MITSUBISHI CNC

700 Series

Simple Programming Function
NAVI LATHE Instruction Manual

Introduction

This manual is an instruction manual for NAVI LATHE for 700 (hereafter NAVI LATHE).
This manual explains how to operate NAVI LATHE, so read this manual thoroughly before use. Be sue 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 Series   Instruction Manual IB-1500042
MITSUBISHI CNC 700 Series   Setup Manual IB-1500124
MITSUBISHI CNC 700 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

Not applicable in this manual.

⚠ 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.

⚠ **CAUTION**

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

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

❶ 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

⚠ When either "TOOL REG No." or "HOLE CYCLE" is input in the hole drilling screen, the feedrate and spindle speed are automatically determined using the data in the tool file screen and the cutting condition file screen. In the same way, when "TOOL REG No." is input in the face cutting screen, the contour cutting screen and the pocket screen, the feedrate and spindle speed are automatically determined. Note that the feedrate and spindle speed of each process determined once will not be changed by changing the data in the tool file screen and the cutting condition file screen.

⚠ 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 #177
MTB macro mode	#450 to #477

When NC program mode is user macro mode, do not use common variables (#150 to #177). 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.

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.

⚠ **CAUTION**

(Continued from previous page)

⚠ 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.

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.

Contents

1. OUTLINE

1.1   System Outline

This manual is an instruction manual for NAVI LATHE for 700 (hereafter NAVI LATHE).
The part program for the turning center (two axes of X and Z) is created with the NAVI LATHE.

(1) The following machining processes can be edited.

- Turning (Outer dia., inner dia., front face)
- Copy cut (Outer dia., inner dia., front face)
- Thread (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

(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.

[LIST VIEW area]
The object of the NAVI LATHE is selected.

[OPERATION VIEW area]
The screen is displayed corresponding to the object selected in the LIST VIEW.

[Cutting conditions automatically determined]
Upon tool registration No. entry, the cutting conditions for each process are automatically determined based on the tool file and cutting condition file.

[Help]
[Guide drawing]

[Menu keys]

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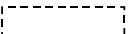
(5) Guidance function provides an operator with error recovery information.

(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 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

Start

File edition
　Tool file
　Cutting condition file

Tool file
(Tool registration No. 101-)

99
et
1　o.
Tool leng. offset　.
　No.　et
Tool diam. offset　o.
　No.　n
Spindle rotation　n
　direction
:

Cut condition file
(Work registration No.1 to 8)

8
able
1　e
Material
Tool applicable
rotation rate
:

Parameter setting
　Parameter file
　(The parameter setting is valid
　even if the parameter is set after
　editing the NC program)

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

NC program selection
　Newly create
　Read out

Process editing

Process editing:
　Initial conditions
　Process mode selection
　Process data input
　　Turning / Copy cutting / Threading
　　/ Grooving / Trapezoidal grooving /
　　Hole drilling / EIA

Program check
　Program Checker is used.

Program check
　(Note) Set the tool
　compensation amount and
　workpiece coordinate system
　offset to perform Program
　Check. This function is realized
　by using the 700 Series graphic
　check function.

NC program operation

END

1.3　Screen Configuration

The screen configuration for the NAVI LATHE is shown below.

```
                  ┌─────────────┐
  ┌──────────┐    │  Program    │
  │ Program  │◄──►│  editing    │
  └──────────┘    │  screen     │
                  └─────────────┘

                  ┌──────────────┐        ┌──────────┐
                  │ Process list │        │ Turning  │────┐   ┌──────────┐
                  │   screen     │        │  screen  │    │   │ Process  │
                  └──────────────┘        └──────────┘    ├──►│ pattern  │
                                          ┌──────────┐    │   │  screen  │
                  ┌──────────────┐        │  Copy-   │    │   └──────────┘
                  │   Initial    │        │ cutting  │────┘
                  │  condition   │        │  screen  │
                  │setting screen│        └──────────┘
                  └──────────────┘        ┌──────────┐
  ┌──────────┐    ┌──────────────┐        │Threading │
  │ Process  │◄──►│ Process mode │        │  screen  │
  └──────────┘    │  selection   │        └──────────┘
                  │   screen     │        ┌──────────┐
                  └──────────────┘        │ Grooving │
                  (For a new process,     │  screen  │
                  select a process from   └──────────┘
                  the process mode.)      ┌──────────┐
                                          │Trapezoidal│
                                          │ grooving │
                                          │  screen  │
                                          └──────────┘
                                          ┌──────────┐
                                          │Hole drilling│
                                          │  screen  │
                                          └──────────┘
                  ┌──────────────┐        ┌──────────┐
                  │   Program    │        │ EIA      │
                  │   checker    │        │ screen   │
                  └──────────────┘        └──────────┘

                  ┌──────────────┐
                  │  Tool file   │
                  │   screen     │
  ┌──────────┐    └──────────────┘
  │  File    │◄──►┌──────────────┐
  └──────────┘    │  Cutting     │
                  │  condition   │
                  │  file screen │
                  └──────────────┘

  ┌──────────┐    ┌──────────────┐        ┌──────────┐
  │Parameter │◄──►│  Parameter   │◄──────►│Preference│
  └──────────┘    │   screen     │        │  screen  │
                  └──────────────┘        └──────────┘

  ┌──────────┐    ┌──────────────┐
  │ Version  │◄──►│   Version    │
  └──────────┘    │   screen     │
                  └──────────────┘
```

Screen name	Details
Program editting 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 machining 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	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.
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	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 ⬚ 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.	D:/NCFILE/NAVI
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.
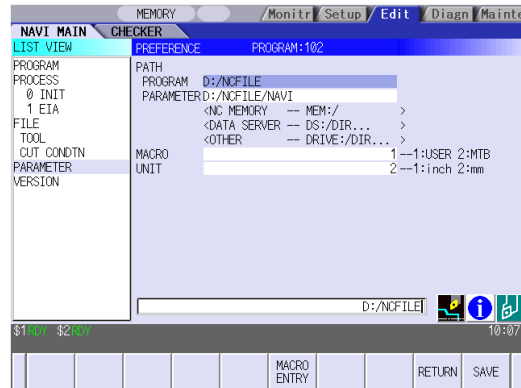
(4) | Select the macro type.
(1:Uer macro 2:MTB macro) |

(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
(2) OPERATION VIEW area
(3) Setting area
(4) Message area
(5) Menu display area

(1) LIST VIEW area

(2) OPERATION VIEW area

(4) Message area

(5) Menu display area

(3) Setting area

<Screen example>
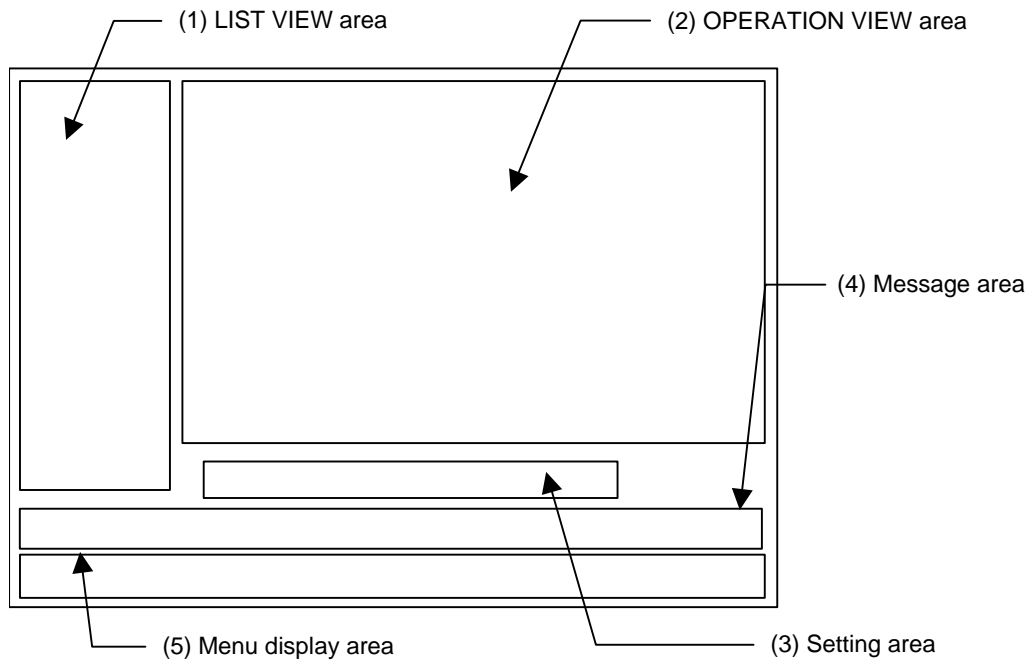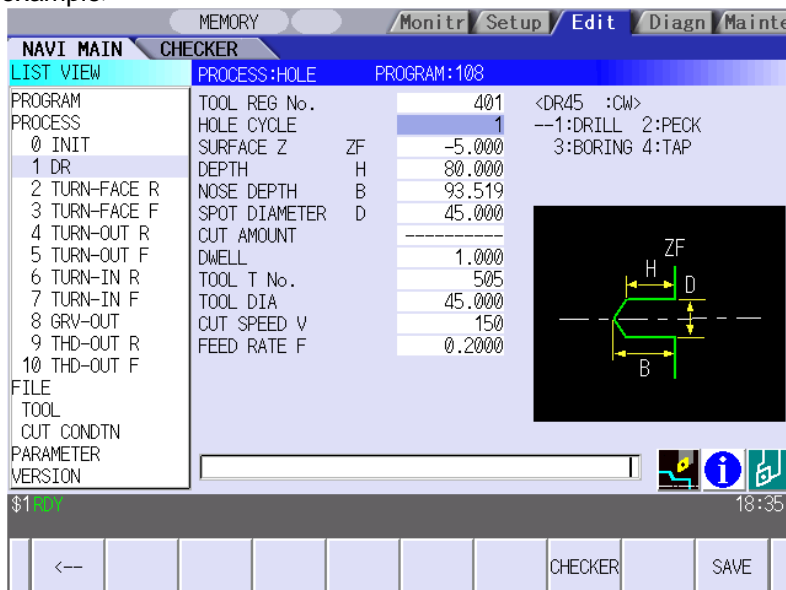
2.1   LIST VIEW Area

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

```
LIST VIEW        ▼                    (1) Area bar
PROGRAM
PROCESS
  0 INIT       ▼
  1 DR
  2 TURN-FACE R
  3 TURN-FACE F
  4 TURN-OUT R                        (2) Object
  5 TURN-OUT F
  6 TURN-IN R
  7 TURN-IN F
  8 GRV-OUT        ◀──                (3) Cursor
  9 THD-OUT R
 10 THD-OUT F
FILE
 TOOL
 CUT CONDTN
PARAMETER
VERSION
```
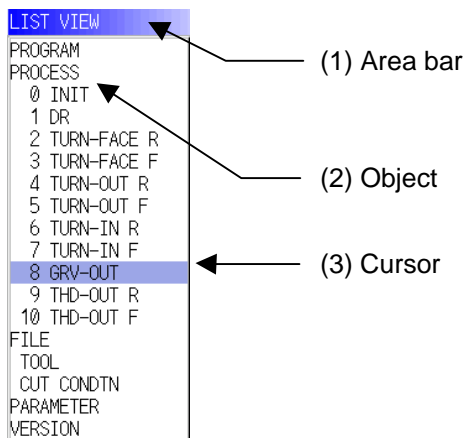
(1) Area bar

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.
	CUT CONDTN	Displays and changes the cutting conditions for each process per tip material or workpiece material.
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

