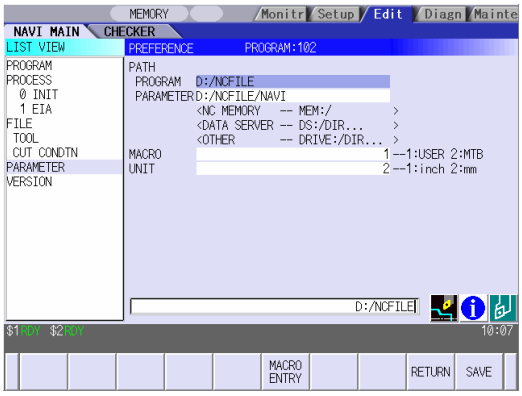
Item	Details	Standard value
PATH PROGRAM	Path to the folder in which NC program is saved.	MEM:/
PATH PARAMETER	Path to the folder in which tool file, cutting condition file and parameter file are saved.	In 700 Series: D:/NCFILE/NAVI In 70 Series: MEM:/
MACRO	Macro program mode 1: User macro mode 2: MTB macro mode	1 (User Macro)
UNIT	Unit for data input 1: inch 2: mm	2 (mm)

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:Uer 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.

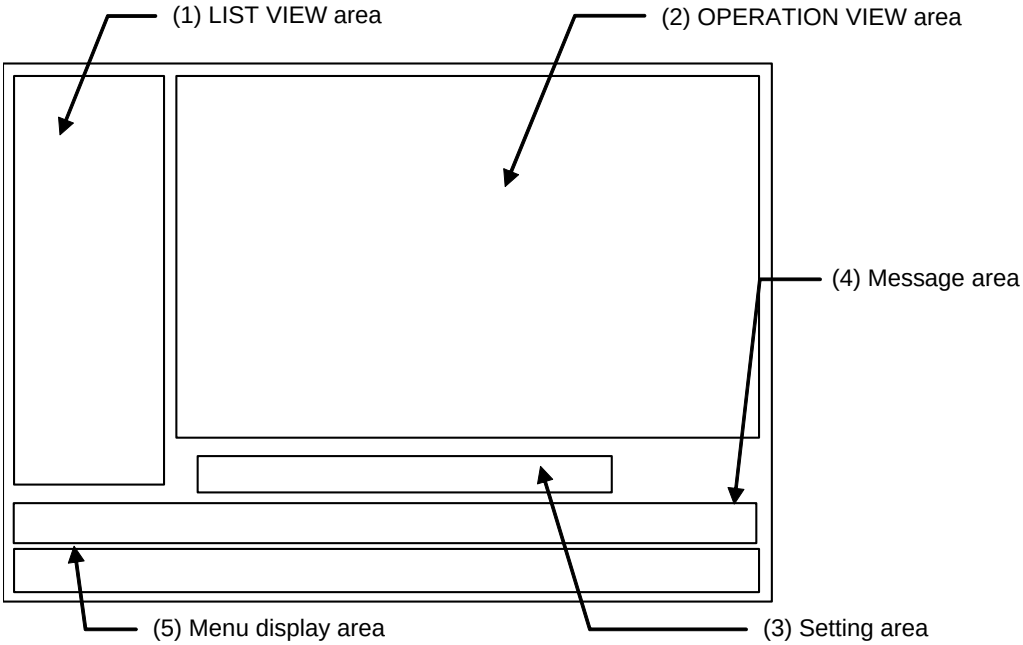
(Addendum)

- 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.

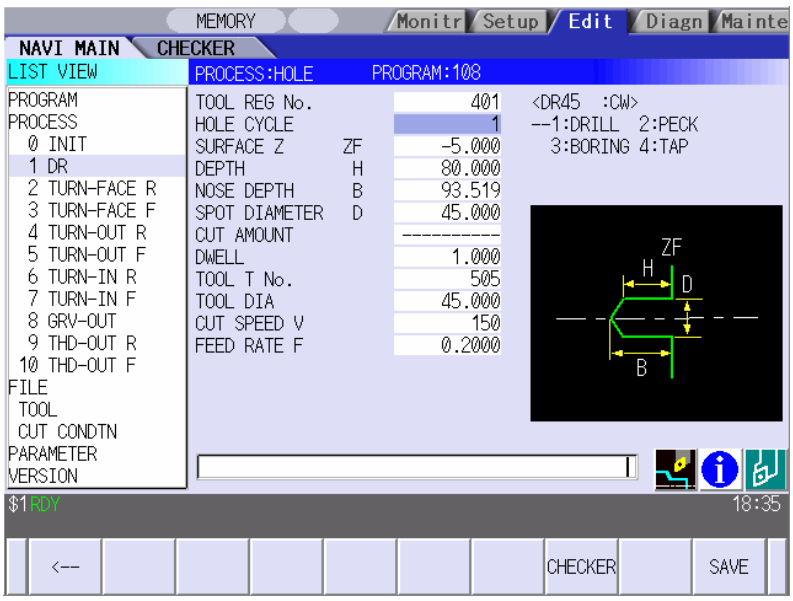
2. FUNCTIONS OF DISPLAY AREA

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")

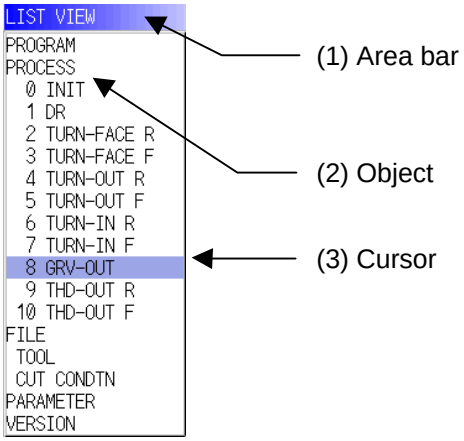


<Screen example>



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.
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.

(3) Cursors

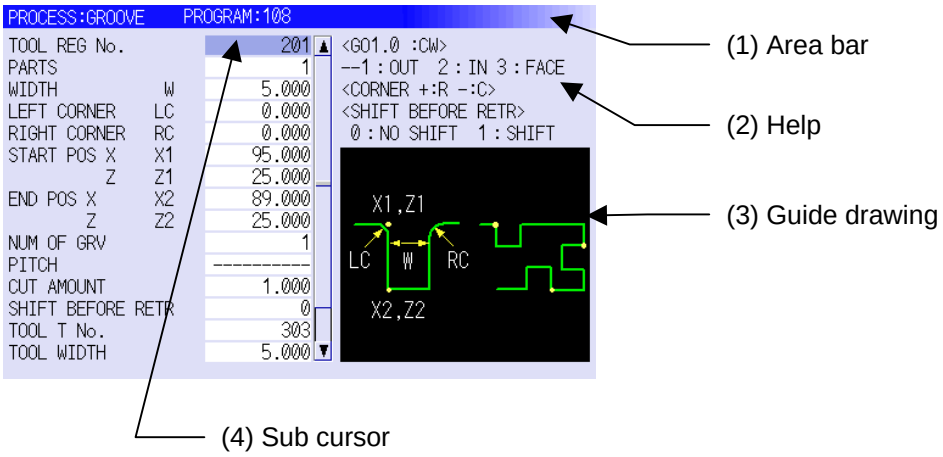
When the LIST VIEW area is active and the object is selected with the cursor, the display in the OPERATION VIEW area and the menu display area will be changed.

<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.

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

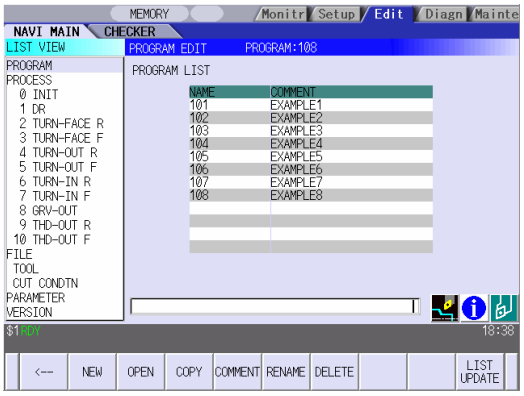
To operate NAVI LATHE, activate either LIST VIEW area or OPERATION VIEW area. When the VIEW is active, the area bar is highlighted and data can be input. Use menu keys [←] and [→] or a pointing device to switch either one of the VIEWs to be activated.

3.2 Changing Screen

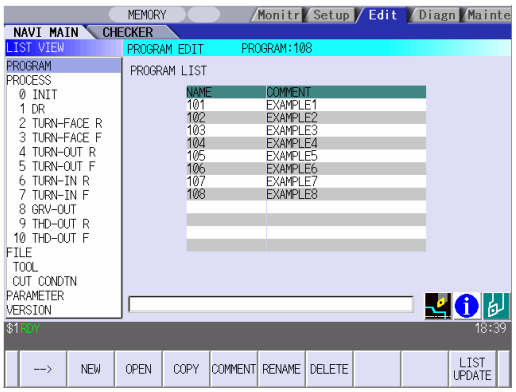
When the object is selected in the LIST VIEW area, the screen (contents in the OPERATION VIEW area) changes. (Refer to the section 2.1 LIST VIEW Area.)
Note that the screen cannot be changed while the OPERATION VIEW area is active.
In such a case, press the [←] menu key or click "LIST VIEW" with a pointing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

Operation example

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

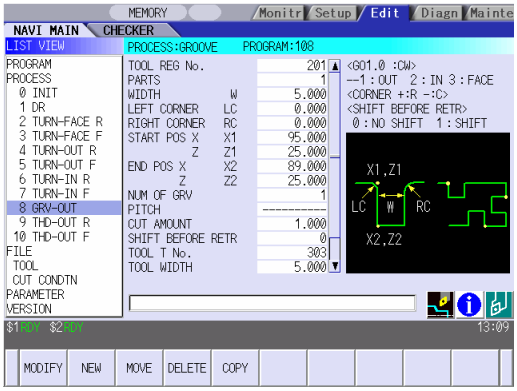


(2) Press the [←] menu key. ➡ The LIST VIEW area will turn active.



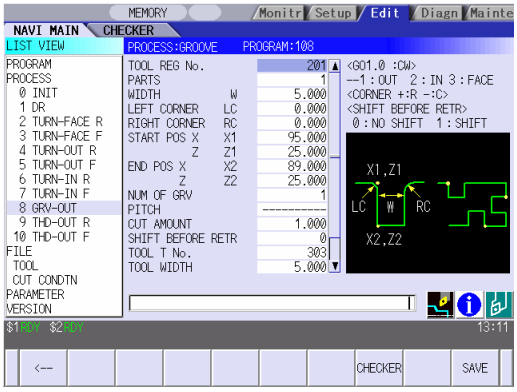
(3) Select the object with the cursor key.

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



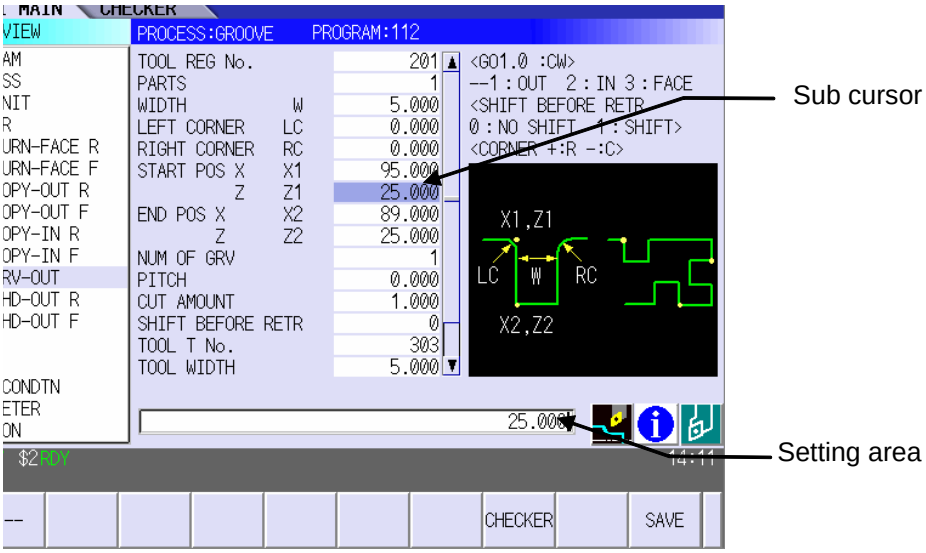
(4) Press the [MODIFY] menu key.

➡ 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

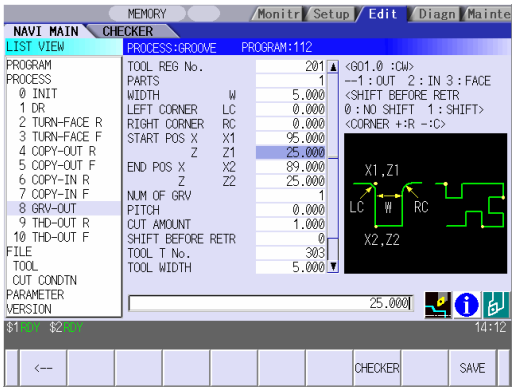
An example for setting the data on the hole drilling screen is shown below.

(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.

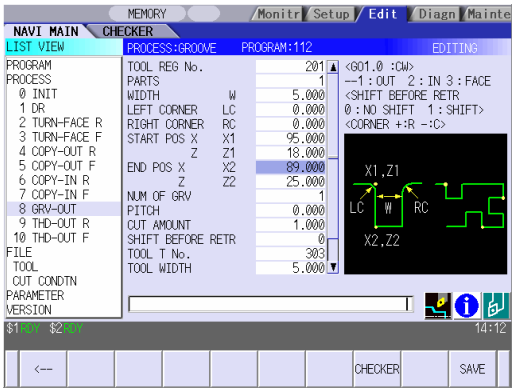


(3) Data key input

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

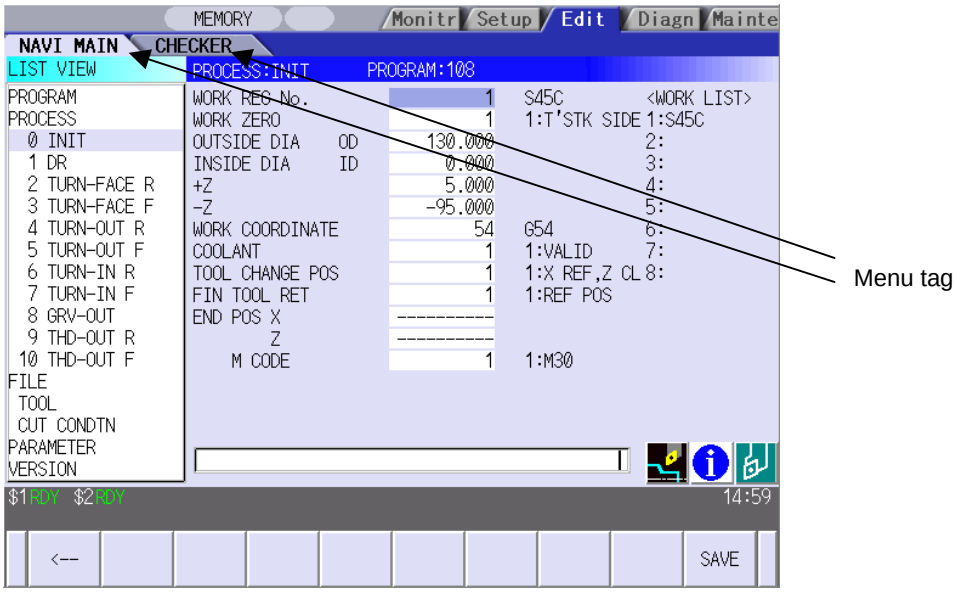
(2) Press the [DETETE] key. ➡ The character in front of the cursor is deleted.
1 2 3 | 5 6

3.4 Switching Windows

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

Button	Application
LIST	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, tag 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

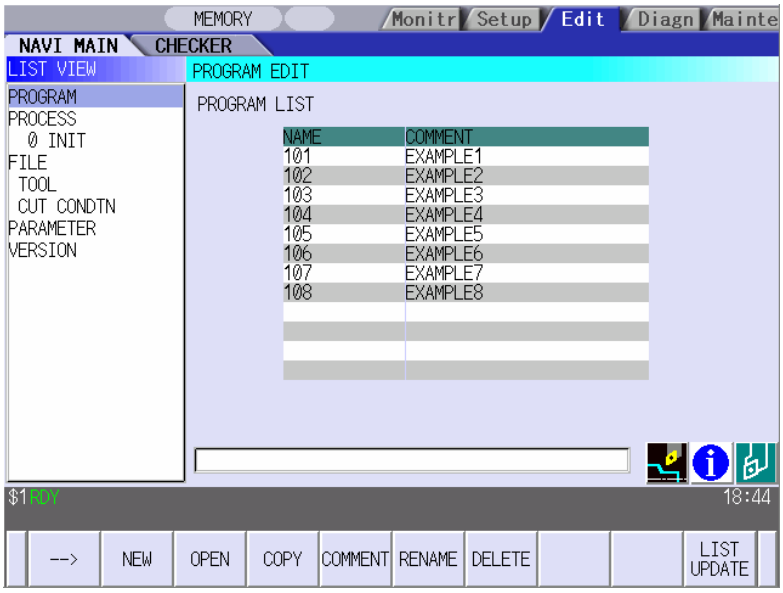
- Always use "=" for the first character.
- Do not use the following characters as the second character or last character.
  Invalid as second character: ∗, /,)
  Invalid as last character: ∗, /, (, +, -
- Make sure that the left parentheses and right parentheses are balanced.
- The 360° limit does not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



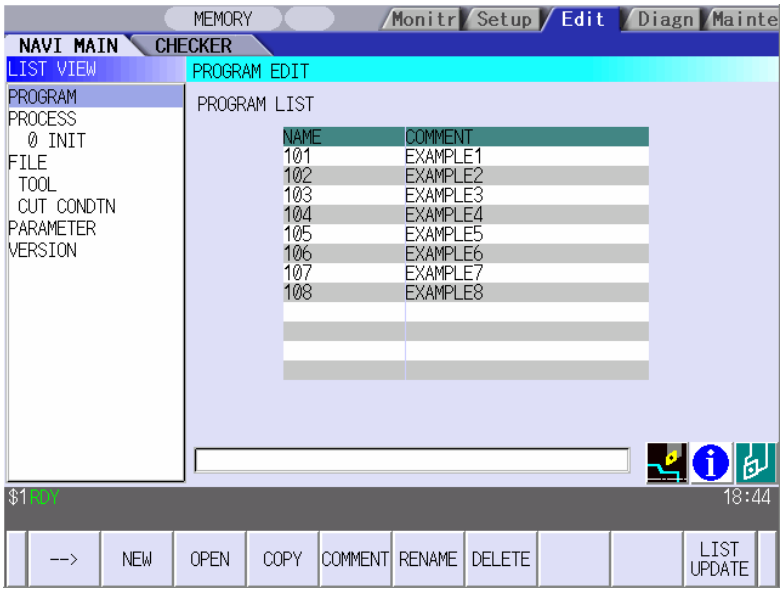
At the initial start up of NAVI LATHE, the cursor is displayed at the position of [PROGRAM] in the LIST VIEW area, and the program edit screen is displayed in the OPERATION VIEW area.
The LIST VIEW area is active.
The process program is not selected.

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 ?	
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
	Inner diameter	THD-IN ?	
	Face	THD-FACE ?	
Groove	Outer diameter	GRV-OUT	
	Inner diameter	GRV-IN	
	Face	GRV-FACE	
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 ?	
Hole drilling	Drill	DR	
	Deep hole	PECK	
	Step	STEP	
	Tapping	TAP	
EIA		EIA	

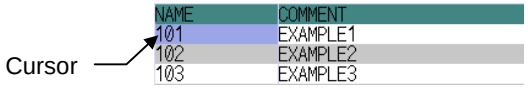
<Milling process displays>

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 ****. ・Front face: FACE ・Outer surface: OUT ・Side surface: SIDE
	Deep hole drilling	M PECK-****	
	Step	M STEP-****	
	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 ?	
Contour cutting	Front face	CONT-FACE ?	
	Outer surface	CONT-OUT ?	
	Side surface	CONT-SIDE ?	

Screen display item

No.	Display item	Details	Setting range
1	PROGRAM LIST	Displays the program number and comment of the NC program that can be currently read out.	-

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( )
7	DELETE	Deletes the NC program. < Display in the setting area when pressing the menu > O( ) to O( )
8	LIST UPDATE	Updates the list display.

