



Changes for the Better

MITSUBISHI CNC

700/70 Series

**Simple Programming Function
NAVI MILL Instruction Manual**








MELDAS is a registered trademark of Mitsubishi Electric Corporation.
Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

Introduction

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

Details described in this manual

CAUTION

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder takes precedence over this manual.
-  Items not described in this manual must be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Confirm with the specifications issued by the machine tool builder before starting to use.
-  Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.
-  Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

Refer to the following documents.

MITSUBISHI CNC 700/70 Series	Instruction Manual	IB-1500042
MITSUBISHI CNC 700/70 Series	Setup Manual	IB-1500124
MITSUBISHI CNC 700/70 Series	Programming Manual (M system)	IB-1500072

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 MILL, 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.

(Continued on next page)

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

- ⚠ Always set the stored stroke limit. Failure to set this could result in collision with the machine end.
- ⚠ Always turn the power OFF before connecting/disconnecting the I/O device cable. Failure to do so could damage the I/O device and NC unit.

4. Items related to screen operation

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

NC program mode	Variables used by NAVI MILL
User macro mode	#150 to #179
MTB macro mode	#450 to #479

When NC program mode is user macro mode, do not use common variables. 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 "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.

5. Items related to operation

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


















(Continued on next page)

CAUTION

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
1.1 System Outline	1
1.2 Input Procedures	2
1.3 Screen Configuration	3
1.4 Starting NAVI MILL	4
1.5 Setting up NAVI MILL	5
2. FUNCTIONS OF DISPLAY AREA	7
2.1 LIST VIEW Area	8
2.2 OPERATION VIEW Area	10
2.3 Setting Area	11
2.4 Message Area	11
2.5 Menu Display Area	11
3. BASIC OPERATIONS	12
3.1 Changing Active View	12
3.2 Changing Screen	12
3.3 Setting Data	14
3.4 Switching Windows	17
3.5 Switching Selection Tags	17
3.6 Inputting Operations	18
4. SCREEN SPECIFICATIONS	19
4.1 Starting NAVI MILL	19
4.2 Screen Related to the Program	20
4.2.1 Program Edit Screen	20
4.3 Screens Related to the Process Edit Functions	24
4.3.1 Process List Screen	24
4.3.2 Multiple Parts Screen	26
4.3.3 Operating Process	30
4.3.4 Process Mode Selection Screen	36
4.3.5 Initial Condition Setting	39
4.3.6 Hole Drilling	43
4.3.7 Face Cutting	55
4.3.8 Contour Cutting	59
4.3.9 Pocket	69
4.3.10 EIA Screen	78
4.4 Screens Related to File Editing	79
4.4.1 Tool File Screen	79
4.4.2 Cutting Condition File Screen	81
4.5 Screen Related to the Parameters	83
4.5.1 Parameter Screen	83
4.5.2 PREFERENCE Screen	86
4.6 Screen Related to the Version	88
4.6.1 Version Screen	88
4.7 Program Checker Screen	89
4.8 Guidance Function	94
4.8.1 Tool Guidance Screen	95

5. PROGRAM SPECIFICATIONS.....	96
5.1 NC Program.....	97
5.1.1 Output Method for NC Program.....	97
5.1.2 Restrictions.....	100
5.2 File Program.....	101
5.3 Parameter Program.....	101
5.4 Macro Program.....	101
6. TOOL FUNCTIONS.....	102
6.1 Tool Change Command (M6) and T-Command.....	102
6.2 Editing Tool Number and Next Tool Number.....	103
7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS.....	104
8. ALARM MESSAGE.....	105
8.1 Error Message.....	105
8.2 Warning Message.....	108
8.3 Operation Message.....	109
APPENDIX 1. VARIABLES USED IN NAVI MILL.....	110
APPENDIX 2. PROGRAMMING EXAMPLE.....	112
Appendix 2.1 Machining Drawing.....	112
Appendix 2.2 Process Table.....	113
Appendix 2.3 Condition Setting.....	113
Appendix 2.4 Creating Program.....	114

1. OUTLINE

1.1 System Outline

This manual is an instruction manual for NAVI MILL for 700/70 (hereafter NAVI MILL).

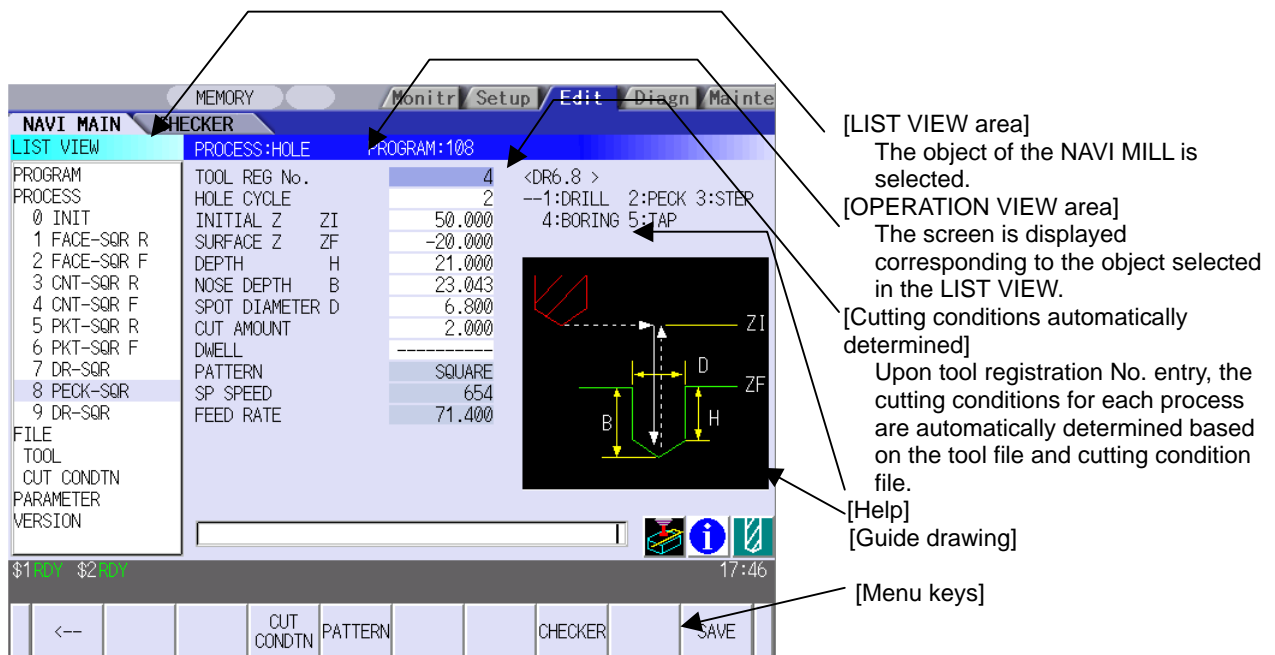
The part program for the vertical machining center (three axes of X, Y and Z) is created with the NAVI MILL.

(1) The following machining processes can be edited.

- Hole drilling (Drilling, pecking, step, boring, tapping, helical boring)
- Face cutting (Circle, square)
- Contour cutting (Circle, square, free)
- Pocket machining (Circle, square, L pattern, U pattern, track)
- EIA

(2) The tool file and the cutting condition file are provided and the cutting conditions 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.



(Note) The operation screen size is fixed to 800(width) x 600(length).

(4) Program Checker enables the tool paths 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.

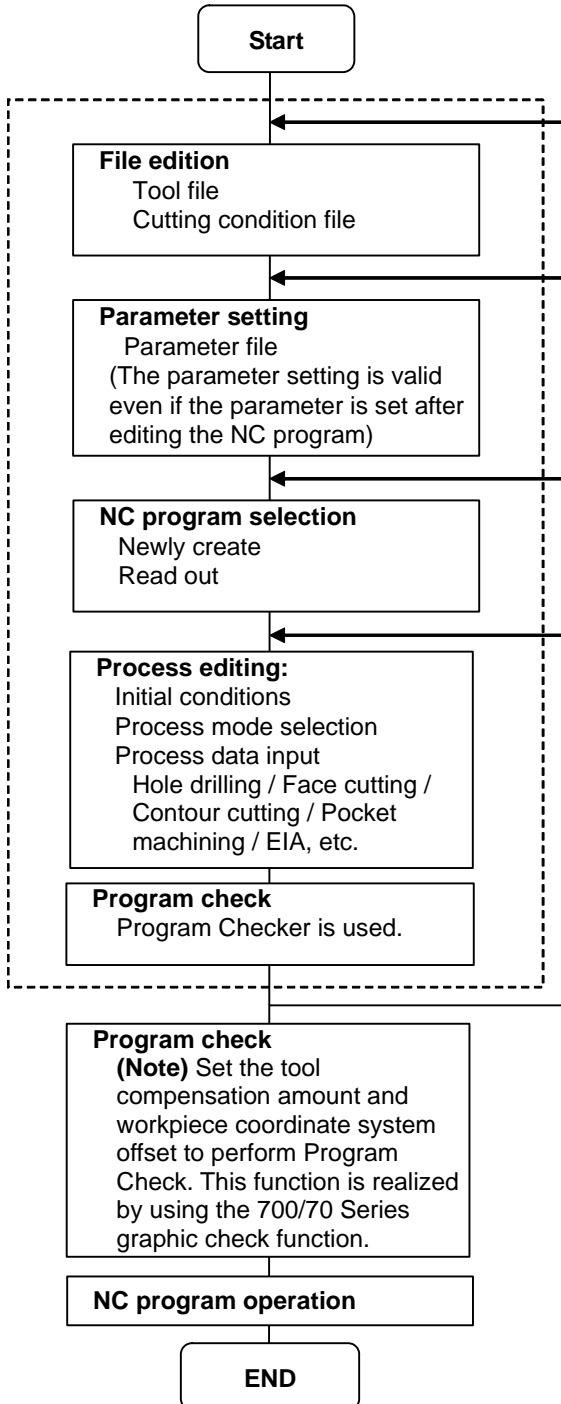
(6) Part program is a macro-program-based NC program. Commands can be added between processes from the edit screen of the standard MELDAS 700/70 Series.

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

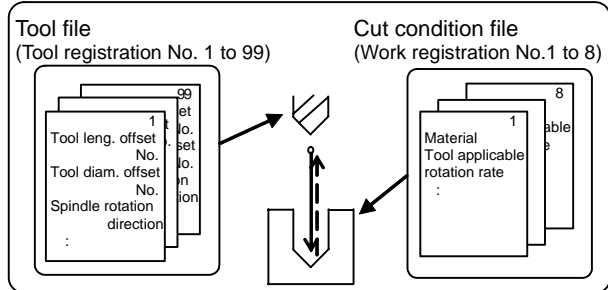
1.2 Input Procedures

The input procedure for the NAVI MILL is shown below.

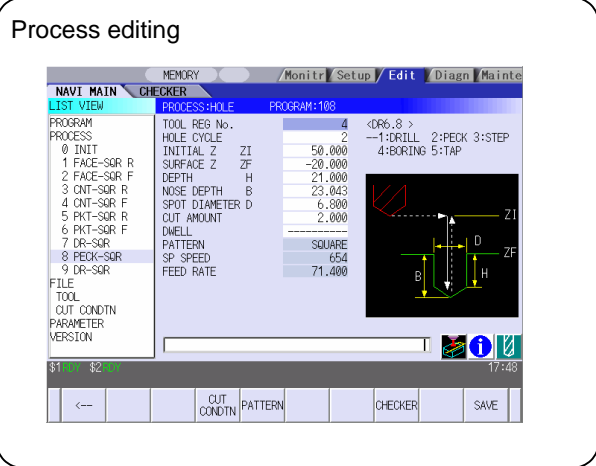
The part  is operated on the NAVI MILL's screen.



Supplements



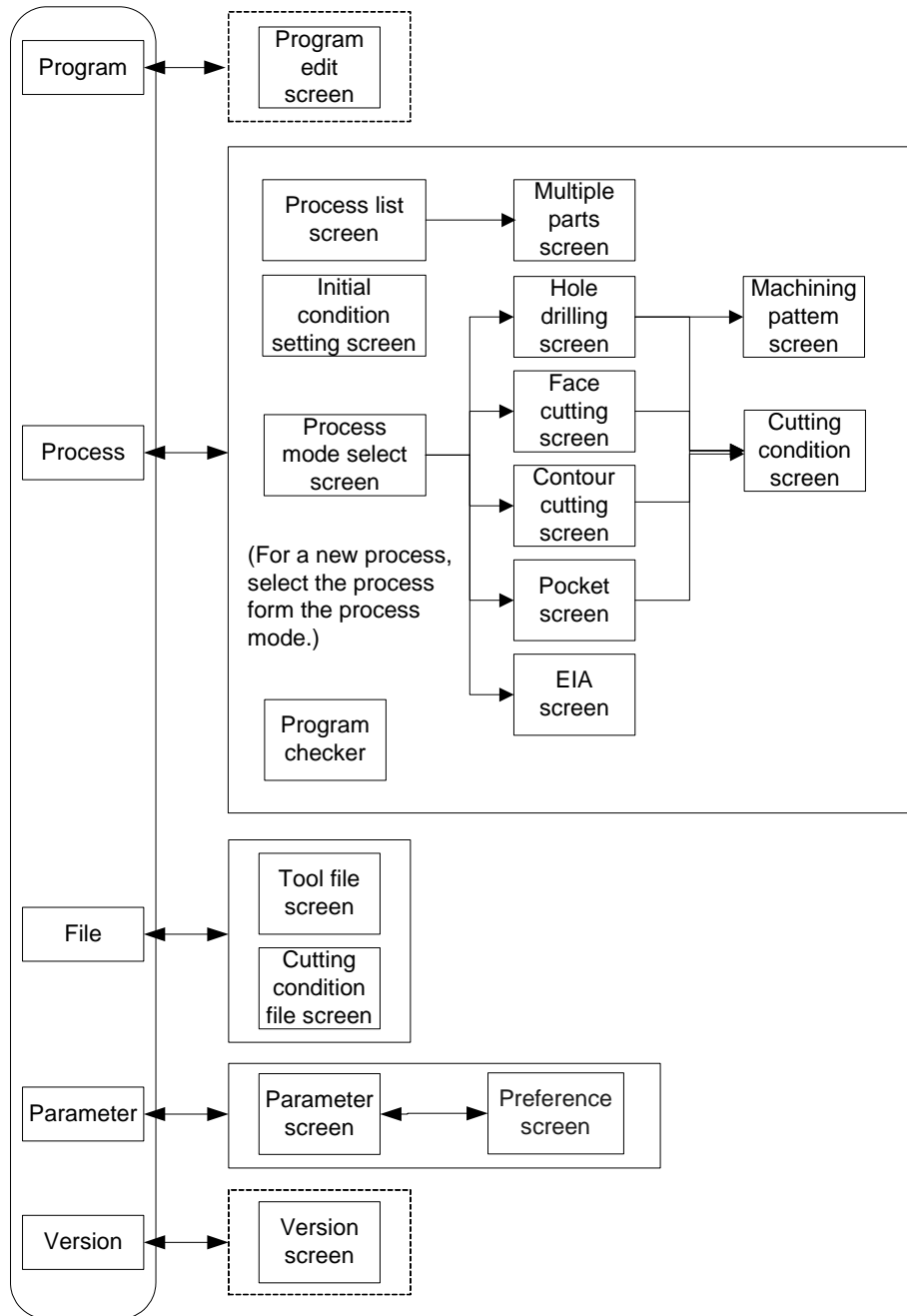
- Parameter setting
- M0 output • M6 output • M19 output • Next tool preparation, etc.
 - Common parameters for hole drilling process (Z clearance, tap selection)
 - Common parameters for face cutting process (XYZ clearance, Z approach G code)
 - Common parameters for contour cutting / pocket machining process



The NC program, tool file, cutting condition file and parameter file transferred to the CNC can be edited on Magicpro-NAVI MILL for 60S.


1.3 Screen Configuration

The screen configuration for the MILL NAVI is shown below.



Screen name	Details
Title screen	This screen is displayed when the power is turned ON.
Program edit screen	The process program is read out and saved, etc.
Process list screen	Tool information and cutting conditions for each process of a machining program are listed.
Multiple parts screen	A NC program for the multiple parts machining is generated.
Process mode select screen	The process mode (hole drilling, etc.) is selected.
Initial conditions setting screen	The initial conditions for the process program are set.
Hole drilling screen	The parameters for the hole drilling process are input.
Hole drilling machining pattern screen	The parameters related to the machining pattern of the hole drilling process are input.
Cutting condition screen	The cutting conditions by the process are input.
Face cutting screen	The parameters for the face cutting process are input.
Contour cutting screen	The parameters for the contour cutting process are input.
Contour cutting pattern screen	The parameters related to the machining pattern of the contour cutting process are input.
Pocket screen	The parameters of the pocket process are input.
Pocket pattern screen	The parameters related to the machining pattern of the pocket 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 (speed rate) by each process are input.
Parameter screen	The tool code and miscellaneous parameter are set.
Preference screen	The system is set up.
Version screen	The version data of the NAVI MILL is displayed.
Program checker	The tool paths of a NC program is graphically traced.