(2) Help

(3) Guide drawing

(4) Sub cursor
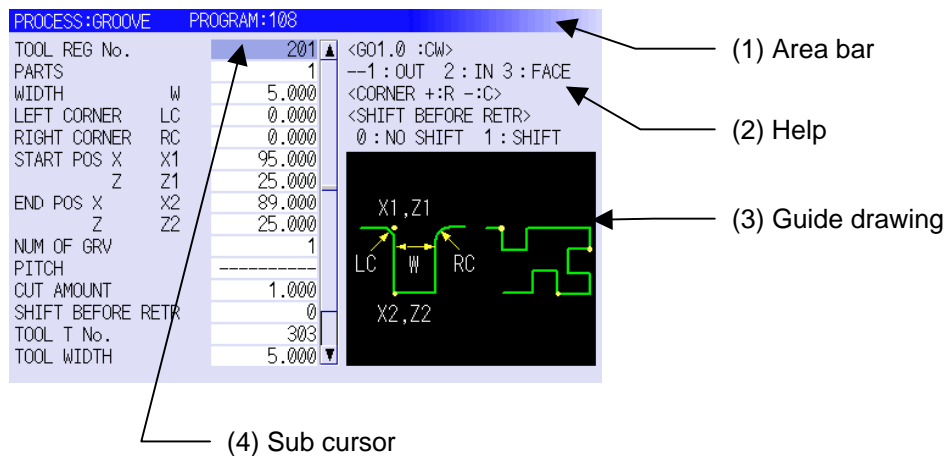
(1) Area bar

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 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) The program edit screen is displayed. ➡ The OPERATION VIEW area is active.
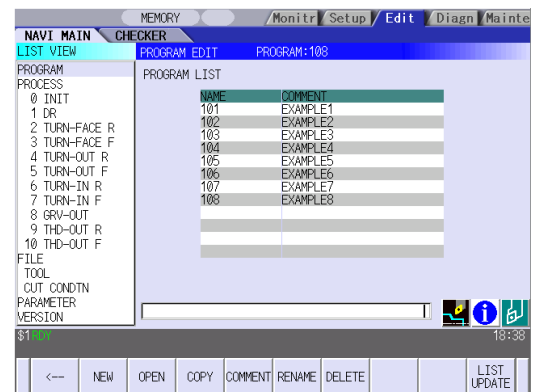
(2) Press the [←] menu key. ➡ The LIST VIEW area will turn active.
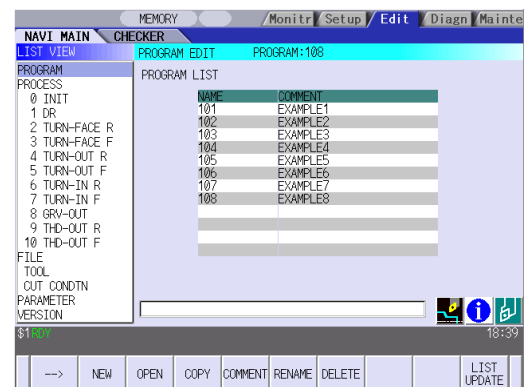
(3) | Select the object with the cursor key. ➡️ The OPERATION VIEW area will change into the screen corresponding to the selected object.
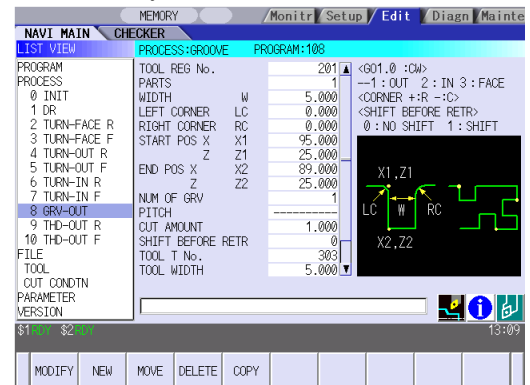
(4) | Press the [MODIFY] menu key. ➡️ The OPERATION VIEW area will turn active.
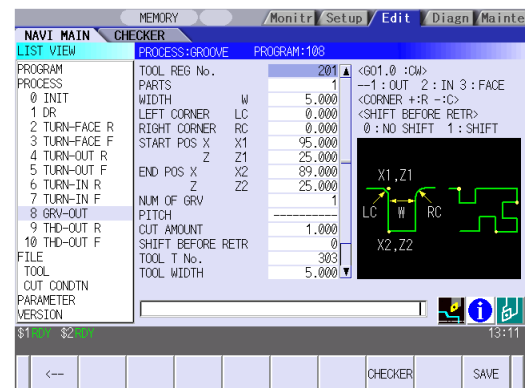
3.3 Setting Data

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.)
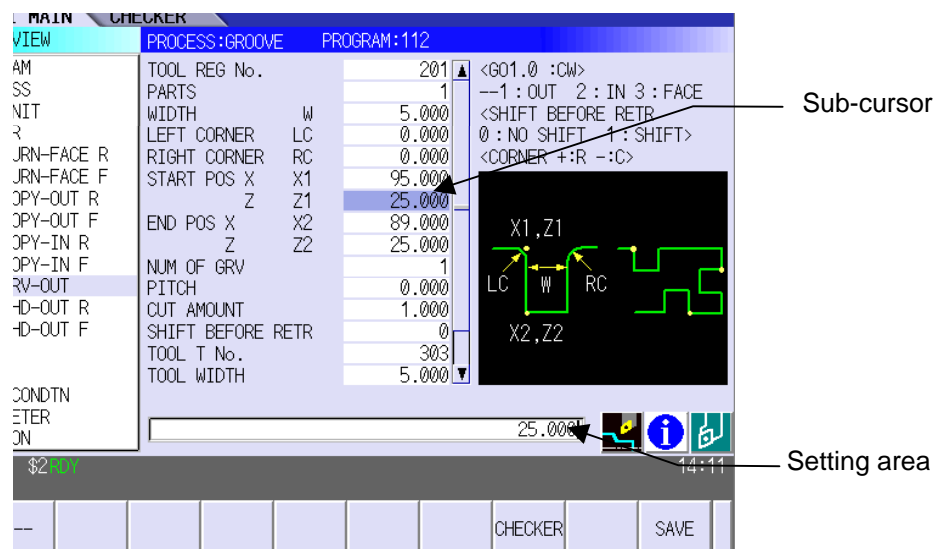
Operation method

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. |
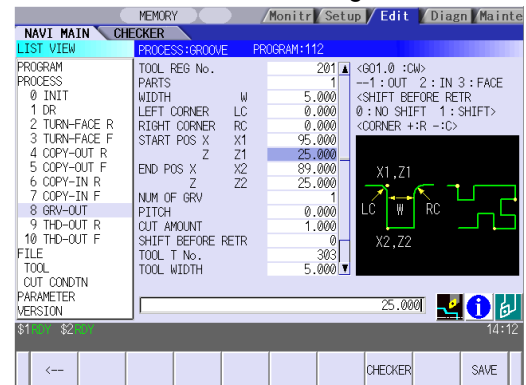
(3) Data key input

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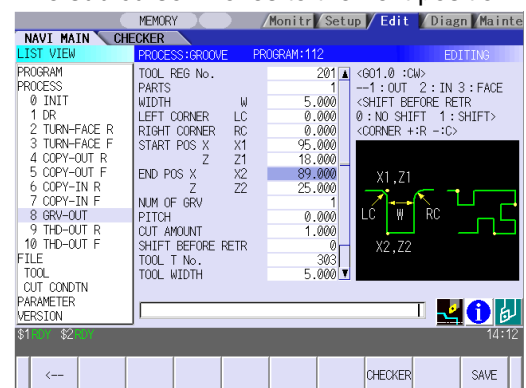
(Note 1) The contents in the data setting area are only displayed when [INPUT] key is not pressed and will be invalidated if the screen is changed at this time. Data for the currently selected setting item will be set when [INPUT] key is pressed.

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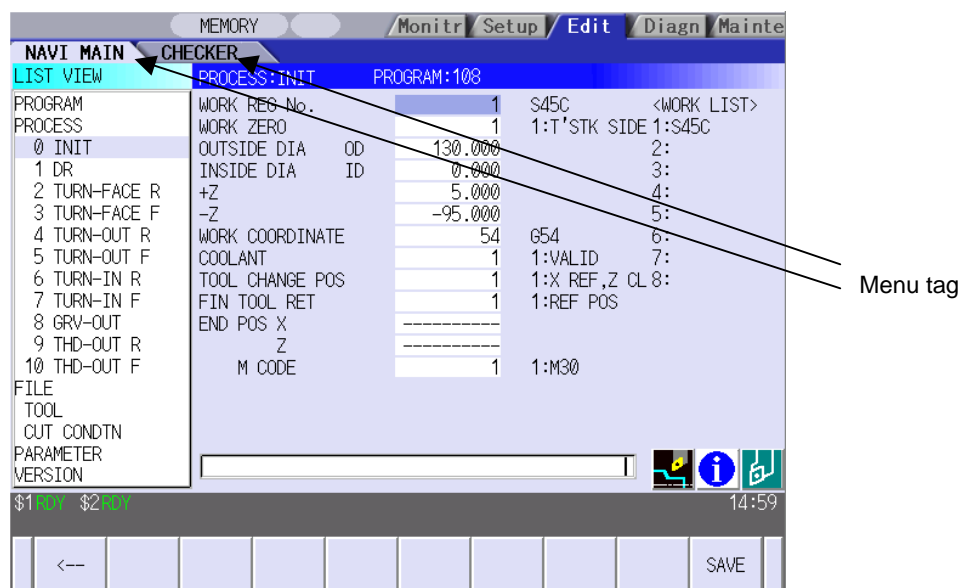
When a tag button on the keyboard is pressed, the main window and checker window can be switched over.

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.

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

(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.

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 (°).
Atangent   : −90 < operation results < 90.

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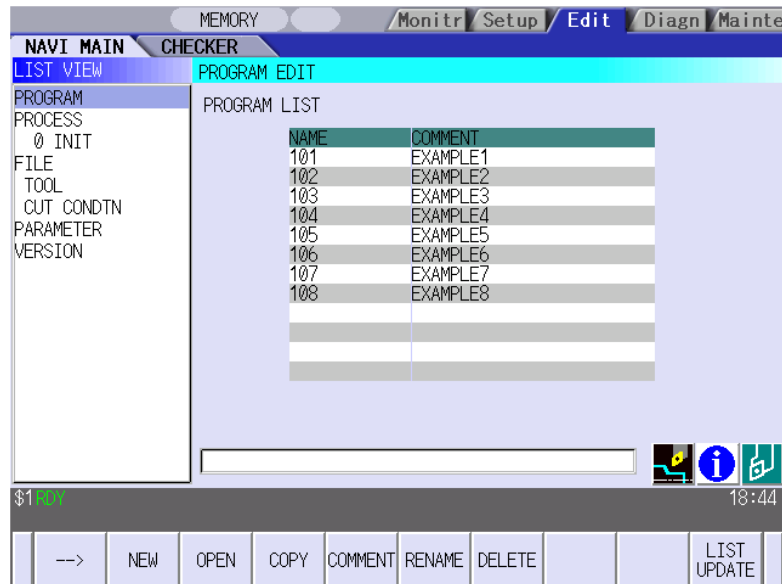
At the initial start up of NAVI LATHE, the cursor is displayed at the position of [PROGRAM] in the LIST VIEW area, and the program edit screen is displayed in the OPERATION VIEW area.
The LIST VIEW area is active.
The process program is not selected.

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
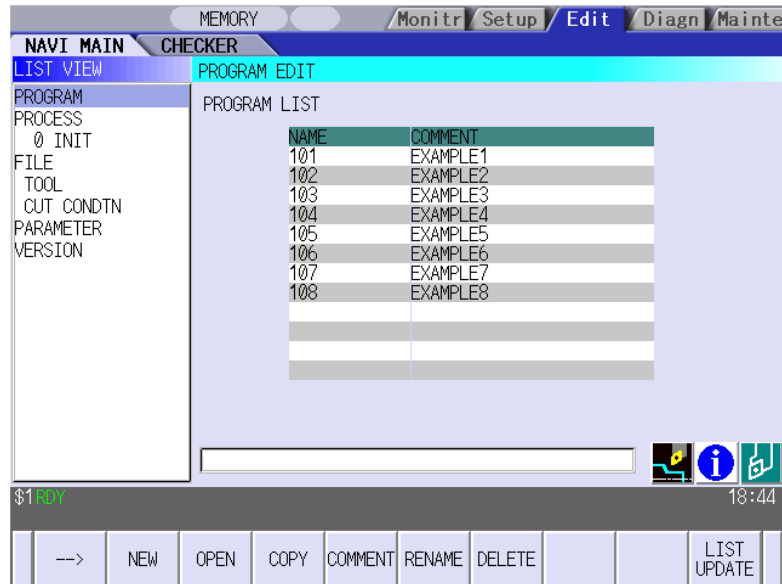
The process list of the currently selected program is displayed in the LIST VIEW area.
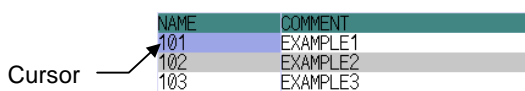
< Process displays >

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 cut	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	

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. Cursor ⟶ [NAME / COMMENT list: 101 EXAMPLE1, 102 EXAMPLE2, 103 EXAMPLE3]
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.
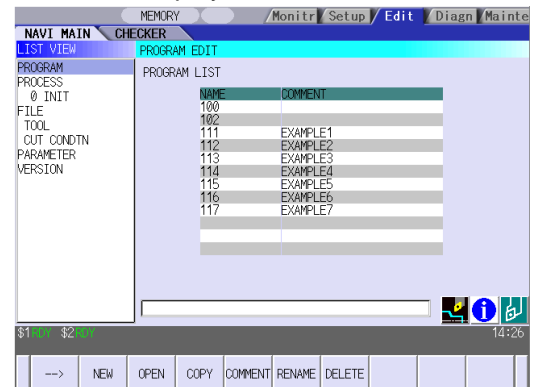
(2) Press the [OPEN] menu key, and input the NC program No. to be read out.