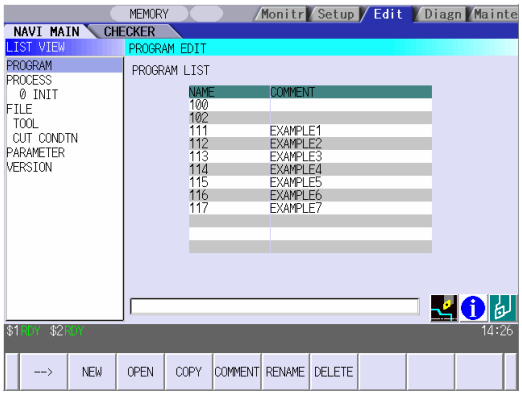
(Note 1) 1 to 7999 or 10000 to 99999999 can be set for the O No, and up to 18 alphanumeric characters can be set for the comment.

(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.

Operation example (Opening the existing NC program)

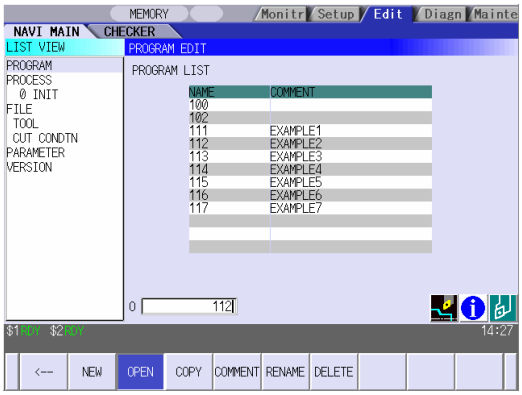
(1) Select the [PROGRAM] in the LIST VIEW area.

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



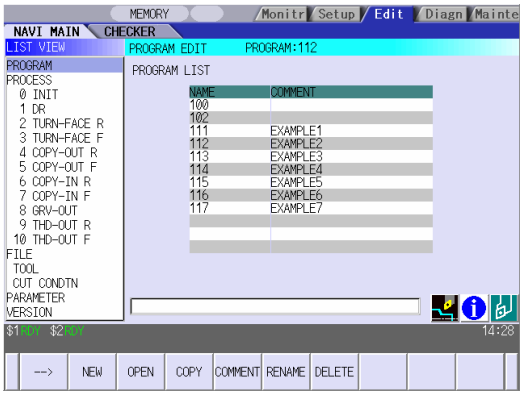
(2) Press the [OPEN] menu key, and input the NC program No. to be read out.

The [OPEN] menu will be highlighted, and the setting area will be displayed.



(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.

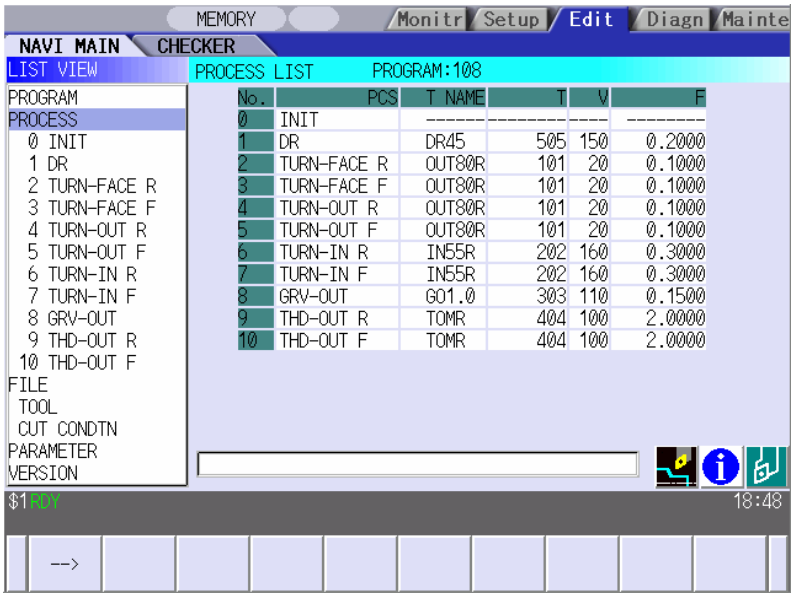


4.3 Screens Related to the Process Edit Functions

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

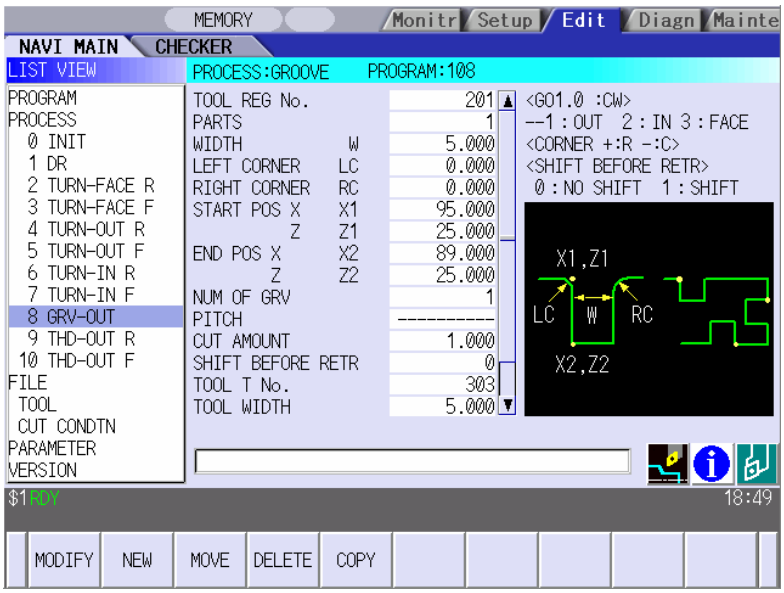
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	SAVE	Saves changes in the process list.

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



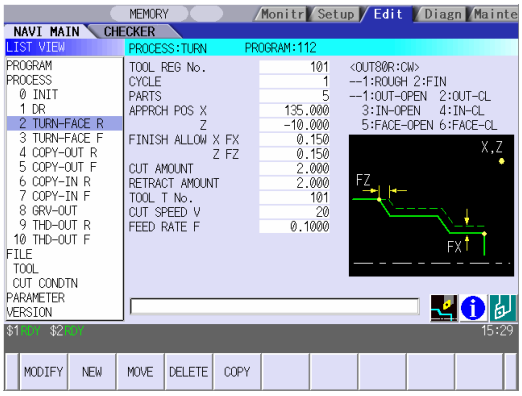
Menus

No.	Menu	Details
1	MODIFY	The OPERATION VIEW area turns active, and the process parameters can be changed.
2	NEW	Adds a new process. The process will be inserted into the cursor position.
3	MOVE	Changes the process position.
4	DELETE	Deletes the process at the cursor position. When performing the deletion, the process under the deleted process will be moved up.
5	COPY	Copies the process at the cursor position. The copied process will be inserted under the cursor position.

Operation example (Selecting the process)

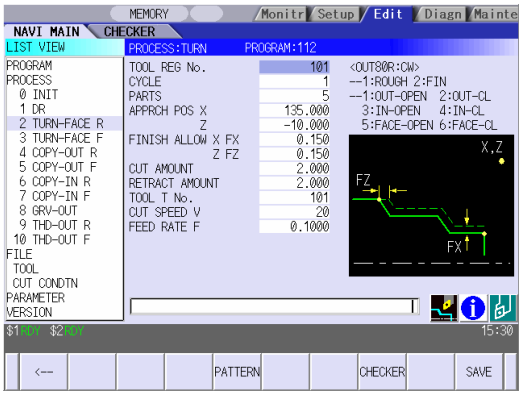
(1) Validate the LIST VIEW area, select the process with the cursor key.

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



(2) Press the [MODIFY] menu key.

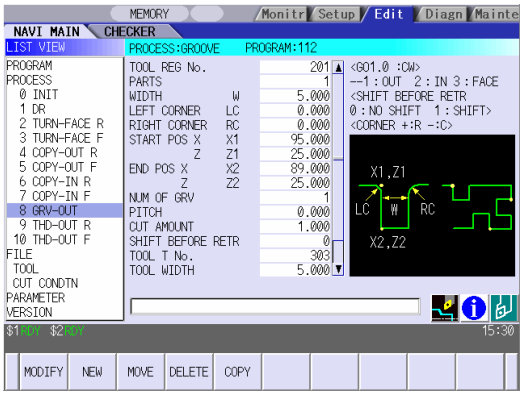
➡ The OPERATION VIEW area will turn active.



Operation example (Deleting the process)

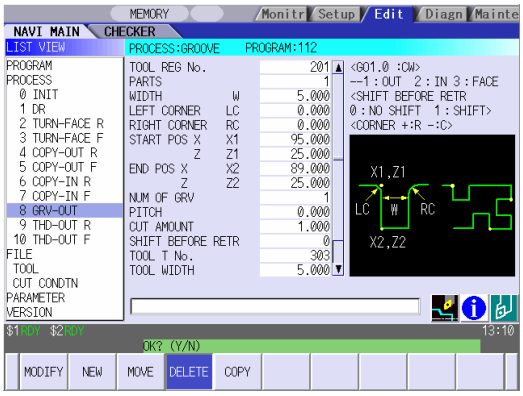
(1) Validate the LIST VIEW area, select the process to be deleted with the cursor key.

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



(2) Press the [DELETE] menu key.

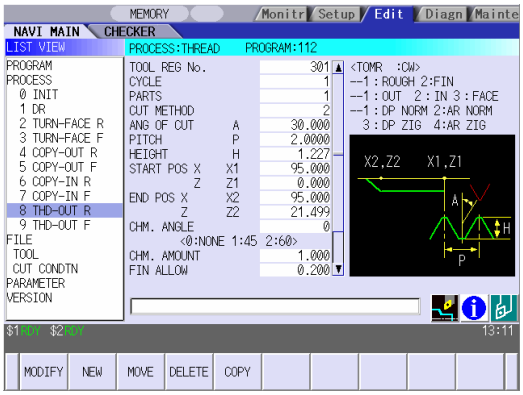
➡ The [DELETE] menu will be highlighted, and a massage confirming the deletion will appear.



(3) Press the [Y] key.

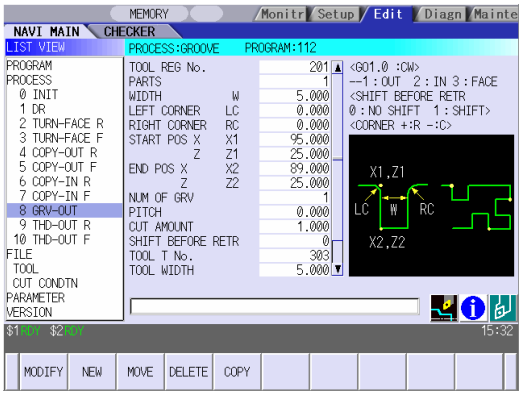
When not deleting the process, press the [N] key

➡ 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.

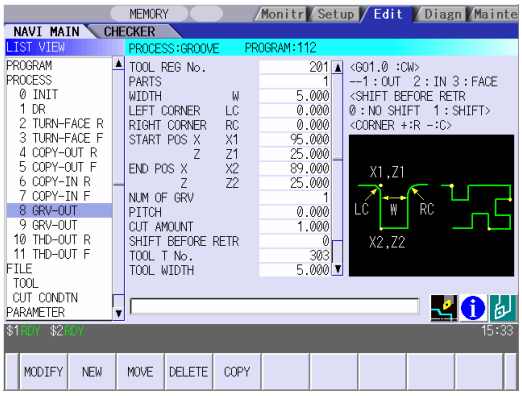


Operation example (Copying the process)

(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.

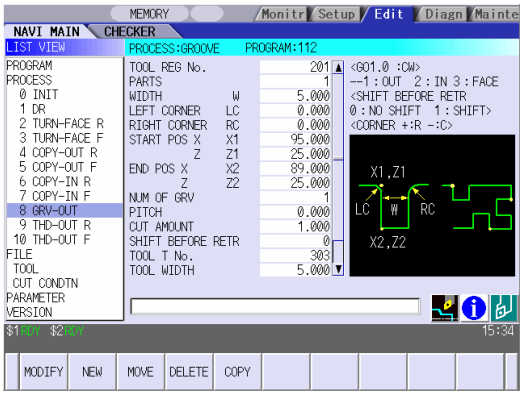


(2) Press the [COPY] menu key. ➡ The copied process will be inserted under the cursor position.

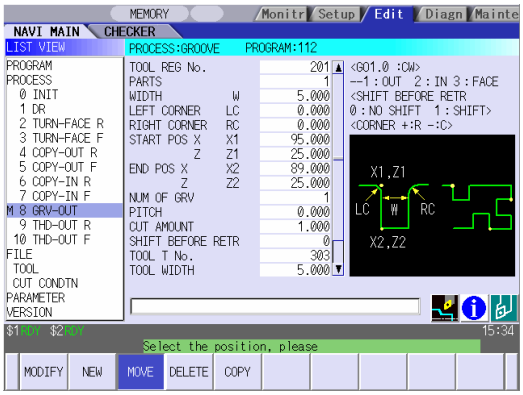


Operation example (Moving the process)

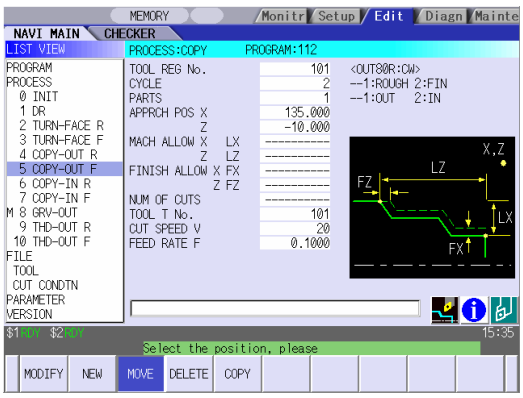
(1) Validate the LIST VIEW area, select the process to be moved with the cursor key. ➡ The contents of the OPERATION VIEW area will change to those of the selected process.



(2) Press the [MOVE] menu key. ➡ The [MOVE] menu will be highlighted. The mark "M" will be displayed beside the process to be moved.



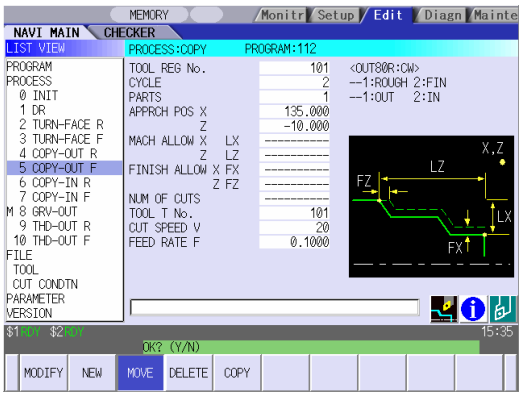
(3) Select the position of the movement destination with the cursor key. ➡



(4) Press the [INPUT] key.

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

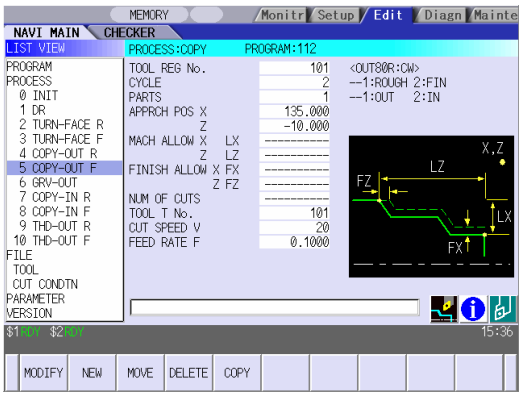
➡ The message to confirm a movement is displayed.



(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.



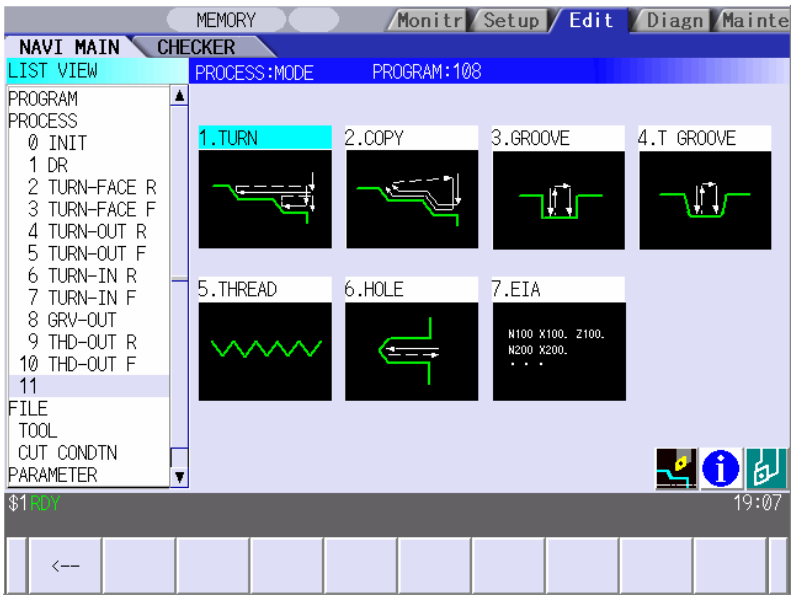
(Note) For the [NEW] menu, refer to the next section.

4.3.3 Process Mode Selection Screen

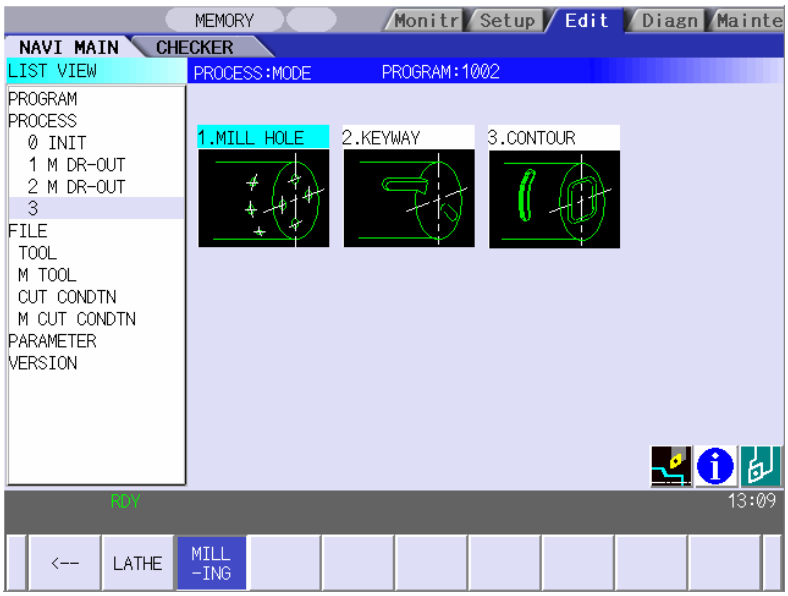
When a new process is added, the process mode is selected on this screen.

Screen layout

• Turning process



• Milling Process



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

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

• Milling Process

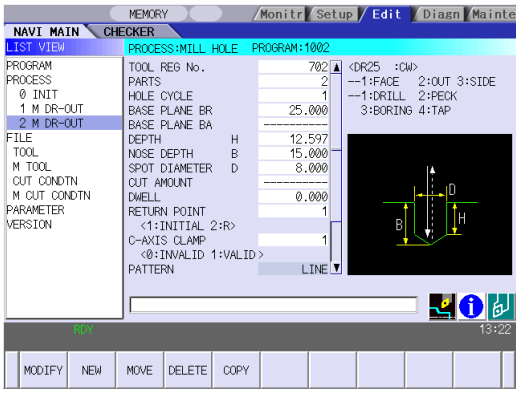
No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected for milling. Select the process mode by moving the sub cursor or inputting numerical values.	1: MILL HOLE 2: KEYWAY 3: CONTOUR

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. **(Note)** This is valid when the milling interporation specifications are provided.
3	MILLING	Displays the process mode for milling. **(Note)** This is valid when the milling interporation specifications are provided.