1.4 Starting NAVI MILL

Select  function, then [NAVI] menu to display NAVI MILL screen.

Program edit screen is displayed once when the power is turned ON. Then, whatever the screen previously selected with NAVI MILL is displayed thereafter.

1.5 Setting up NAVI MILL

Part program output from NAVI MILL 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 MILL's reference files are saved, as well as the unit for data input, have to be specified prior to NAVI MILL operations.

NAVI MILL setup items

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

NAVI MILL setup procedures

(1) Open PARAMETER screen.

(2) Set "999 MAINTEN" to 1.

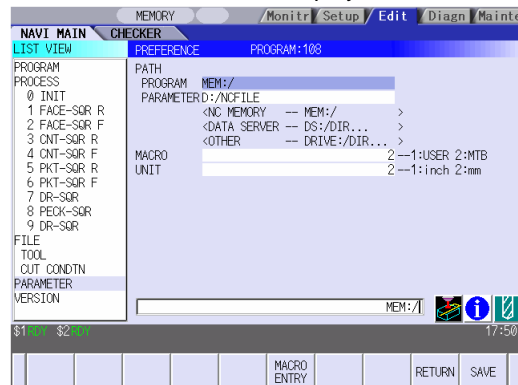


[PREFERENCE] menu is displayed.

(3) Press [PREFERENCE] menu.



PREFERENCE screen is displayed.



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

- (5) ➡ "OK?(Y/N)" message is displayed.
- (6) ➡ Macro program is registered in NC system.
- (7)
- (8)
- (9) ➡ When the unit is changed, turn the power OFF and ON again.

(Addendum)

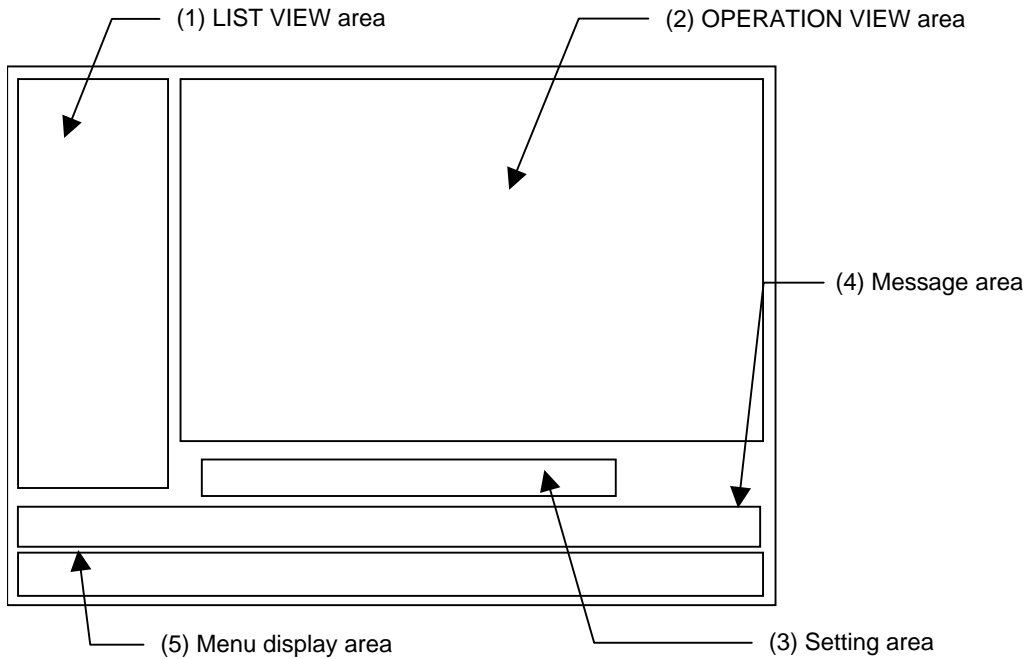
- Always carry out a macro program registration when setting up NAVI MILL or switching "MACRO" types.
- Change "PROGRAM PATH" and "PARAMETER PATH" when necessary.
- When "UNIT" is changed, turn the power OFF and ON again.
- If the tool file, cutting condition file and parameter file do not exist in "PARAMETER PATH" folder when the power is turned ON, those files are created by the system.

2. FUNCTIONS OF DISPLAY AREA

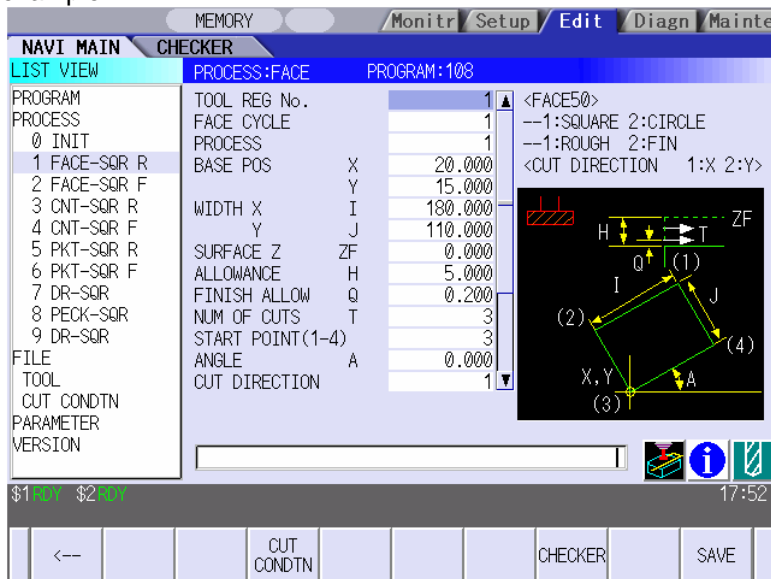
2. FUNCTIONS OF DISPLAY AREA

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

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

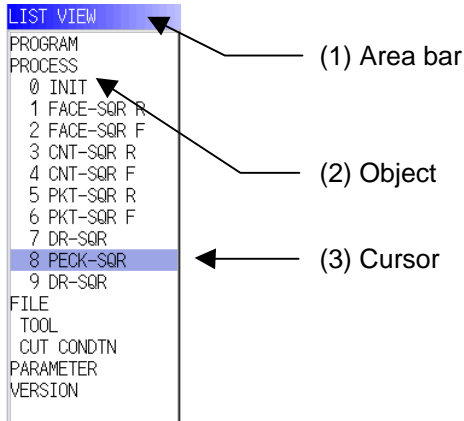


<Screen example>



2.1 LIST VIEW Area

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



(1) Area bar

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

(2) Objects

The list of objects that can be selected are displayed. The object is composed of the main object and the sub object that the main object is detailed. 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-LINE :	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 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 MILL.

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

(3) Cursors

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

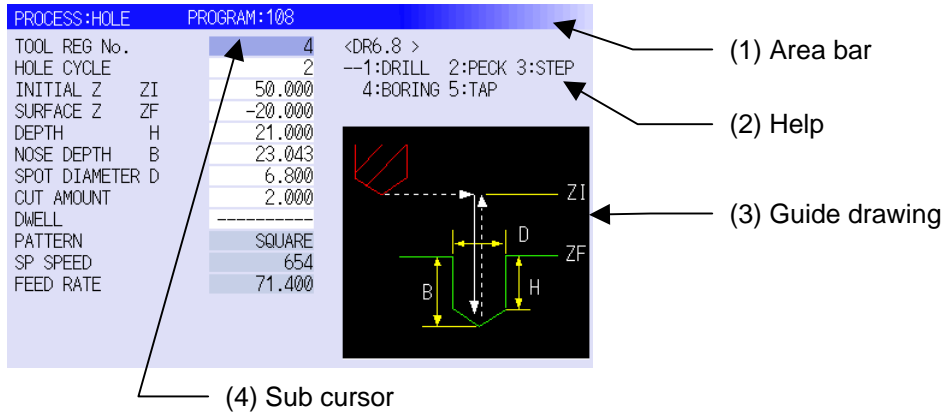
<Cursor Movement>

The cursor is moved using the cursor keys or a pointing device.

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

2.2 OPERATION VIEW Area

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



(1) Area bar

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

(2) Help

Quick reference on the setting items is displayed.

(3) Guide drawing

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

(4) Sub cursor

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

2.3 Setting Area

The value to be set to data is input.

2.4 Message Area

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

2.5 Menu Display Area

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

3. BASIC OPERATIONS

3.1 Changing Active View

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

3.2 Changing Screen

When the object is selected in the LIST VIEW area, the screen (contents in the OPERATION VIEW area) changes. (Refer to the section 2.1 LIST VIEW Area.)

Note that the screen cannot be changed while the OPERATION VIEW area is active.

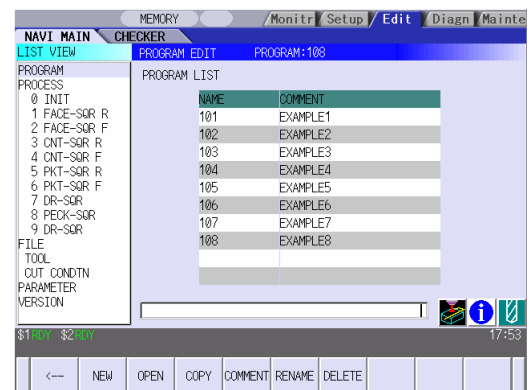
In such a case, press the [←] menu key or click "LIST VIEW" with a pointing device to turn the LIST VIEW area active.

Operation example

(1) Open the program edit screen.



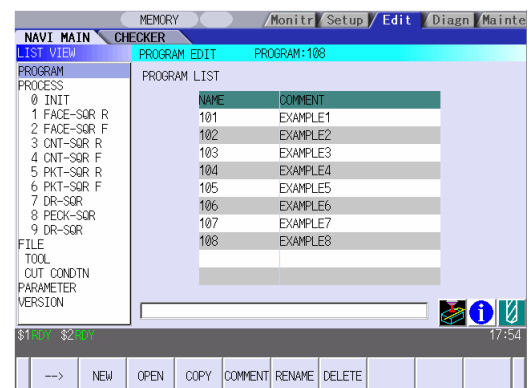
The OPERATION VIEW area is active.



(2) Press the [←] menu key.

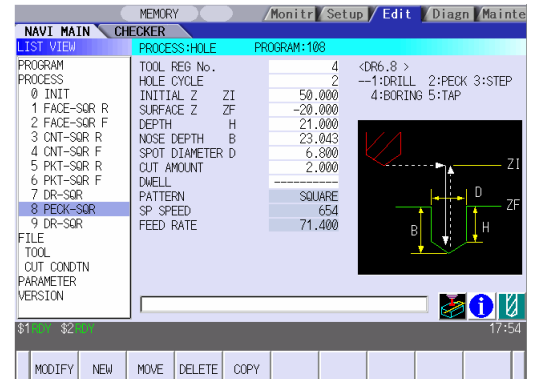


The LIST VIEW area will turn active.



(3) Select the object with the cursor key.

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



(4) Press the [MODIFY] menu key.

The OPERATION VIEW area will turn active.



3.3 Setting Data

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

The screenshot shows a CNC control interface with the following components:

- Top Menu:** MEMORY, Monitr, Setup, Edit, Diagn, Mainte
- NAVI MAIN CHECKER:** PROGRAM:108
- Left Menu:** LIST VIEW, PROGRAM, PROCESS, FILE, TOOL, CUT CONDTN, PARAMETER, VERSION
- Main Data Table:**

PROGRAM	TOOL REG No.	4	<DR6.8 >
PROCESS	HOLE CYCLE	2	--1:DRILL 2:PECK 3:STEP
0 INIT	INITIAL Z ZI	50.000	4:BORING 5:TAP
1 FACE-SQR R	SURFACE Z ZF	-20.000	
2 FACE-SQR F	DEPTH H	21.000	
3 CNT-SQR R	NOSE DEPTH B	23.043	
4 CNT-SQR F	SPOT DIAMETER D	6.800	
5 PKT-SQR R	CUT AMOUNT	2.000	
6 PKT-SQR F	DWELL	-----	
7 DR-SQR	PATTERN	SQUARE	
8 PECK-SQR	SP SPEED	654	
9 DR-SQR	FEED RATE	71.400	
- Diagram:** A technical drawing of a hole with dimensions ZI, ZF, D, B, and H. A sub-cursor is positioned over the 'DEPTH' field in the table.
- Bottom Bar:** \$1, \$2, 17:55, and buttons for CUT CONDTN, PATTERN, CHECKER, and SAVE.

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] [2] [3] [.] [4] [5] [6]

→ 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 moves in the data setting area.




1	2	3	4		5	6
---	---	---	---	--	---	---

(2) Press the [DETETE] key.  The character in front of the cursor is deleted.

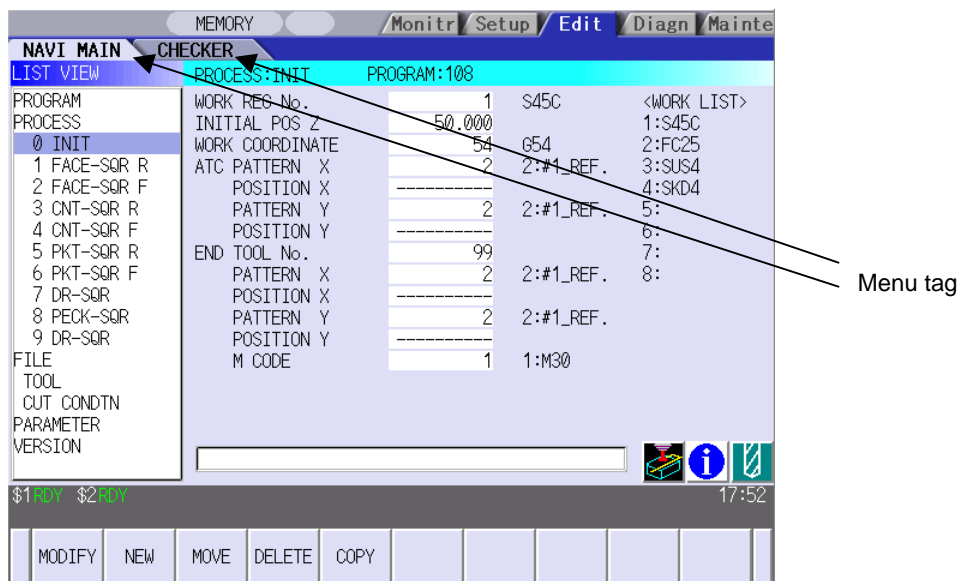
1	2	3		5	6
---	---	---	--	---	---

3.4 Switching Windows



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

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

3.5 Switching Selection Tags



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

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

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

3.6 Inputting Operations

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

Input method

Numeric values, function symbols, operators and parentheses () are combined and set in the data setting area.

The operation results appear when the [INPUT] key is pressed. Data for the currently selected setting item will be set when [INPUT] key is pressed again.

The contents in the data setting area are erased.

Examples of operator settings, and results		
Operation	Setting example	Operation results
Addition	=100+50	150.000
Subtraction	=100-50	50.000
Multiplication	=12.3*4	49.200
Division	=100/3	33.333
Function	=1.2*(2.5+SQRT(4))	5.4

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

Operation examples

- (1) Set as shown below, and press the [Input] key.
 =12*20 [Input] → The operation results appear in the data setting area.
 240 |
- (2) Press the [Input] key again. → Data for the selected setting item is set. The cursor moves to the next position.

Notes for using operators and functions

- Division : Zero division causes an error.
- Square root : If the value in the parentheses is negative, an error occurs.
- Triangle function : The unit of angle θ is degree ($^{\circ}$).
- Arc tangent : $-90 < \text{operation results} < 90$.

Restrictions

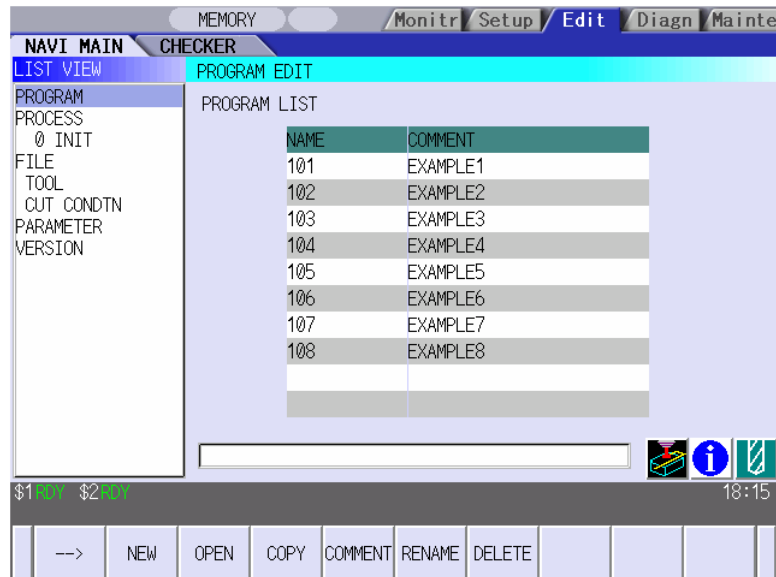
- Always use "=" for the first character.
- Do not use the following characters as the second character or last character.
 Invalid as second character: *, /,)
 Invalid as last character: *, /, (, +, -
- Make sure that the left parentheses and right parentheses are balanced.
- The 360° limit does not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

4. SCREEN SPECIFICATIONS

4.1 Starting NAVI MILL

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

Screen layout



At the initial start up of NAVI MILL, 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.