➡ The [OPEN] menu will be highlighted, and the setting area will be displayed.
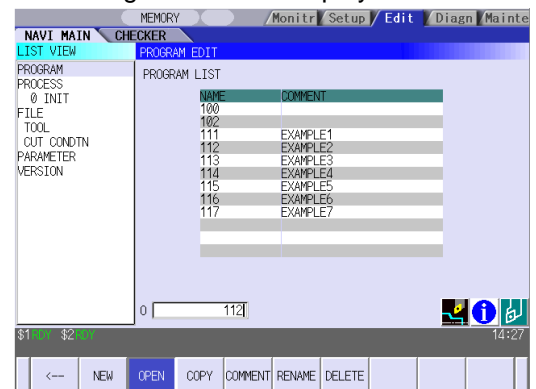
(3) Press the [INPUT] key.

➡ 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.
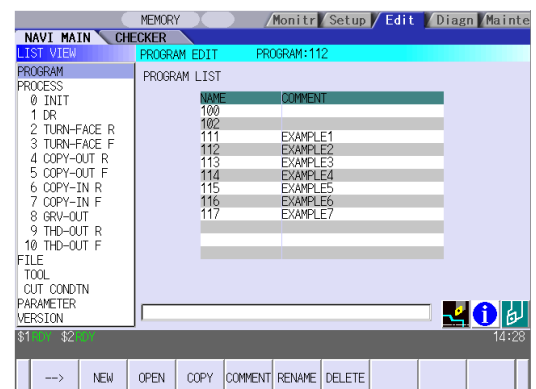
4.3   Screens Related to the Process Edit Functions

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
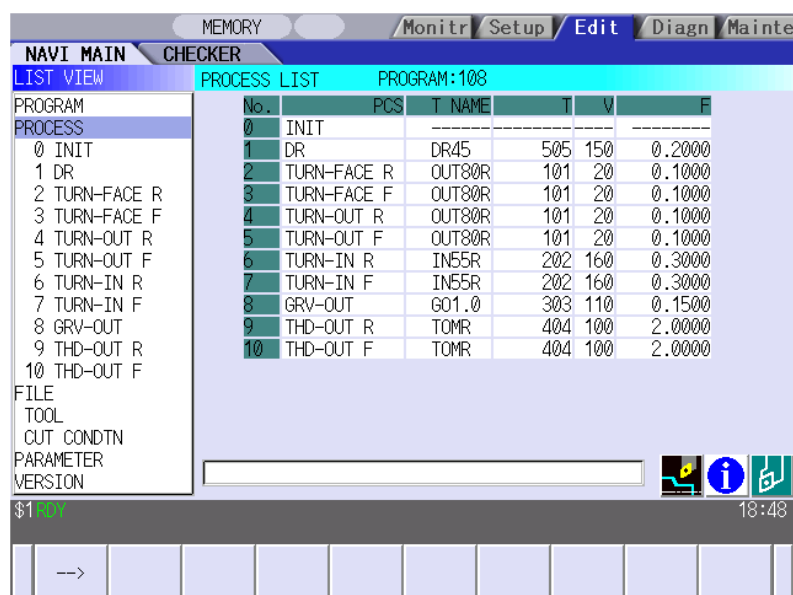
Screen display items

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.	0 to 99999999
4	V	The cutting speed is displayed. The cutting speed can be changed.	1 to 9999 m/min 1 to 9999feet/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
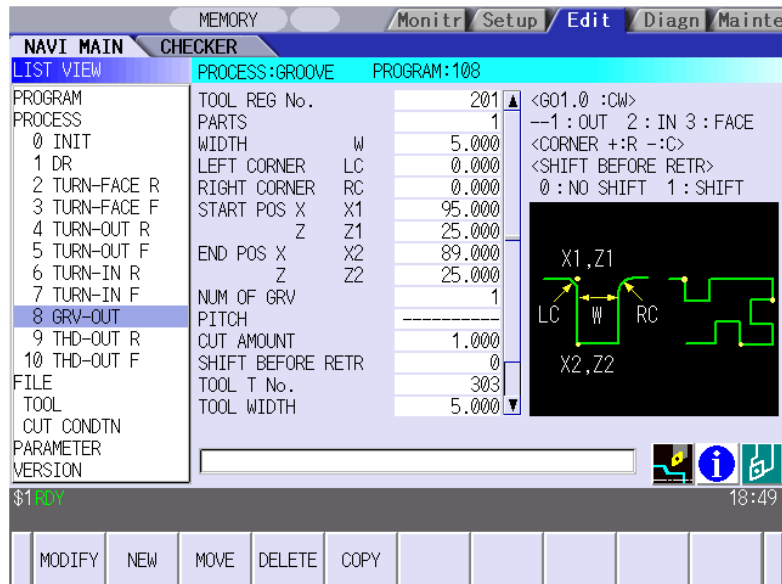
Menus

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.
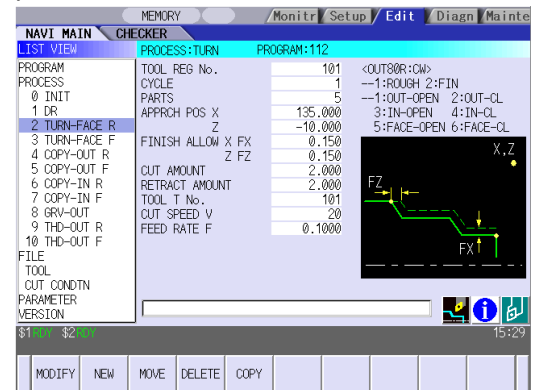
(2) | Press the [MODIFY] menu key. | ➡ The OPERATION VIEW area will turn active.

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.
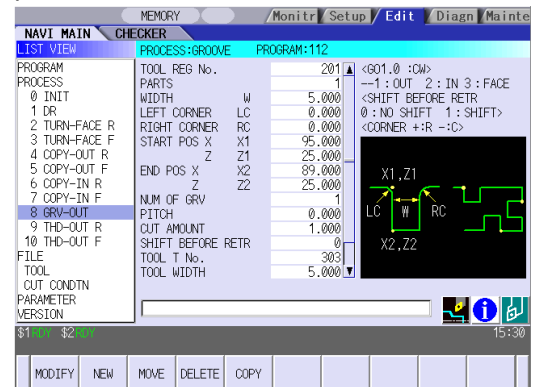
(2) | Press the [DELETE] menu key. | ➡ | The [DELETE] menu will be highlighted, and a massage confirming the deletion will appear.

(3) | Press the [Y] key. | ➡ | The highlight of the [DELETE] menu will turn OFF, and the process at the cursor position will be deleted.

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

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.
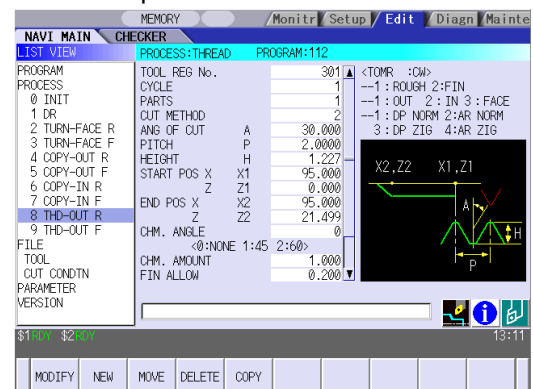
Operation example (Copying the process)

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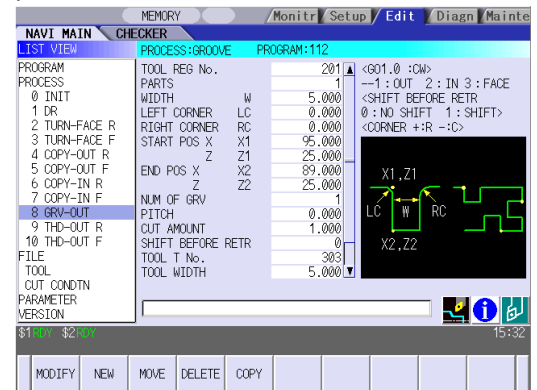
(2) | Press the [COPY] menu key. | ➡ The copied process will be inserted under the cursor position.
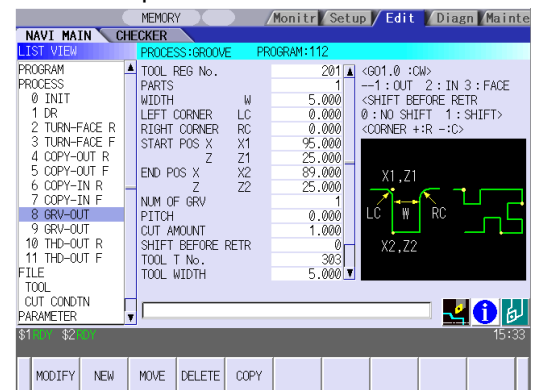
Operation example (Moving the process)

(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.
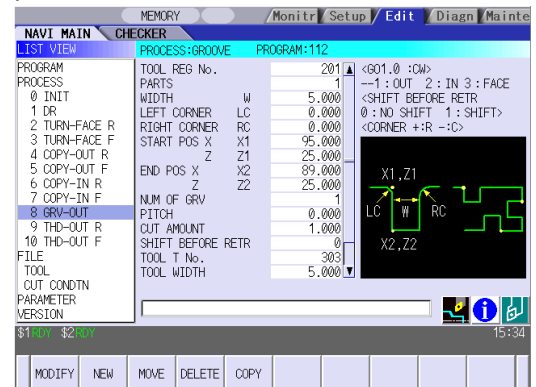
(2) Press the [MOVE] menu key. ➡ The [MOVE] menu will be highlighted.

The mark "M" will be displayed beside the process to be moved.
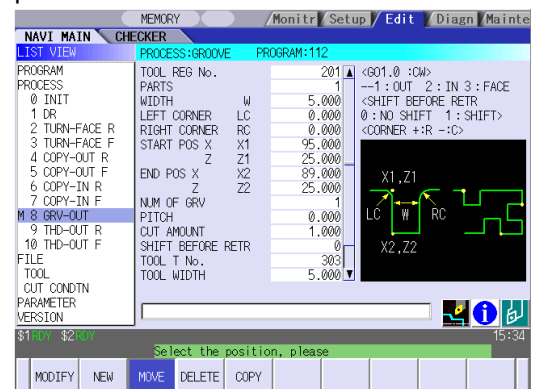
(3) Select the position of the movement destination with the cursor key. ➡
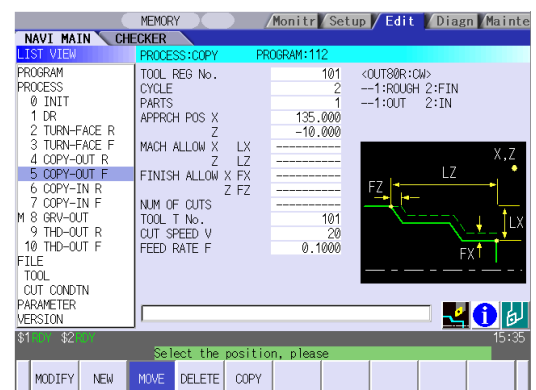
(4) Press the [INPUT] key. ➡ The message to confirm a movement is displayed.

If the [MOVE] menu key is pressed again during the movement operation, the movement operation will be canceled.
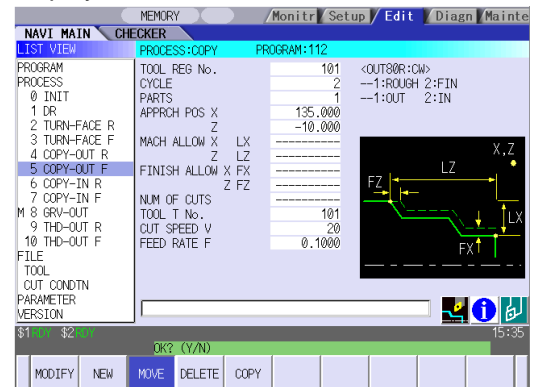
(5) Press the [Y] key.

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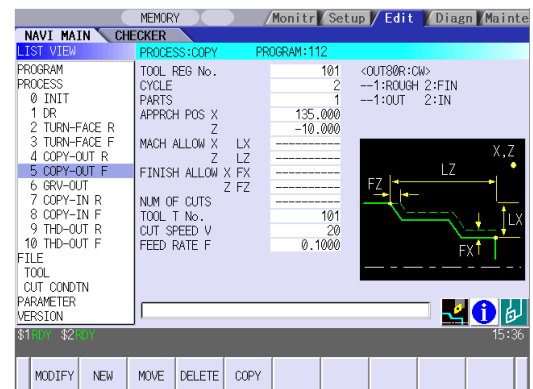
(Note) For the [NEW] menu, refer to the next section.

4.3.3   Process Mode Selection Screen

When a new process is added, the process mode is selected on this screen. When the [NEW] menu key is pressed at the position of the [PROCESS] in the LIST VIEW area, this screen is displayed.