Operation example (Adding a new process)

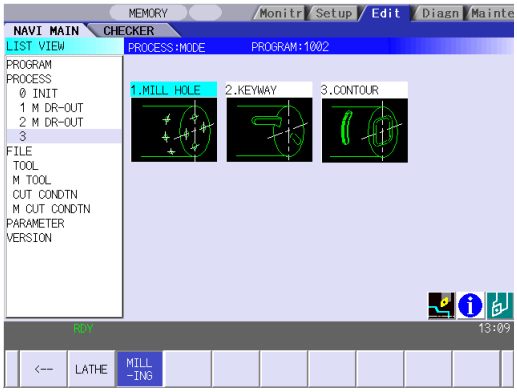
(1) Validate the LIST VIEW area, and select the position where the process is added with the cursor key.



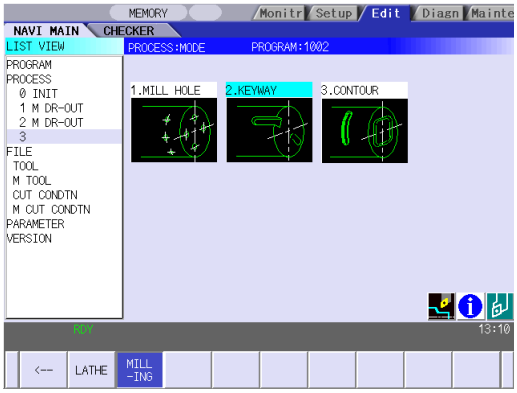
(2) Press the [NEW] menu key.

Select turning mode or milling mode by pressing [LATHE] or [MILLING] respectively.

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.

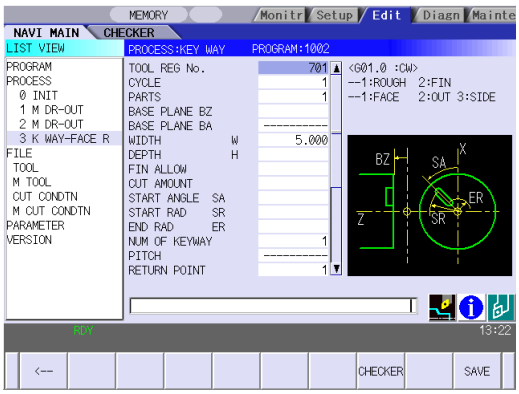


(3) Select the process mode with the cursor or the numerical value input.



(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.

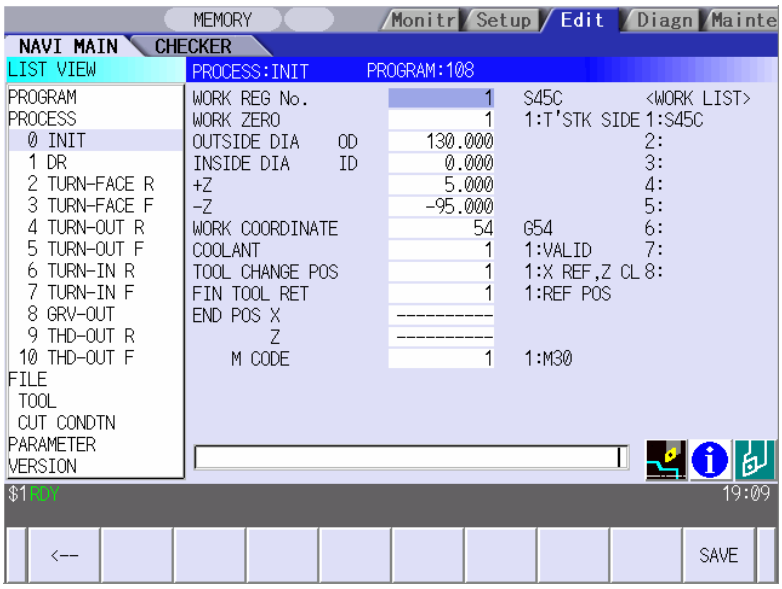


(Note) If the [←] menu key is pressed during adding the process, the screen will return to the state before pressing the [NEW] menu key (state of the 1).

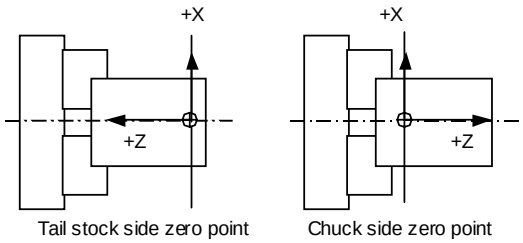
4.3.4 Initial Condition Setting Screen

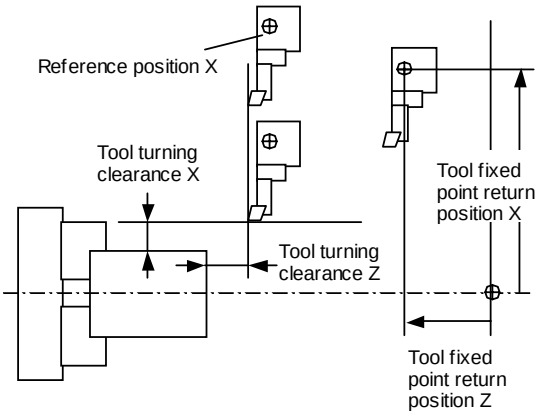
The initial conditions for the program are set on this screen. When the [INIT] is selected in the LIST VIEW area, this screen is displayed.

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

No.	Display item	Details	Setting range
3	OUTSIDE DIA	Input the workpiece outer diameter.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
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	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  	C1 C1 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		

No.	Display item	Details	Setting range
13	END M CODE	At the program end, select the M command to be output. 1 : M30 2 : M02 3 : M99	1 to 3

Menus

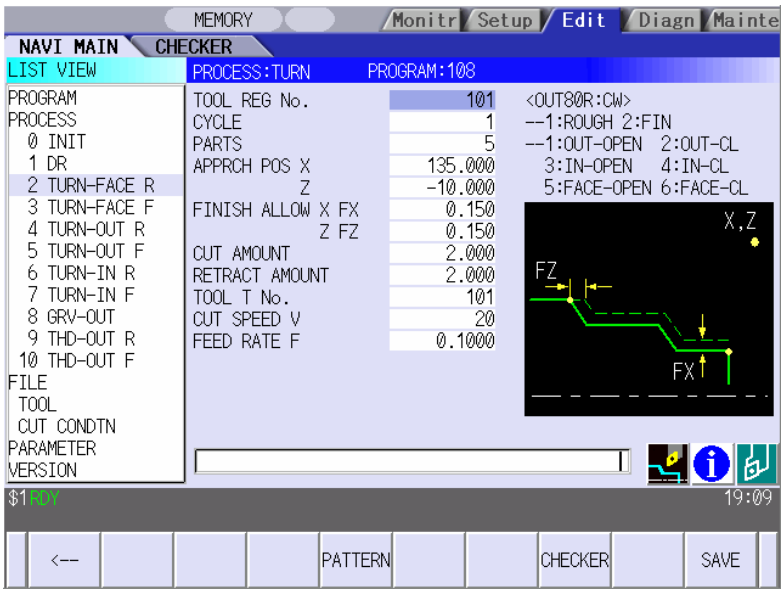
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	SAVE	Saves the changes in the initial conditions.

4.3.5 Turning Screen

(1) Turning screen

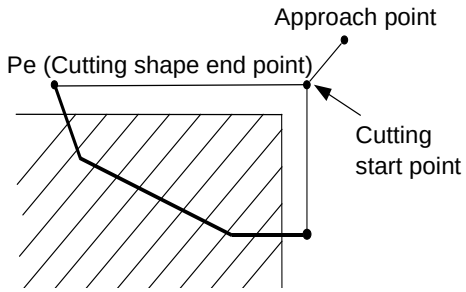
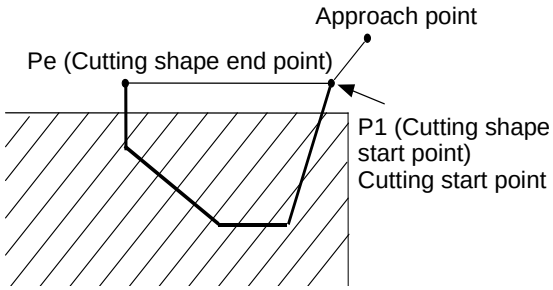
The parameters for the turning process are input on this screen.

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.	101 to 150 601 to 650
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

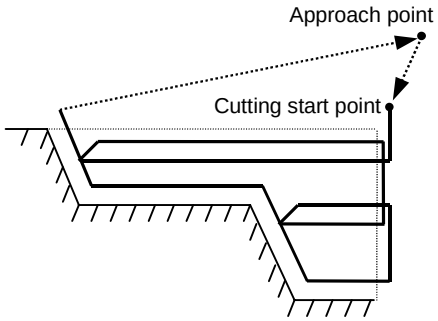
No.	Display item	Details	Setting range
3	PARTS	Input the machining area. <1: OD OPEN> Machines the outer diameter area from the front face of workpiece. <2: OD CLOSE> Machines the outer diameter area from the halfway of workpiece. <3: ID OPEN> Machines the inner diameter area from the front face of workpiece. <4: ID CLOSE> Machines inner area from the halfway of workpiece. <5: FACE OPEN> Machines the front face of workpiece. <6: FACE CLOSE> Machines the front face from the halfway of workpiece. [OPEN type]  [CLOSE type]  When the cutting shape is not incremented or decremented monotonously, CLOSE type is selected.	1 to 6
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.	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
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

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

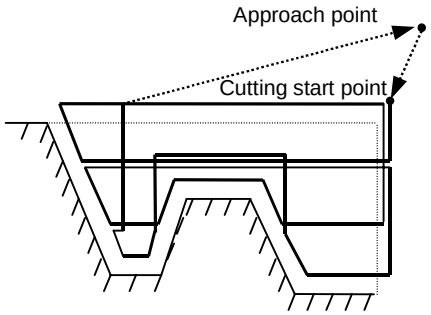
[OPEN type]

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.

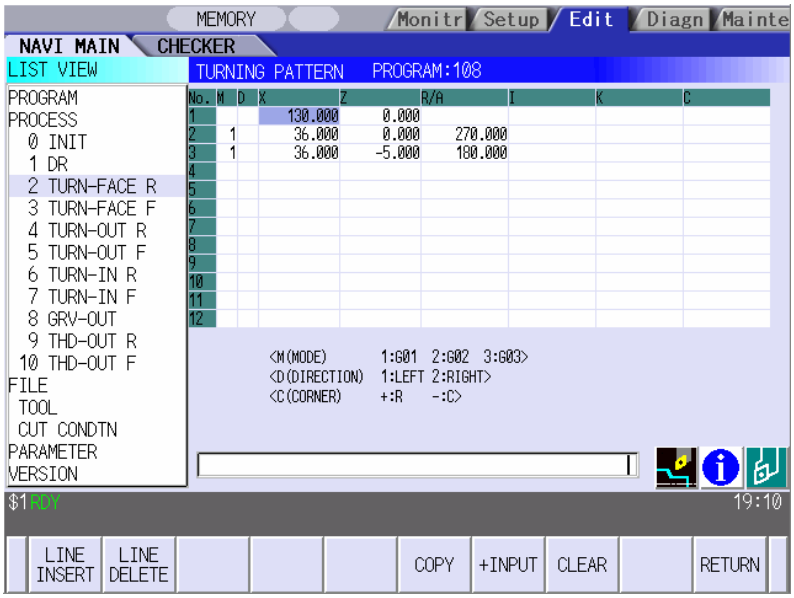
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	PATTERN	Machining pattern selection screen is displayed.
3	CHECKER	Displays the checker screen. Select this to check the set data.
4	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

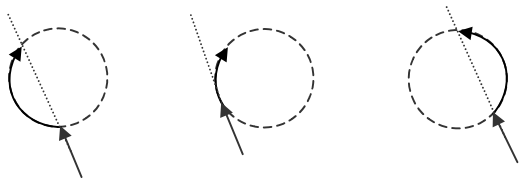
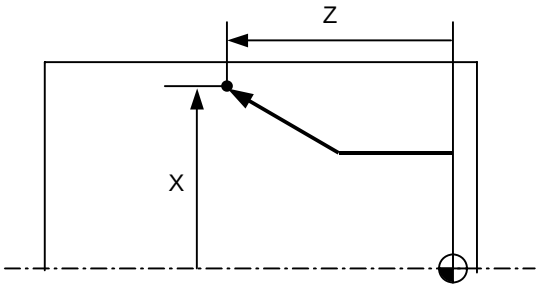
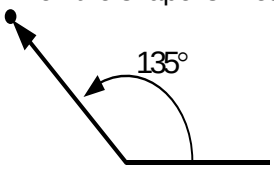
The cutting shapes for the turning process are input on this screen.

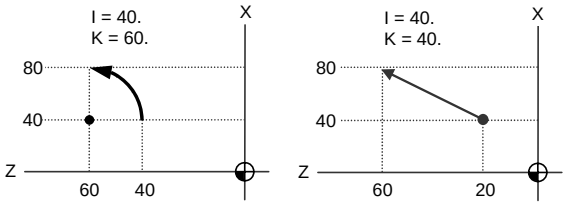
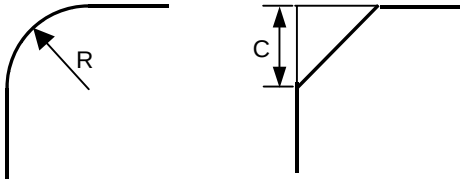
Screen layout



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

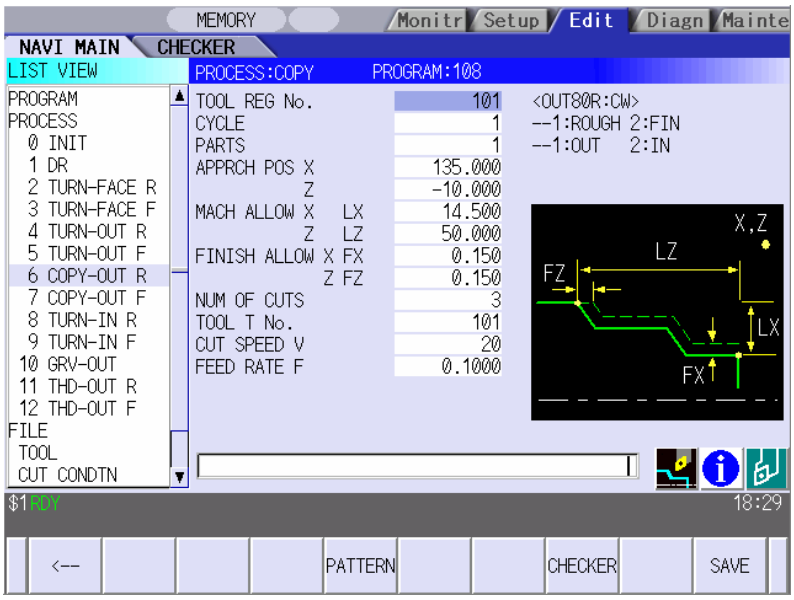
No.	Menu	Details
1	LINE INSERT	Inserts the shape data in front of the cursor position. **(Note)** This menu is not available when the cursor is at No.1 (machining start point).
2	LINE DELETE	Deletes the shape data at the cursor position. **(Note)** This menu is not available when the cursor is at No.1 (machining start point).
3	COPY	Copies the previous line data at the cursor position.
4	+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.
5	CLEAR	Clears the data at the cursor position.
6	RETURN	Returns to the turning screen.

4.3.6 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.	101 to 150 601 to 650
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
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
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

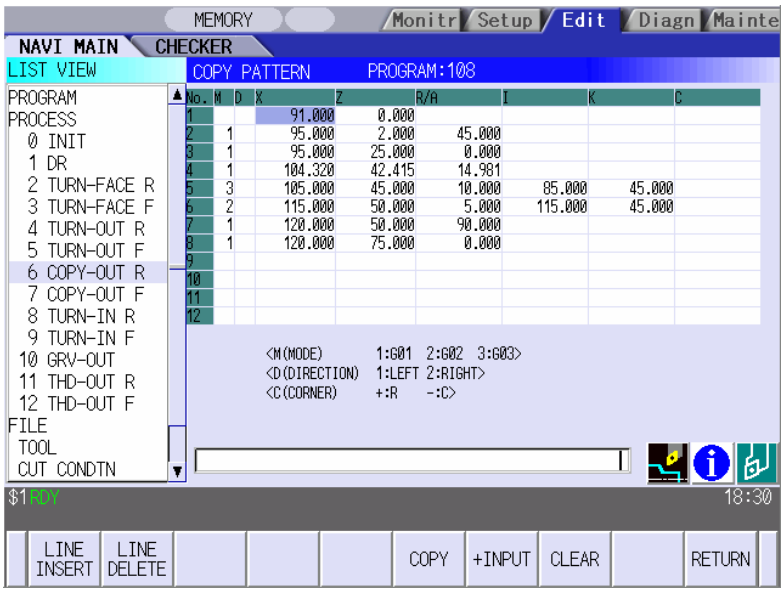
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	PATTERN	Displays the machining pattern selection screen.
3	CHECKER	Displays the checker screen. Select this to check the set data.
4	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".

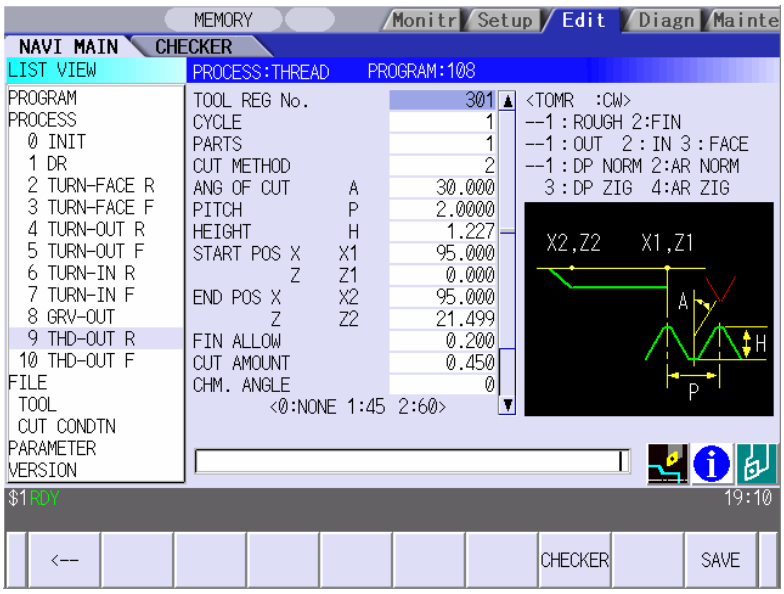
Menus

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).
3	COPY	Copies the previous line data at the cursor position.
4	+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.
5	CLEAR	Clears the data at the cursor position.
6	RETURN	Returns to the copy cutting screen.

4.3.7 Threading Screen

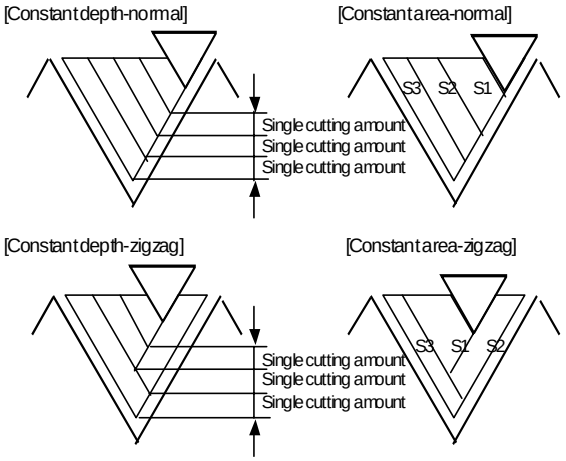
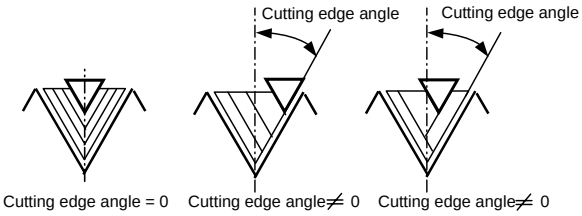
The parameters for the thread process are input on this screen.