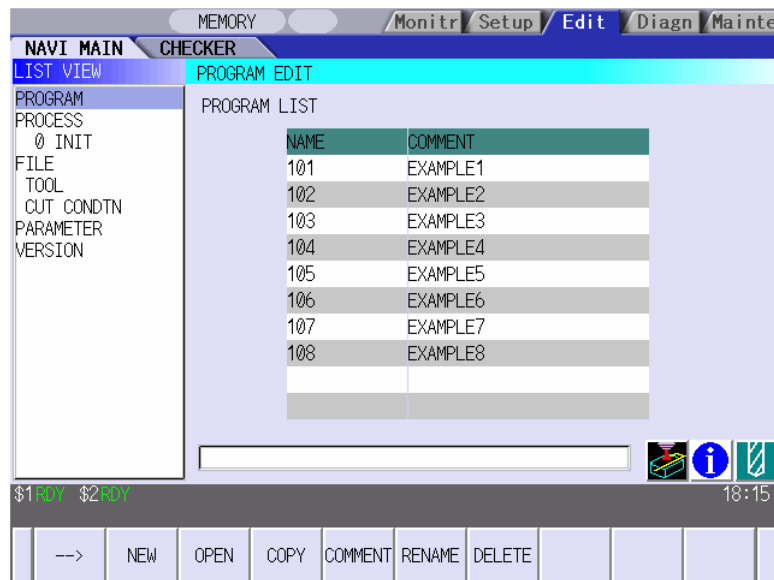
The process program is not selected.

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.

4. SCREEN SPECIFICATIONS

4.2 Screen Related to the Program

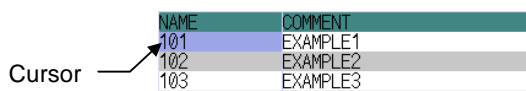
< Process displays >

Process name		Display character	Remarks
Hole drilling	Drilling	DR-****	The symbol (abbrev.) which indicates the machining pattern is applied to the "****" part. <ul style="list-style-type: none"> ● Random : RNDM ● Linear : LINE ● Arc : ARC ● Circle : CIR ● Square : SQR ● Grid : GRID
	Pecking	PECK-****	
	Step	STEP-****	
	Boring	BORE-****	
	Tapping	(TAP-****)	
	Helical boring	HLX-BORE ?	The symbol which indicates the machining type (rough / finishing) is applied to the "?" part. <ul style="list-style-type: none"> ● Rough : R ● Finishing : F
Face cutting	Square	FACE-SQR ?	The symbol which indicates the machining type (rough / finishing) is applied to the "?" part. <ul style="list-style-type: none"> ● Rough : R ● Finishing : F
	Circle	FACE-CIR ?	
Contour cutting	Square	CNT-SQR ?	
	Circle	CNT-CIR ?	
	Free	CNT-FREE ?	
Pocket	Square	PKT-SQR ?	
	Circle	PKT-CIR ?	
	L pattern	PKT-LPT ?	
	U pattern	PKT-UPT ?	
	Track	PKT-TRK ?	
EIA		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 →
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.

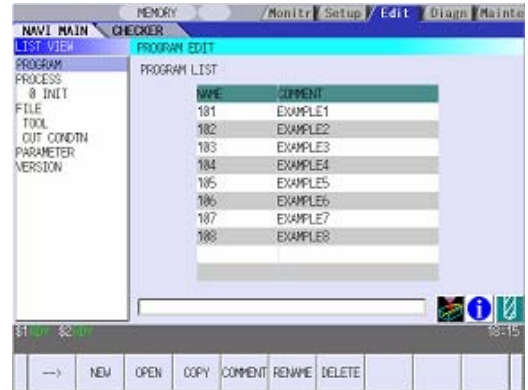
(Note 1) 1 to 7999 or 10000 to 99999999 can be set for the O No, and up to 18 alphanumeric characters can be set for the comment.

(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)

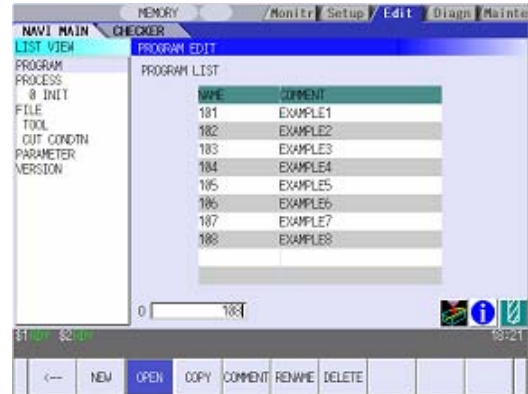
- (1) Select the [PROGRAM] in the LIST VIEW area.

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

The screenshot shows a software interface with a menu on the left and a data table on the right. The menu includes options like PROGRAM, PROCESS, FILE, TOOL, CUT COND TN, PARAMETER, and VERSION. The data table has columns for No., PCS, T NAME, T, NT, H, D, S, and F. The table contains 10 rows of process data.

No.	PCS	T NAME	T	NT	H	D	S	F
0	INIT	-----	---	0	---	---	---	---
1	FACE-SQR R	FACE50	1	0	1	1	720	360.000
2	FACE-SQR F	FACE50	1	0	1	1	720	360.000
3	CNT-SQR R	EM20	8	0	8	8	640	64.000
4	CNT-SQR F	EM20	8	0	8	8	640	64.000
5	PKT-SQR R	EM20	8	0	8	8	640	64.000
6	PKT-SQR F	EM20	8	0	8	8	640	64.000
7	DR-SQR	CD3	5	0	5	5	1050	70.000
8	PECK-SQR	DR6.8	4	0	4	4	654	71.400
9	DR-SQR	DC20	3	0	3	3	112	56.000

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

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	Specify the No. of tool to be used.	0 to 9999
4	NT	Specify the No. of tool to be used in the next process. According to the specified tool No., the tool is determined after the tool change. (Note) When "1" is set in the parameter "103 NEXT TOOL PREP", this data is valid.	0 to 9999
5	H	Specify the tool length offset No. The maximum value for the H is changed according to the specifications.	1 to number of tool sets
6	D	Specify tool diameter offset No. The maximum value for the D is changed according to the specifications.	1 to number of tool sets
7	S	Input the spindle rotation speed.	1 to 99999 rev/min
8	F	Input the feedrate. When the hole cycle type is "TAP", input the pitch (mm/rev).	0.001 to 60000.000 mm/min 0.001 to 999.999 mm/rev

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	NEXT T PRESET	Sets the next tool No. automatically. For the next tool No., the tool No. of the next process is set.
3	MULTI PT	Multiple Parts screen is displayed. When using the Multiple Parts function, press "MULTI PT".
4	SAVE	Saves changes in the process list.

4.3.2 Multiple Parts Screen

Multiple Parts function enables you to generate a NC program that allows one designated machining process to be completed at once for multiple workpieces. This NC program can be generated based on the NC programs generated for a single part machining with the appropriate arrangement of the multiple workpieces.

When working on multiple workpieces, there are two options available to specify their positions:

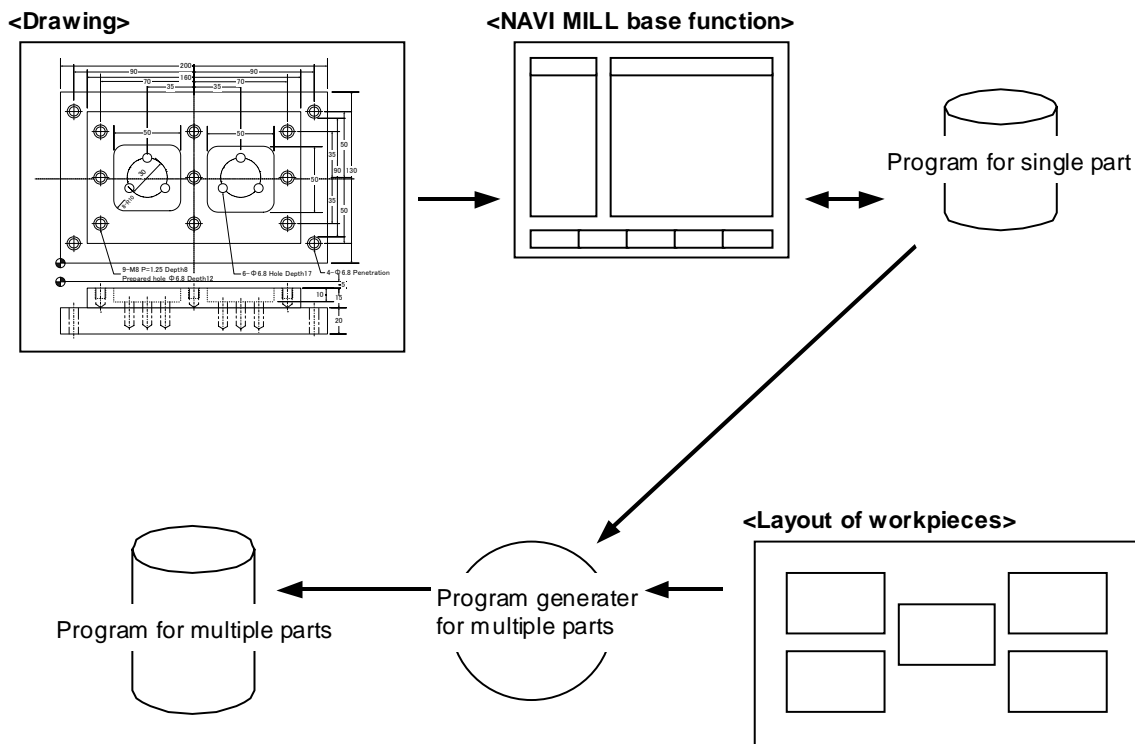
- by identifying the values of the work-coordinate system per each workpiece.
- by determining each offset amount from one specific work coordinate.

As for machining pattern, there are two options available:

- to have one machining process completed for all workpieces on the table before moving on to the next machining process (the number of times to execute tool-changes can be reduced by choosing this option).
- to have entire machining processes completed per each workpiece.

(Note 1) The NC program generated with the Multiple Parts function is differed from its original NC program.

(Note 2) The NC program generated with the Multiple Parts function cannot be edited with NAVI MILL. If editing is attempted, the operation message “No init process, Create OK? (Y/N)” appears.



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

To view the "Multiple Parts Screen", go to the "Process List Screen" and press [MULTI PT] menu.

Screen Layout

The screenshot displays the 'Multiple Parts' screen. At the top, there are menu options: MEMORY, Monitr, Setup, Edit, Diagn, and Mainte. Below this, the screen is divided into several sections:

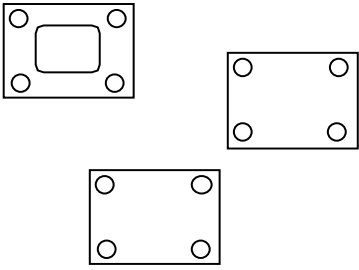
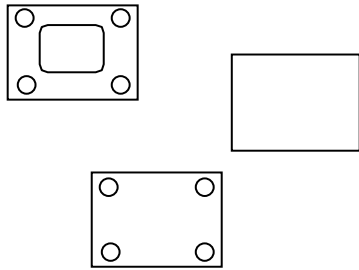
- NAVI MAIN CHECKER**: A menu on the left with options like LIST VIEW, PROGRAM, PROCESS, FILE, TOOL, CUT CONDTN, PARAMETER, and VERSION.
- MULTIPLE PARTS PROGRAM:108**: A header for the current screen.
- PROGRAM 0 1108**: Program identification.
- NUM OF PARTS 5**: Number of parts.
- MACHINING 1 --1:EVERY PROCESS 2:EVERY PART**: Machining mode.
- COORDINATE 1 --1:WC 2:POSITION**: Coordinate system.
- Table**: A table with columns No., X, Y, and WC.

No.	X	Y	WC
1	*****	*****	54<WC>
2	*****	*****	5554 :654
3	*****	*****	56 :
4	*****	*****	5759 :659
5	*****	*****	58P1 :654.1P1
6	*****	*****	***** :
7	*****	*****	*****P48:654.1P48
8	*****	*****	*****
- 0 [] COMMENT []**: A comment field.
- Icons**: Three small icons (a cube, an 'i', and a document) are visible.
- Status Bar**: Shows '\$1 REV \$2 REV' on the left and '18:28' on the right.
- Buttons**: A row of buttons at the bottom, including 'GEN PRGM' and 'RETURN'.

4. SCREEN SPECIFICATIONS

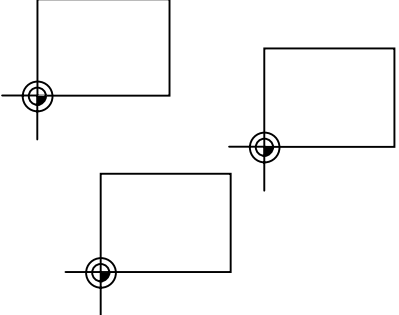
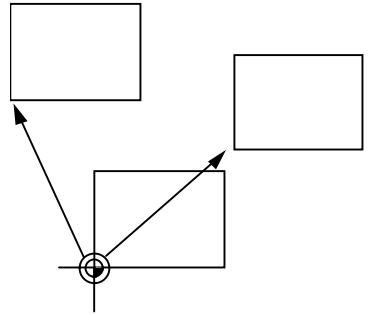
4.3 Screens Related to the Process Edit Functions

Screen Details

No.	Display item	Details	Setting range
1	PROGRAM ○	<p>Program No. and comments are input.</p> <p>Avoid using the same program No. for the Multiple Parts NC program and the currently editing program. If a duplicated program No. is used for both programs, an error message occurs.</p>	—
2	NUM OF PARTS	Number of parts is input.	2 to 8
3	MACHINING	<p>Machining pattern is selected.</p> <p>1: A pattern that has one machining process completed for all workpieces before moving on to the next machining process</p>  <p>2: A pattern that has entire machining processes completed per one workpiece before moving on to the next workpiece</p> 	1 to 2

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

No.	Display item	Details	Setting range
4	COORDINATE	<p>The method to specify the position of multiple workpieces is selected.</p> <p>1: A method that identifies the values of the work-coordinate system per each workpiece</p>  <p>2: A method that determines each offset amount from one specific work coordinate</p> 	1 to 2
5	X Y	<p>The value of each workpiece position is input depending on the offset amount from one specific work coordinate.</p> <p>This is valid when selecting "2" in the COORDINATE screen.</p>	-99999.999 to 99999.999mm
	WC	<p>The value of the work-coordinate system per each workpiece is input.</p> <p>This is valid when selecting "1" in the COORDINATE screen.</p>	54 to 59 P1 to P48

(Note) Program No. can be selected from 1 to 7999 or 10000 to 99999999.
Comment section allows up to 18 characters/numbers.

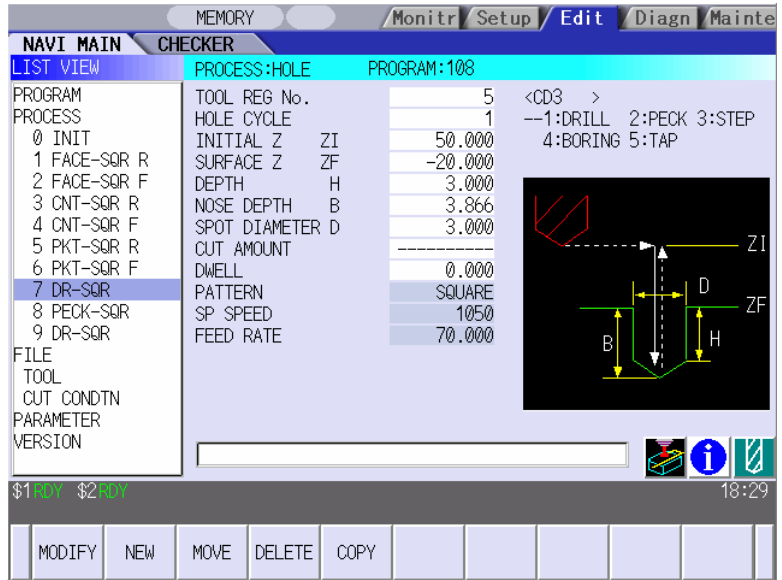
Menu

No.	Menu	Details
1	GEN PRGM	Generate a NC program for the Multiple Parts machining.
2	RETURN	Return to the Process List Screen.

4.3.3 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.

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

Operation example (Selecting the process)

- (1) Validate the LIST VIEW area and 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.



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

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 message confirming the deletion will appear.



- (3) Press the [Y] key.
When not deleting the process, press the [N] key



The highlight of the [DELETE] menu will turn OFF, and the process at the cursor position will be deleted.

The process under the deleted process will be moved up one.

The contents in the OPERATION VIEW area will change to those of the process at the cursor position.



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

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.