Screen layout
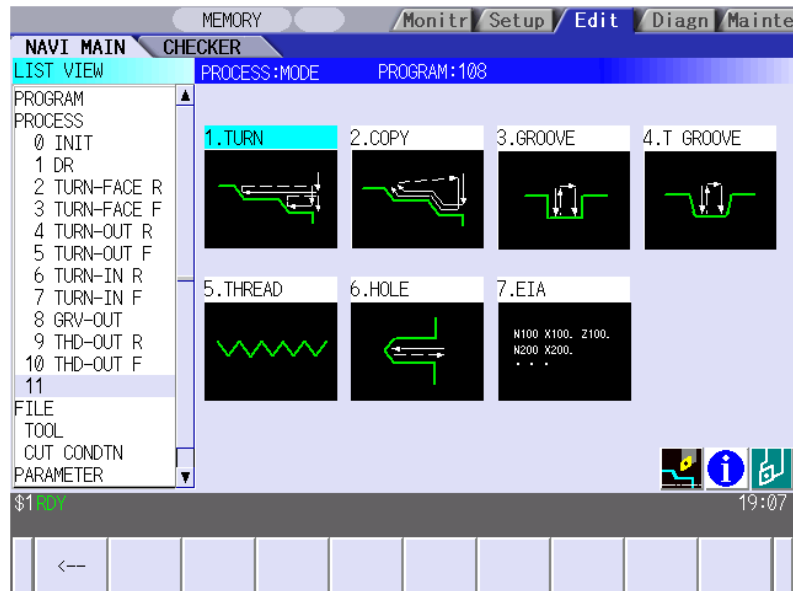
Screen display item

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected. Select the process mode by moving the sub-cursor or inputting numerical values.	1 to 7

Menu

No.	Menu	Details
1	←	Cancels adding a new process. The LIST VIEW area will turn active after cancel.

Operation example (Adding a new process)

(1) Validate the LIST VIEW area, and select the position where the process is added with the cursor key. ➡️
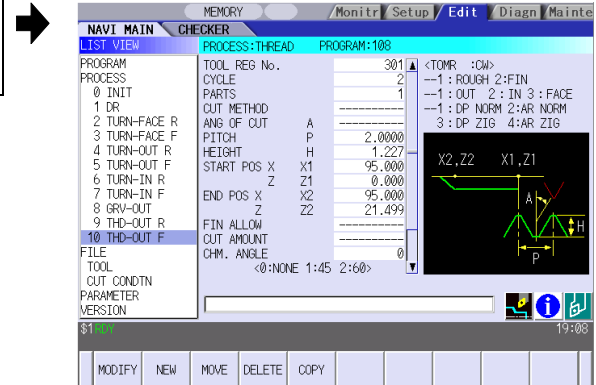
(2) Press the [NEW] menu key. ➡️ A blank process will be inserted into the cursor position.

The process mode selection screen will be displayed in the OPERATION VIEW area, and the OPERATION VIEW area will turn active.
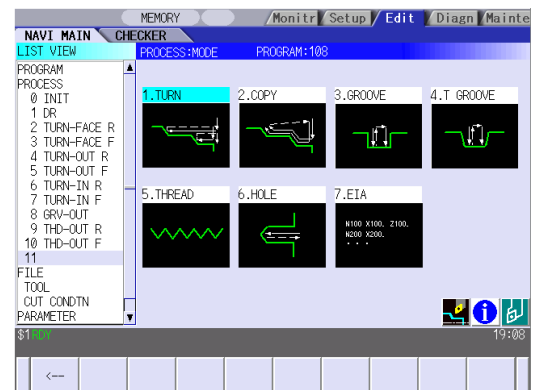
(3) Select the process mode with the cursor or the numerical value input. ➡️
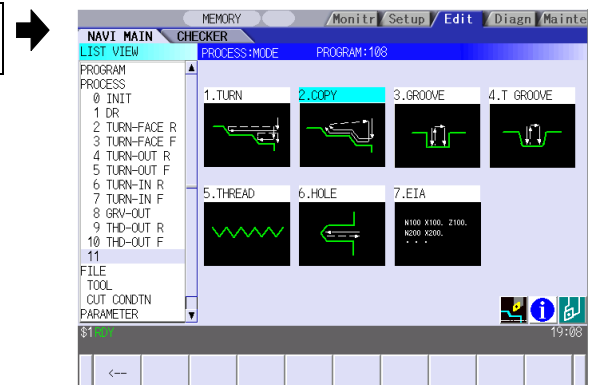
(4) | Press the [INPUT] key. ➡ The contents in the OPERATION VIEW area will change into those of the selected process mode.

The selected process mode will be displayed at the cursor position in the LIST VIEW area.
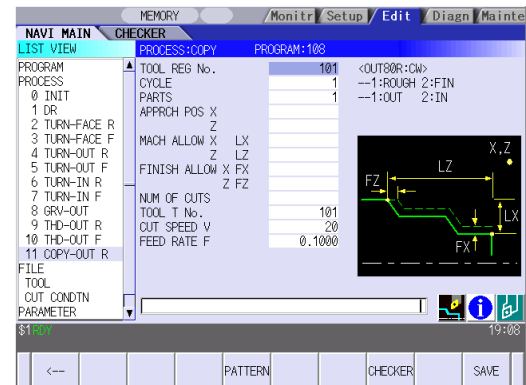
(Note) If the [←] menu key is pressed during adding the process, the screen will return to the state before pressing the [NEW] menu key (state of the 1).

4.3.4   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	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

No.	Display item	Details	Setting range
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
12	END POS Z		-9999.9999 to 9999.9999inch

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 Process

(1) Turning process screen
The parameters for the turning process are input on this screen.
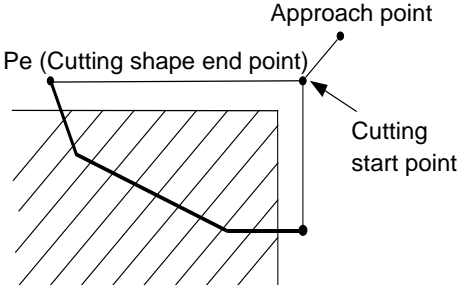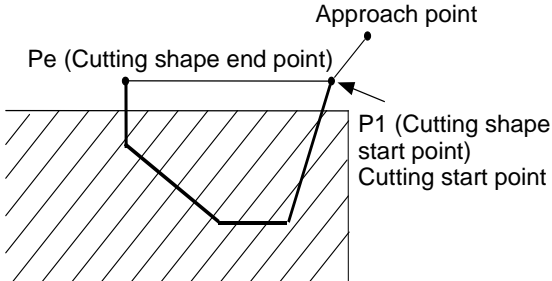
Screen layout

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
5	APPRCH POS Z		-9999.9999 to 9999.9999inch

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
7	FINISH ALLOW Z (FZ)		0.0000 to 9999.9999inch
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.
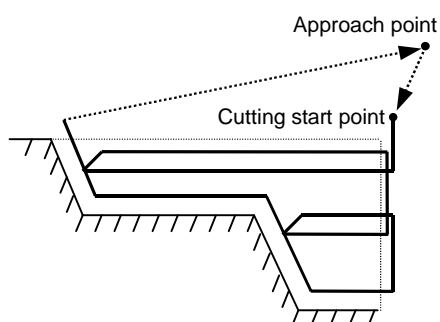
[CLOSE type]
The tool is retracted tracing the cutting shape.
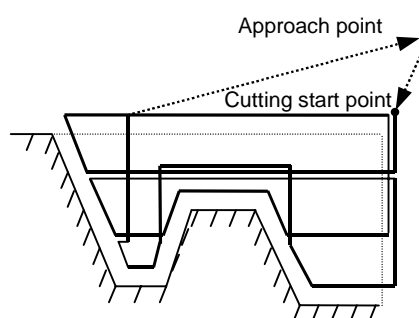
(Note) Tool path is not provided based on the tool shape (tool nose angle, front edge angle, etc.) Therefore, when the cutting shape is not incremented or decremented monotonously, take the tool shape into consideration to input the cutting shape.

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
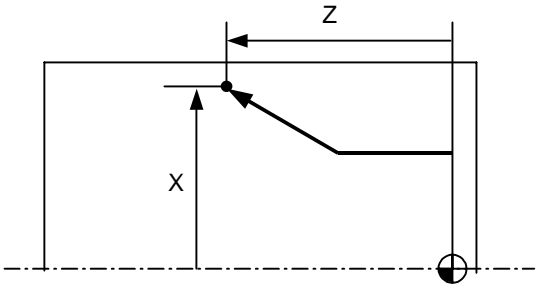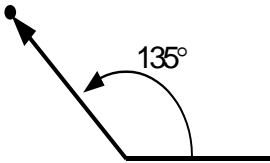
Screen display items

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
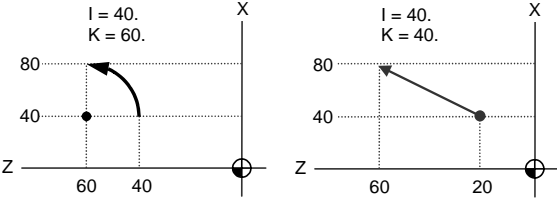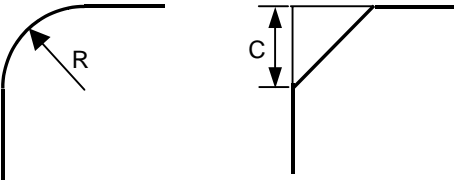
No.	Display item	Details	Setting range
3	D	Input right turn or left turn in respect to the vector at the end of the previous shape. 1: Left turn 2: Right turn (Note 1) When nothing is input, it is regarded as "contacting". (Note 2) Omittable. However, when the end point of the previous line, X and Z, 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.
		
		135°
		
		(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). I = 40. K = 60. I = 40. K = 40. (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 R 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	Insert the shape data in front of the cursor position.
		(Note) Not operatable at No.1 (machining start point).
2	LINE DELETE	Delete the shape data at the cursor position.
		(Note) Not operatable at No.1 (machining start point).
3	COPY	Copy 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	Clear the data at the cursor position.
6	RETURN	Returns to the turning screen.

4.3.6   Copy Cut Process

(1) Copy cutting screen
   The parameters for the copy cut process are input on this screen.

Screen layout

Screen display items

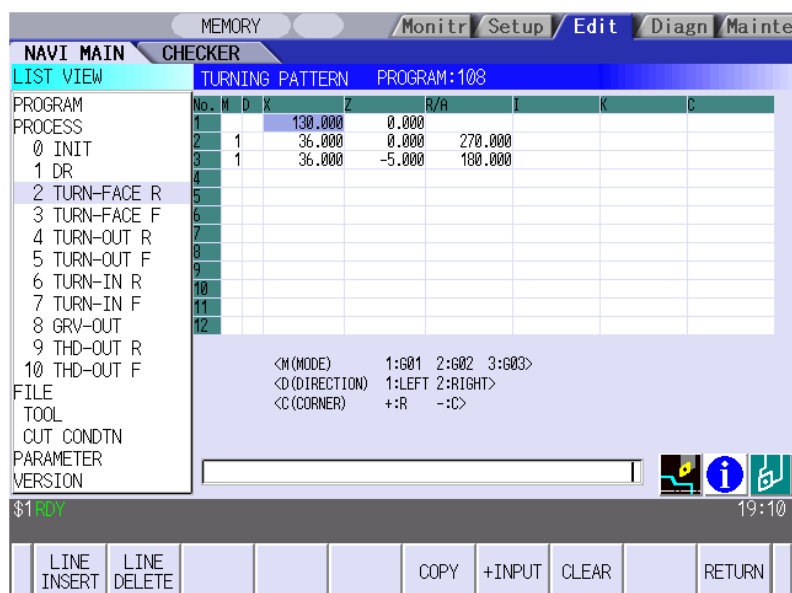| No.	Display item	Details	Setting range
1	TOOL REG No.	Input the registration No. of the tool to be used. Use the No. registered in the tool file.	101 to 150 601 to 650
2	CYCLE	Input the machining method. <1: Rough machining> Cuts into the cutting area gradually. Leaves the finishing allowance for the 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
5	APPRCH POS Z		-9999.9999 to 9999.9999inch
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
7	MACH ALLOW Z (LZ)	Input the allowance in Z axis direction for the rough machining.	0.0001 to 9999.9999inch
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
9	FINISH ALLOW FZ (FZ)		0.0000 to 9999.9999inch
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	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 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.
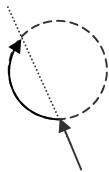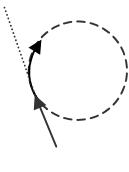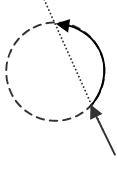
Screen layout
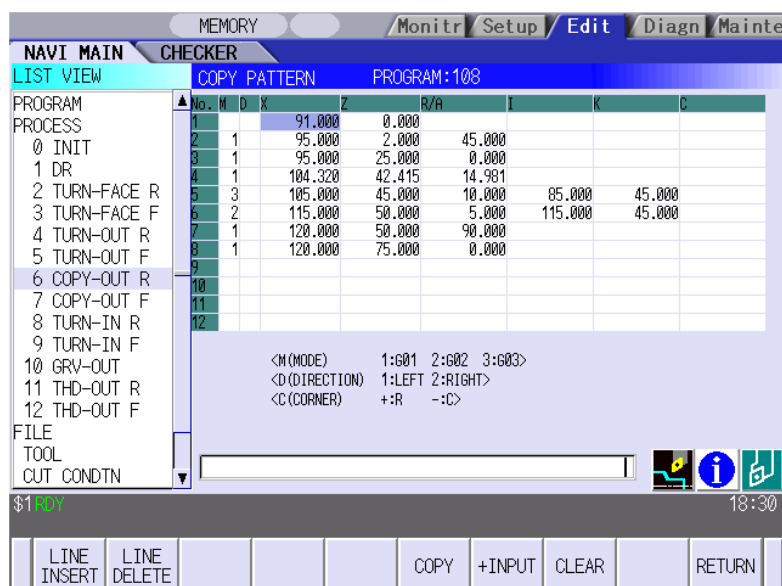
Screen display items