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.	301 to 350
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the thread shape gradually. Leaves the finishing allowance for the thread shape. <2: Finishing machining> Machines the thread shape in one cycle.	1,2
3	PARTS	Input the machining area. <1: Outer diameter> Thread the outer diameter area of the workpiece. <2: Inner diameter> Thread the inner diameter area of the workpiece. <3: Face> Thread the front area of the workpiece.	1 to 3

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
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

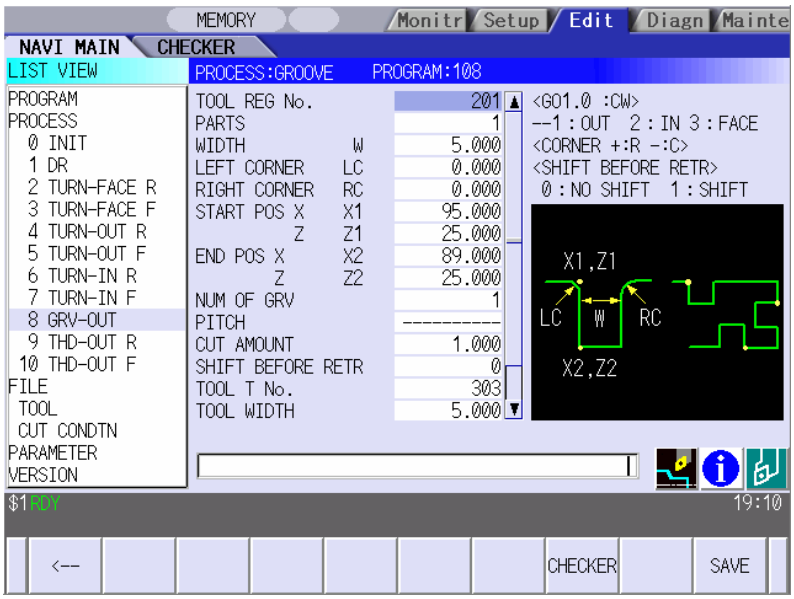
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	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.

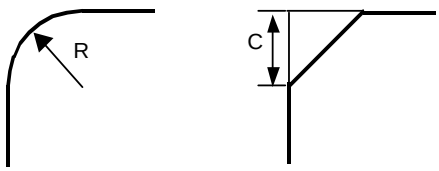
4.3.8 Grooving Screen

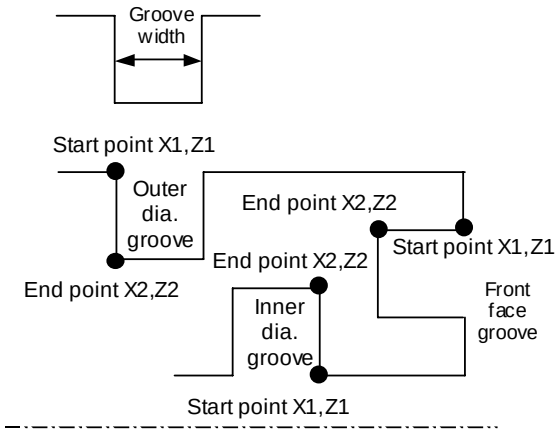
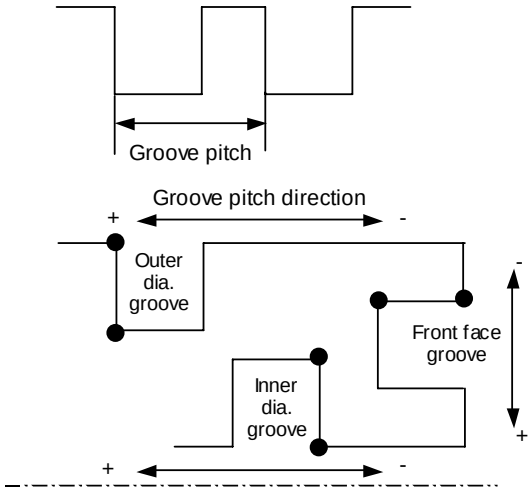
The parameters for the groove process are input on this screen.

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.	201 to 250
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.	1 to 3
3	WIDTH (W)	Input the groove width.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
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

No.	Display item	Details	Setting range
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
12	CUT AMOUNT	Input the cut amount.	0.001 to 99999.999mm 0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
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

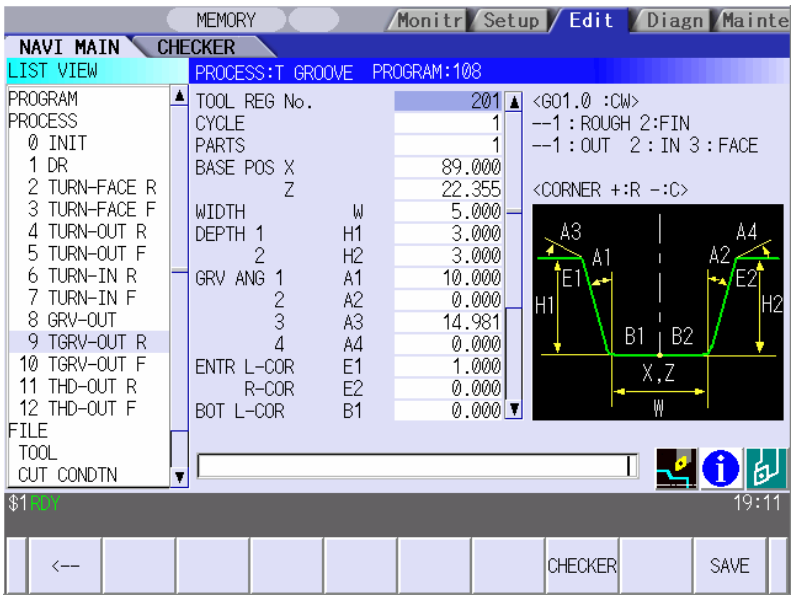
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	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 Trapezoidal Grooving Screen

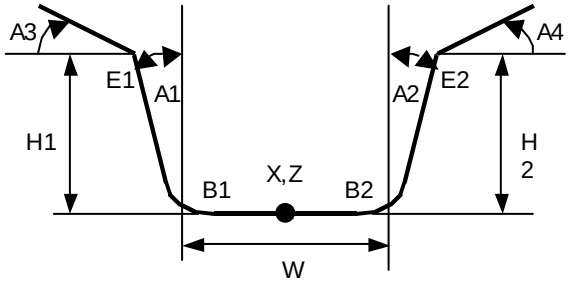
The parameters for the trapezoidal groove process are input on this screen.

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.	201 to 250
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
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.	1 to 3
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

No.	Display item	Details	Setting range
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 -9999.9999 to 9999.9999inch
14	ENTR R-COR (E2)	Input the right corner amount of trapezoidal groove entrance. Positive value: Corner R Negative value: Corner C	
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	
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

No.	Display item	Details	Setting range
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

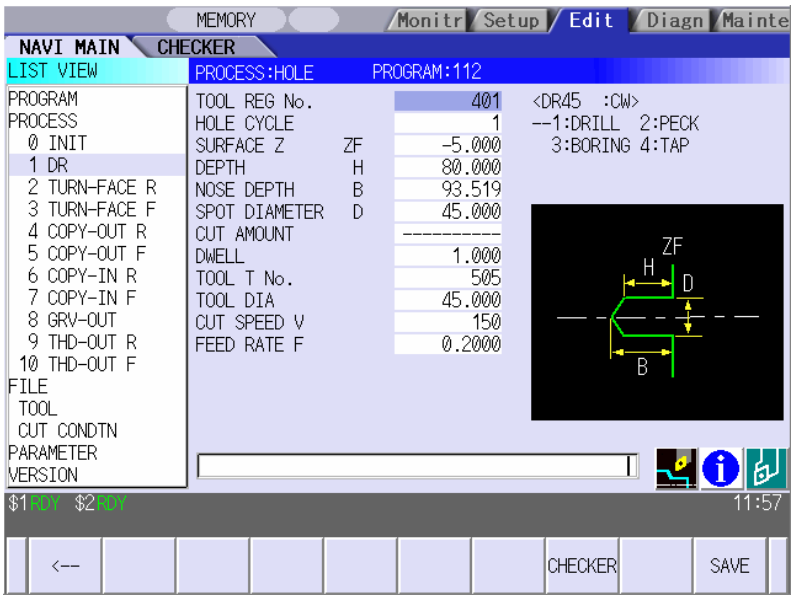
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	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 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.	401 to 450 501 to 550
2	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: Deep hole> (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 cycle> (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: Tapping> (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

No.	Display item	Details	Setting range
3	SURFACE Z (ZF)	Input the top surface position of the hole.	-99999.999 to 99999.999mm
4	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
5	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
6	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
7	CUT AMOUNT	When selecting the hole cycle type C=2(deep hole), input the cut amount per cut.	0.001 to 99999.999mm
8	DWELL	Input the dwell time at the bottom of the hole.	0.0 to 99.999sec
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.	1 to 999999
10	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
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 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

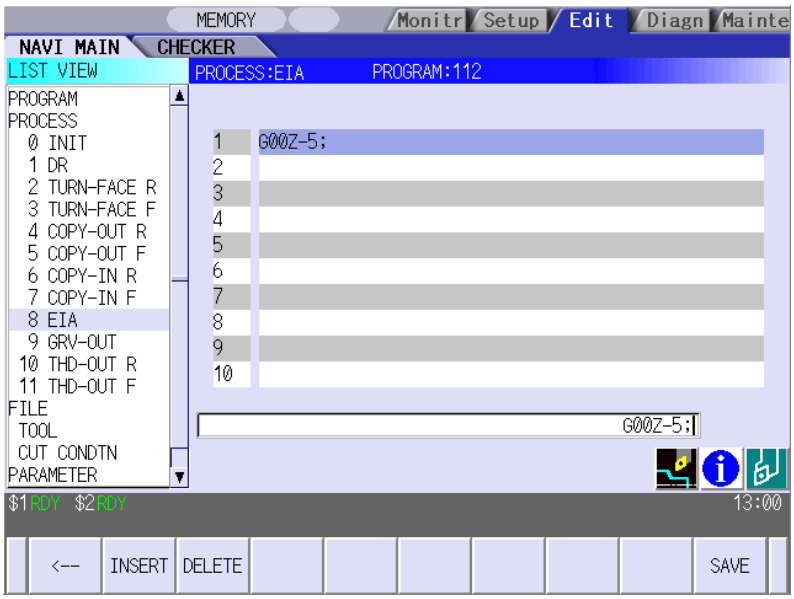
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	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 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

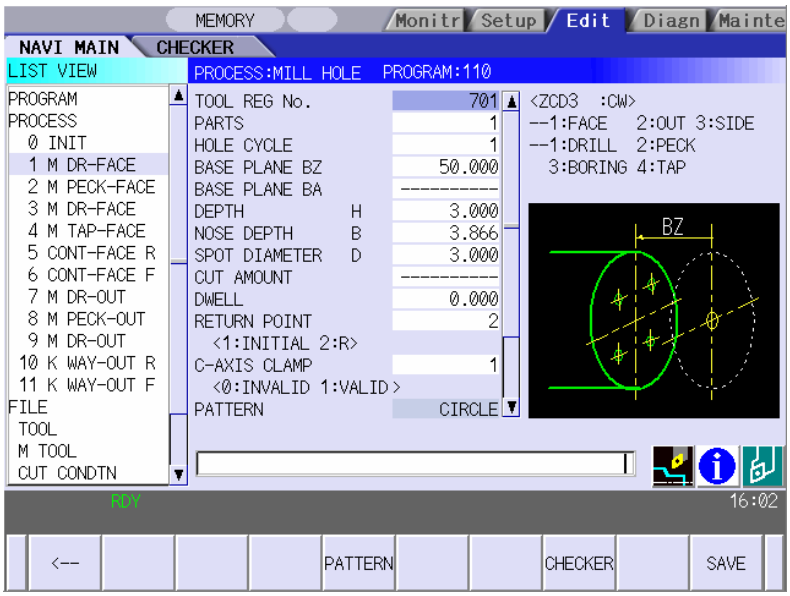
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	INSERT	Inserts a blank block before the block where the cursor exists.
3	DELETE	Deletes the data of the block where the cursor exists.
4	SAVE	Saves the changes in the process.

4.3.12 Milling Hole Drilling Screen

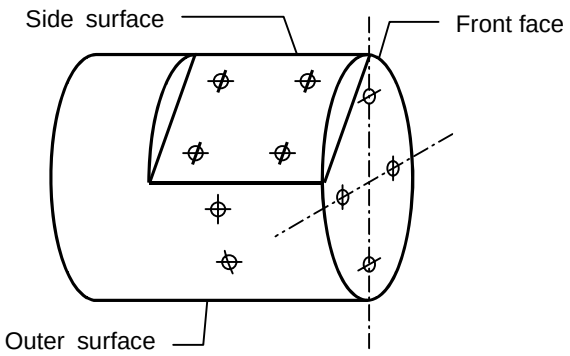
(1) Milling hole drilling screen

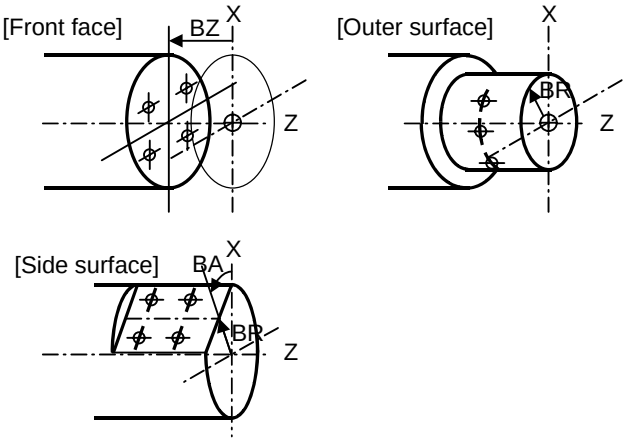
The parameters for the milling hole drilling are input on this screen.

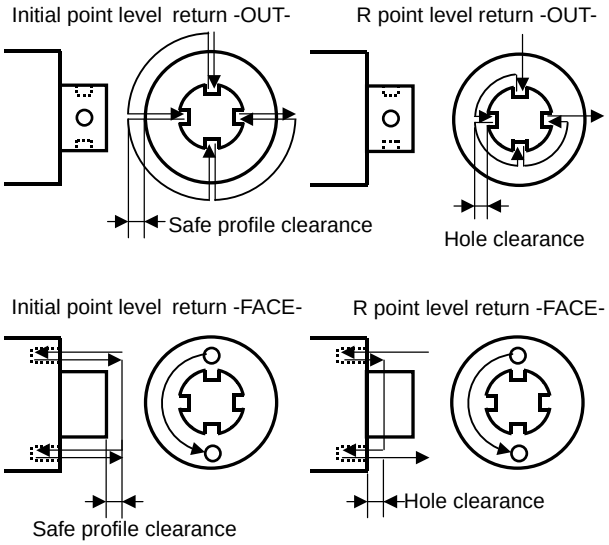
Screen layout



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.	701 to 799
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. Y-axis specifications are required for the side cutting.  	1 to 3

No.	Display item	Details	Setting range
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: BORE>(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
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

No.	Display item	Details	Setting range
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
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
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
12	PATTERN	The machining pattern is displayed. RANDOM LINE ARC CIRCLE SQUARE GRID Change the machining pattern on the machining pattern screen.	-
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

No.	Display item	Details	Setting range
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
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

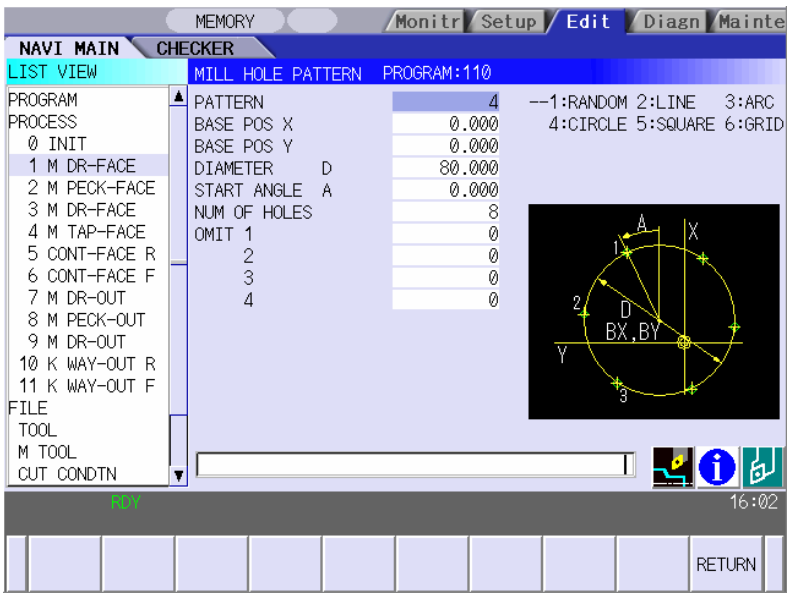
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	PATTERN	The machining pattern selection screen is displayed.
3	CHECKER	Displays the checker screen. Select this to check the set data.
4	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	○	○	○	○	○	○

○: Selectable, ×: Not selectable

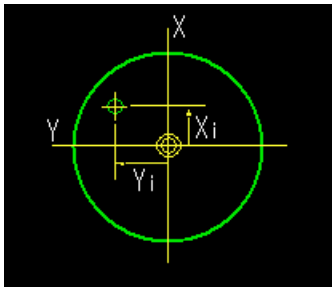
Screen display items

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

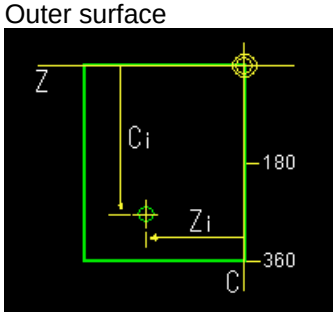
(Note) Display items No.2 and later differ according to each pattern.