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

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. →



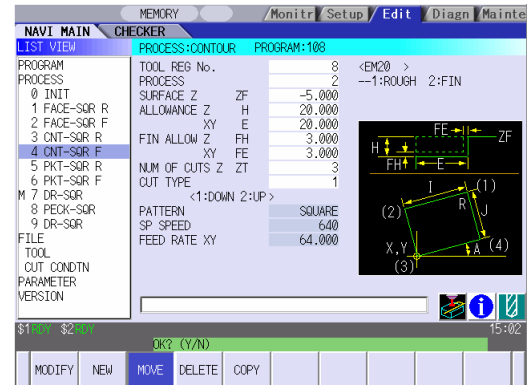
4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

- (4) Press the [INPUT] key.

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

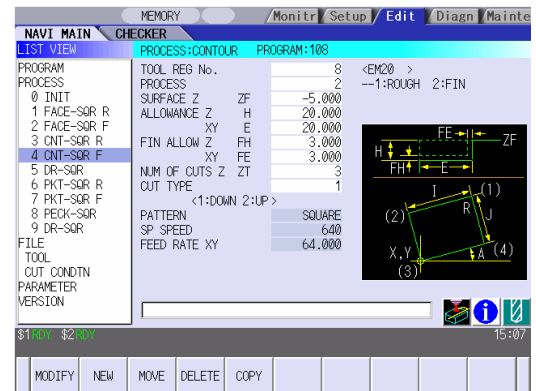
→ The message to confirm a movement is displayed.



- (5) Press the [Y] key.

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

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

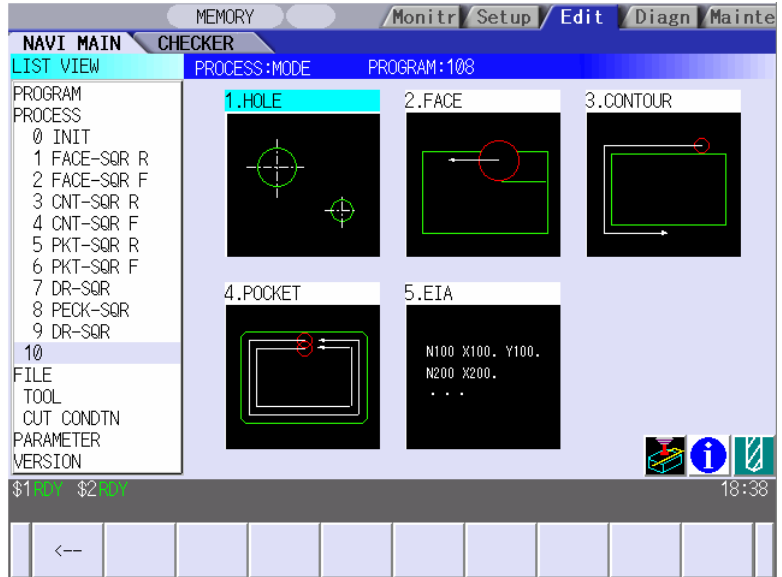


(Note) For the [NEW] menu, refer to the next section.

4.3.4 Process Mode Selection Screen

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

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 5

Menu

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

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

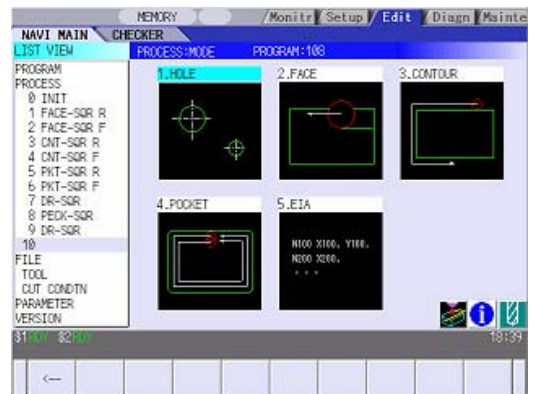
Operation example(Adding a new process)

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

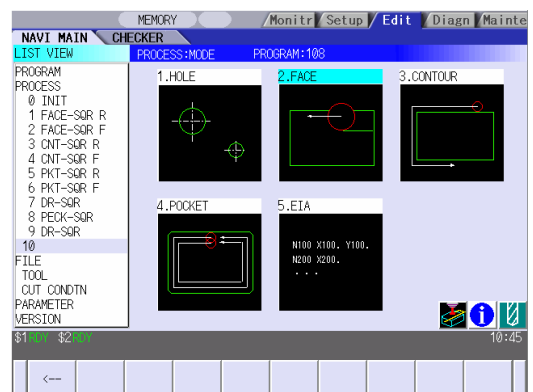


- (2) Press the [NEW] menu key.

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. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(4) Press the [INPUT] key.



The contents in the OPERATION VIEW area will change into those of the selected process mode.

The selected process mode will be displayed at the cursor position in the LIST VIEW area.



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

4.3.5 Initial Condition Setting

(1) 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	INITIAL POS Z	Input the initial position Z. In the workpiece coordinate system, input the Z axis position where the workpiece or jig does not interfere with the tool even if the table is moved.	-99999.999 to 99999.999mm

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Continued from the previous page)

No.	Display item	Details	Setting range
3	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
4	ATC PATTERN X ATC PATTERN Y	Select the table position at the tool change (ATC) with the following No.'s. 1 : No specification (The table is not moved.) 2 : 1st zero point 3 : 2nd zero point 4 : Specified position (The table is moved to the specified ATC position.)	1 to 4
5	ATC POSITION X ATC POSITION Y	In the machine coordinate system, input the table position at the tool change. This is valid when "4" is set in the "ACT PATTERN".	-99999.999 to 99999.999mm
6	END TOOL No.	At the program end, input the tool No. that you want to call. If 0 is input, the tool used at the machining end will be stopped with that attached to the spindle. If the tool No. is input, the tool will be changed at the machining end and then stopped.	0 to 9999
7	END PATTERN X END PATTERN Y	Select the table position at the program end with the following No.'s. 1 : No specification (The table is not moved.) 2 : 1st zero point 3 : 2nd zero point 4 : Specified position (The table is moved to the specified end position.)	1 to 4
8	END POSITION X END POSITION Y	In the machine coordinate system, input the table position at the program end. This is valid when "4" is set in the "END PATTERN".	-99999.999 to 99999.999mm
9	END M CODE	At the program end, select the M command to be output. 1 : M30 2 : M02 3 : M99	1 to 3

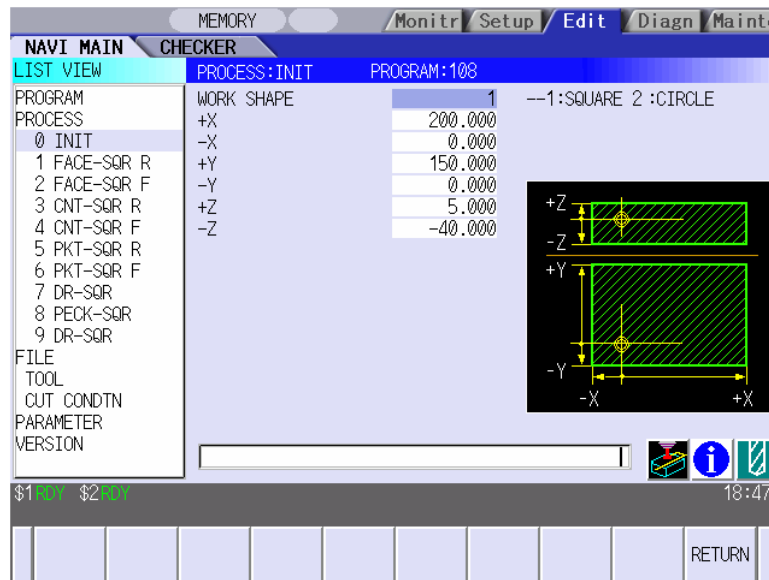
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	WORK SHAPE	Workpiece Size Setting screen is displayed.
3	SAVE	Saves the changes in the initial conditions.

(2) Workpiece Size Setting Screen

The size of workpiece is set on this screen. Parameters on this screen are used to display the size of the workpiece and the tool paths during Program Checker.

Screen layout



Screen display items

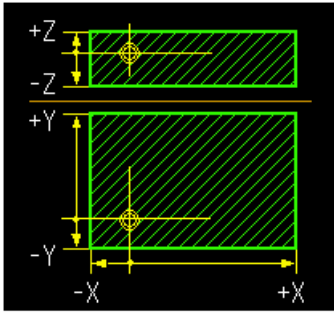
No.	Display item	Details	Setting range
1	WORK SHAPE	Input the shape of workpiece. 1 : SQUARE 2 : CIRCLE	1, 2

(Note) Display items No.2 and later differ according to each shape of workpiece.

4. SCREEN SPECIFICATIONS

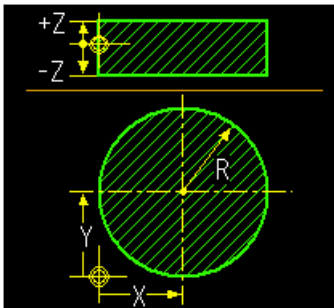
4.3 Screens Related to the Process Edit Functions

- Parameters for SQUARE



No.	Display item	Details	Setting range
2	+X	Input +X position based on the work coordinate zero point.	-99999.999 to 99999.999mm
3	-X	Input -X position based on the work coordinate zero point.	-99999.999 to 99999.999mm
4	+Y	Input +Y position based on the work coordinate zero point.	-99999.999 to 99999.999mm
5	-Y	Input -Y position based on the work coordinate zero point.	-99999.999 to 99999.999mm
6	+Z	Input +Z position based on the work coordinate zero point.	-99999.999 to 99999.999mm
7	-Z	Input -Z position based on the work coordinate zero point.	-99999.999 to 99999.999mm

- Parameters for CIRCLE



No.	Display item	Details	Setting range
2	CENTER X	Input the center of the circle. (X)	-99999.999 to 99999.999mm
3	CENTER Y	Input the center of the circle. (Y)	-99999.999 to 99999.999mm
4	RADIUS R	Input the radius of the circle.	0.001 to 99999.999mm
5	+Z	Input -Y position based on the work coordinate zero point.	-99999.999 to 99999.999mm
6	-Z	Input +Z position based on the work coordinate zero point.	-99999.999 to 99999.999mm

Menu

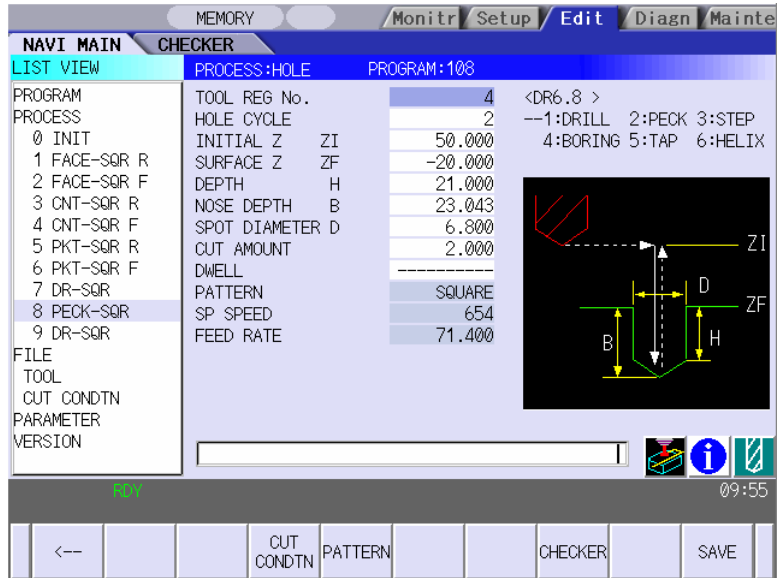
No.	Menu	Details
1	RETURN	Returns to the initial condition setting screen.

4.3.6 Hole Drilling

(1) Hole Drilling Screen

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

Screen layout



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it with the No. registered in the tool file.	1 to 99
2	HOLE CYCLE	<p>Input the type of the hole machining cycle.</p> <p><1: DRILL> (G81, G82) 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.</p> <p><2: PECK> (G83) The machining is performed as far as the middle of the hole, and the tool is returned to the higher position than the hole top each time. The machining is performed as far as the hole bottom with such operation repeatedly executed.</p> <p><3: STEP> (G73) The machining is performed as far as the middle of the hole, and the tool is returned each time by the G73 return amount. The machining is performed as far as the hole bottom with such operations repeatedly executed.</p> <p><4: BORE> (G85, G89) 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.</p> <p><5: TAP> (G84, G74) 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.</p> <p><6: HELIX> The machining is performed with helical interpolation as far as the hole bottom, and then the tool is lifted up.</p>	1 to 5

(Note) Display items No.3 and later differ according to each type of hole machining cycle.

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

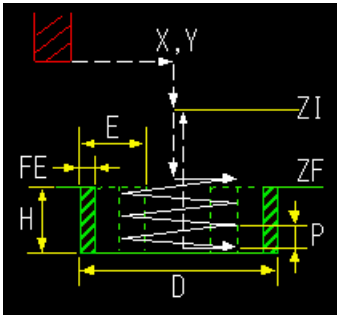
- When HOLE CYCLE is set from the types between 1 to 5 (DRILL, PECK, STEP, BORE or TAP)

3	INITIAL Z (ZI)	Input the initial position. The tool is returned to the initial position after the machining has been finished.	-99999.999 to 99999.999mm
4	SURFACE Z (ZF)	Input the workpiece top surface.	-99999.999 to 99999.999mm
5	DEPTH (H)	Input the hole depth from the workpiece top surface with an 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 over will occur.	-99999.999 to 99999.999mm
6	NOSE DEPTH (B)	Input the tool nose depth from the workpiece top surface with an addition input method. When the tool nose depth is changed, the hole depth will be automatically updated.	0.001 to 99999.999mm
7	SPOT DIAMETER (D)	Input the tool radius of the workpiece face. When the tool radius of the workpiece face is input, DEPTH and NOSE DEPTH will be automatically updated.	0.001 to Tool diameter
8	CUT AMOUNT	When the hole cycle type C=2 (PECK) or C=3 (STEP) is selected, input the cutting amount for one time. If a value other than 0.000 is input when selecting the hole cycle type C=5 (TAP), the pecking tap process will be applied.	0.000 to 99999.999mm
9	DWELL	When the hole cycle type C=1 (Drilling), C=3 (Step), C=4 (Boring), C=5 (Tap) is selected, input the dwell.	0.000 to 99999.999sec
10	PATTERN	The machining pattern is displayed. RANDOM LINE ARC CIRCLE SQUARE GRID Perform changing the machining pattern on the machining pattern screen.	-
11	SP SPEED	The spindle rotation speed is displayed. Perform changing the spindle rotation speed on the cutting condition screen.	1 to 99999rev/min
12	FEED RATE	The feedrate is displayed. When the hole cycle type is "TAP", the pitch is displayed. Perform changing the feedrate on the cutting condition screen.	0.001 to 60000.000 mm/min 0.001 to 999.999 mm/rev

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

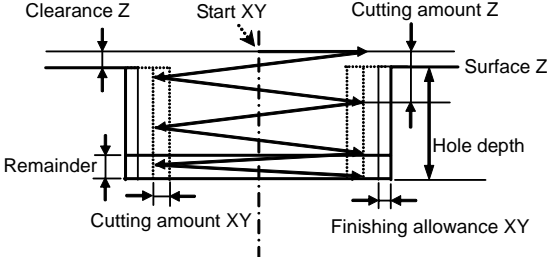
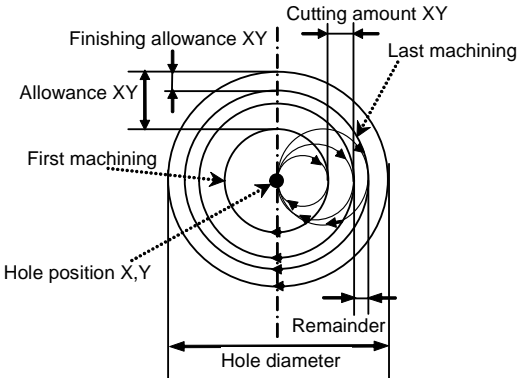
- When HOLE CYCLE is set to 6 (HELIX)



No.	Display item	Details	Setting range
3	PROCESS	<p>Input the machining type.</p> <p><1: ROUGH > The rough machining is performed and the finishing allowance remains.</p> <p><2: FIN> The finishing machining is performed to the machining surface.</p>	1,2
4	INITIAL Z	<p>Input the initial position.</p> <p>After machining, the tool returns to the initial position.</p>	-99999.999mm to 99999.999mm
5	SURFACE Z	<p>Input the workpiece top surface position.</p>	-99999.999mm to 99999.999mm
6	POS X	<p>Input the hole position (X).</p>	-99999.999mm to 99999.999mm
7	POS Y	<p>Input the hole position (Y).</p>	-99999.999mm to 99999.999mm
8	HOLE DIAMETER	<p>Input the hole diameter.</p>	0.001mm to 99999.999mm
9	DEPTH	<p>Input the hole depth from the workpiece top surface with an addition input method.</p>	0.001mm to 99999.999mm
10	ALLOWANCE XY	<p>Input the allowance of the side surface.</p> <p>This data is valid for the rough machining only.</p>	0.001mm to 99999.999mm
11	FIN ALLOW XY	<p>Input the finishing allowance of the side surface.</p> <p>The rough machining leaves the finishing allowance in respect to the side surface.</p> <p>This data is valid for the rough machining only.</p>	0.000mm to 99999.999mm