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

Menus

No.	Menu	Details
1	LINE INSERT	Insert the shape data in front of the cursor position.
		(Note) Not operatable at No.1 (machining start point).
2	LINE DELETE	Delete the shape data at the cursor position.
		(Note) Not operatable at No.1 (machining start point).
3	COPY	Copy 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	Clear 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.
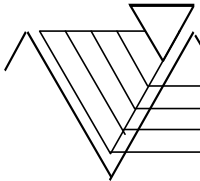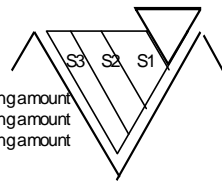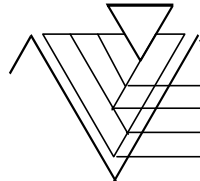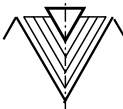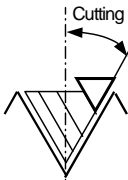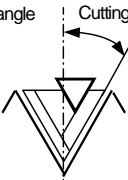
Screen layout

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 [Constant depth-normal] [Constant area-normal] Single cutting amount Single cutting amount Single cutting amount S3 S2 S1 [Constant depth-zigzag] [Constant area-zigzag] Single cutting amount Single cutting amount Single cutting amount S3 S1 S2	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. Cutting edge angle Cutting edge angle Cutting edge angle = 0 Cutting edge angle ≠ 0 Cutting edge angle ≠ 0	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.<table><tr><td>M METER</td><td>UN UNIFY</td><td>W WIT</td><td>PF PT PS PIPING</td><td>NPT PIPING</td><td>TM TRAP.30°</td><td>TW TRAP.29°</td></tr></table>	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. $d_n = d_1(\sqrt{n} - \sqrt{(n-1)})$ d_1: 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.
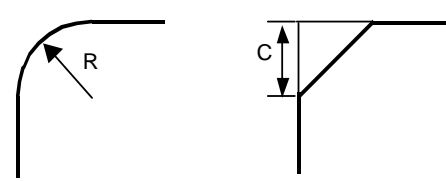
4.3.8   Grooving Screen

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 R 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.
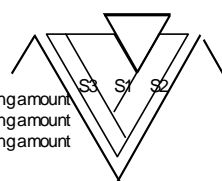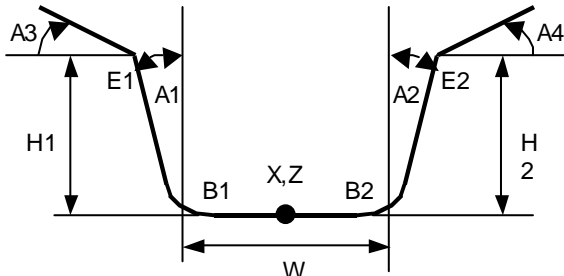
Screen layout

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
14	ENTR R-COR (E2)	Input the right corner amount of trapezoidal groove entrance. Positive value: Corner R Negative value: Corner C	-9999.9999 to 9999.9999inch
15	BOT L-COR (B1)	Input the left corner amount of trapezoidal groove bottom. Positive value: Corner R Negative value: Corner C	
16	BOT R-COR (B2)	Input the right corner amount of trapezoidal groove bottom. Positive value: Corner R Negative value: Corner C	
17	FIN ALLOW	Input the finishing allowance of the groove for the rough machining.	0.000 to 99999.999mm 0.0000 to 9999.9999inch

No.	Display item	Details	Setting range
18	CUT AMOUNT	Input the cut amount.	0.001 to 99999.999mm 0.0001 to 9999.9999inch
19	TOOL T No.	Input the turret No. (or ATC No.) of the tool being set, as well as the compensation No. When tool registration No. is specified, tool No. registered in the tool file is automatically set.	1 to 999999
20	TOOL WIDTH	Input the tool width of the respective tool. When tool registration No. is specified, tool width registered in the tool file is automatically set.	0.001 to 999.999mm 0.0001 to 99.9999inch
21	CUT SPEED V	Input the cutting speed. When tool registration No. is specified, cutting speed is automatically set based on the contents in the tool file and cutting condition file.	1 to 9999 m/min 1 to 9999 feet/min
22	FEED RATE F	Input the feedrate. When tool registration No. is specified, feedrate is automatically set based on the contents in the tool file and cutting condition file.	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 inch/rev

Menus

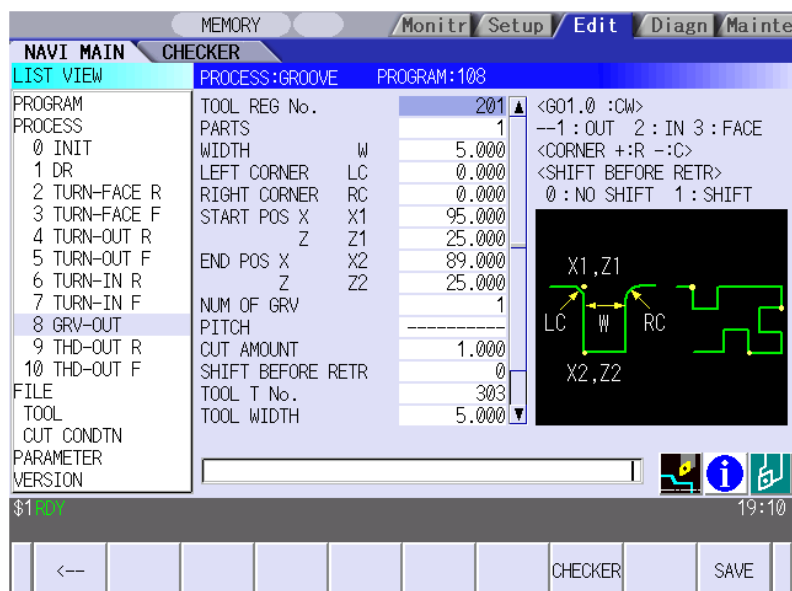| No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to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

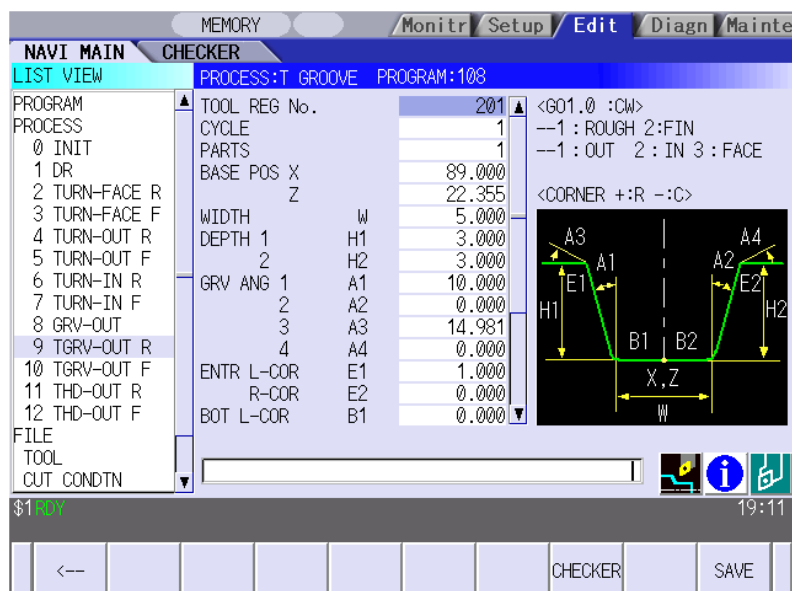| No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CHECKER	Displays the checker screen. Select this to check the set data.
3	SAVE	Saves the changes in the process. If illegal parameters are found in saving, an error will be displayed. When a parameter is incorrectly input, the cursor moves to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

4.3.11   EIA Screen

The EIA process is input on this screen.

Screen layout
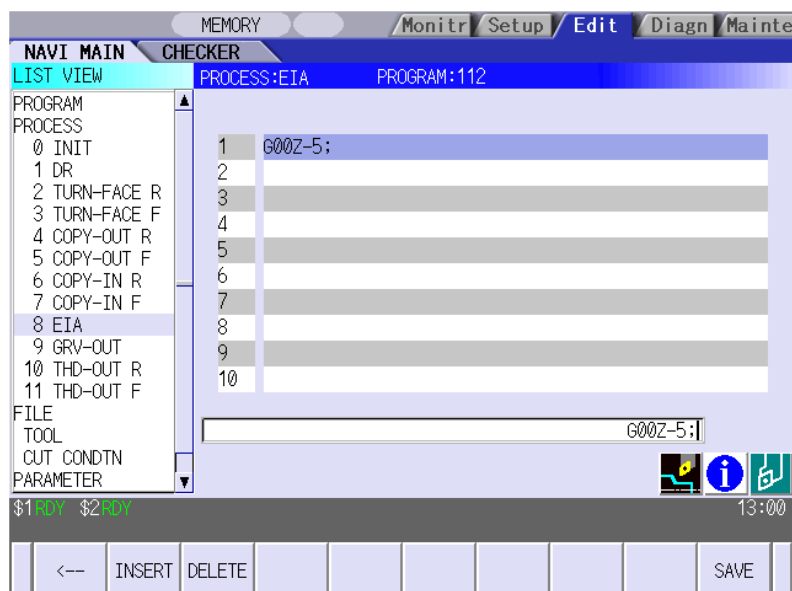
Screen display item

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.

4.4   Screens Related to File Editing

4.4.1   Tool File Screen

The tool data is registered on this screen. When [TOOL] is selected in the LIST VIEW area, this screen is displayed. The tool data includes the followings. Use the menu key to select one.
- • TURNING TOOLS
- • GROOVING TOOLS
- • THREADING TOOLS
- • DRILLS
- • TAPS
- • BUTTON TOOLS
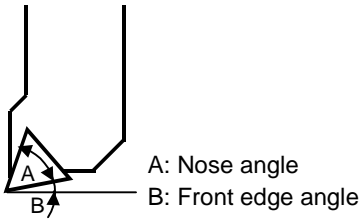
Screen layout
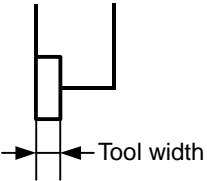
(Note) Menu for the currently selected tool is highlighted.

Screen display items
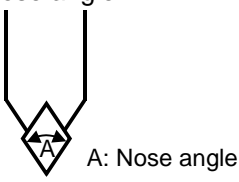
• TURNING TOOLS

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. A: Nose angle B: Front edge angle	0.001 to 180.000°
7	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW
8	L/R HAND	Input left/right hand for the tool.	1: Right 2: Left
9	TIP MATERIAL	Input the tip material.	Max. 4 alphanumerical characters

• 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. ←Tool 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 99.9999 inch
5	NOSE ANGLE	Input the tool nose angle.	0.001 to 180.000°
6	PITCH	Input the pitch.	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 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 sceren.
5	DRILL	Displays the drilling input screen.
6	TAP	Displays the tapping input screen.
7	BUTTON	Displays the bottun tool input screen.
8	SAVE	Saves the changes in the tool file.

4.4.2   Cutting Condition File Screen

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
4		F	Input the feedrate for the rough turning machining.	1.00 to 9999.00feet/min
5	TURN F	V	Input the cutting speed for the finishing turning machining.	
6		F	Input the feedrate for the finishing turning machining.	Feedrate:
7	GRV R	V	Input the cutting speed for the rough grooving machining.	0.0001 to 999.9999
8		F	Input the feedrate for the rough grooving machining.	mm/rev
9	GRV F	V	Input the cutting speed for the finishing grooving machining.	0.00001 to 99.99999 inch/rev
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.

4.5   Screen Related to the Parameters

4.5.1   Parameter Screen

The parameters for the machining program are input on this screen. When [PARAMETER] is selected in the LIST VIEW area, this screen is displayed.