・Parameters for RANDOM

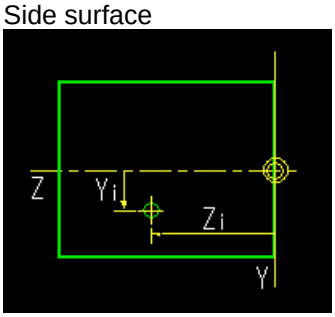
Front face



Outer surface



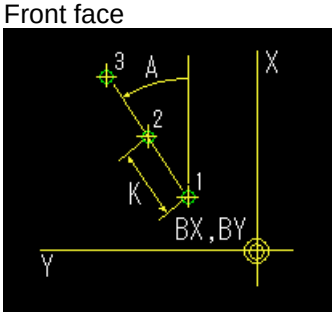
Side surface



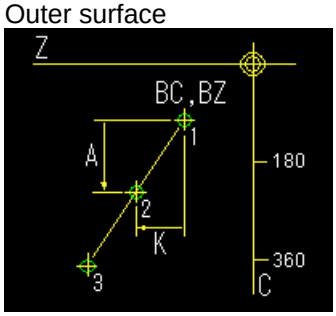
No.	Display item	Details	Setting range
2	HOLE No.	Input the hole No.	1 to 35
3	FACE: POS X POS Y OUT: POS C POS Z SIDE: POS Y POS Z	Input the hole position. [Front face] [Outer circumference] [Side face]	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch C: -359.999 to 360.000

・Parameters for LINE

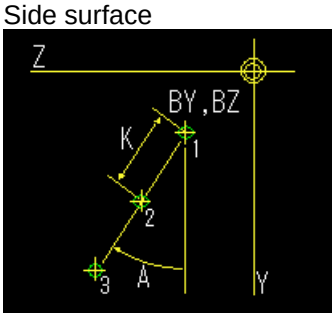
Front face



Outer surface



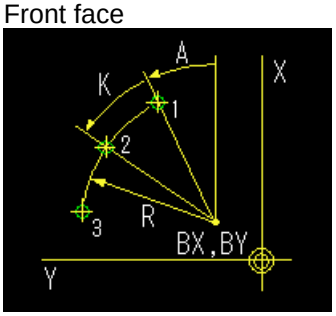
Side surface



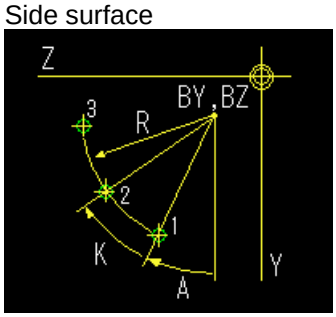
No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y OUT: BASE POS C BASE POS Z SIDE: BASE POS Y BASE POS Z	Set the first hole position for the machining area.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch C: -359.999° to 360.000°
3	ANGLE (A) PITCH (A)	Front face: Input 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.	1 to number of holes

・Parameters for ARC

Front face



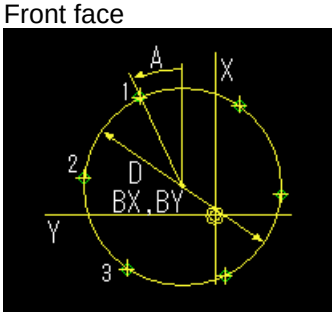
Side surface



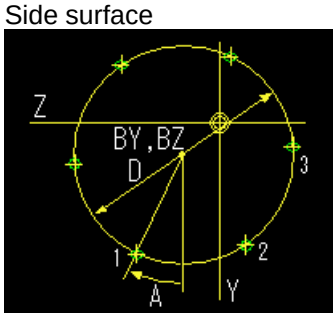
No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the arc center position.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
3	RADIUS R	Input the arc radius.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
4	START ANGLE A	Front face: Input 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.	1 to number of holes

・Parameters for CIRCLE

Front face



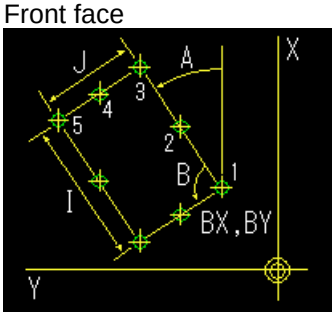
Side surface



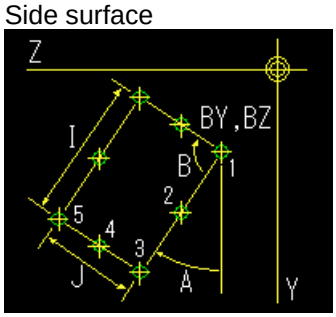
No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the circular center position.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch
3	DIAMETER D	Input the circular diameter.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
4	START ANGLE A	Front 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.	1 to number of holes

・Parameters for SQUARE

Front face



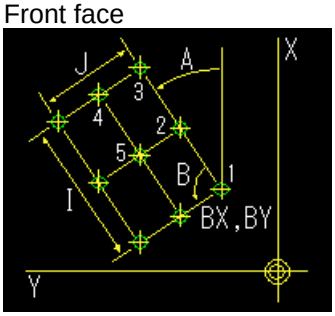
Side surface



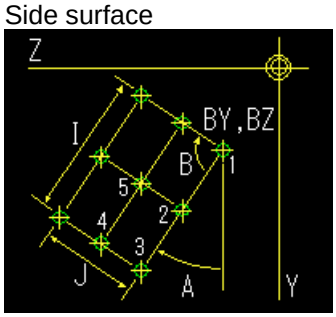
No.	Display item	Details	Setting range
2	FACE: BASE POS X BASE POS Y SIDE: BASE POS Y BASE POS Z	Input the position of the machining start point.	X,Y,Z: -99999.999mm to 99999.999mm -9999.9999 to 9999.9999inch
3	X WIDTH I	Input the width of the machining point in the X axis direction.	-99999.999mm to 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°
9	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	1 to number of holes

・Parameters for GRID

Front face



Side 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°
9	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	1 to number of holes

Menus

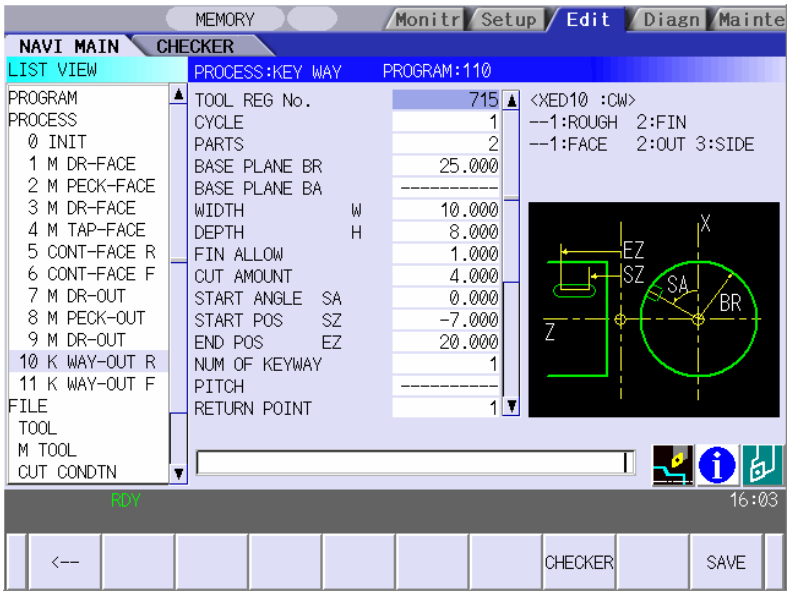
No.	Menu	Details
1	LINE INSERT	Inserts the hole position in front of the cursor position. This is available only for the RANDOM pattern.
2	LINE DELETE	Deletes the hole position at the cursor position. This is available only for the RANDOM pattern.
3	COPY	Copies the previous line data above cursor to the setting area. This is available only for the RANDOM pattern.
4	+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.
5	RETURN	Returns to the hole drilling screen.

4.3.13 Keyway Cutting Screen

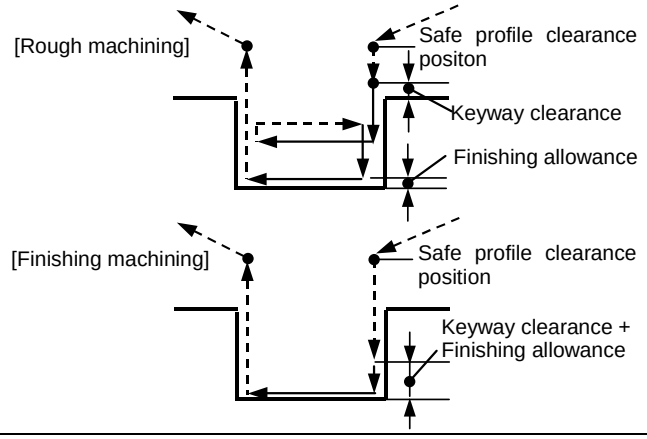
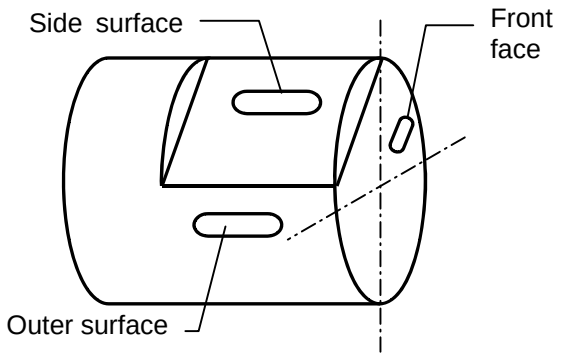
(1) Keyway Cutting Screen

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

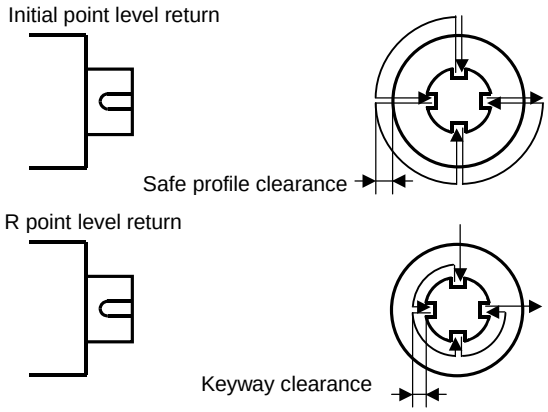
Screen layout

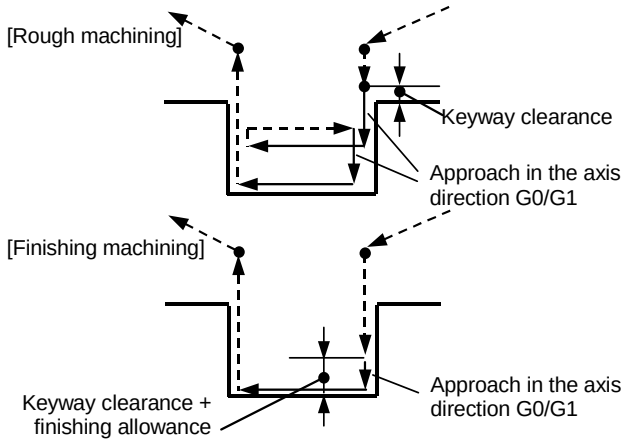


Screen display items

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.	701 to 799
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the keyway shape gradually. Leaves the finishing allowance in respect to the keyway shape. <2: Finishing machining> Machines the keyway shape in one cycle.  	1,2
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. Y-axis specifications are required for the side cutting.  	1 to 3

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] 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		
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

No.	Display item	Details	Setting range
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
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
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

No.	Display item	Details	Setting range
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
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
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
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

Menus

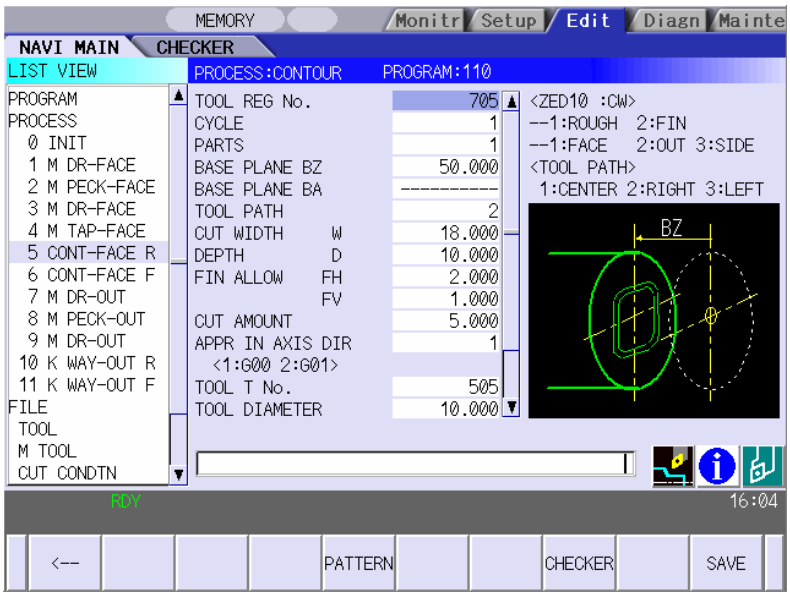
No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to that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

4.3.14 Contour Cutting Screen

(1) Contour Cutting Screen

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

Screen layout

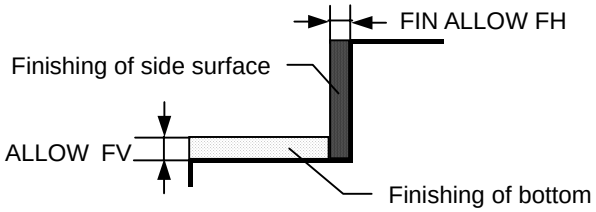
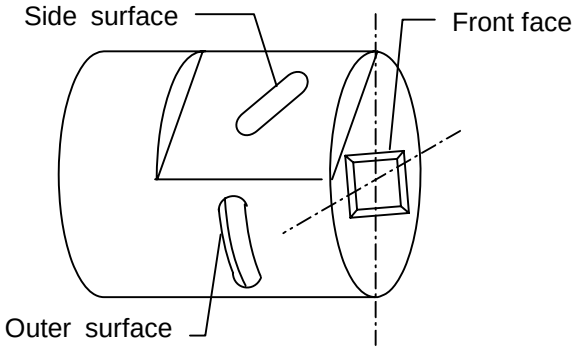


Screen display items

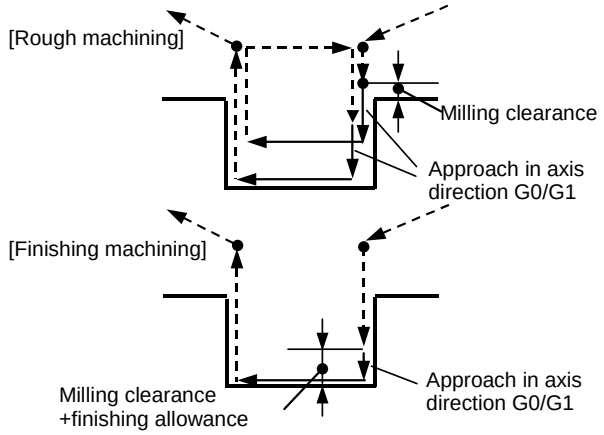
No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	701 to 799

(Continued to the next page)

(Continued from the last page)

No.	Display item	Details	Setting range
2	CYCLE	Input the machining method. <1: 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: 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
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. Y-axis specifications are required for the side cutting.  	1 to 3

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] BZ X Z [Outer surface] X BR Z [Side surface] BA X BR Z 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
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
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

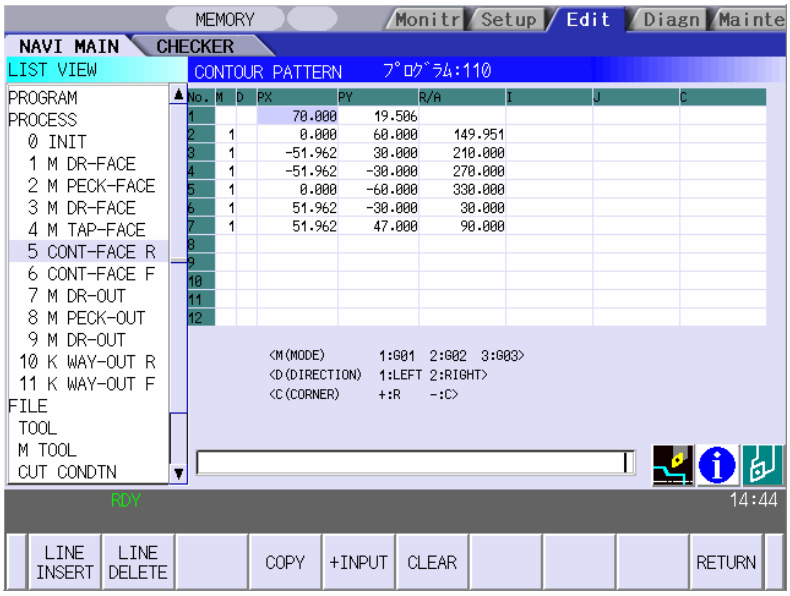
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	PATTERN	Displays the machining pattern selection screen.
3	CHECKER	Displays the checker screen. Select this to check the set data.
4	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.

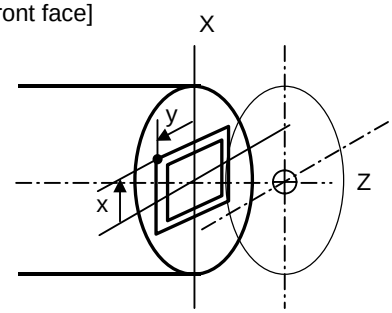
Screen layout



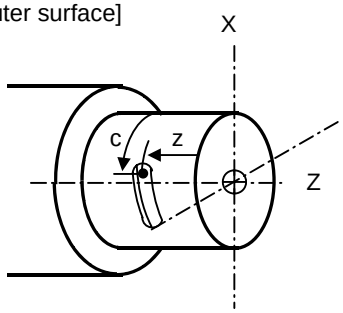
Input coordinate system of contour machining shape

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	

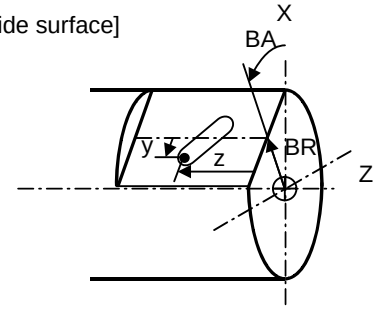
[Front face]



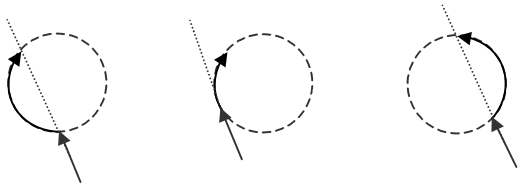
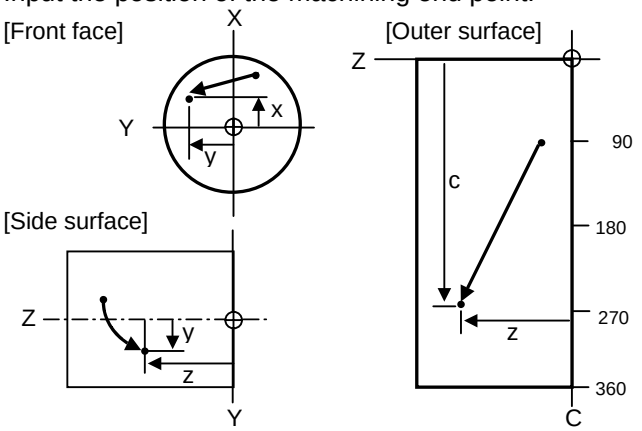
[Outer surface]

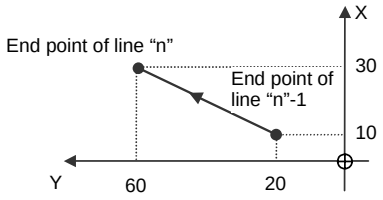
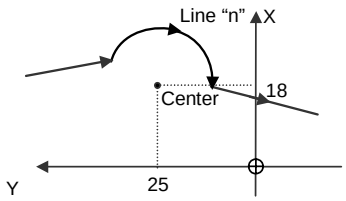
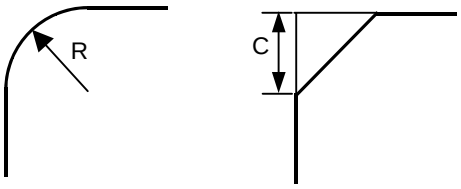


[Side surface]



Screen display items

No.	Display item		Details	Setting range
Shape				1 to 35
1	1	M	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.	1 to 3
	2	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)** This data, although omittable, must be input when the end points X,Y of the previous line are uncertain.  Left turn Tangent Right turn	1,2
	3	FACE: PX,PY OUT: PC,PZ PY,PZ SIDE: PY,PZ	Input the position of the machining end point.  **(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.	X,Y,Z: -99999.999 to 99999.999mm -9999.9999 to 9999.9999inch C: -99999.999° to 99999.000°

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).
3	COPY	Copies the previous line data above cursor to the setting area.
4	+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).
5	CLEAR	Clears the data at the cursor position.
6	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.
7	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.
8	RETURN	Returns to the contour cutting screen.

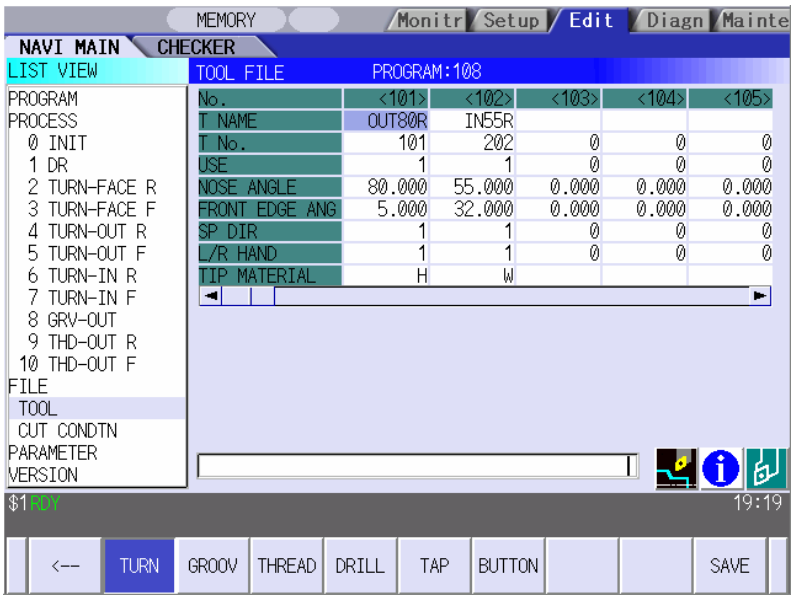
4.4 Screens Related to File Editing

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

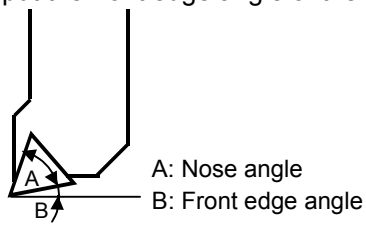
Screen layout



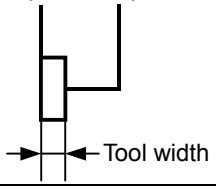
(Note) Menu for the currently selected tool is highlighted.