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

No.	Display item	Details	Setting range
12	CUT AMOUNT Z	<p>Input the cutting amount in Z-direction. The remainder, if left after "DEPTH + CLEARANCE Z / CUT AMOUNT" has been applied, will be the last cutting amount.</p> 	0.001mm to 99999.999mm
13	CUT AMOUNT XY	<p>Input the cutting amount in XY-direction. This data is valid for the rough machining only.</p> 	0.001mm to 99999.999mm
14	CUT TYPE	<p>Input the cutting type (up-cutting, down-cutting). 1: DOWN 2: UP When the HOLE CYCLE is set to 6 (HELIX), the cutting type is set to 1 (DOWN) as a default value.</p>	1,2
15	SP SPEED	<p>The spindle rotation speed is displayed. The spindle rotation speed is changed on the setting screen for cutting conditions.</p>	1 to 99999 (r/min)
16	FEED RATE F	<p>The feedrate is displayed. The feedrate is changed on the setting screen for cutting conditions.</p>	0.001 to 60000.000 mm/min 0.01 to 999.99 mm/rev

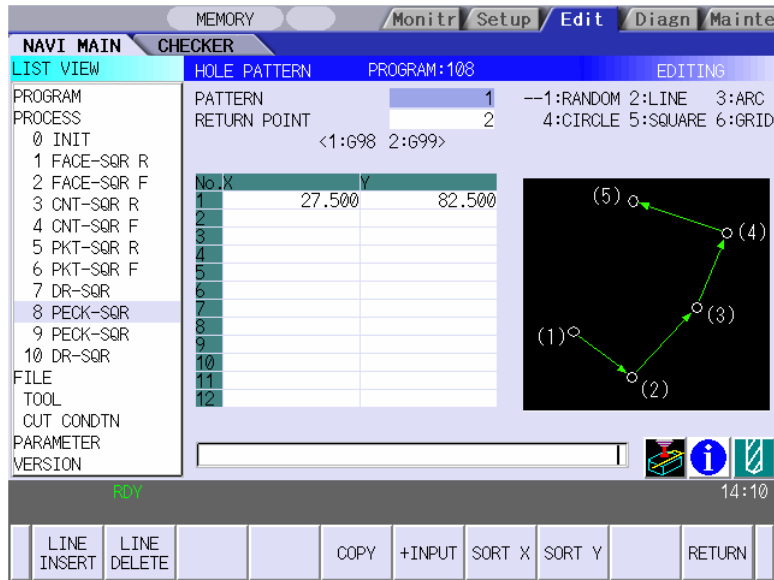
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CUT CONDTN	Displays the setting screen for the cutting conditions.
3	PATTERN	Displays the selection screen for the machining pattern. This menu is not displayed when HOLE CYCLE is set to 6 (HELIX).
4	CHECKER	Displays the checker screen. Selects this to check the set data.
5	SAVE	Saves the changes in the process. If an illegal parameter exists when saving, an error message will appear. If the input parameter is illegal, the cursor moves to the illegal parameter position. If the input parameters for the pattern input screen or cutting condition screen are illegal, the screen name and error message will be displayed.

(2) Hole Drilling Machining Pattern Screen

The parameters for the hole drilling machining pattern are input on this screen. When the [PATTERN] menu is pressed on the hole drilling screen, this screen is displayed. This screen is not available when HOLE CYCLE is set to 6 (HELIX).

Screen layout

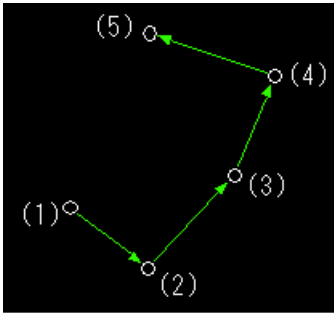


Screen display items

No.	Display item	Details	Setting range
1	PATTERN	Input the type of the hole machining pattern. <1: RANDOM> The machining points are randomly arranged. <2: LINE> The machining points are equally spaced on a line. <3: ARC> The machining points are equally spaced on an arc. <4: CIRCLE> The machining points are equally spaced on a circle. <5: SQUARE> The machining points are squarely arranged. <6: GRID> The machining points are arranged in grid.	1 to 6

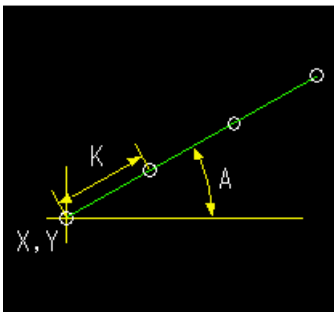
(Note) Display items No.2 and later differ according to each pattern.

• Parameters for RANDOM



No.	Displayed item	Details	Setting range
2	RETURN POINT	Specify the return point when the hole drilling is completed. 1: Initial point level return (G98) 2: R point level return (G99)	1,2
3	No.	Input the hole No.	1 to 50
4	POS X POS Y	Input the hole position.	-99999.999mm to 99999.999mm

• Parameters for LINE

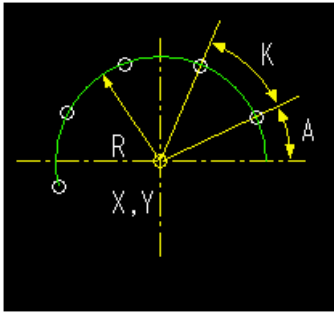


No.	Display item	Details	Setting range
2	ANGLE (A)	Input the angle formed with the machining direction and the positive direction of the X-axis.	-359.999 to 360.000°
3	PITCH (K)	Input the space from the machining point to the next machining point.	0.000 to 99999.999mm
4	NUM OF HOLES	Input the number of holes.	2 to 999
5	BASE POS X	Input the 1st hole position (X).	-99999.999 to 99999.999mm
6	BASE POS Y	Input the 1st hole position (Y).	-99999.999 to 99999.999mm
7	RETURN POINT	Specify the return point that is applied after the hole machining. 1: Initial point level return (G98) 2: R point level return (G99)	1, 2

4. SCREEN SPECIFICATIONS

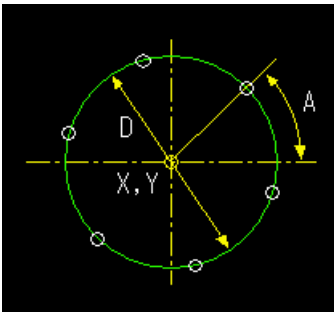
4.3 Screens Related to the Process Edit Functions

• Parameters for ARC



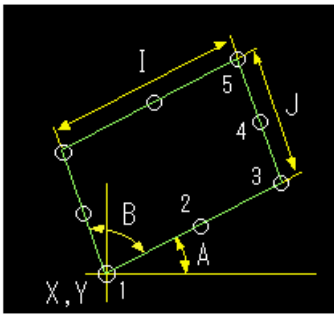
No.	Display item	Details	Setting range
2	RADIUS (R)	Input the arc radius.	0.001 to 99999.999mm
3	START ANGLE (A)	Input the angle formed with the first machining point and the X-axis direction.	-359.999 to 360.000°
4	PITCH (K)	Input the angle from the previous machining point to the next machining point.	-359.999 to 360.000°
5	NUM OF HOLES	Input the number of holes.	2 to 999
6	BASE POS X	Input the arc center position (X).	-99999.999 to 99999.999mm
7	BASE POS Y	Input the arc center position (Y).	-99999.999 to 99999.999mm
8	RETURN POINT	Specify the return point that is applied after the hole machining. 1: Initial point level return (G98) 2: R point level return (G99)	1, 2

• Parameters for CIRCLE



No.	Display item	Details	Setting range
2	DIAMETER (D)	Input the circular diameter.	0.001 to 99999.999mm
3	START ANGLE (A)	Input the angle formed with the first machining point and the positive direction of the X-axis.	-359.999 to 360.000°
4	NUM OF HOLES	Input the number of holes.	1 to 999
5	BASE POS X	Input the circular center position (X).	-99999.999 to 99999.999mm
6	BASE POS Y	Input the circular center position (Y).	-99999.999 to 99999.999mm
7	RETURN POINT	Specify the return point that is applied after the hole machining. 1: Initial point level return (G98) 2: R point level return (G99)	1, 2

• Parameters for SQUARE

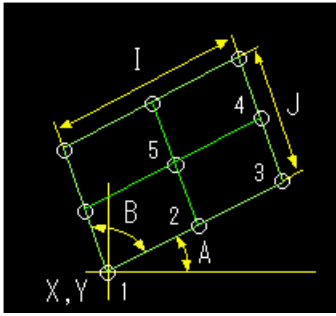


No.	Display item	Details	Setting range
2	X WIDTH (I)	Input the width of the machining point in the X-axis direction.	-99999.999 to 99999.999mm
3	X NUM OF HOLES	Input the number of machining points in the X-axis direction.	2 to 999
4	Y WIDTH (J)	Input the width of the machining point in the Y-axis direction.	-99999.999 to 99999.999mm
5	Y NUM OF HOLES	Input the number of machining points in the Y-direction.	2 to 999
6	BASE POS X	Input the position (X) of the machining start point.	-99999.999 to 99999.999mm
7	BASE POS Y	Input the position (Y) of the machining start point.	-99999.999 to 99999.999mm
8	RETURN POINT	Specify the return point that is applied after the hole machining. 1: Initial point level return (G98) 2: R point level return (G99)	1, 2
9	ANGLE (A)	Input the workpiece's inclination angle between the machining start direction and the X axis.	-359.999° to 360.000°
10	ANGLE (B)	Input the interior angle. Default value is 90°.	0.001° to 179.999°
11	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	1 to number of holes

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

- Parameters for GRID



No.	Display item	Details	Setting range
2	X WIDTH (I)	Input the width of the machining point in the X-axis direction.	-99999.999 to 99999.999mm
3	X NUM OF HOLES (J)	Input the number of machining points in the X-axis direction.	2 to 999
4	Y WIDTH	Input the width of the machining point in the Y-axis direction.	-99999.999 to 99999.999mm
5	Y NUM OF HOLES	Input the number of machining points in the Y-direction.	2 to 999
6	BASE POS X	Input the position (X) of the machining start point.	-99999.999 to 99999.999mm
7	BASE POS Y	Input the position (Y) of the machining start point.	-99999.999 to 99999.999mm
8	RETURN POINT	Specify the return point that is applied after the hole machining. 1: Initial point level return (G98) 2: R point level return (G99)	1, 2
9	ANGLE (A)	Input the workpiece's inclination angle between the machining start direction and the X axis.	-359.999° to 360.000°
10	ANGLE (B)	Input the interior angle. Default value is 90°.	0.001° to 179.999°
11	OMIT 1 to 4	Specify the hole No. to be omitted (deleted). Maximum hole No. that can be specified is 127.	1 to number of holes

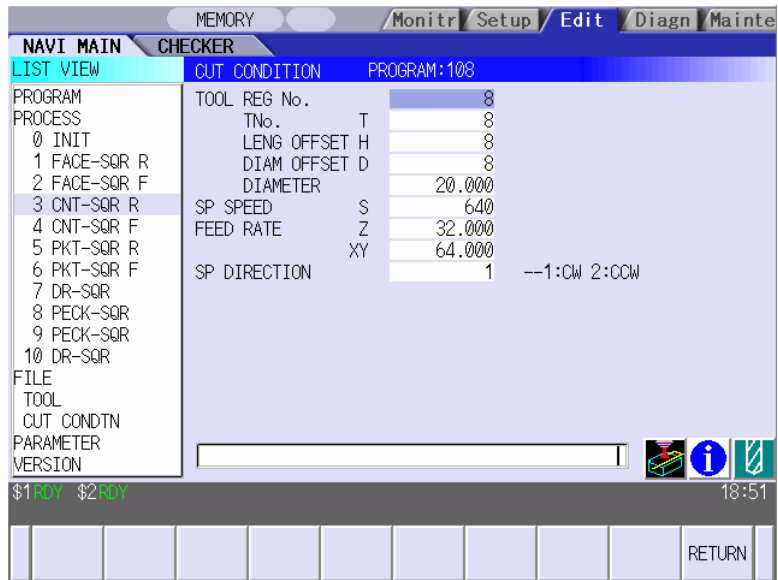
Menus

No.	Menu	Details
1	RETURN	Returns to the hole drilling screen.
2	LINE INSERT	Inserts the hole position in front of the cursor position. This is available for the RANDOM pattern only.
3	LINE DELETE	Deletes the hole position at the cursor position. This is available for the RANDOM pattern only.
4	COPY	Copies the previous line data above cursor to the setting area. This is available for the RANDOM pattern only.
5	+INPUT	Adds the previous line data above cursor to the setting data, and enters the value to the setting area. This is available for the RANDOM pattern only.
6	SORT X	Sorts POS X and Y in ascending order based on POS X. This is available for the RANDOM pattern only.
7	SORT Y	Sorts POS X and Y in ascending order based on POS Y. This is available for the RANDOM pattern only.

(3) Cutting Condition Screen

The cutting conditions are set on this screen. When the [CUT CONDTN] menu is pressed on each process edit screen, this screen is displayed.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it with the No. registered in the tool file.	1 to 99
2	TOOL T No. (T)	Specify the tool No. to be used. (T function code data output as the NC data) When the tool registration No. is specified, the tool No. registered in the tool file will be automatically set. If 0 is set, the T function code will not be output.	0 to 9999
3	TOOL LENG OFFSET (H)	Specify the tool length offset No. The maximum value for the H is changed according to the specifications. When the tool registration No. is specified, the tool length offset No. registered in the tool file will be automatically set.	1 to the number of tool sets

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Continued from the previous page)

No.	Display item	Details	Setting range
4	TOOL DIAM OFFSET (D)	Specify tool diameter offset No. The maximum value for the D is changed according to the specifications. When the tool registration No. is specified, the tool diameter offset No. registered in the tool file will be automatically set. That can be also rewritten manually.	1 to the number of tool sets
5	TOOL DIAMETER	Input the tool diameter. When the tool registration No. is specified, the tool diameter registered in the tool file will be automatically set.	0.001 to 99999.999 mm
6	SP SPEED (S)	Input the spindle rotation speed. The spindle rotation speed is automatically set based on the standard rotation speed registered in the tool file and the rate registered in the cutting condition file. Spindle rotation speed = Standard rotation speed × rate	1 to 99999 rev/min
7	FEED RATE (Z) (XY)	Input the feedrate. Specify the pitch with mm/rev when tapping is selected in the hole drilling machining mode. Specify the feedrate with mm/min in the other machining mode. The feedrate is automatically set based on the standard feedrate registered in the tool file and the rate registered in the cutting condition file. Feedrate = Standard feedrate × rate (Note) The setting items are changed according to the process type. <Hole drilling process, face cutting process> FEED RATE F <Contour cutting process, pocket process> FEED RATE Z FEED RATE XY	0.001 to 60000.000 mm/min 0.001 to 999.999 mm/rev
8	SP DIRECTION	Input the spindle rotation direction. When the tool registration No. is specified, the tool rotation direction registered in the tool file will be automatically set. 1: CW 2: CCW	1, 2

Menu

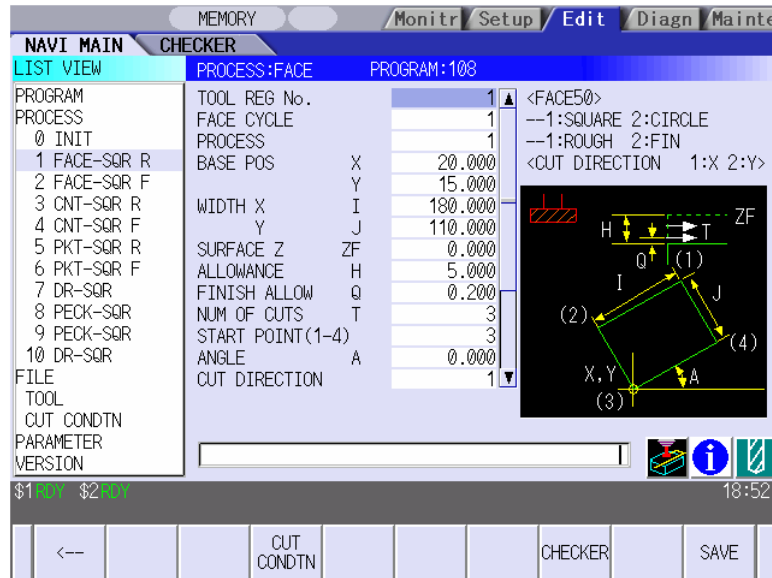
No.	Menu	Details
1	RETURN	Returns the screen to the input screen for the hole process parameter.