Screen layout

Screen display items

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
5 (105)	TOOL FIX RET POS X	Input the tool change position in the machine coordinate system.	-99999.999 to 99999.999mm
6 (106)	TOOL FIX RET POS Z	This is valid when fixed point is selected for the tool change position. 	-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

No.	Display item	Details	Setting range
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
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

No.	Display item	Details	Setting range
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
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	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
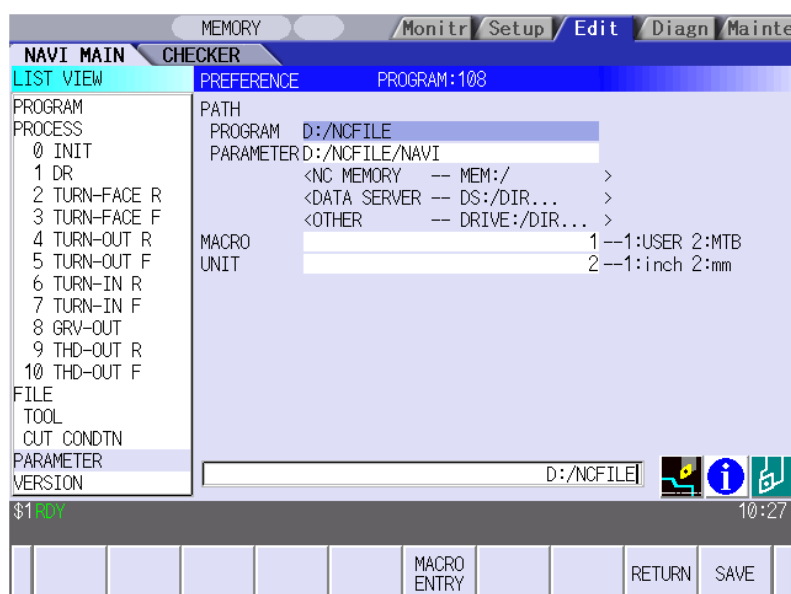
Screen display items

No.	Display item	Details	Setting range
1	PATH PROGRAM	Set the path to the folder in which NC program is saved.	(Drive name) : (Folder name)
2	PATH PARAMETER	Set the path to the folder in which tool file, cutting condition file and parameter file are saved.	
3	MACRO	Set the macro program mode. 1: User Macro 2: MTB Macro	1,2
4	UNIT	Set the unit for data input. 1: inch 2: mm	1,2

Menus

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

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

Version

4.7   Program Checker Screen

Tool paths of a NC program are graphically displayed on this screen.

Program Checker screen will be appeared by pressing ☐ or ☐ when MAIN screen is displayed. Program

Checker screen will also be appeared by clicking the checker icon ☐ .

Screen layout

GRAPHIC AREA

PROCESS

MESSAGE AREA   SCALE
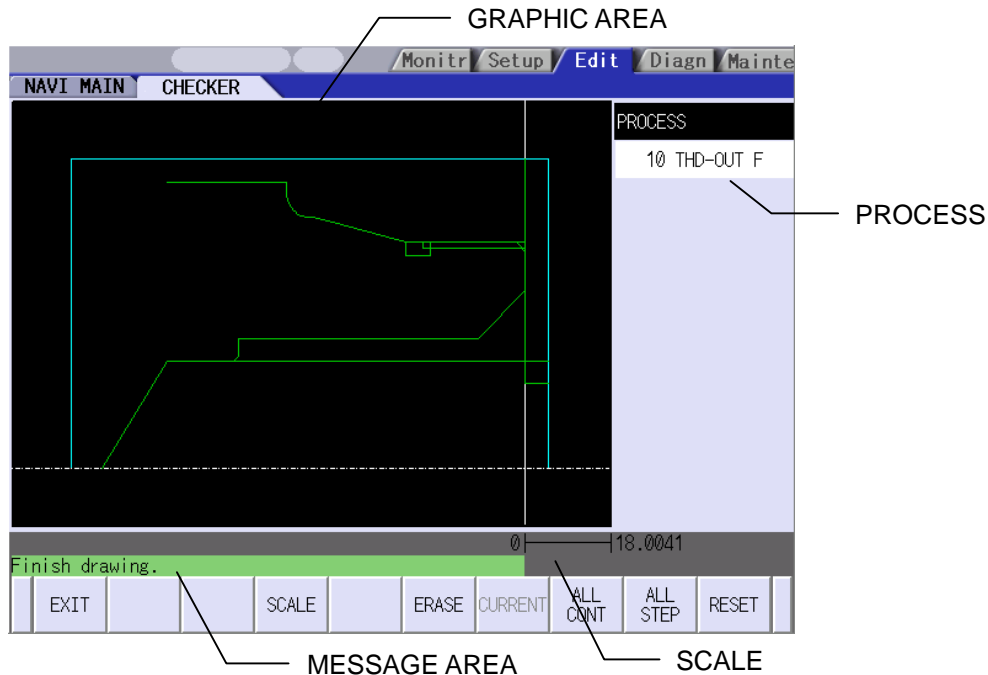
Screen display items

No.	Display item	Details
1	GRAPHIC AREA	The workpiece shape and the machining shape are graphically displayed within this area. Items and their display colors appeared on the screen are as follows: 1) Machining shape --- Green 2) Workpiece --- Light blue
2	PROCESS	The name of the process of which machining shape are currently displayed is indicated here.
3	SCALE	Scale value of the graphic display area is indicated.
4	MESSAGE AREA	Messages on graphic display of the machining shape are appeared here.

Main menus

No.	Menu	Details
1	EXIT	Terminates the Program Checker, and then closes the screen.
2	SCALE	This menu is used 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.
3	ERASE	Deletes the drawing data.
4	CURRENT	Tool paths of the currently selected process are displayed.
5	ALL CONT	Tool paths of the entire processes are displayed.
6	ALL STEP	Tool paths of each process are displayed one at a time.
7	RESET	Reset the graphic display of the tool paths.

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.
6	↓	Moves down the scale frame. The same function can also be achieved by pressing ↓ key.
7	←	Moves the scale frame toward the left. The same function can also be achieved by pressing ← key.
8	→	Moves the scale frame toward the right. The same function can also be achieved by pressing → key.
9	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.
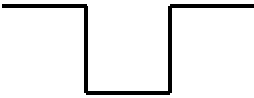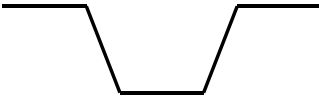
(Note 2) The changed position data will be stored while the power is ON. Therefore, it is possible to perform another graphic display at the same position.

(Note 3) 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 cut machining, the shape data is displayed up to the error point.

Examples of graphic drawings

[Turning / Copy cut]	[Threading]
[Grooving]	[Trapezoidal grooving]
[Hole drilling --- Drilling ---]	[Hole drilling --- Tapping ---]
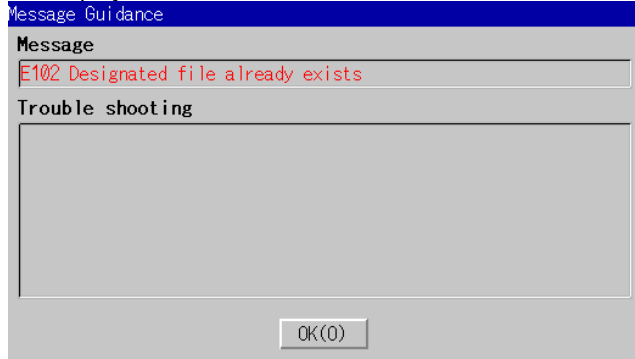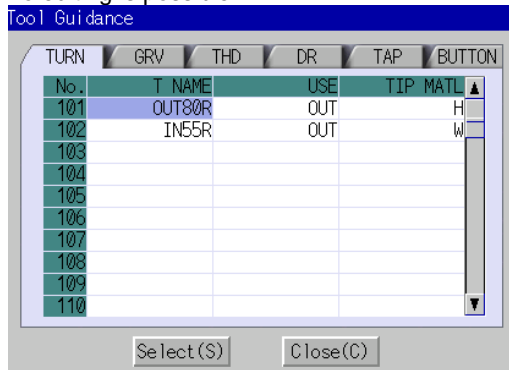
4.8   Guidance Function

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 ⬚ 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
Primary data of the tool data registered in the tool file is displayed on this screen.

Screen layout
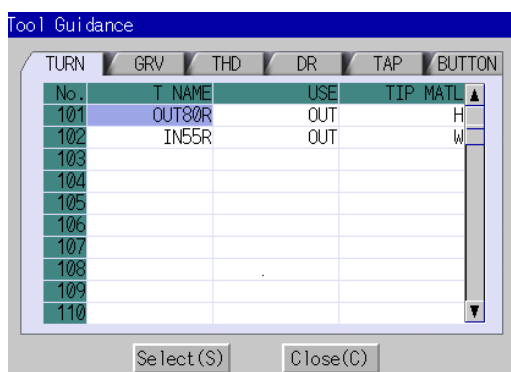
Screen display items

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	This 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

*Macro program is registered only in the memory of 700 series in which NAVI LATHE is installed.

NC program

NC program

O100 ( )
:
G65 P○○○
G65 P×××
G65 P△△△
:
:

Macro program

For grooving
O○○○○ ( )
G0 X#1 Y#2;
:
:

For threading
O××× ( )
G0 X#1 Y#2;
:
:

For hole drilling
O△△△ ( )
G0 X#1 Y#2;
:
:
:

Macro call

For tool change
O****( )
T#1;
:
:

Macro call

NAVI LATHE

Program input/output

File program

Tool file

Cutting condition file

Parameter program

#○○=1;
:
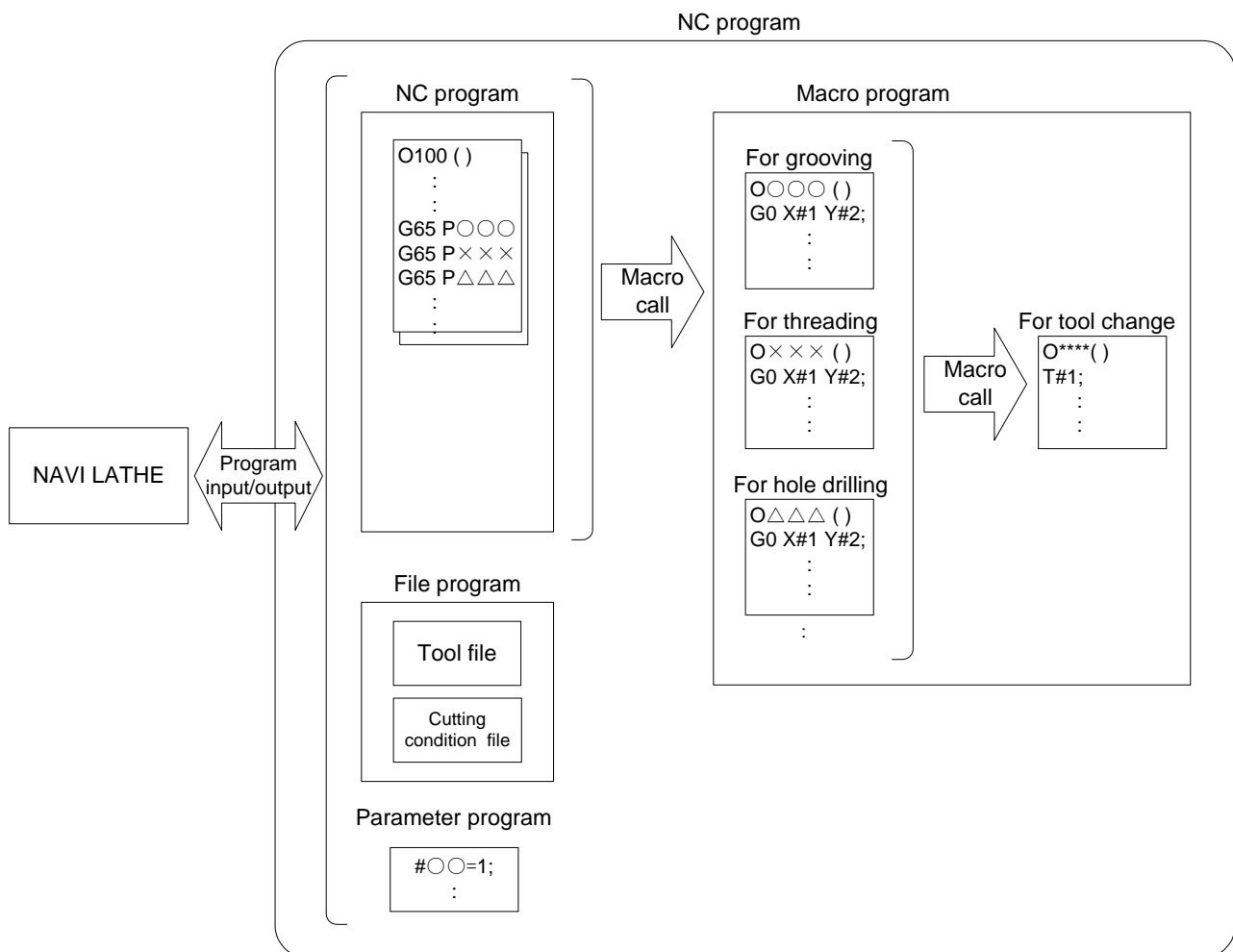
5.1   NC Program

NC program generated by NAVI LATHE is limited with its program No. to 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); ● ● ●	