Screen display items

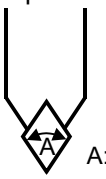
• TURNING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	101 to 150
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.  	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

• GROOVING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	201 to 250
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

• THREADING TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	301 to 350
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

• DRILLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	401 to 450
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

• TAPS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	501 to 550
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

• BUTTON TOOLS

No.	Display item	Details	Setting range
1	No.	Tool registration No.	601 to 650
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

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.
8	SAVE	Saves the changes in the tool file.

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	No.	Tool registration No. (701～799)	701 to 799
2	T NAME	Input the tool name.	Max. 6 alphanumeric 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 diameter.	0.001 to 999.999mm 0.0001 to 99.9999 inch
5	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000°
6	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
7	SP DIR	Input the spindle rotation direction.	1:CW 2:CCW
8	TIP MATERIAL	Input the tip material.	Max. 4 alphanumeric characters

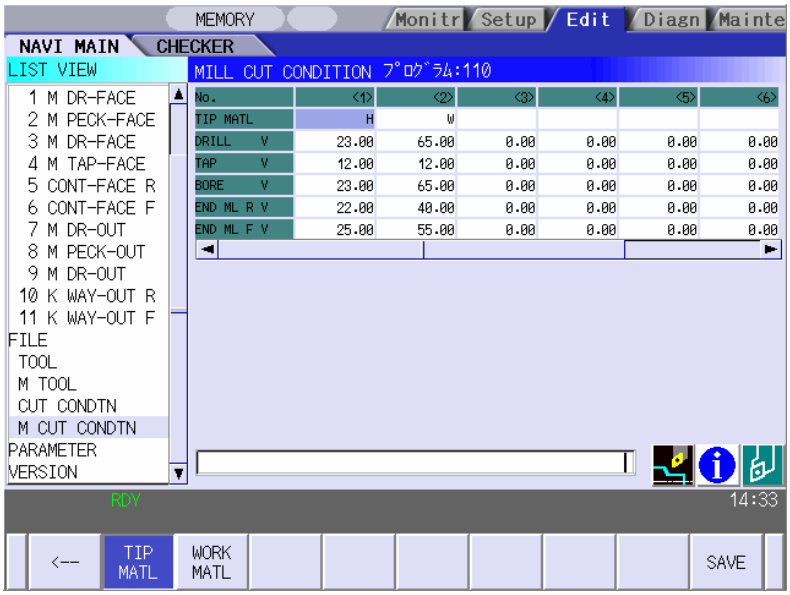
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	SAVE	Saves the changes in the tool file.

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.
4	SAVE	Saves the changes in the cutting condition file.

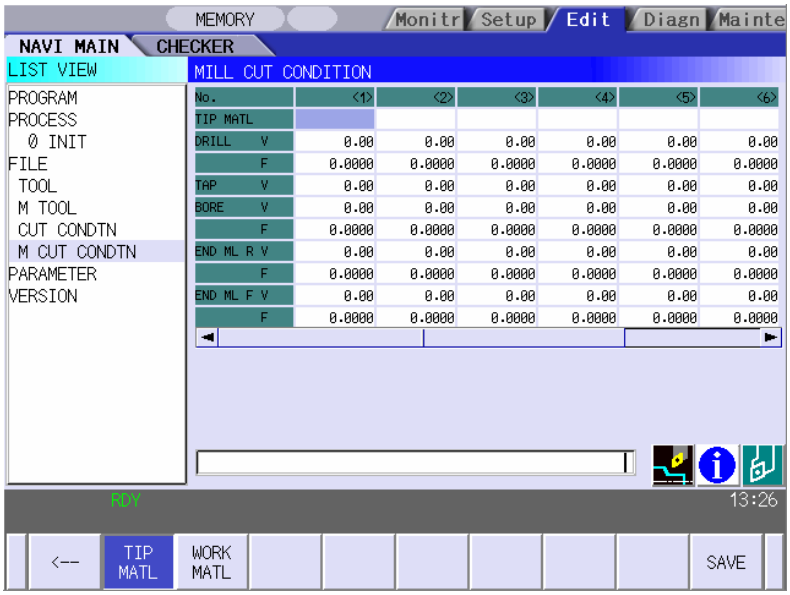
⚠ CAUTION

⚠ 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.
4	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.

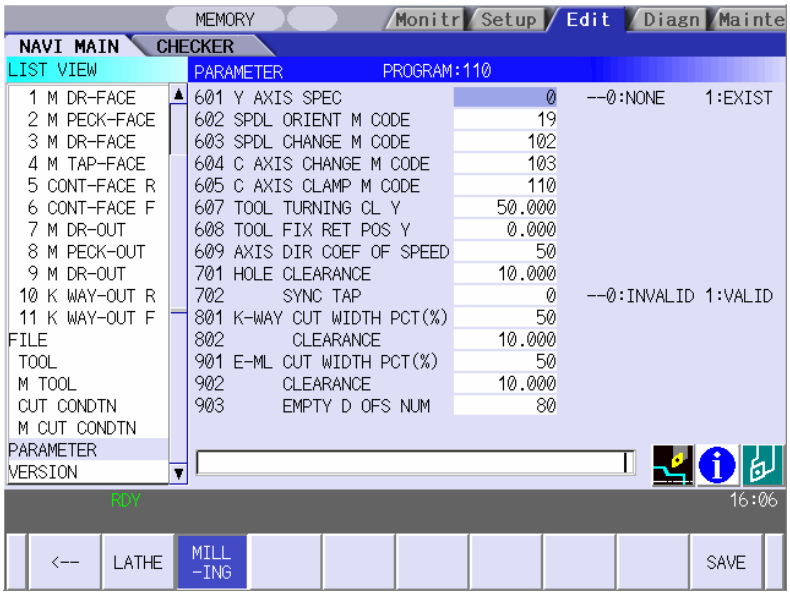
4.5 Screen Related to the Parameters

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.

4.5.1.1 Parameters for Turning

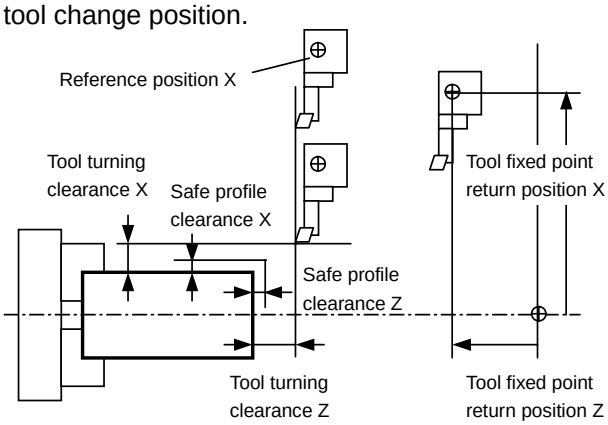
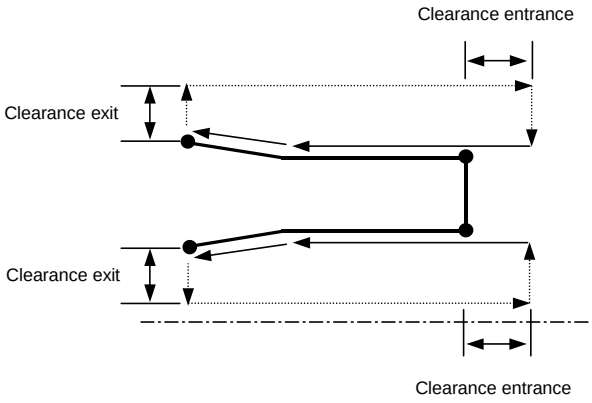
Screen layout

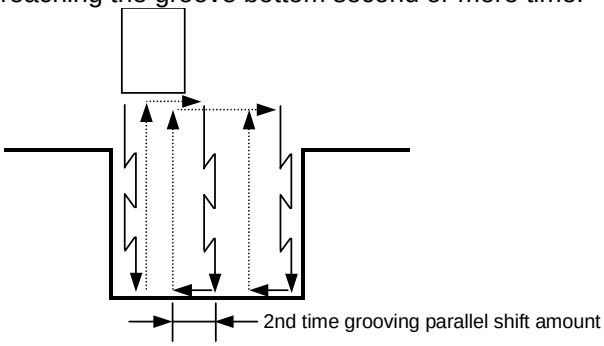
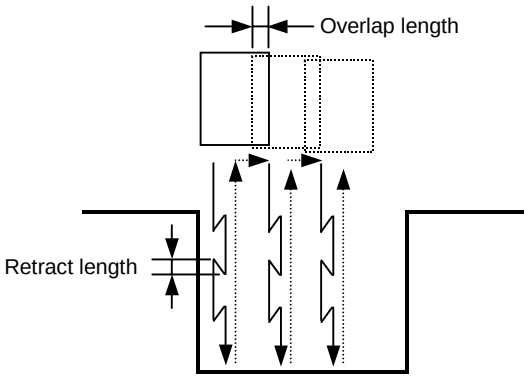
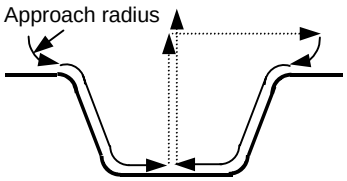


Screen display items

• Parameters for turning

No.	Display item	Details	Setting range
1 (101)	M1 OUTPUT	Specify whether to output the M1 code before tool indexing command. 0: Not output 1: Output	0,1
2 (102)	SPDL CLAMP SPEED	Input the maximum spindle clamp speed of a machining program.	1 to 99999 rev/min
3 (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
4 (104)	TOOL TURNING CL Z		0.0001 to 9999.9999inch

No.	Display item	Details	Setting range
5 (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
6 (106)	TOOL FIX RET POS Z		-9999.9999 to 9999.9999inch
7 (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.	0.001 to 99999.999mm
8 (108)	SAFE PROFILE CL FACE	Input the clearance for the front area in radius value when the approaching/escaping path is used between processes.	0.0001 to 9999.9999inch
9 (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
10 (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
11 (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

No.	Display item	Details	Setting range
12 (301)	GRV DWELL	Input the dwell value at the bottom of the groove.	0.000 to 99.999sec
13 (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
14 (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
15 (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
16 (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
17 (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

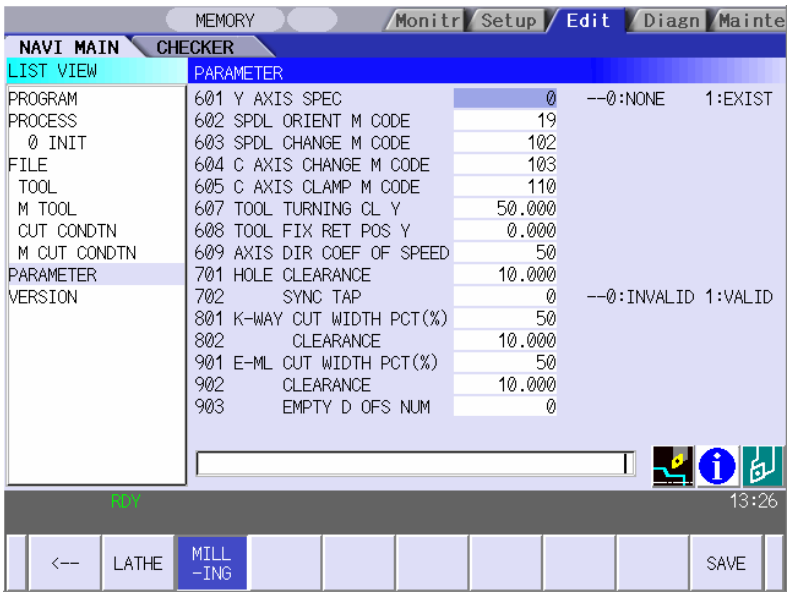
No.	Display item	Details	Setting range
18 (401)	HOLE CLEARANCE	The distance from the R-point, where the cutting feed begins, to the hole top position is set in the radius value.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
19 (402)	HOLE SYNC TAP	Set valid or invalid of synchronous tapping for tapping cycle machining. 0: INVALID (ASYNC) 1: VALID (SYNC)	0 to 1

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	SAVE	Saves the changes in the parameters.

4.5.1.2 Parameters for Milling

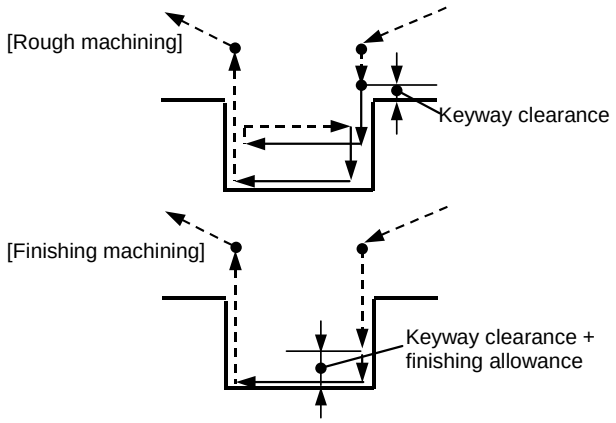
Screen layout

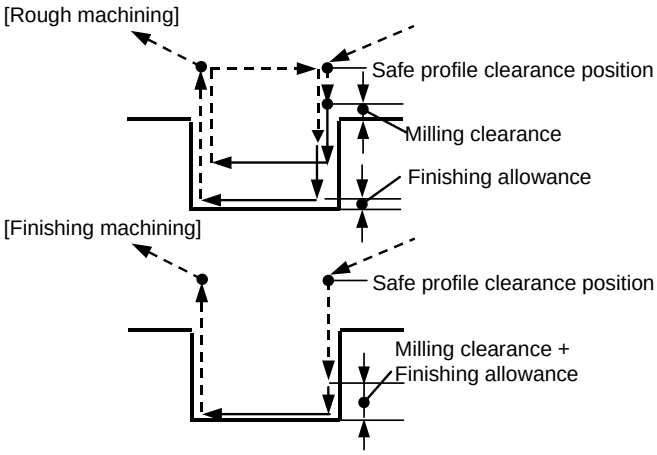


Screen display items

• 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.	1 to 4
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

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	SAVE	Saves the changes in the parameters.

4.5.2 PREFERENCE Screen

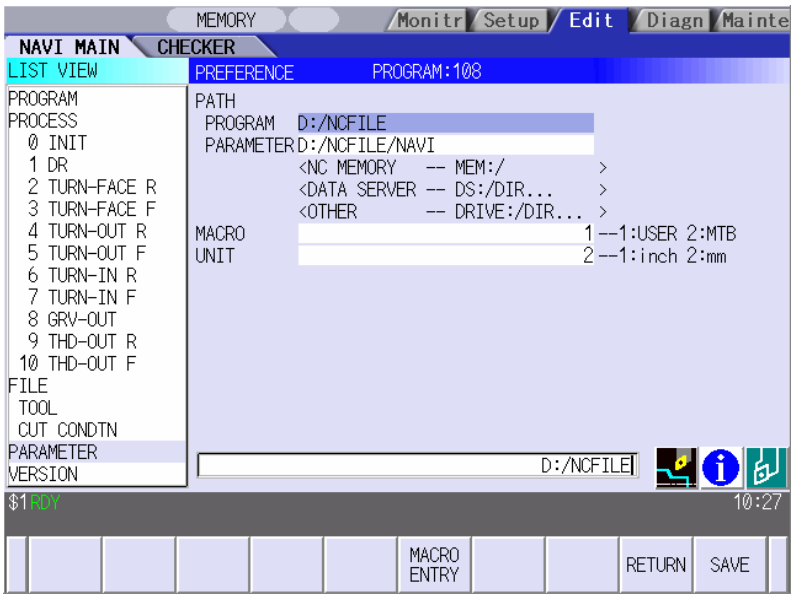
Prior to the NAVI LATHE operation, system setups are done on this screen.
The followings are the items to be setup.

- 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)

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

Screen layout

<When NAVI LATHE for 700 is used>



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) (Example) • D:/NCFILE • MEM:/
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

(Note) The drive names available in the path are different between 700 Series and 70 Series. Refer to the table below.

[Devices and drives available in the program path and parameter path]

Type	Device	Drive name	Input example	Remarks
700 Series	NC memory	MEM	MEM:/	Always input “MEM:/” for NC memory.
	HD	D	D:/NCFILE	This is an example to input the folder name “NCFILE”. Drive name of HD is fixed to “D”.
	Data server	DS	DS:/NAVI/PARA	This is an example to input the folder name “NAVI/PARA”.
	IC card	IC	IC:/	This is an example to input the folder name as root folder.
70 Series	NC memory	MEM	MEM:/	Always input “MEM:/” for NC memory.
	Memory card	MC	MC:/	This is an example to input the folder name as root folder.

Menus

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

(Note) The PREFERENCE data is saved to the PREFERENCE file (navi.ini) in the following folder.

Type	Folder to save the data
700 Series	C:\ncsys\navi_lathe\
70 Series	/PRG/MMACRO

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

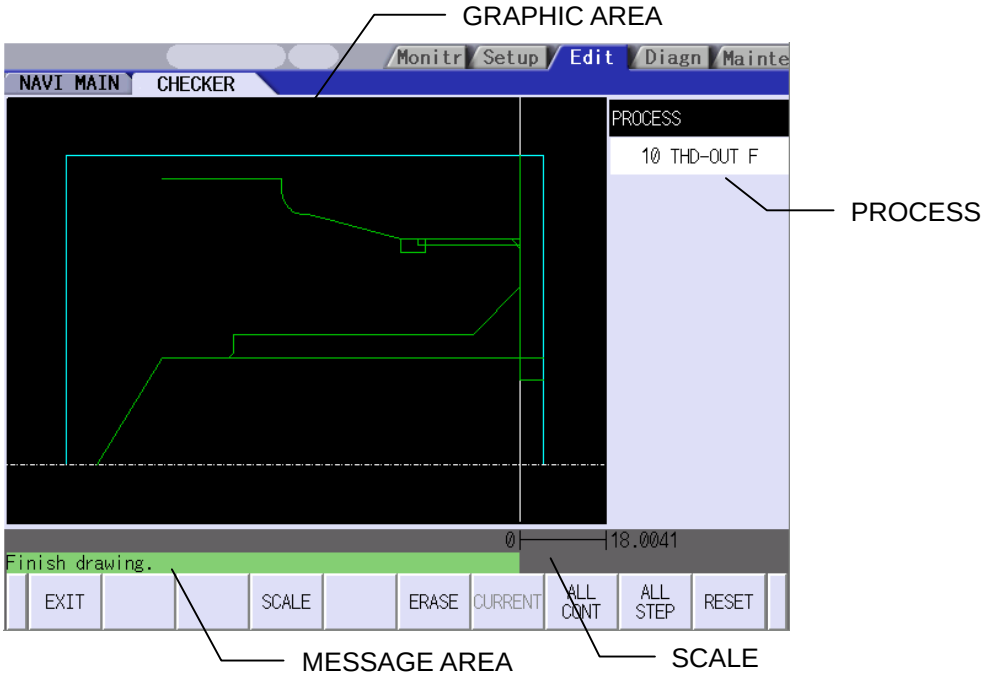
<When NAVI LATHE for 700 is used>

4.7 Program Checker Screen

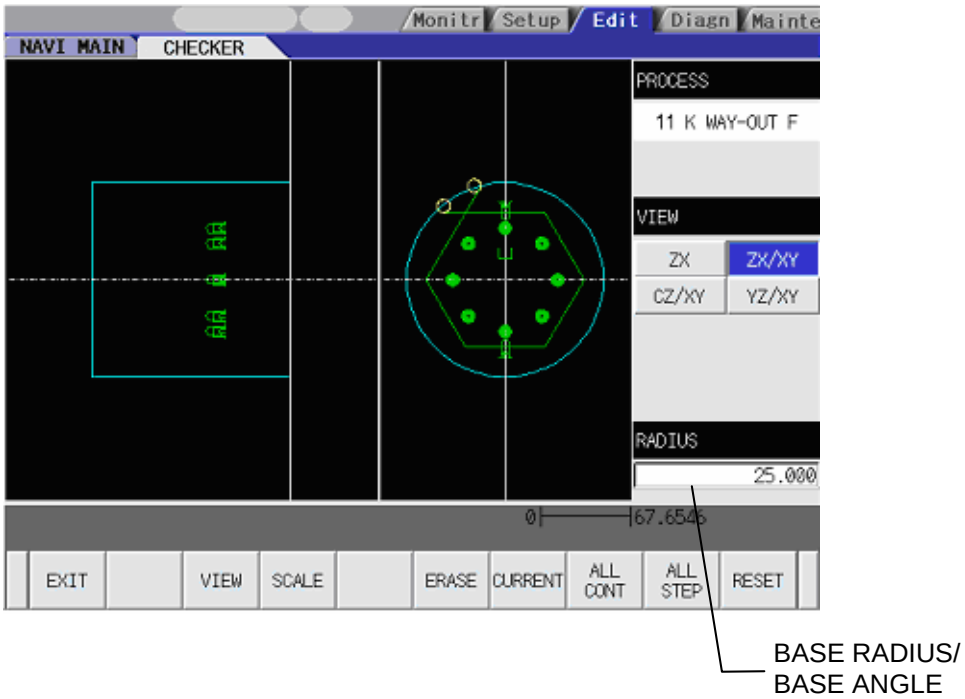
Machining shapes of a NC program are graphically displayed on this screen.
Program Checker screen appears when or is pressed while MAIN screen is displayed.
Program Checker screen also appear when the checker icon is clicked.