4.3.7 Face Cutting

(1) Face Cutting Screen

The parameters for the face cutting process are set on this screen.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it with the No. registered in the tool file.	1 to 99
2	FACE CYCLE	Input the type of the face cutting cycle. <1: SQUARE> The face cutting machining in the square area is performed. <2: CIRCLE> The face cutting machining in circular area is performed.	1, 2
3	PROCESS	Input the type of the machining. <1: ROUGH> The rough machining is performed with cuttings of the number of rough machinings, remaining the finishing allowance. <2: FIN> The finishing machining is performed to the finishing surface (Height of machining top surface - allowance) at a stretch.	1, 2

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Continued from the previous page)

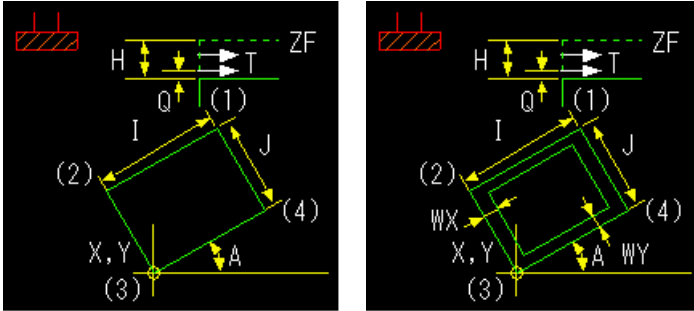
No.	Display item	Details	Setting range
4	BASE POS X	Input the position (X) of the base point.	-99999.999 to 99999.999mm
5	BASE POS Y	Input the position (Y) of the base point.	-99999.999 to 99999.999mm
6	SURFACE Z (ZF)	Input the workpiece top surface position.	-99999.999 to 99999.999mm
7	ALLOWANCE (H)	Input the size of the allowance from the height of the machining top surface.	0.001 to 99999.999mm
8	FINISH ALLOW (Q)	Input the size of the finishing allowance from the finishing surface. The rough machining is performed to the machining surface, remaining the finishing allowance. (Note) This data is valid only when the rough machining is specified.	0.000 to 99999.999mm
9	NUM OF CUTS (T)	Input how many times the allowance is cut in the Z-axis direction when performing the rough cutting. The cutting amount for one time is as follows. Allowance – finishing allowance / Number of rough machinings (Note) This can be input only when the rough machining is specified.	1 to 99999 times
10	START POINT	Input form which corner ((1) to (4)) in the four screen guide drawings the machining is started.	1 to 4
11	CUT DIRECTION	Input the direction (X-direction or Y-direction) in which the machining is performed. 1: X 2: Y	1, 2
12	PATH	Input the machining path (reciprocation machining, single-direction machining, shape machining) 1: RECIP 2: SINGLE 3: SHAPE	1 to 3
13	SP SPEED	The spindle rotation speed is displayed. Perform changing the spindle rotation speed on the cutting condition screen.	1 to 99999rev/min
14	FEED RATE	The feedrate is displayed. Perform changing the feedrate on the cutting condition screen.	0.001 to 60000.000 mm/min

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

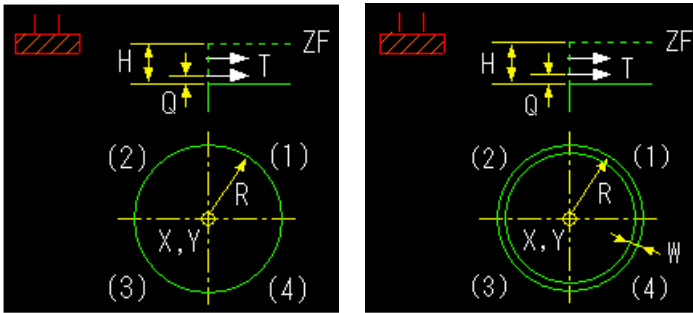
(Note) Display items No.15 and later differ according to each type of the face cutting cycle.

- Parameter used when the face cutting cycle is SQUARE



No.	Display item	Details	Setting range
15	WIDTH X (I)	Input the machining width of the machining surface in the X-direction (horizontal direction). This is valid when the face cutting cycle is SQUARE.	0.001 to 99999.999mm
16	WIDTH Y (J)	Input the machining width of the machining surface in the Y-direction (vertical direction). This is valid when the face cutting cycle is SQUARE.	0.001 to 99999.999mm
17	ANGLE (A)	Input the inclination angle of the machining surface. This is valid when the face cutting cycle is SQUARE.	-359.999 to 360.000°
18	CUT WIDTH X (WX)	Input the width of machining part in the X-direction during the square machining. This is valid when the face cutting cycle is SQUARE.	0.000 to 99999.999mm
19	CUT WIDTH Y (WY)	Input the width of machining part in the Y-direction during the square machining. This is valid when the face cutting cycle is SQUARE.	0.000 to 99999.999mm

- Parameter used when the face cutting cycle is CIRCLE



No.	Display item	Details	Setting range
15	WORK RADIUS (R)	Input the radius of the workpiece surface. This is valid when the face cutting cycle is CIRCLE.	0.001mm to 99999.999mm
16	CUT WIDTH (W)	Input the width of machining part during the circular machining. This is valid when the face cutting cycle is SQUARE.	0.000 to 99999.999mm

Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CUT CONDTN	Displays the setting screen for the cutting conditions.
3	SAVE	Saves the changes in the process. If an illegal parameter exists when saving, an error message will appear. If the input parameter is illegal, the cursor moves to the illegal parameter position. If the input parameters for the cutting condition screen are illegal, the screen name and error message will be displayed.

(2) Cutting Condition Screen

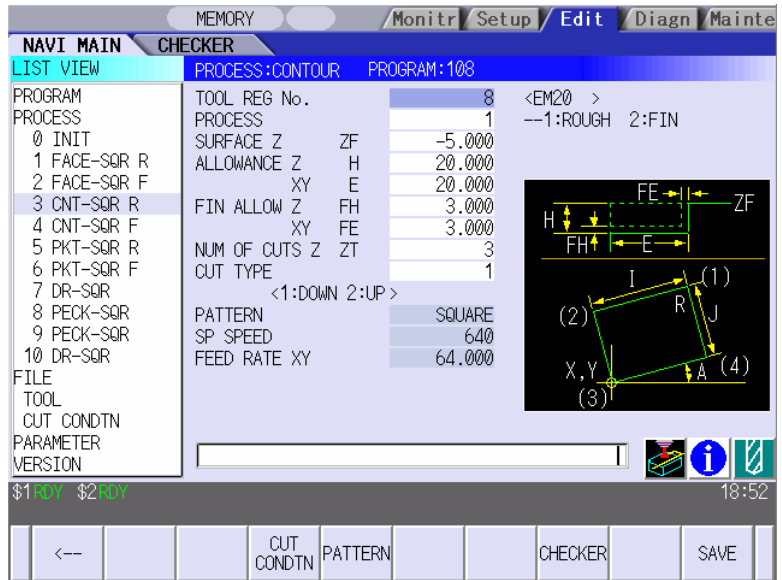
The cutting conditions are set on this screen. When the [CUT CONDTN] menu is pressed on each process edit screen, this screen is displayed. Refer to "(3) Cutting Condition Screen" in the section "4.3.6 Hole Drilling" for details.

4.3.8 Contour Cutting

(1) Contour Cutting Screen

The parameters for the contour cutting process are set on this screen.

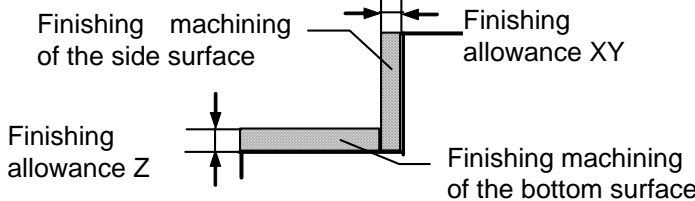
Screen layout



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it with the No. registered in the tool file.	1 to 99
2	PROCESS	<p>Input the machining type.</p> <p><1: ROUGH> Z-direction : The machining is performed with cuttings of the number of cuttings, remaining the finishing allowance Z. XY-direction: The machining is performed with shifting, remaining the finishing allowance XY.</p> <p><2: FIN> First, the finishing machining of the bottom surface is performed and then that of the side surface.</p>  <p>[Finishing machining of the bottom surface]</p> <p>Z-direction : The machining of the finishing allowance Z is performed at a stretch. XY-direction: The machining is performed with the shift of the allowance, remaining the finishing allowance XY. When the finishing allowance Z is 0, the finishing machining of the bottom surface is not performed.</p> <p>[Finishing machining of the side surface] Z-direction : The machining is performed with cuttings of the number of cuttings. XY-direction: The machining of the finishing allowance XY is performed at a stretch. When the finishing allowance XY is 0, the finishing machining of the side surface is not performed.</p>	1, 2
3	SURFACE Z (ZF)	Input the workpiece top surface position.	-99999.999 to 99999.999mm

(Continue to the next page)

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Continued from the previous page)

No.	Display item	Details	Setting range
4	ALLOWANCE Z (H)	Input the machining depth from the workpiece top surface with the addition input method.	0.001 to 99999.999mm
5	ALLOWANCE XY (E)	Input the allowance of the side surface. (Note) This data is valid when the pattern type is CIRCLE, SQUARE or FREE and RIGHT/LEFT is specified for the tool path.	0.001 to 99999.999mm
6	FIN ALLOWANCE Z (FH)	Input the finishing allowance of the bottom surface. The rough machining is performed to the bottom surface, remaining the finishing allowance.	0.000 to 99999.999mm
7	FIN ALLOWANCE XY (FE)	Input the finishing allowance of the side surface. The rough machining is performed to the side surface, remaining the finishing allowance. (Note) This data is valid when the pattern type is CIRCLE, SQUARE or FREE and RIGHT/LEFT is specified for the tool path.	0.000 to 99999.999mm
8	NUM OF CUTS Z (ZT)	Input how many times the workpiece is cut in the Z-axis direction when performing the machining. The cutting amount for one time is as follows. Rough machining: (Allowance – Finishing allowance) / Number of cuttings Finishing machining of side surface: Allowance/Number of cuttings	1 to 99999 times
9	CUT TYPE	Input the cutting type (up-cutting, down-cutting). 1: DOWN 2: UP (Note) This data is valid only when the pattern type is CIRCLE or SQUARE.	1, 2
10	PATTERN	The machining pattern is displayed. CIRCLE SQUARE FREE	-
11	SP SPEED	The spindle rotation speed is displayed. The spindle rotation speed is changed on the setting screen for cutting conditions.	1 to 99999 rev/min
12	FEED RATE (XY)	The feedrate in the XY-direction is displayed. The feedrate is changed on the setting screen for cutting conditions.	0.001 to 60000.000 mm/min

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Note) Depending on the machining pattern, the actual cutting amount of the side surfaces is differed as shown below.

CIRCLE/ SQUARE	FREE (right-side/left-side)	FREE (center)

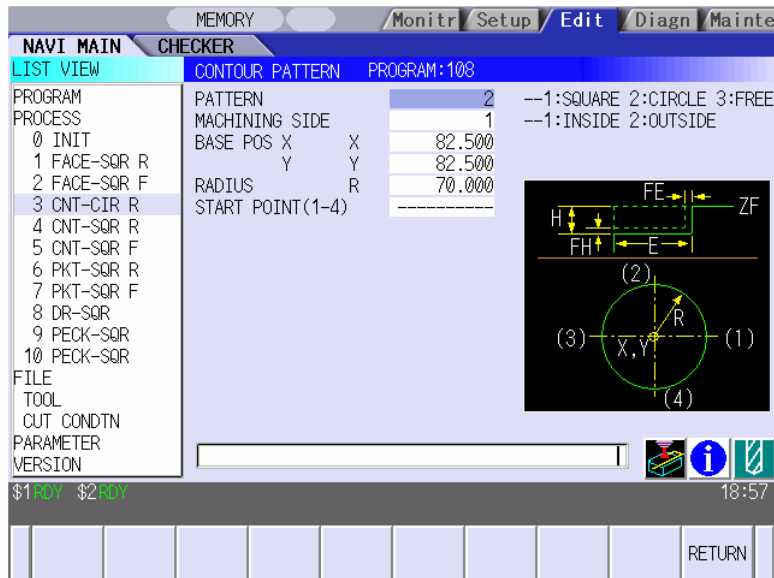
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CUT CONDTN	Displays the setting screen for the cutting conditions.
3	PATTERN	Displays the selection screen for the machining pattern.
4	SAVE	Saves the changes in the process. If an illegal parameter exists when saving, an error message will appear. If the input parameter is illegal, the cursor moves to the illegal parameter position. If the input parameters for the pattern input screen or cutting condition screen are illegal, the screen name and error message will be displayed.

(2) Contour Cutting Pattern Screen

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

Screen layout



Screen display items

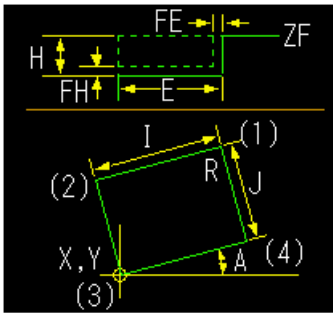
No.	Display item	Details	Setting range
1	PATTERN	Input the type of the contour cutting pattern. <1: SQUARE > The machining pattern of square shape <2: CIRCLE > The machining pattern of circular shape <3: FREE > The machining pattern of arbitrary shape that consists of the straight line and arc	1 to 3

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

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

- Parameters for SQUARE

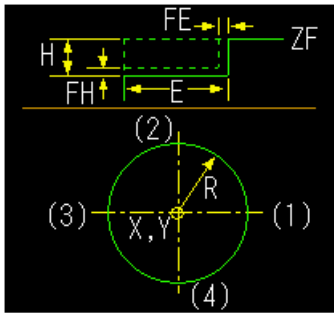


No.	Display item	Details	Setting range
2	MACHINING SIDE	Input the machining position. <1: INSIDE> The machining is performed to the inside of the shape. <2: OUTSIDE> The machining is performed to the outside of the shape.	1, 2
3	BASE POS X (X)	Input the base position X of the square.	-99999.999 to 99999.999mm
4	BASE POS Y (Y)	Input the base position Y of the square.	-99999.999 to 99999.999mm
5	WIDTH (I)	Input the width of the square in the X-direction.	0.001 to 99999.999mm
6	WIDTH (J)	Input the width of the square in the Y-direction.	0.001 to 99999.999mm
7	CORNER SIZE (R)	Input the corner size. Positive value : Corner R, Negative value: Corner C	-99999.999 to 99999.999mm
8	ANGLE (A)	Input the workpiece's inclination angle between the square and X-axis.	-359.999 to 360.000°
9	START POINT	Input the machining start position (1 to 4). (Note) This data is valid when the machining position is "OUTSIDE".	1 to 4

4. SCREEN SPECIFICATIONS

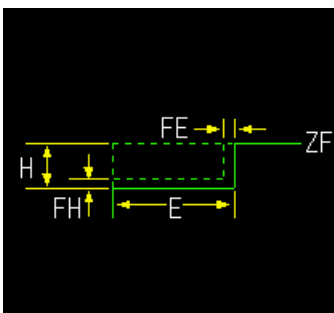
4.3 Screens Related to the Process Edit Functions

• Parameters for CIRCLE



No.	Display item	Details	Setting range
2	MACHINING SIDE	Input the machining position. <1: INSIDE> The machining is performed to the inside of the shape. <2: OUTSIDE> The machining is performed to the outside of the shape.	1, 2
3	BASE POS X (X)	Input the center position X of the circle.	-99999.999 to 99999.999mm
4	BASE POS Y (Y)	Input the center position Y of the circle.	-99999.999 to 99999.999mm
5	RADIUS (R)	Input the radius of the circle.	0.001 to 99999.999mm
6	START POINT	Input the machining start position (1 to 4). (Note) This data is valid when the machining position is "OUTSIDE".	1 to 4

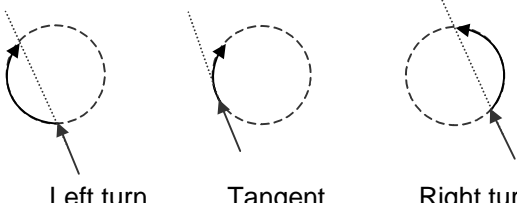
• Parameters for FREE



No.	Display item	Details	Setting range
2	TOOL PATH	Input the tool path. <1: CENTER> The machining is performed along the tool path without the diameter offset. <2: RIGHT> The machining is performed along the tool path with the offset to the right side (G42). <3: LEFT> The machining is performed to the tool path with the offset in the right side (G41).	1 to 3