Process start comment

Process		Comment	Remarks
Initial setting		(NAVI-INIT);	The symbol which indicates the
Turning		(NAVI-TURN-****)	machining area is set in the **** part.
Copy cutting		(NAVI-COPY-****)	OUT: Outer diameter
Threading		(NAVI-THD-****)	IN: Inner diameter
Trapezoidal grooving		(NAVI-TGRV-****)	FACE: Front face
Hole drilling	Drilling	(NAVI-HOLE-DRILL);	
	Pecking	(NAVI-HOLE-PECK);	
	Boring	(NAVI-HOLE-BORE);	
	Tapping	(NAVI-HOLE-TAP);	
EIA process		(NAVI-EIA);	
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;	Movement to the tool change position, T command
		G96 S_ M3(4) ;	
		G0 X_ Z_ F_;	Movement to the approach point
		G41(42);	Nose R compensation mode ON
		G71(72) U(W)_ R_ H_;	
		G71(72) P_ Q_ U_ W_;	
		N_ G1 X_ Z_;	Start point of the cutting shape
		• • •	
		N_ G1 X_ Z_;	End point of the cutting shape
		N_ G65 P9105 C;	Move. to the safe profile clearance pos
		G40;	Nose R compen. mode cancel
	FIN	G65 P9102 A B C;	Movement to the tool change position, T command
		G96 S_ M3(4) ;	
		G0 X_ Z_ F_;	Movement to the approach point
		G41(42);	Nose R compensation mode ON
		G70 P_ Q_;	
		GOTO N_	Start point of the cutting shape
		N_ G1 X_ Z_;	
		• • •	End point of the cutting shape
		N_ G1 X_ Z_;	
		N_ G65 P9105 C;	Move. to the safe profile clearance pos
		G40;	Nose R compen. mode cancel

Copy cutting	ROUGH	G65 P9102 A_ B_ C_;	Movement of the tool change position, T command
		G96 S_ M3(4) ;	
		G0 X_ Z_ F_;	Movement to the approach point
		G41(42);	Nose R compensation Mode ON
		G73 U_ W_ R_;	
		G73 P_ Q_ U_ W_;	
		N_ G1 X_ Z_;	Start point of the cutting shape
		• • •	
		N_ G1 X_ Z_;	End point of the cutting shape
		N_ G65 P9105 C;	Move. to the safe profile clearance pos
		G40;	Nose R compen. mode cancel
	FIN	G65 P9102 A_ B_ C_;	Movement of the tool change position, T command
		G96 S_ M3(4) ;	
		G0 X_ Z_ F_;	Movement to the approach point
		G41(42);	Nose R compensation Mode ON
		G70 P_ Q_;	
		GOTO N_	
		N_ G1 X_ Z_;	Start point of the cutting shape
		• • •	
		N_ G1 X_ Z_;	End point of the cutting shape
		N_ G65 P9105 C;	Move. to the safe profile clearance pos
		G40;	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	G65 P9120 A_ B_ C_ D_ E_ F • • • Z_;	Common in drilling, pecking, boring and tapping.
	Pecking		
	Boring		
	Tapping		
EIA process		• • • ;	
End process		G65 P9190;	
		M#158;	

(Note) Macro program No. (P***) in the table is used when user macro is selected. For the macro program No. used when manufacturer macro is selected, refer to the section "5.4 Macro Program".

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

Deleting a block in the NC program process unit (process start comment to end comment) is no problem. Note that editing the program with NAVI LATHE may be disabled, if a block of the process start comment, process data or process end comment is deleted. Therefore, do not delete a block of the process start comment, process data or process end comment.

(2) Inserting block

Inserting a block into the processes of the NC program (between the process end comment and next process start comment) is no problem.
When a block is inserted into the process of the NC program (between the process start comment and process end comment), the process can be edited with the NAVI LATHE. However, the inserted block is not recognized in most cases. Therefore, if the process into which a block has been inserted is edited with the NAVI LATHE, the block may be lost.

In response to the operating process (moving process, deleting process, copying process) with NAVI LATHE, an inserted block is operated as follows.

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

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

Before movement   After 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.
───────►

(NAVI-THD-OUT); G65 P9130 ····;   Process B (/NAVI); (NAVI-GRV-OUT); G65 P9140 ····;   Process A M50; ◄──Inserted block (/NAVI);
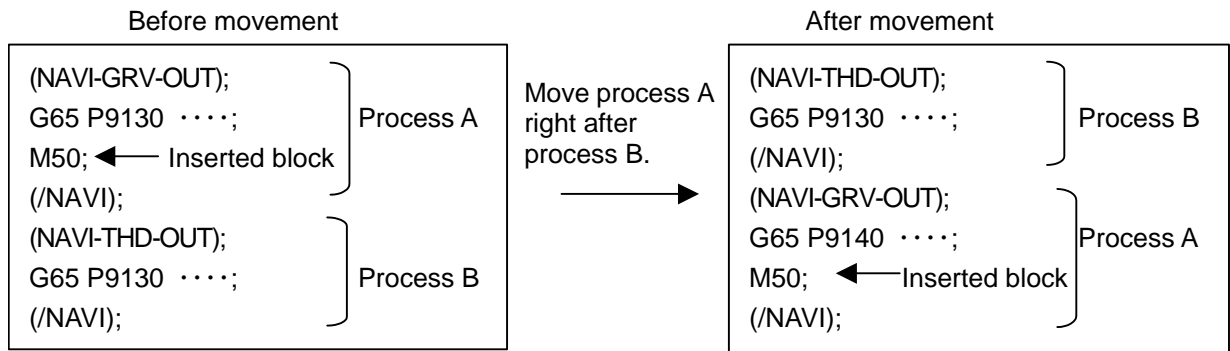
(Example2)   Moving process (An inserted block exists between the processes.)

Before movement   After 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.
───────►

M50; ◄── Inserted block (NAVI-THD-OUT); G65 P9130 ····;   Process B (/NAVI); (NAVI-GRV-OUT); G65 P9140 ····;   Process A (/NAVI);
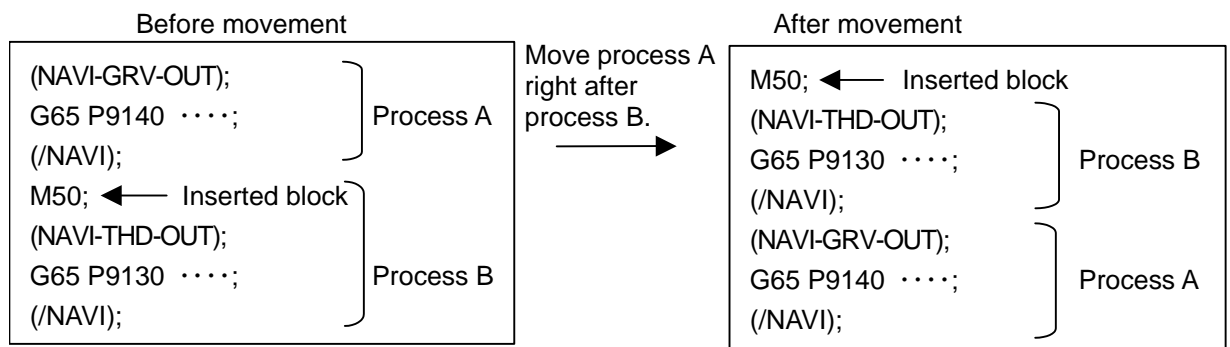
(3) Changing process data

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

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

5.4   Macro Program

This program is called from the NC program.
(Macro program will be registered in the memory of 700 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	100019160	TGROOVE MACRO
7	Macro program for hole drilling process	9170	100019170	HOLE MACRO
8	Macro program for tool change	9102	100019102	TOOL CHANGE
9	Macro program for end process	9190	100019190	END-MACRO
10	Macro program for parameter setting	-	100019103	PARAM-SET-MACRO
11	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

In operating the NAVI LATHE and the format of the machining program created with the NAVI LATHE, the following restrictions are applied to the 700 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 turning machining		
	Compound type fixed cycle for turning machining (Type II)		
	Variable command 200 sets or more		
	Conner chamfering / Corner R		
Parameter specifications	Parameter name	Setting details	Remarks
	#1026 base-I	X	Address of the axes configuring a plane is specified.
	#1028 base-K	Z	
	#1265 ext01	bit0: 0 bit2: 0	Select the conventional format for the following command format. • Compound type fixed cycle for turning machining • Hole drilling fixed cycle MITSUBISHI CNC special format cannot be used.
	#1076 AbsInc	1	Absolute command and incremental command are switched by the address code.
	#1013 axname	1:X 2:Z	Address of each axis name is specified.
	#1014 incax	1:U 2:W	Specify the incremental command axis name address for each axis.
	#1019 dia	1	The diameter specification axis is selected by the X axis.

6. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

Division	Specifications		Remarks
Parameter specifications	#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	
	#1228 aux12(bit5)	0	Select the workpiece coordinate system for the coordinates during constant surface speed.
	#1181 G96_ax	1	Specify the 1st axis for the axis to be targeted for constant surface speed control.
	#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.
	#8112 G04P DECIMAL PNT-P	1	The decimal point command for G04 address P is validated.
	#8102 COLL. ALM OFF	1	This is validated when executing the machining program created with NAVI LATHE.

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.

(Continued to the next page)

(Continued from the previous page)

Division	Message	Details
Process editing	E201 Process number over	The number of processes exceeded 100.
Turning/ Copy cutting	E211 Geometry record number entry over	Exceeded the number of records currently registered.
	E212 Geometry maximum record number over	The maximum number of records (35) is exceeded.
	E213 Geometry record number entry over	The record No. is illegal.
	E214 I,K agreement with angle (line number)	Linear I,K and angle are contradictory.
	E215 No end point on circumference (line number)	The end point does not exist on the circumference.
	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.
	E218 Corner C error (line number)	Corner C error
	E219 Corner R error (line number)	Corner R error
	E220 shape input error (line number)	Shape input error
	E221 Last line has corner R/C (line number)	There is corner R/C in the last line.
	E222 Start point error (line number)	Start point error
	E223 Corner no move	The block following corner R/C is not a movement command.
	E224 Corner short	When issuing corner C/R command, the movement distance in the next block is smaller than corner C/R.
	E225	
	E226	
	E227	
	E228	
	E229	
Threading	E231 H < FIN ALLOW	"Thread height < finishing allowance" is applied.
	E232 H < CUT AMOUNT	"Thread height < cut 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 < cut 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.

(Continued from the previous page)

Division	Message	Details
Trapezoidal grooving	E251 W < TOOL WIDTH	"Groove width < tool width" is applied.
	E252 H< CUT AMOUNT	"Groove height < cut 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	
	E257	
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	Cut 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.
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 in Contour – Free occurs, 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.		Code	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	0	Variable for operation
#155	#455		END POS Z	-9999.9999 to 9999.9999inch	0	Variable for operation
#156	#456		END M CODE	1 to 3	1	Variable for operation
#157	#457		OUTSIDE DIA	0.001 to 99999.999mm	100	Variable for operation
#158	#458		+Z	-99999.999 to 99999.999mm	100	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	50.000mm	Common
#163	#463	104	TOOL TURNING CL Z	0.0001 to 9999.9999inch	1.9685inch	Common
#164	#464	105	TOOL FIX RET POS X	-99999.999 to 99999.999mm	0	Common
#165	#465	106	TOOL FIX RET POS Z	-9999.9999 to 9999.9999inch	0	Common
#166	#466	107	SAFE PROFILE CL OD	0.001 to 99999.999mm	2.000mm	Common
#167	#467	108	SAFE PROFILE CL FACE	0.0001 to 9999.9999inch	0.0787inch	Common
#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

Variable No.		Para No.	Parameter name	Setting range	Standard value	Remarks
User macro mode	MTB macro mode					
#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

⚠ 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 #177
MTB macro mode	#450 to #477

When NC program mode is user macro mode, do not use common variables (#150 to #177). 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