Screen layout

Turning



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



(Note) 2-plane display is available when the milling interporation specifications are provided.

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: • Machining shape --- Green • Workpiece --- Light blue • Cutting plane on Y-Z view --- White
2	PROCESS	Indicates the name of the process of which machining shape are currently displayed.
3	VIEW	Displays the currently selected view. This is available when the milling interporation specifications are provided. Not available unless the milling interporation 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. When the [R/A] menu in the VIEW change menu is pressed, the cursor appears to set base radius and base angle. Base radius is indicated when C-Z view is selected, while base angle is shown when Y-Z view is selected. This is not available unless C-Z view or Y-Z view is displayed. 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.

Main menus

No.	Menu	Details
1	EXIT	Terminates the Program Checker and then closes the screen.
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. Not available unless the milling interporation 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	Machining shapes of the currently selected process are displayed. The views and scales are set for CURRENT display.
6	ALL CONT	Machining shapes of the entire processes are displayed. The views and scales are set for ALL CONT display.
7	ALL STEP	Machining shapes of each process are displayed one at a time.
8	RESET	Reset 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 and the views selected in the ALL CONT or ALL STEP display are retained for the ALL CONT display. This views and scales are retained for the ALL CONT display until the application is closed.

View Change Menu

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

No.	Menu	Details
1	CANCEL	Returns to the main menu.
2	ZX	Converts the view into the Z-X view and returns to the main menu.
3	ZX/XY	Converts the view into the 2-plane display of Z-X and X-Y, and then returns to the main menu.
4	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.
5	YZ/XY	Converts the view into the 2-plane display of Y-Z and X-Y, and then returns to the main menu. In ALL CONT and ALL STEP display, Y-Z view only displays the shapes made upon the fixed base radius for the machining process.
6	R/A	Set the base radius and the base angle. This is available 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 is not available when Z-X or ZX/XY view is selected, nor when the CURRENT display is performed.

(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 may be deleted upon any change of plane.
(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
Keyway cutting	Front face	ZX/XY
	Outer diameter	CZ/XY
	Side surface	YZ/XY
Contour cutting	Front face	ZX/XY
	Outer diameter	CZ/XY
	Side surface	YZ/XY

(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. Workpiece is displayed in the center of the screen.
3	ENLARGE	Enlarges scale. The same function can be achieved by pressing – key.
4	REDUCE	Reduces scale. The same function can be achieved by pressing + key. The solid scale frame will be drawn in dotted lines when its size exceeding 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.

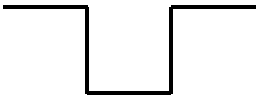
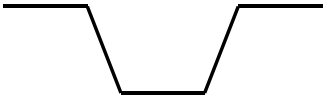
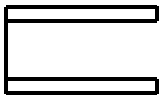
(Note 1) Display area is shown with a white frame.
(Note 2) The displayed machining shape will be deleted upon change of display scale or position.

Restrictions on the graphic display function

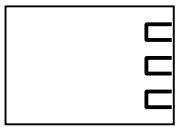
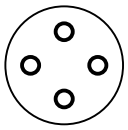
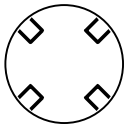
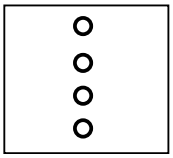
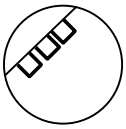
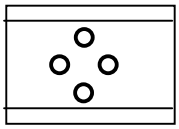
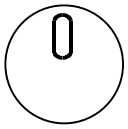
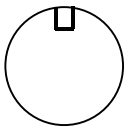
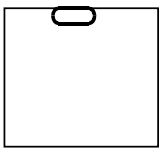
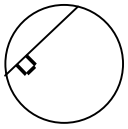
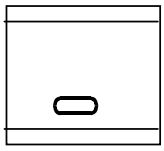
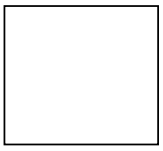
- Graphic display is not available for the EIA process.
- When there is an error in the specified shape data for the turning/copy cutting, the shape data is displayed up to the error point.

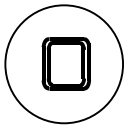
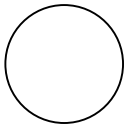
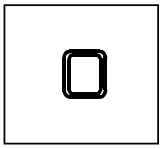
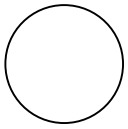
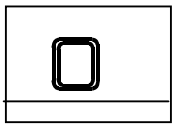
Examples of graphic drawings

[Turning]
Only ZX view is displayed for turning.

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

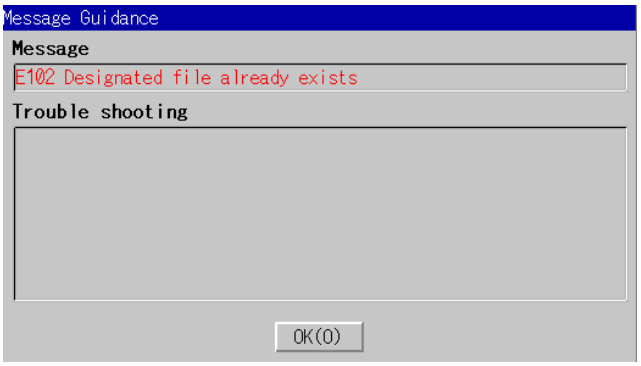
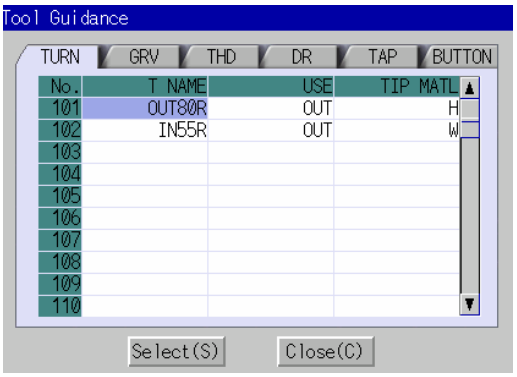
[Milling]
For milling process, machining shapes are displayed on the views that correspond to each machining area.

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

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 . Guidance window will be closed by clicking [OK].

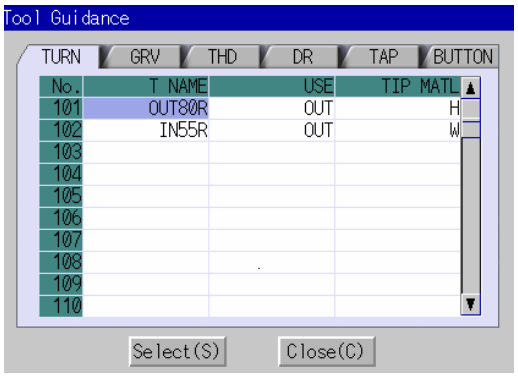
Guidance Type	Starting method		Details
	Key-board	Icon	
Message Guidance	?		Details or countermeasures related to the current error and message are displayed. 
Tool Guidance	LIST		A segment of tool data registered in the tool file is displayed. Note that no editing is possible. 

4.8.1 Tool Guidance Screen

Tool guidance is provided for the turning and the milling machining.
The tool guidance is displayed according to the machining process selected in the LIST VIEW area.
Primary data of turning tool is displayed when turning process is selected in the LIST VIEW area.

4.8.1.1 Tool Guidance for Turning

Screen layout



Screen display items

Turning

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (101 to 150)
2	T NAME	Displays the tool name.
3	USE	Displays the application of tool.
4	TIP MATL	Displays the tip material.

Grooving

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (201 to 250)
2	T NAME	Displays the tool name.
3	USE	Displays the application of tool.
4	TOOL WIDTH	Displays the tip width.
5	TIP MATL	Displays the tip material.

Threading

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (301 to 350)
2	T NAME	Displays the tool name.
3	USE	Displays the application of tool.
4	NOSE ANGLE	Displays the tool nose angle.
5	TIP MATL	Displays the tip material.

Drilling

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (401 to 450)
2	T NAME	Displays the tool name.
3	DIA	Displays the tool diameter.
4	NOSE ANGLE	Displays the tool nose angle.
5	TIP MATL	Displays the tip material.

Tapping

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (501 to 550)
2	T NAME	Displays the tool name.
3	DIA	Displays the tool diameter.
4	PITCH	Display the pitch.
5	TIP MATL	Displays the tip material.

Button

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (601 to 650)
2	T NAME	Displays the tool name.
3	USE	Displays the application of tool.
4	TIP DIA	Displays the tip diameter.
5	TIP MATL	Displays the tip material.

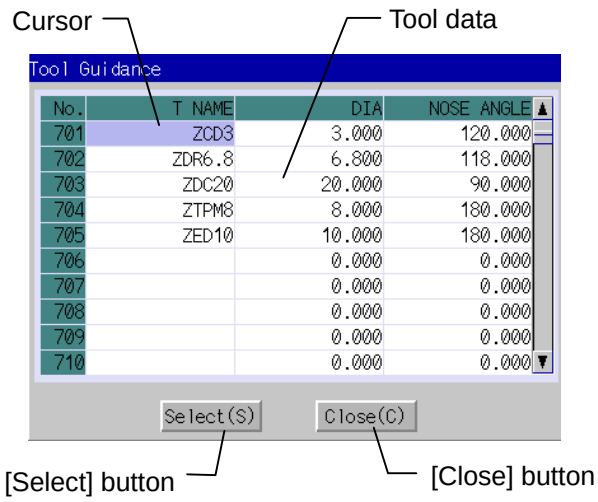
Buttons

No.	Button	Application
1	Select	The tool registration No. at the cursor position is set to "TOOL REG No." in each process screen. This button is valid only when the tool guidance screen is opened while the cursor is at the "tool registration No." in each process screen.
2	Close	Closes the tool guidance screen.

4.8.1.2 Tool Guidance for Milling

Primary data of the milling tool is displayed when milling process is selected in the LIST VIEW area.

Screen layout



Screen display items

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition.(701～799)
2	T NAME	Displays the tool name.
3	DIA	Displays the tool diameter.
4	NOSE ANGLE	Displays the tool nose angle.

Buttons

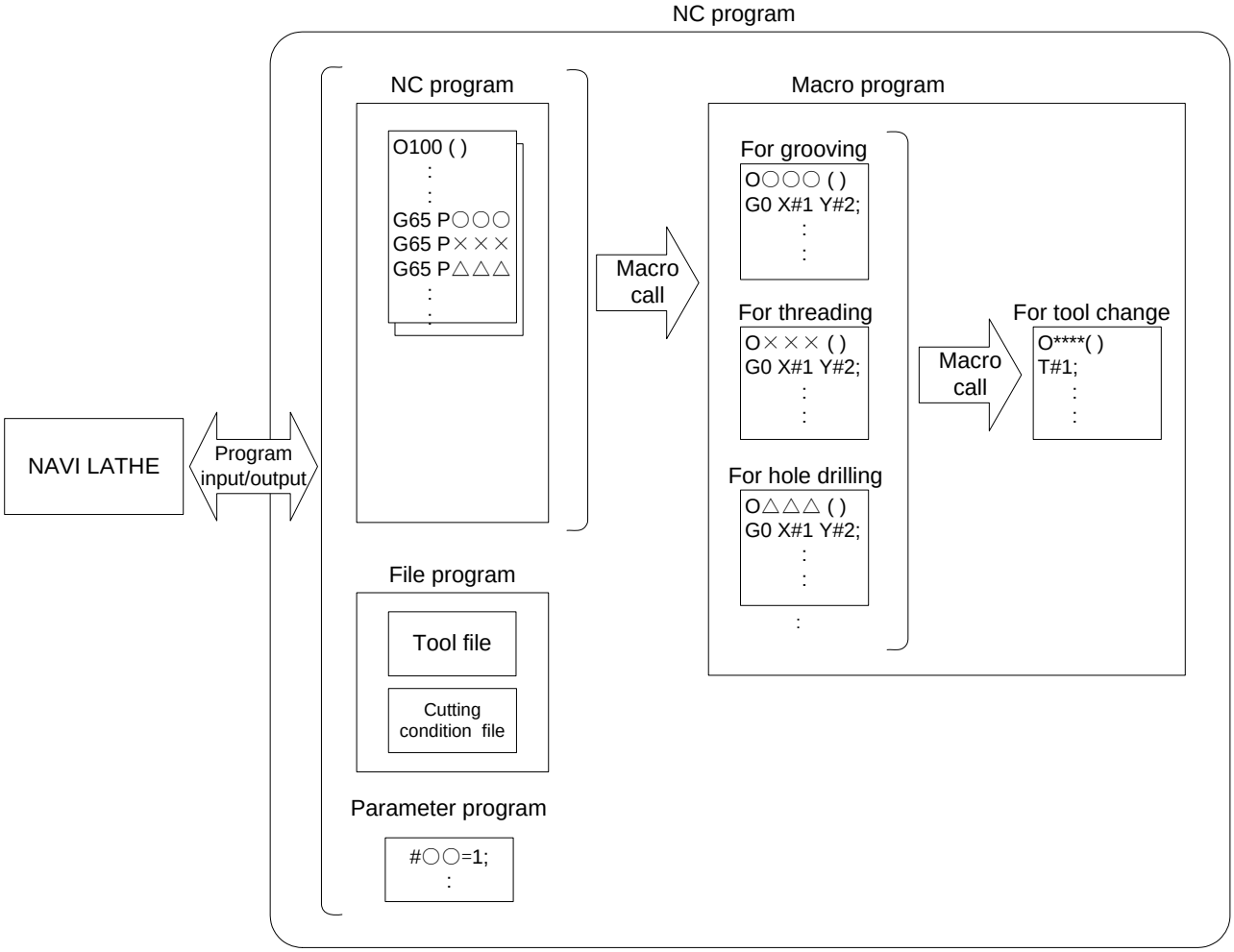
No.	Button	Application
1	Select	The tool registration No. at the cursor position is set to “TOOL REG No.” in each process screen. This button is valid only when the tool guidance screen is opened while the cursor is at the “TOOL REG No.” in each process screen.
2	Close	Closes the tool guidance screen.

5. 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.



5.1 NC Program

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

5.1.1 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
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 process		(NAVI-EIA);	
Milling hole drilling	Drilling	(NAVI-M HOLE-****-DRILL);	The symbol which indicates the machining area is set in the **** part. FACE: Front face OUT: Outer surface SIDE: Side 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-****);	
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, workpiece coordinate system setting
Turning	ROUGH	G65 P9102 A B C; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G71(72) U(W)_ R_ H_; G71(72) P_ Q_ U_ W_; N_ G1 X_ Z_; • • • N_ G1 X_ Z_; N_ G65 P9105 C; G40;	Movement to the tool change position, T command 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 pos Nose R compen. mode cancel
	FIN	G65 P9102 A B C; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G1 X_ Z_; • • • N_ G1 X_ Z_; N_ G65 P9105 C; G40;	Movement to the tool change position, T command 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 pos Nose R compen. mode cancel

Process		Program block	Remarks
Copy cutting	ROUGH	G65 P9102 A B C; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G73 U_ W_ R_; G73 P_ Q_ U_ W_; N_ G1 X_ Z_; • • • N_ G1 X_ Z_; N_ G65 P9105 C; G40;	Movement of the tool change position, T command 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 pos Nose R compen. mode cancel
	FIN	G65 P9102 A B C; G96 S_ M3(4) ; G0 X_ Z_ F_; G41(42); G70 P_ Q_; GOTO N_ N_ G1 X_ Z_; • • • N_ G1 X_ Z_; N_ G65 P9105 C; G40;	Movement of the tool change position, T command 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 pos Nose R compen. mode cancel
Threading		G65 P9130 A B C D E F • • • Z;	
Grooving		G65 P9140 A B C D E F • • • Z;	
Trapezoidal grooving		G65 P9150 A B C D E F • • • Z;	
Hole drilling	Drilling Pecking Boring Tapping	G65 P9120 A B C D E F • • • Z;	Common in drilling, pecking, boring and tapping.
EIA process		• • • ;	
Milling hole drilling	Drilling Deep hole drilling Boring Tapping	G65 P9171 A B C D E F • • • Z;	Common in drilling, deep hole drilling, boring and tapping.
Keyway cutting		G65 P9155 A B C D E F • • • Z;	
Contour cutting		G65 P9180 A B C D E F • • • Z;	
End process		G65 P9190; M#158;	

(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 "5.4 Macro Program".

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

Process end comment

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

5.1.2 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

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.
Deleting a block in the NC program process unit (process start comment to end comment) is no problem.

(2) Inserting block

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.
Inserting a block between the processes of the NC program (between the process end comment and next process start comment) is no problem.
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.)

Before movement

```
(NAVI-GRV-OUT);
G65 P9130 ····;        Process A
M50; ← Inserted block
(/NAVI);
(NAVI-THD-OUT);
G65 P9130 ····;        Process B
(/NAVI);
```

Move process A right after process B. →

After movement

```
(NAVI-THD-OUT);
G65 P9130 ····;        Process B
(/NAVI);
(NAVI-GRV-OUT);
G65 P9140 ····;        Process A
M50; ← Inserted block
(/NAVI);
```

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

Before movement

```
(NAVI-GRV-OUT);
G65 P9140 ····;        Process A
(/NAVI);
M50; ← Inserted block
(NAVI-THD-OUT);
G65 P9130 ····;        Process B
(/NAVI);
```

Move process A right after process B. →

After movement

```
M50; ← Inserted block
(NAVI-THD-OUT);
G65 P9130 ····;        Process B
(/NAVI);
(NAVI-GRV-OUT);
G65 P9140 ····;        Process A
(/NAVI);
```

(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.

5.2 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) Tool files and cutting condition files are saved via "parameter path" specified in the PREFERENCE screen.

5.3 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) Parameters are saved via "parameter path" specified in the PREFERENCE screen.

5.4 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 INIT process	9110	100019110	INIT MACRO
2	Macro program for turning process	9120	100019120	TURN MACRO
3	Macro program for copy-cutting process	9130	100019130	COPY MACRO
4	Macro program for threading process	9140 to 9145	100019140 to 100019145	THREAD MACRO
5	Macro program for grooving process	9150 to 9154	100019150 to 100019154	GROOVE MACRO
6	Macro program for trapezoidal grooving process	9160 to 9166	100019160 to 100019166	TGROOVE MACRO
7	Macro program for hole drilling process	9170	100019170	HOLE MACRO
8	Macro program for milling hole drilling process	9171 to 9177	100019171 to 100019177	MILL HOLE MACRO
9	Macro program for keyway cutting process	9155 to 9158	100019155 to 100019158	KEYWAY MACRO
10	Macro program for contour cutting process	9180	100019180	CONTOUR MACRO
11	Macro program for tool change	9102	100019102	TOOL CHANGE
12	Macro program for end process	9190	100019190	END-MACRO
13	Macro program for parameter setting	-	100019104	PARAM-SET-MACRO
14	Macro program for variable control	9105	100019105	VARIABLE-CTRL-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.".