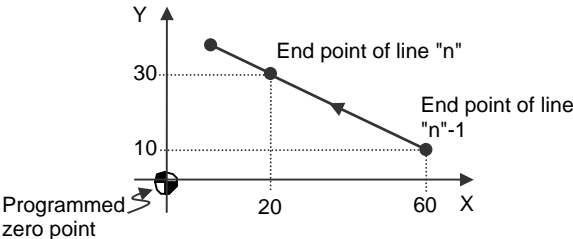
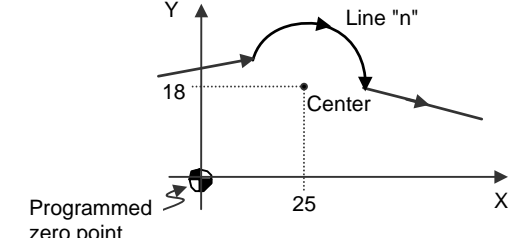
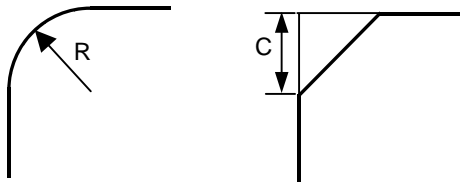
4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

No.	Display item	Details	Setting range
Shape			1 to 35
3	1	M Input the shape. <1> The linear (G01) machining is performed. <2> The CW arc (G02) machining is performed. <3> The CCW arc (G03) machining is performed. (Note) Omission is not possible.	1 to 3
	2	D Input right turn or left turn in respect to the vector at the end of the previous shape. 1: Left turn 2: Right turn (Note 1) When nothing is input, it is regarded as "contacting". (Note 2) Omittable. However, when the end point of the previous line, X and Z, is uncertain, always input. 	1,2
	3	X Y Input the position of the machining end point. (Note 1) Always input if the line is the last one. If it is not the last one, omission is possible. (Note 2) Always input if the corner shape dimensions are input in the previous line.	-99999.999mm to 99999.999mm
	4	R/A • When the shape is arc, the radius of arc is entered. Positive value: Arc command (less than 180°) Negative value: Arc command (more than 180°) • When the shape is line, angle is entered. (Note 1) When the shape is arc, always input. (Note 2) When the shape is line, and the position X, Y or vector I, J are input, the data is invalid.	Radius: -999999.999mm to -0.001mm, 0.001mm to 999999.999mm Angle: -359.999 to 360.000

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

No.	Display item	Details	Setting range
5	I J	<ul style="list-style-type: none"> When the shape is line, the gradient (vector) is entered.  <ul style="list-style-type: none"> When the shape is arc, the position of the arc center is entered.  <p>(Note 1) When the shape is arc, and only I or J is entered, the other is regarded as 0.</p> <p>(Note 2) When the shape is line, and the position X, Y or angle is input, this data will be invalid.</p>	1 to 35 -99999.999mm to 99999.999mm
6	C	<p>Input corner size. Positive value: Corner R, Negative value: Corner C</p>  <p>(Note 1) When corner dimensions are specified, the end points (X,Y) are entered for the following line in principle.</p>	-99999.999mm to 99999.999mm

(Note) The first point is a machining start point, so only the position X and Y can be input.

Menus

- For CIRCLE or SQUARE

No.	Menu	Details
1	RETURN	Returns the screen to the contour cutting screen.

- For FREE

No.	Menu	Details
1	LINE INSERT	Inserts the shape data before the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
2	LINE DELETE	Deletes the shape data at the cursor position. (Note) This menu is not available when the cursor is at No.1 (machining start point).
3	COPY	Copies the same data as that of previous line for the 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 position X and Y.
5	CLEAR	Clears the data at the cursor position.
6	RETURN	Returns the screen to the contour cutting screen.

(3) Cutting Condition Screen

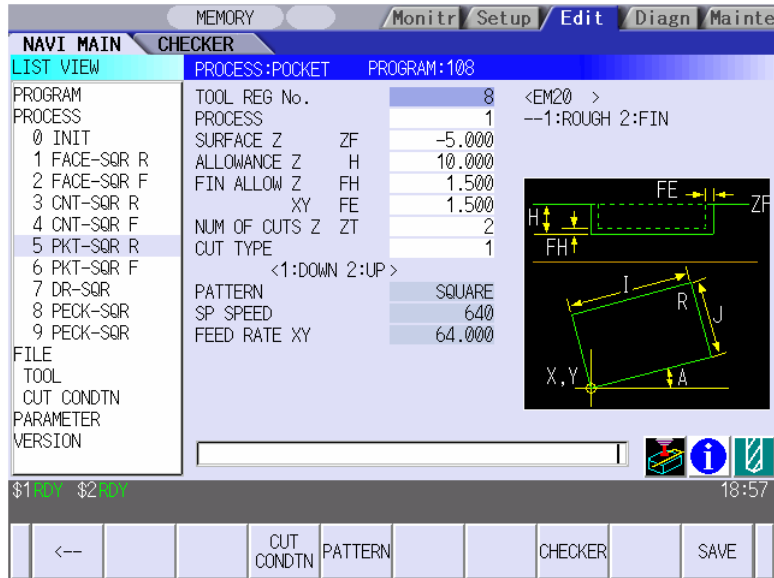
The cutting conditions are set on this screen. When the [CUT CONDTN] menu is pressed on each process edit screen, this screen is displayed. Refer to "(3) Cutting Condition Screen" in the section "4.3.6 Hole Drilling" for details.

4.3.9 Pocket

(1) Pocket Screen

The parameters for the pocket process are set on this screen.

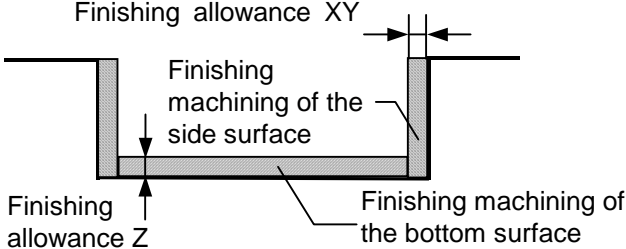
Screen layout



4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it with the No. registered in the tool file.	1 to 99
2	PROCESS	<p>Input the machining type.</p> <p><1: ROUGH> Z-direction : The machining is performed with cuttings of the number of cuttings, remaining the finishing allowance Z. XY-direction: The machining is performed with shifting, remaining the finishing allowance XY.</p> <p><2: FIN> First, the finishing machining of the bottom surface is performed and then that of the side surface.</p>  <p>[Finishing machining of the bottom surface] Z-direction : The machining of the finishing allowance Z is performed at a stretch. XY-direction: The machining is performed with the shift of the allowance, remaining the finishing allowance XY.</p> <p>When the finishing allowance Z is 0, the finishing machining of the bottom surface is not performed.</p> <p>[Finishing machining of the side surface] Z-direction : The machining is performed with cuttings of the number of cuttings. XY-direction: The machining of the finishing allowance XY is performed at a stretch.</p> <p>When the finishing allowance XY is 0, the finishing machining of the side surface is not performed.</p>	1, 2
3	SURFACE Z (ZF)	Input the workpiece top surface position.	-99999.999 to 99999.999mm
4	ALLOWANCE Z (H)	Input the machining depth from the workpiece top surface with the addition input method.	0.001 to 99999.999mm
5	FIN ALLOW Z (FH)	Input the finishing allowance of the bottom surface. The rough machining is performed to the bottom surface, remaining the finishing allowance.	0.000 to 99999.999mm

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.3 Screens Related to the Process Edit Functions

(Continued from the previous page)

No.	Display item	Details	Setting range
6	FIN ALLOW XY (FE)	Input the finishing allowance of the side surface. The rough machining is performed to the side surface, remaining the finishing allowance.	0.000 to 99999.999mm
7	NUM OF CUTS Z (ZT)	Input how many times the workpiece is cut in the Z-axis direction when performing the machining. The cutting amount for one time is as follows. Rough machining: (Allowance – Finishing allowance) / Number of cuttings Finishing machining of side surface: Allowance/Number of cuttings	1 to 99999 times
8	CUT TYPE	Input the cutting type (down-cutting, up-cutting). 1: DOWN 2: UP	1, 2
9	PATTERN	The machining pattern is displayed. SQUARE CIRCLE L PATTERN U PATTERN TRACK The machining pattern is changed on the machining pattern screen.	-
10	SP SPEED	The spindle rotation speed is displayed. The spindle rotation speed is changed on the setting screen for cutting conditions.	1 to 99999 rev/min
11	FEED RATE (XY)	The feedrate in the XY-direction is displayed. The feedrate is changed on the setting screen for cutting conditions.	0.001 to 60000.000 mm/min

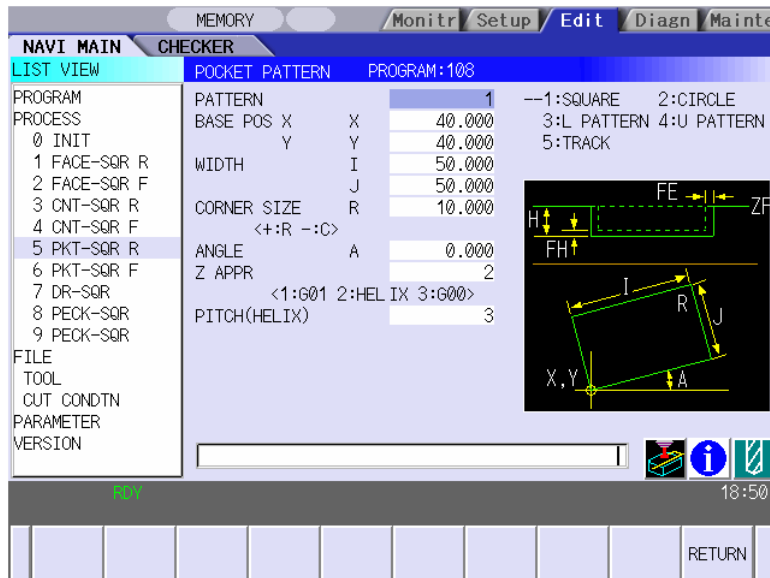
Menus

No.	Menu	Details
1	←	Turns the LIST VIEW area active.
2	CUT CONDTN	Displays the setting screen for the cutting conditions.
3	PATTERN	Displays the selection screen for the machining pattern.
4	SAVE	Saves the changes in the process. If an illegal parameter exists when saving, an error message will appear. If the input parameter is illegal, the cursor moves to the illegal parameter position. If the input parameters for the pattern input screen or cutting condition screen are illegal, the screen name and error message will be displayed.

(2) Pocket Pattern Screen

The parameters for the pocket pattern are set on this screen. When the [PATTERN] menu is pressed on the pocket screen, this screen is displayed.

Screen layout



Screen display items

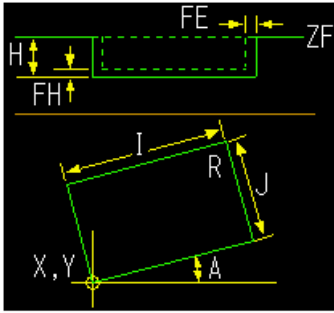
No.	Display item	Details	Setting range
1	PATTERN	Input the type of the pocket pattern. <1: SQUARE > The cutting pattern of square shape <2: CIRCLE > The cutting pattern of circular shape <3: L PATTERN> The cutting pattern of L-shape <4: U PATTERN> The cutting pattern of U-shape <5. TRACK> The cutting pattern of track shape	1 to 5

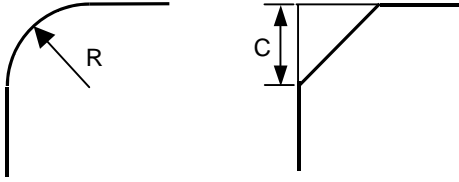
(Note) Display items No.2 and later differ according to each pattern.

4. SCREEN SPECIFICATIONS

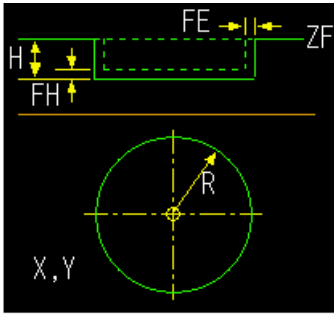
4.3 Screens Related to the Process Edit Functions

- Parameters for SQUARE



No.	Display item	Details	Setting range
2	BASE POS X (X)	Input the base position X of the square.	-99999.999 to 99999.999mm
3	BASE POS Y (Y)	Input the base position Y of the square.	-99999.999 to 99999.999mm
4	WIDTH (I)	Input the width of the X-direction in the square shape machining.	0.001 to 99999.999mm
5	WIDTH (J)	Input the width of the Y-direction in the square shape machining.	0.001 to 99999.999mm
6	CORNER SIZE (R)	Input the corner size. Positive value: Corner R, Negative value: Corner C 	-99999.999 to 99999.999mm
7	ANGLE (A)	Input the workpiece's inclination angle between the square and X-axis.	-359.999 to 360.000°
8	Z APPR	Select the G code that is commanded in the Z approach. 1: Linear interpolation (G01) 2: Helical (G02/G03) 3: Positioning (G00)	1 to 3
9	PITCH (HELIX)	The helical operations of the times specified in the pitch are operated. "Tool diameter/2" is applied to the helical radius at this time. (Note) This data is valid only when "HELIX" is selected with the Z approach method.	1 to 99

• Parameters for CIRCLE

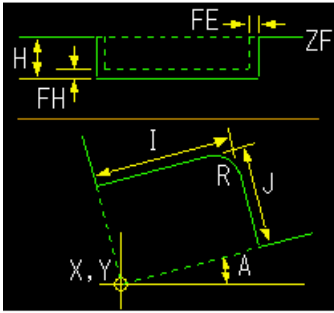


No.	Display item	Details	Setting range
2	BASE POS X (X)	Input the center position X of the circle.	-99999.999 to 99999.999mm
3	BASE POS Y (Y)	Input the center position Y of the circle.	-99999.999 to 99999.999mm
4	RADIUS (R)	Input the radius of the circle.	0.001 to 99999.999mm
5	Z APPR	Select the G code that is commanded in the Z approach. 1: Linear interpolation (G01) 2: Helical (G02/G03) 3: Positioning (G00)	1 to 3
6	PITCH (HELIX)	The helical operations of times specified in the pitch are operated. "Tool diameter/2" is applied to the helical radius at this time. (Note) This data is valid only when "HELIX" is specified with the Z approach method.	1 to 99

4. SCREEN SPECIFICATIONS

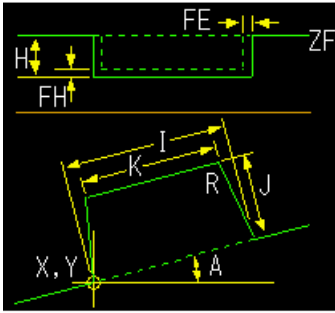
4.3 Screens Related to the Process Edit Functions

- Parameters for L PATTERN

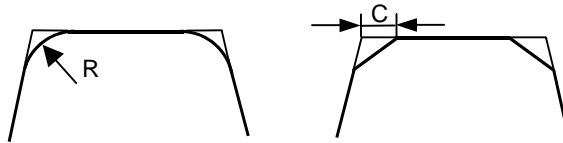


No.	Display item	Details	Setting range
2	BASE POS X (X)	Input the base position X.	-99999.999 to 99999.999mm
3	BASE POS Y (Y)	Input the base position Y.	-99999.999 to 99999.999mm
4	WIDTH (I)	Input the width of the X-direction in the L-shape.	0.001 to 99999.999mm
5	WIDTH (J)	Input the width of the Y-direction in the L-shape.	0.001 to 99999.999mm
6	CORNER SIZE (R)	Input the corner size. Positive value: Corner R, Negative value: Corner C	-99999.999 to 99999.999mm
7	ANGLE (A)	Input the workpiece's inclination angle of the L-shape as seen from the X-axis.	-359.999 to 360.000°

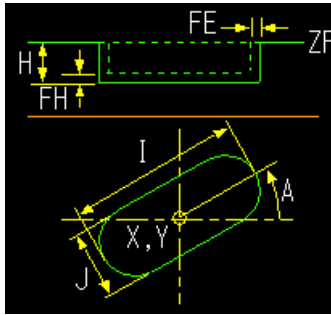
• Parameters for U PATTERN



No.	Display item	Details	Setting range
2	BASE POS X (X)	Input the base position X.	-99999.999 to 99999.999mm
3	BASE POS Y (Y)	Input the base position Y.	-99999.999 to 99999.999mm
4	WIDTH (I)	Input the width of the opening part in the U-shape.	0.001 to 99999.999mm
5	WIDTH (J)	Input the depth in the U-shape.	0.001 to 99999.999mm
6	WIDTH (K)	Input the length of the opposite side of the opening side in the U-shape.	0.001 to 99999.999mm
7	CORNER SIZE (R)	Input the corner size. Positive value: Corner R, Negative value: Corner C	-99999.999 to 99999.999mm
8	ANGLE (A)	Input the workpiece's inclination angle of the U-shape as seen from the X-axis.	-359.999 to 360.000°



• Parameters for TRACK



No.	Display item	Details	Setting range
2	BASE POS X (X)	Input the center position X of the track shape.	-99999.999mm to 99999.999mm
3	BASE POS Y (Y)	Input the center position Y of the track shape.	-99999.999mm to 99999.999mm
4	WIDTH (I)	Input the width of the track machining operation in the X axis direction.	0.001mm to 99999.999mm
5	WIDTH (J)	Input the width of the track machining operation in the Y axis direction.	0.001mm to 99999.999mm
6	ANGLE (A)	Input the inclination angle between the track shape and X-axis.	-359.999° to 360.000°
7	Z APPR	Select the G code that is commanded in the Z approach. 1: Linear interpolation (G01) 2: Helical (G02/G03) 3: Positioning (G00)	1 to 3
8	PITCH (HELIX)	The helical operations of the times specified in the pitch are operated. "Tool diameter/2" is applied to the helical radius at this time. (Note) This data is valid only when "HELIX" is selected with the Z approach method.	1 to 99

Menus

No.	Menu	Details
1	RETURN	Returns the screen to the pocket screen.