Appendix 2.1 Machining Drawing
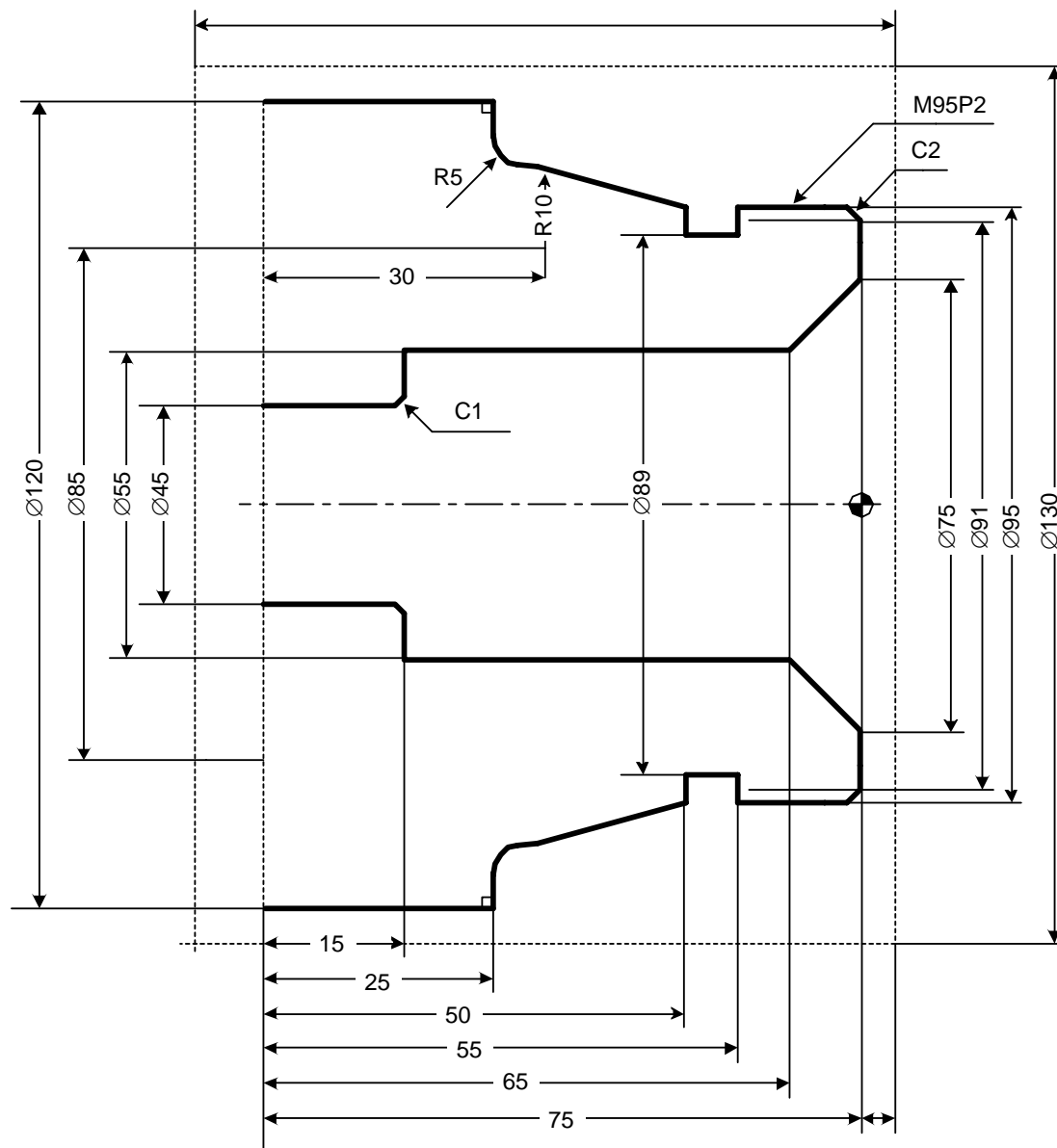
Appendix 2.2   Process Table

Processes are shown below.

Process	Machining	Tool
1	DR	DR
2	TURN-FACE R	OUTR
	TURN-FACE F	OUTR
3	TURN-OUT R	OUTR
	TURN-OUT F	OUTR
4	TURN-OUT R	INR
	TURN-OUT F	INR
5	GRV-OUT	GO
6	THD-OUT R	TOMR
	THD-OUT F	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.

LIST VIEW
PROGRAM
PROCESS
0   INIT
FILE

LIST VIEW
PROGRAM
PROCESS
0   INIT
FILE

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.

LIST VIEW
PROGRAM
PROCESS
0   INIT
1   DR
FILE

6. Process 2 Turning face rough machining (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	

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.

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

7. Process 2 Turning face finishing machining (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

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.

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

8. Process 3 Turning outer diameter rough machining (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 outer diameter finishing machining (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.

10. Process 4 Turning inner diameter rough machining (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
FILE

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 inner diameter finishing machining (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 outer diameter machining (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	

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

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.

13. Process 6 Threading outer diameter rough machining (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 outer diameter finishing machining (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

Revision History

Date of revision	Manual No.	Revision details
Nov. 2005	IB(NA)1500146-A	First edition created.

Global service network

NORTH AMERICA FA Center
EUROPEAN FA Center
CHINA FA Center
KOREAN FA Center
ASEAN FA Center
HONG KONG FA Center
TAIWAN FA Center
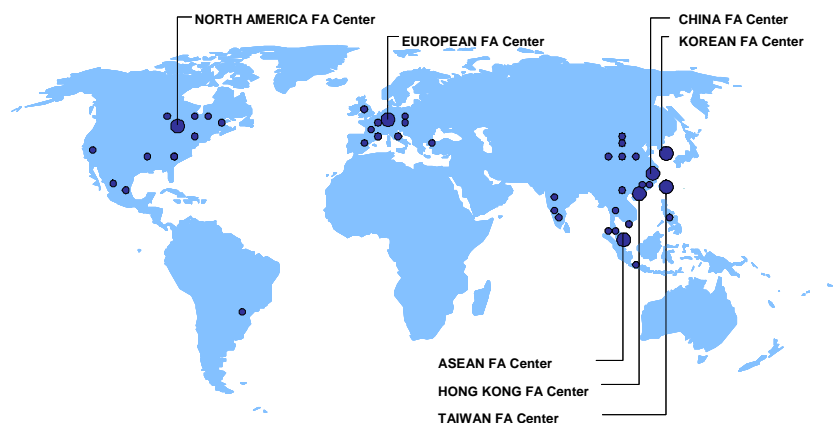
North America FA Center (MITSUBISHI ELECTRIC AUTOMATION INC.)

Illinois CNC Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL. 60061, U.S.A.
TEL: +1-847-478-2500 (Se   FAX: +1-847-478-2650 (Se

California CNC Service Center
5665 PLAZA DRIVE, CYPRESS, CA. 90630, U.S.A.
TEL: +1-714-220-4796   FAX: +1-714-229-3818

Georgia CNC Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GA., 30097, U.S.A.
TEL: +1-678-258-4500   FAX: +1-678-258-4519

New Jersey CNC Service Center
200 COTTONTAIL LANE SOMERSET, NJ. 08873, U.S.A.
TEL: +1-732-560-4500   FAX: +1-732-560-4531

Kentucky CNC Service Center
8025 PRODUCTION DRIVE, FLORENCE, KY., 41042, U.S.A.
TEL: +1-859-342-1700   FAX: +1-859-342-1578

Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500   FAX: +1-269-673-4092

Ohio CNC Service Satellite
62 W. 500 S., ANDERSON, IN., 46013, U.S.A.
TEL: +1-847-478-2608   FAX: +1-847-478-2690

Texas CNC Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TX. 76051, U.S.A.
TEL: +1-817-251-7468   FAX: +1-817-416-1439

Canada CNC Service Center
4299 14TH AVENUE MARKHAM, ON. L3R OJ2, CANADA
TEL: +1-905-475-7728   FAX: +1-905-475-7935

Mexico CNC Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662   FAX: +52-55-9171-7698

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

Brazil MITSUBISHI CNC Agent Service Center
(AUTOMOTION IND. COM. IMP. E EXP. LTDA.)
ACESSO JOSE SARTORELLI, KM 2.1 18550-000 BOITUVA – SP, BRAZIL
TEL: +55-15-3363-9900   FAX: +55-15-3363-9911

European FA Center (MITSUBISHI ELECTRIC EUROPE B.V.)

Germany CNC Service Center
GOTHAER STRASSE 8, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0   FAX:+49-2102486-591

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

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

Lyon CNC Service Satellite

U.K CNC Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-282-846   FAX:-44-1707-278-992

Italy CNC Service Center
ZONA INDUSTRIALE VIA ARCHIMEDE 35 20041 AGRATE BRIANZA, MILANO ITALY
TEL: +39-039-60531-342   FAX: +39-039-6053-206

Spain CNC Service Satellite
CTRA. DE RUBI, 76-80 -APDO.420 08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236   FAX:

Turkey MITSUBISHI CNC Agent Service Center
(GENEL TEKNIK SISTEMLER LTD. STI.)
DARULACEZE CAD. FAMAS IS MERKEZI A BLOCK NO.43 KAT2 80270 OKMEYDANI ISTANBUL, TURKEY
TEL: +90-212-320-1640   FAX: +90-212-320-1649

Poland MITSUBISHI CNC Agent Service Center (MPL Technology Sp. z. o. o)
UL SLICZNA 34, 31-444 KRAKOW, POLAND
TEL: +48-12-632-28-85   FAX:

Wroclaw MITSUBISHI CNC Agent Service Satellite (MPL Technology Sp. z. o. o)
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53   FAX: +48-71-333-77-53

Czech MITSUBISHI CNC Agent Service Center
(AUTOCONT CONTROL SYSTEM S.R.O.   )
NEMOCNICNI 12, 702 00 OSTRAVA 2 CZECH REPUBLIC
TEL: +420-596-152-426   FAX: +420-596-152-112

ASEAN FA Center (MITSUBISHI ELECTRIC ASIA PTE. LTD.)

Singapore CNC Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE   159943
TEL: +65-6473-2308   FAX: +65-6476-7439

Thailand MITSUBISHI CNC Agent Service Center (F. A. TECH CO., LTD)
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 12,14   RAMA III RD BANGPONGPANG, YANNAWA, BANGKOK 10120. THAILAND
TEL: +66-2-682-6522   FAX: +66-2-682-6020

Malaysia MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN MALAYSIA
TEL: +60-3-5631-7605   FAX: +60-3-5631-7636

JOHOR MITSUBISHI CNC Agent Service Satellite
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
NO. 16, JALAN SHAHBANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218   FAX: +60-7-557-3404

Indonesia MITSUBISHI CNC Agent Service Center
(PT. AUTOTEKNINDO SUMBER MAKMUR)
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144   FAX: +62-21-3917-164

India MITSUBISHI CNC Agent Service Center (MESSUNG SALES & SERVICES PVT. LTD.)
B-36FF, PAVANA INDUSTRIAL PREMISES M.I.D.C., BHOASRI PUNE 411026, INDIA
TEL: +91-20-2711-9484   FAX: +91-20-2712-8115

BANGALORE MITSUBISHI CNC Agent Service Satellite
(MESSUNG SALES & SERVICES PVT. LTD.)
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119   FAX: +91-80-532-0480

Delhi MITSUBISHI CNC Agent Parts Center (MESSUNG SALES & SERVICES PVT. LTD.)
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32ND MILESTONE   GURGAON 122001, INDIA
TEL: +91-98-1024-8895   FAX:

Philippines MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM CORPORATION)
UNIT No.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416   FAX: +63-2-807-2417

Vietnam MITSUBISHI CNC Agent Service Center (SA GIANG TECHNO CO., LTD)
47-49 HOANG SA ST. DAKAO WARD, DIST.1 HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763   FAX: +84-8-910-2593

China FA Center (MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD.)

China CNC Service Center
2/F., BLOCK 5 BLDG.AUTOMATION INSTRUMENTATION PLAZA, 103 CAOBAO RD.   SHANGHAI 200233, CHINA
TEL: +86-21-6120-0808   FAX: +86-21-6494-0178

Shenyang CNC Service Center
TEL: +86-24-2397-0184   FAX: +86-24-2397-0185

Beijing CNC Service Satellite
9/F, OFFICE TOWER1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,   DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830   FAX: +86-10-6518-8030

China MITSUBISHI CNC Agent Service Center
(BEIJING JIAYOU HIGHTECH TECHNOLOGY DEVELOPMENT CO.)
RM 709, HIGH TECHNOLOGY BUILDING NO.229 NORTH SI HUAN ZHONG ROAD, HAIDIAN DISTRICT , BEIJING 100083, CHINA
TEL: +86-10-8288-3030   FAX: +86-10-6518-8030

Tianjin CNC Service Satellite
RM909, TAIHONG TOWER, NO220 SHIZILIN STREET, HEBEI DISTRICT, TIANJIN, CHINA 300143
TEL: -86-22-2653-9090   FAX: +86-22-2635-9050

Shenzhen CNC Service Satellite
RM02, UNIT A, 13/F, TIANAN NATIONAL TOWER, RENMING SOUTH ROAD, SHENZHEN, CHINA 518005
TEL: +86-755-2515-6691   FAX:+86-755-8218-4776

Changchun Service Satellite
TEL: +86-431-50214546   FAX: +86-431-5021690

Hong Kong CNC Service Center
UNIT A, 25/F RYODEN INDUSTRIAL CENTRE, 26-38 TA CHUEN PING STREET, KWAI CHUNG, NEW TERRITORIES, HONG KONG
TEL: +852-2619-8588   FAX: +852-2784-1323

Taiwan FA Center (MITSUBISHI ELECTRIC TAIWAN CO., LTD.)

Taichung CNC Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688   FAX: +886-4-2359-0689

Taipei CNC Service Satellite
TEL: +886-4-2359-0688   FAX: +886-4-2359-0689

Tainan CNC Service Satellite
TEL: +886-4-2359-0688   FAX: +886-4-2359-0689

Korean FA Center (MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD.)

Korea CNC Service Center
DONGSEO GAME CHANNEL BLDG. 2F. 660-11, DEUNGCHON-DONG KANGSEO-KU SEOUL, 157-030 KOREA
TEL: +82-2-3660-9607   FAX: +82-2-3663-0475

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

MITSUBISHI CNC

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

MODEL	700 Series
MODEL CODE	008—366
Manual No.	IB-1500146 (ENG)