6. 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 or MTB macro		
	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 interporation		These are necessary for milling.
	Multiple-spindle control II		
	Spindle position control (spindle/C axis control)		
Parameter specifications	**Parameter name**	**Setting details**	**Remarks**
	#1013 axname	1:X 2:Z 3:C 4:Y	Address of each axis name is specified. Select C for the address of the 3rd axis when performing milling machining. Select Y for the address of the 4th axis when performing side cutting in the milling machining.
	#1014 incax	1:U 2:W 3:H 4:V	Specify the incremental command axis name address for each axis. Select H for the address of the 3rd axis when performing milling machining. Select V for the address of the 4th axis when performing side cutting in the milling machining.

Division	Specifications		Remarks
Parameter specifications	#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	
	#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	Specifications		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.
	#1516 mill_ax	C	Select C for the name of the rotary axis used in milling interporation.
	#1517 mill_C	0	Specify Y axis as the hypothetical axis for milling interporation.

Division	Specifications		Remarks
Parameter specifications	#8102 COLL. ALM OFF	1	This is validated when executing the machining program created with NAVI LATHE.
	#8111 Milling Radius	0	Select all axes radius command to set the linear axis for milling interporation.
	#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.

Recommended specifications

Division	Specifications	Remarks
Additional specifications	Graphic check	
	Graphic trace	

7. ALARM MESSAGE

7.1 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.
	E198 Program format error	Program format is illegal.
	E199 File system error	An error occurred during file input or output.
Process editing	E201 Process number over	The number of processes exceeded 100.

Division	Message	Details
For turning: Turning/ Copy cutting For milling: Contour 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 agreement with angle (line number)	Linear I,K and angle are contradictory. Address I,K is changed according to the machining pattern.
	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.
Turning /Copy cutting	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.
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.
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).
Keyway cutting	E611 W < TOOL WIDTH	"Groove width < tool width" is applied.
	E612 GRV Height < CUT AMOUNT	"Groove height < cutting amount" is applied.
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.
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.

(Note) When data error occurs in Turning, Copy cutting and Contour cutting, line No. of the shape data is displayed following "L".

7.2 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.
	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…
	Data protect	Saving of the program, file, parameters is prohibited because the data protect key is validated. Reconsider the data protect key setting.

APPENDIX 1. VARIABLES USED IN NAVI LATHE

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

(1) Operation variables during program operation

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 interporation specifications	0: Not provided 1: Provided	0	Variable for operation

(2) Parameter variables during program operation

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/min0.001 to 99999.999mm	2000rev/min	Common
#162	#462	103	TOOL TURNING CL X	0.001 to 99999.999mm 0.0001 to 9999.9999inch	50.000mm 1.9685inch	Common
#163	#463	104	TOOL TURNING CL Z			Common
#164	#464	105	TOOL FIX RET POS X	-99999.999 to 99999.999mm -9999.9999 to 9999.9999inch	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

Variable No. User macro mode	Variable No. MTB macro mode	Para No.	Parameter name	Setting range	Standard value	Remarks
#168	#468	201	THD CLEARANCE EXIT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	2.000mm 0.0787inch	THD
#169	#469	202	THD CLEARANCE ENTR	0.000 to 99999.999mm 0.0000 to 9999.9999inch	2.000mm 0.0787inch	THD
#170	#470	301	GRV DWELL	0.001 to 99.999sec	1.000sec	GRV
#171	#471	302	GRV 2nd SHIFT AMOUNT	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV
#172	#472	303	GRV CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	1.000mm 0.0394inch	GRV
#173	#473	304	GRV RETRACT LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.2mm 0.0079inch	GRV
#174	#474	305	GRV OVERLAP LENGTH	0.001 to 99999.999mm 0.0001 to 9999.9999inch	0.1mm 0.0039inch	GRV
#175	#475	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	SYNC TAP	0: Invalid 1: Valid	0	HOLE
#180	#480	601	Y AXIS SPEC	0: Not provided 1: Provided	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	604	C AXIS CHANGE M CODE	0 to 9999	103	Common
#184	#484	605	C-AXIS CLAMP	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
#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

Variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#189	#489	701	HOLE CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	10.000mm 0.3937inch	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
#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	10.000mm 0.3937inch	K WAY
#193	#493	901	E-ML CUT WIDTH PCT(%)	1 to 100%	50%	CONT
#194	#494	902	E-ML CLEARANCE	0.001 to 99999.999mm 0.0001 to 9999.9999inch	10.000mm 0.3937inch	CONT
#195	#495	903	E-ML EMPTY D OFS NUM	1 to tool sets	0	CONT
#196	#496	903	TAP ON M CODE	0 to 9999	0	M HOLE
#197	#497	903	TAP OFF M CODE	0 to 9999	0	M HOLE

⚠ CAUTION

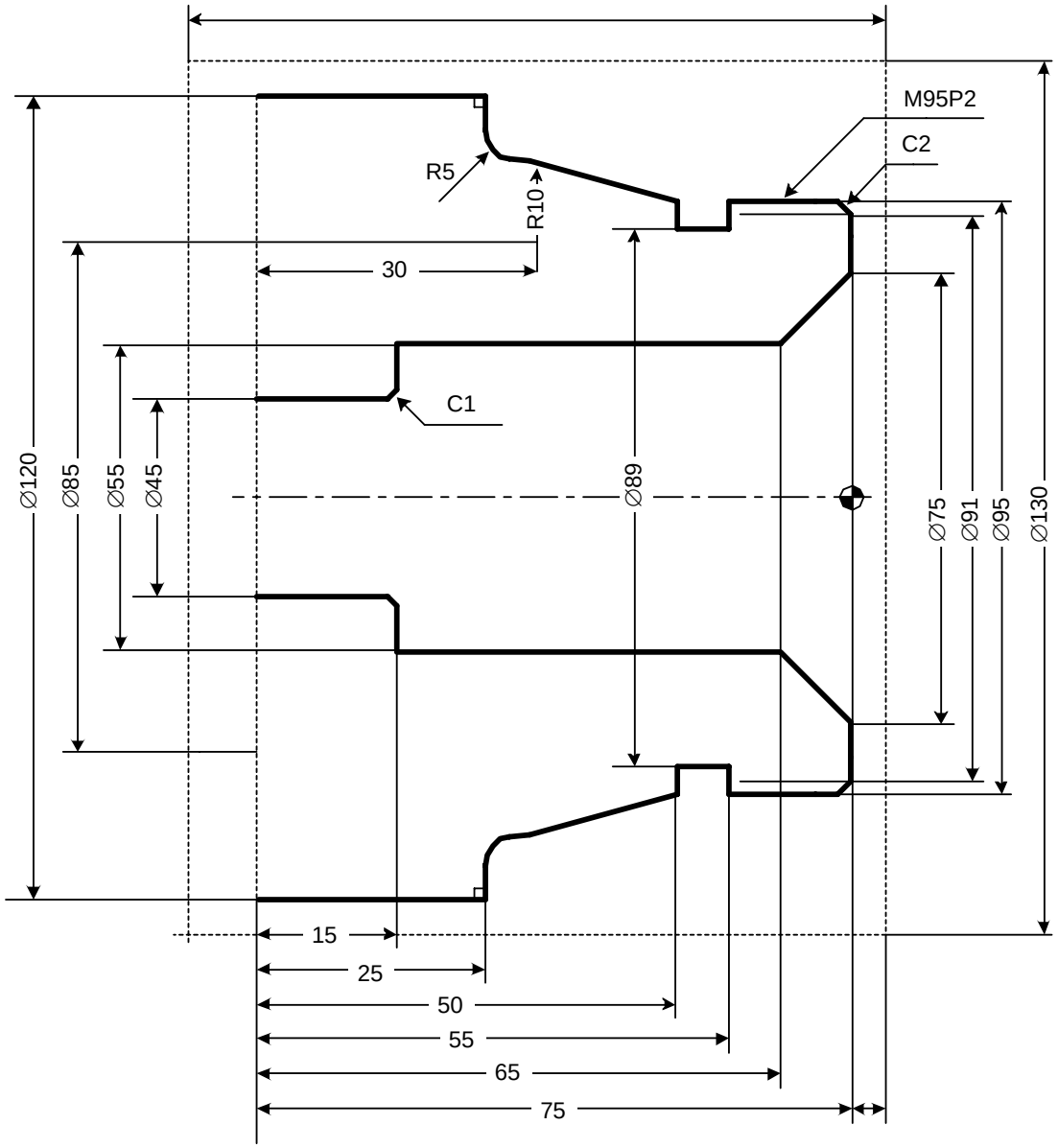
⚠ NAVI LATHE uses the following variables in order to operate the NC program.

NC program mode	Variables used by NAVI LATHE
User macro mode	#150 to #197
MTB macro mode	#450 to #497

When NC program mode is user macro mode, do not use common variables (#150 to #197). 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 the power is turned 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.

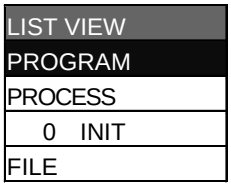
2. Press the [NEW] menu and create a new NC program.

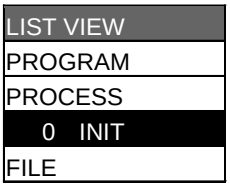
3. Move the cursor to "0 INIT" and press the [MODIFY] menu.

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.



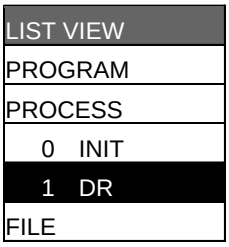


5. Process 1 Drilling machining (DR)

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.

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

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

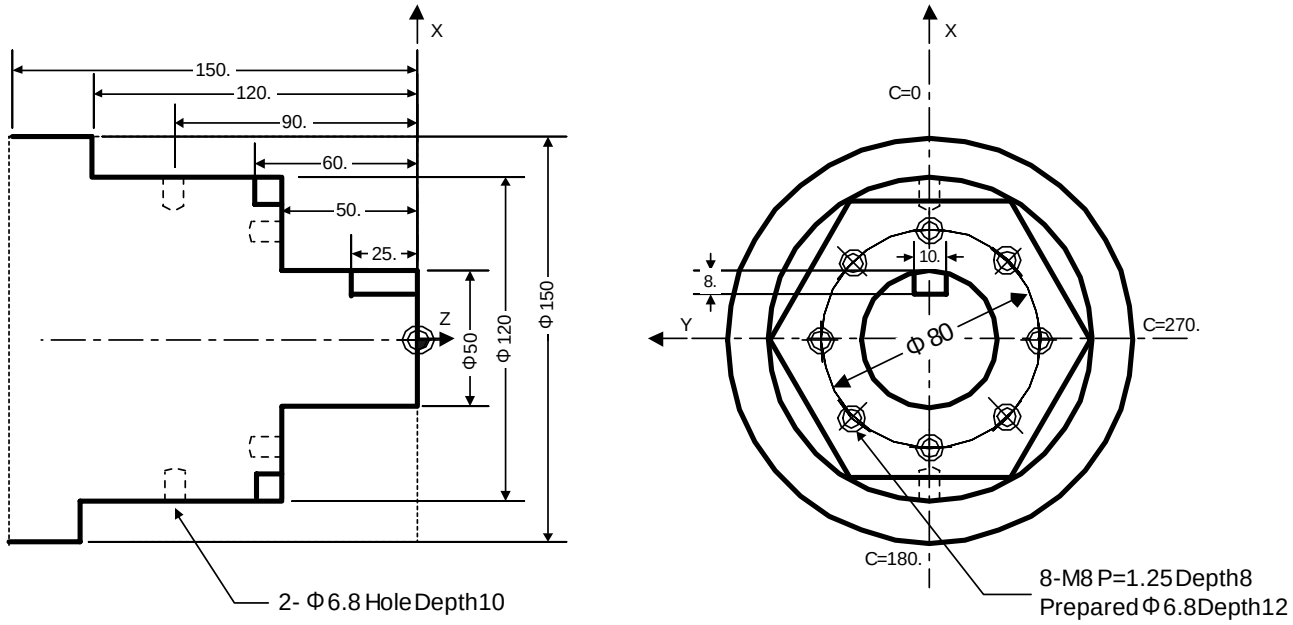
14.3 Save the data of the threading outer diameter finishing machining by pressing the [SAVE] menu.

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.

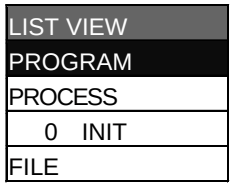
2. Press the [NEW] menu and create a new NC program.

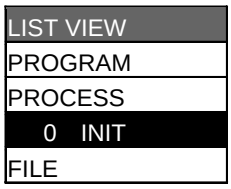
3. Move the cursor to "0 INIT" and press the [MODIFY] menu.

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK -FACE
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
7 M DR -OUT
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
7 M DR -OUT
8 M PECK-OUT
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
7 M DR -OUT
8 M PECK-OUT
9 M DR -OUT
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
7 M DR -OUT
8 M PECK-OUT
9 M DR -OUT
10 K WAY-OUT R
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 M DR -FACE
2 M PECK-FACE
3 M DR -FACE
4 M TAP -FACE
5 CONT -FACE R
6 CONT -FACE F
7 M DR -OUT
8 M PECK-OUT
9 M DR -OUT
10 K WAY-OUT R
11 K WAY-OUT F
FILE

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.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

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

Western Michigan Service Satellite
ALLEGAN, MICHIGAN., 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-269-673-4092

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

Minnesota Service Satellite
RICHFIELD, MINNESOTA, 55423, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Western Region Service Center
5665 PLAZA DRIVE, CYPRESS, CALIFORNIA, 90630, U.S.A.
TEL: +1-714-220-4796 / FAX: +1-714-229-3818

Eastern Region Service Center
200 COTTONTAIL LANE SOMERSET, NEW JERSEY, 08873, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Western Pennsylvania Service Satellite
ERIE, PENNSYLVANIA, 16510, U.S.A.
TEL: +1-814-897-7820 / FAX: +1-814-987-7820

Southern Region Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GEORGIA, 30097, U.S.A.
TEL: +1-678-258-4500 / FAX: +1-678-258-4519

Northern Texas Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TEXAS, 76051, U.S.A.
TEL: +1-817-251-7468 / FAX: +1-817-416-5000

Southern Texas Service Satellite
FRIENDSWOOD, TEXAS, 77546, U.S.A
TEL: +1-832-573-0787 / FAX: +1-678-573-8290

Central Florida Service Satellite
SATELLITE BEACH, FLORIDA, 32937, U.S.A.
TEL : +1-321-610-4436 / FAX : +1-321-610-4437

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

Mexico City Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 / FAX: +52-55-9171-7649

Monterrey Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 / FAX: +52-81-8365-4171

Brazilian Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

Brazilian's Sites Service Center
CITIES OF PORTO ALEGRE AND CAXIAS DO SUL BRAZIL
CITIES OF SANTA CATARINA AND PARANA STATES
TEL: +55-15-3363-9927

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V. (EUROPE FA CENTER)
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-3270-010 / FAX: +49-711-3270-0141

France Service Center
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
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI 7-PALAZZO SIRIO CENTRO DIREZIONALE COLLEONI,
20041 AGRATE BRIANZA MILANO ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

Italy (Padova) Service Sattelite
VIA SAVELLI 24 - 35129 PADOVA ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-27-6100 / FAX: +44-1707-27-8992

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08190 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: +48-12-630-4727

Poland (Wroclaw) Service Center
UL KOBIERZYCKA 23,52-315 WROCLAW,POLAND
TEL: +48-71-333-77-53 / FAX: +48-71-333-77-53

Turkey Service Center
BAYRAKTAR BULVARI, NUTUK SOKAK NO:5, YUKARI DUDULLU
ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
TECHNOLOGICKA 374/6,708 00 OSTRAVA-PUSTKOVEC, 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

Sweden Service Center
STRANDKULLEN, 718 91 FROVI, SWEDEN
TEL: +46-581-700-20 / FAX: +46-581-700-75

Bulgaria Service Center
4 A. LYAPCHEV BOUL., 1756 - SOFIA, BULGARIA
TEL: +359-2-8176000 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY PEREULOK 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +38-57-732-7744 / FAX: +38-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +38-044-494-3355 / FAX: +38-044-494-3366

Belarus Service Center
703, OKTYABRSKAYA STR., 16/5, 220030 MINSK, BELARUS
TEL: +375-17-210-4626 / FAX: +375-17-227-5830

South Africa Service Center
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625 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

Indonesia Service Center
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 / FAX: +62-21-3917-164

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
No. 16, JALAN SHAH BANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Vietnam Service Center-1
47-49 HOANG SA ST. DAKAO WARD, DIST. 1, HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 / FAX: +84-8-910-2593

Vietnam Service Center-2
THUAN KIEN PLAZA 190 HONG BANG ROAD.TOWER C. SUITE 3002. DIST.5,
HO CHI MINH CITY,VIETNAM
TEL: +84-8-240-3587 / FAX: +84-8-726-7968

Vietnam (Hanoi) Service Center
5TH FL., 93B KIM LIEN ST., PHUONG LIEN WARD, DONG DA DIST.
HN, VIETNAM
TEL: +84-8-573-7646 / FAX: +84-4-573-7650

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

MITSUBISHI ELECTRIC AUTOMATION (THAILAND) CO., LTD. (THAILAND FA CENTER)

BANG-CHAN INDUSTRIAL ESTATE No.111 SOI SERITHAI 54
T.KANNAYAO, A.KANNAYAO, BANGKOK 10230, THAILAND
TEL: +66-2906-8255 / FAX: +66-2906-3239

Thailand Service Center
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 7
RAMA III RD BANGPONGPANG,YANNAWA,BANGKOK 10120. THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-9750

INDIA

MITSUBISHI ELECTRIC ASIA PVT LTD

FIRST & SECOND FLOOR, AVR BASE, MUNICIPAL No.BC-308,
HENNURE BANASWADI ROAD, HRBR RING ROAD, BANGALORE-560 043,INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699

India (Pune) Service Center
EL-3, J BLOCK, M.I.D.C., BHOSARI PUNE 411026, INDIA
TEL: +91-20-2710-2000 / FAX: +91-20-2710-2185

India (Bangalore) Service Center
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 / FAX: +91-80-532-0480

India (Delhi) Service Center
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32nd MILESTONE
GURGAON 122001, INDIA
TEL: +91-98-1024-8895

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA

CHINA

MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD. (CHINA FA CENTER)

China (Shanghai) Service Center
4/F ZHI FU PLAZA, NO. 80 XIN CHANG ROAD,
SHANGHAI 200003,CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-2800
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTRE, 18 JIANGUOMENNEI AVENUE
DONGCHENG DISTRICT, BEIJING, CHINA 100005
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030
China (Beijing) Service Dealer

China (Tianjin) Service Center
B-2-801-802, YOUYI BUILDING. 50 YOUYI ROAD, HEXI DISTRICT
TIANJIN, CHINA 300061
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite

China (Chengdu) Service Center
BLOCK B-1, 23F, CHUAN XIN MANSION, 18 SECTION 2
RENMIN ROAD (SOUTH), CHENGDU, SICHUAN, CHINA 610016
TEL: +86-28-8619-9730 / FAX: +86-28-8619-9805

China (Changchun) Service Satellite
BING 3RD ROAD., INDUSTRIAL&ECONOMICAL DEVELOPMENT ZONE, CHAOYANG DISTRICT
TEL : +86-0431-5021-546 / FAX : +86-0431-5021-690

China (Shenzhen) Service Center
ROOM 2512-2516, GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN, CHINA 518034
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776

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 Taegu Service Satellite
603 CRYSTAL BUILDING 1666, SANBYEOK-DONG, BUK-KU, DAEGU, 702-010, KOREA
TEL: +82-53-604-6047 / FAX: +82-53-604-6049

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD(TAIWAN FA CENTER)

Taiwan (Taichung) Service Center
No.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center
3TH. FLOOR, No.122 WUKUNG 2ND RD., WU-KU HSIANG, TAIPEI HSIEN, TAIWAN R.O.C
TEL: +886-2-2299-2205 / FAX: +886-2-2298-1909

Taiwan (Tainan) Service Center
2F(C),1-1, CHUNGHWA-RD, YONGKANG CITY, TAINAN HSIEN, TAIWAN R.O.C
TEL: +886-6-313-9600 / FAX: +886-6-313-7713

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

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M700/M70 Series
MODEL CODE	008—366
Manual No.	IB-1500146

IB(NA)1500146(ENG)-C(1004)MEE