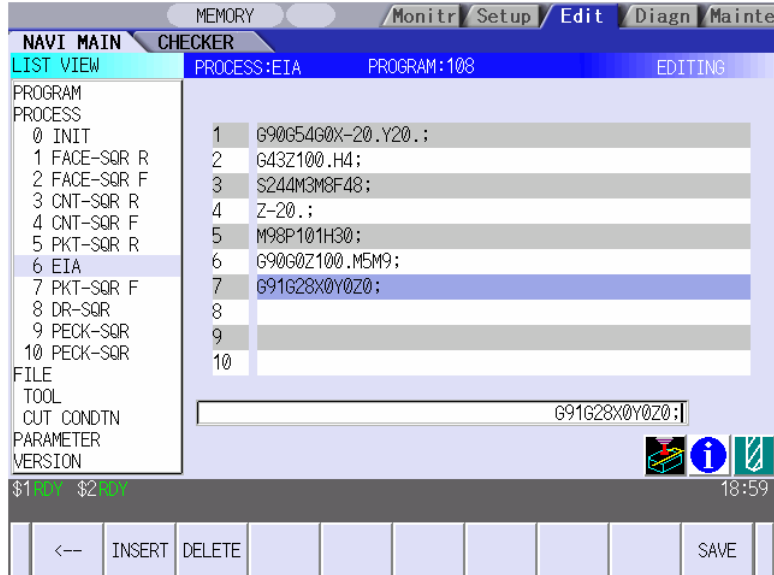
(3) Cutting Condition Screen

The cutting conditions are set on this screen. When the [CUT CONDTN] menu is pressed on each process edit screen, this screen is displayed. Refer to "(3) Cutting Condition Screen" in the section "4.3.6 Hole Drilling" for details.

4.3.10 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> <ul style="list-style-type: none"> Characters that can be input into the EIA block are up to 50 characters. 	EIA code Max. 10 blocks

Menus

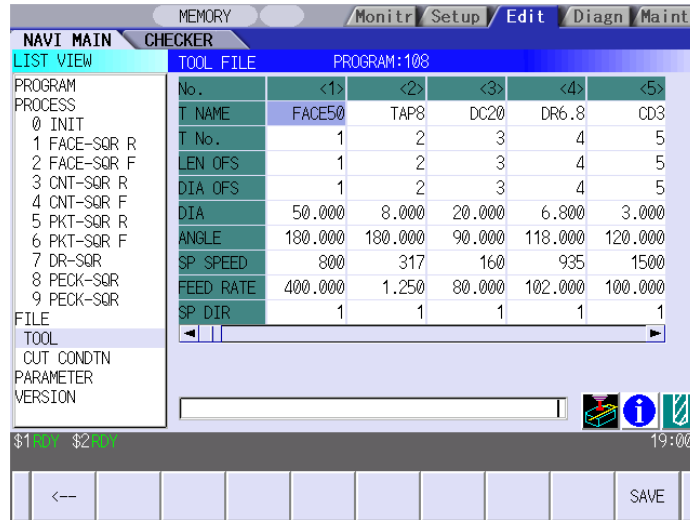
No.	Menu	Details
1	<--	Turns the LIST VIEW area active.
2	INSERT	Inserts a blank block before the block where the cursor exists.
3	DELETE	Deletes the data of the block where the cursor exists.
4	SAVE	Saves the changes in the process.

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.

Screen layout



Screen display items

No.	Display item	Details	Setting range
1	No.	This is the tool registration No. set with the machining condition. (1 to 99)	-
2	T NAME	Specify the tool name.	Max. 6 alphanumeric characters
3	T No.	Specify the tool No. to be used. (T function code data output as the NC data)	0 to 9999
4	LEN OFS	Specify the tool length offset No. The maximum value for the H is changed according to the specifications.	1 to number of tool sets
5	DIA OFS	Specify the tool diameter offset No. The maximum value for the D is changed according to the specifications.	1 to number of tool sets
6	DIA	Input the tool diameter.	0.001 to 9999.999 mm
7	ANGLE	Input the tool nose angle.	0.001 to 180.000°
8	SP SPEED	Input the standard rotation speed of the spindle.	1 to 99999 rev/min

(Continued to the next page)

(Continued from the previous page)

No.	Display item	Details	Setting range
9	FEED RATE	Input the standard feedrate. When tapping, input the pitch (mm/rev).	0.001 to 60000.000 mm/min 0.001 to 999.999 mm/rev
10	SP DIR	Input the spindle rotation direction.	1: CW 2: CCW

Menus

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

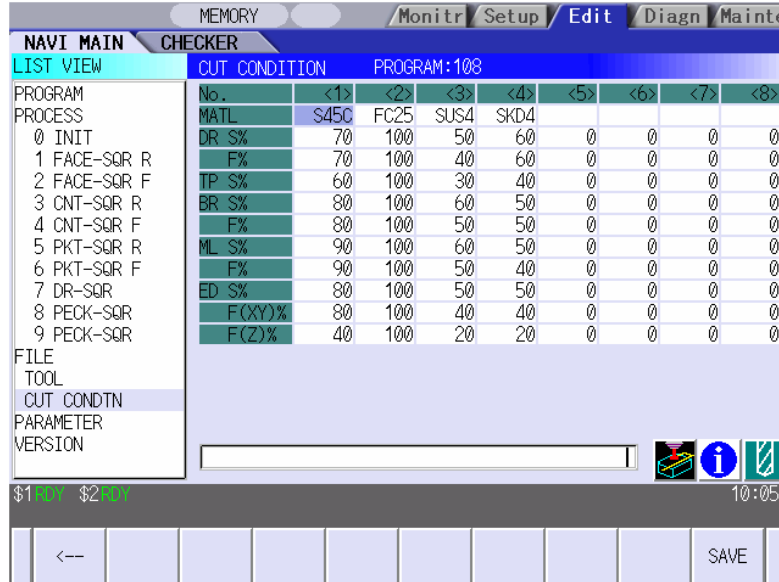
⚠ CAUTION

⚠ 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.

4.4.2 Cutting Condition File Screen

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




Screen display items

No.	Display item	Details	Setting range
1	No.	This is the workpiece registration No. input in the initial condition setting screen. (1 to 8)	-
2	MATL	Input the name that indicates the material of the workpiece.	Max. 5 alphanumeric characters
3	S%	Input the rate (%) of the applicable rotation speed to the standard rotation speed of the tool file by the tool type. <Tool type> DR: For drilling TP: For tapping BR: For boring ML: For face cutting ED: For contour cutting and pocket	0 to 200 %
4	F%	Input the rate (%) of the applicable feedrate to the standard feedrate of the tool file by the tool type. (The tool type is the same as the S%.)	0 to 200 %

Menus

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

 **CAUTION**

-  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.

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)	M0 OUTPUT	Specify whether to output the M0 code at the beginning of each process (right before the execution of the tool function). 0: INVALID 1: VALID	0, 1
2 (102)	M6 OUTPUT	Specify whether to output the M6 code when "NEXT TOOL PREP" is invalid. 0: INVALID 1: VALID	0, 1
3 (103)	NEXT TOOL PREP	Specify whether to prepare the tool for the next process program beforehand or not. 0: INVALID 1: VALID (M6T□ is commanded in one block.) 2: VALID (M6T□ is commanded in two blocks.)	0, 1, 2
4 (104)	M19 OUTPUT	Specify whether to validate the orientation function after the spindle is stopped. 0: INVALID 1: VALID	0, 1

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.5 Screen Related to the Parameters

(Continued from the previous page)

No.	Display item	Details	Setting range
5 (105)	ATC POS Z	Select a number to determine the position of table when ATC is performed. 0: 1st zero point 1: 2nd zero point	1, 2
6 (106)	SEQUENCE No. OUTPUT	Specify whether to output sequence No. in each process of the machining program. 0: Do not output 1: Output	0, 1
7 (201)	HOLE Z CLEARANCE	The distance from the R-point, where the cutting feed begins, to the hole top position is set.	0.001 to 99999.999mm
8 (202)	SYNC TAP	Select "Asynchronous tap: 0" or "Synchronous tap: 1" in the "Tap cycle (C=5)" machining. 0: INVALID (ASYNC) 1: VALID (SYNC)	0, 1
9 (301)	FACE CUT WIDTH PCT (%)	In the face cutting machining, when the machining is performed to the second step after the machining for the first step, the machining is performed with the tool overlapping the machining width of the first step. Set such overlap of the tool ("Overlap percentage") with "%". For example, if the overlap percentage is 70% when the machining is performed with the tool of $\phi 100$, the machining is performed to the second line in the width of maximum 70mm. When this data is not input, 70% will be applied.	1 to 100%
10 (302)	FACE Z CLEARANCE	After the first machining is completed, set the return point of the Z-axis with the distance from the machining surface height. At the second time or later, the Z-axis will be returned in this clearance amount from the machining surface.	0.001 to 99999.999mm
11 (303)	FACE XY CLEARANCE	Set the cutting start position in the face machining, and set the clearance amount from the workpiece with the distance from the workpiece face.	0.001 to 99999.999mm
12 (304)	FACE Z APPROACH	When the positioning is performed in the face machining, the Z-axis is moved to the position set in the FACE Z CLEARANCE with the rapid traverse. Set which the rapid traverse or the cutting feed is performed from that set position to the cutting start position. 1: RAPID (G00) 2: CUT (G01)	1, 2

(Continued to the next page)

4. SCREEN SPECIFICATIONS

4.5 Screen Related to the Parameters

(Continued from the previous page)

No.	Display item	Details	Setting range
13 (401)	E-ML CUT WIDTH PCT(%)	In the contour/pocket machining, when the machining is performed to the second step after the machining for the first step, the machining is performed with the tool overlapping the machining width of the first step. Set such overlap of the tool ("Overlap percentage") with "%". For example, if the overlap percentage is 70% when the machining is performed with the tool of $\phi 100$, the machining is performed to the second line in the width of maximum 70mm. When this data is not input, 50% will be applied.	1 to 100%
14 (402)	E-ML Z CLEARANCE	After the first machining is completed, set the return point of the Z-axis with the distance from the machining surface height. At the second time or later, the Z-axis will be returned in this clearance amount from the machining surface.	0.001 to 99999.999mm
15 (403)	E-ML XY CLEARANCE	Set the cutting start position in the contour/pocket machining, and set the clearance amount from the workpiece with the distance from the workpiece face.	0.001 to 99999.999mm
16 (404)	E-ML SHIFT FEED RATE PCT(%)	Set the speed coefficient for the feed of the shift process (first cutting of the side surface) in the contour/pocket machining.	1 to 100%
17 (405)	E-ML EMPTY D OFS NUM	Set the temporary offset No. to set the offset of the tool diameter in the contour/pocket machining.	1 to tool sets
18 (406)	E-ML Z APPROACH	In "Contour cutting" CIRCLE or SQUARE, select "cutting feed" or "rapid traverse" as speed rate for which a movement from the positioning point to the cutting start position is carried out. 1: Cutting feed (G01) 2: Rapid traverse (G00)	1,2
19 (999)	MAINTE	Specify whether to validate the maintenance operations. 0: INVALID 1: VALID	0,1

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 MILL 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 MAINTEN", is pressed.

Screen layout

<When NAVI MILL for 700 is used>



Screen display items

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

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

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

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

Menu

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 of the PREFERENCE data. (Note)

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

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

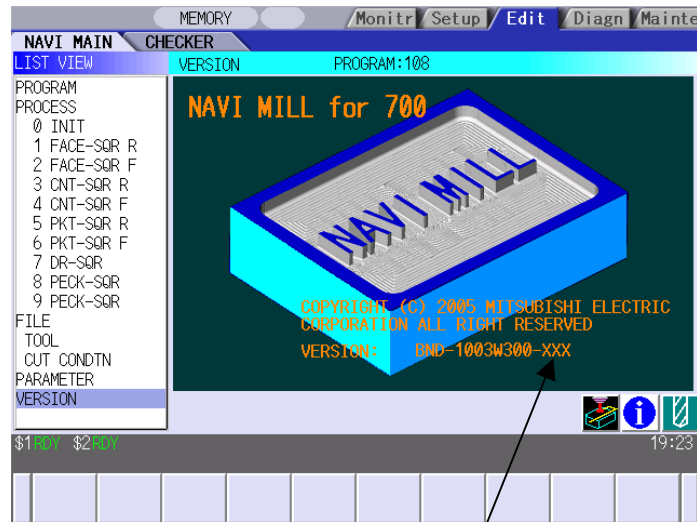
4.6 Screen Related to the Version

4.6.1 Version Screen

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

Screen layout



<When NAVI MILL for 700 is used>



Version

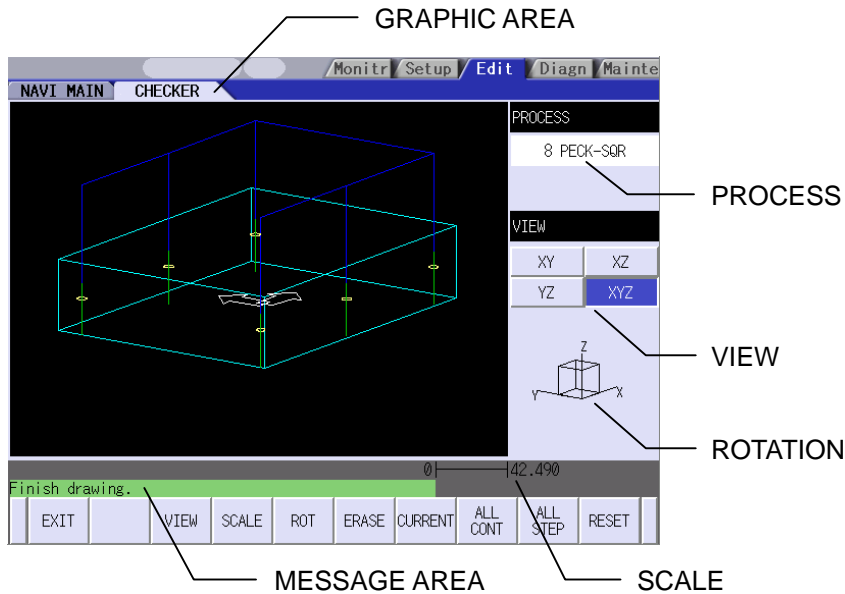
4.7 Program Checker Screen

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

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

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

Screen layout



Screen display items

No.	Display item	Details
1	GRAPHIC AREA	The workpiece shape and the tool paths are graphically displayed within this area. Items and their display colors appeared on the screen are as follows: <ul style="list-style-type: none"> • Tool paths --- Blue (Rapid traverse), Green (Cutting feed) • Program paths (Note 1) --- White • Workpiece --- Light blue • Tool mark (Cutting start point) (Note 2) --- Yellow (Note 1) Program paths instead of tool path will be displayed in the Cutter Radius Compensation mode. (Note 2) When performing hole drilling, the spot diameter will be marked on the SURFACE Z.
2	PROCESS	The name of the process of which tool paths are currently displayed is indicated here.
3	VIEW	Currently selected VIEW is displayed.
4	ROTATION	Viewpoint angle is displayed. This drawing is appeared when selecting XYZ for VIEW.
5	SCALE	Scale value of the graphic display area is indicated.
6	MESSAGE AREA	Messages on graphic display of the tool paths are appeared here.

Main menus

No.	Menu	Details
1	EXIT	Terminates the Program Checker, and then closes the screen.
2	VIEW	This menu is used when changing planes of the workpiece. Select a view from XY, YZ, XZ or XYZ. The menu will be changed to VIEW change menu by pressing this menu.
3	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.
4	ROTATION	This menu is used when changing the viewpoint angle of the XYZ (3D) drawing. The viewpoint can be moved upward, downward, or toward the left/right. The menu will be changed to ROTATION change menu by pressing this menu.
5	ERASE	Deletes the drawing data.
5	CURRENT	Tool paths of the currently selected process are displayed.
6	ALL CONT	Tool paths of the entire processes are displayed.
7	ALL STEP	Tool paths of each process are displayed one at a time.
8	RESET	Reset the graphic display of the tool paths.

VIEW change menu

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

No.	Display Item	Details
1	CANCEL	Returns to the main menu.
2	XY	Convert VIEW into the X-Y plane display format, and returns to the main menu.
3	YZ	Convert VIEW into the Y-Z plane display format, and returns to the main menu.
4	XZ	Convert VIEW into the Z-X plane display format, and returns to the main menu.
5	XYZ	Convert VIEW into the 3 dimensional display format, and returns to the main menu.

(Note 1) [VIEW] menu is not available while graphic display is performed; press [RESET] menu and cancel the graphic display in advance.

(Note 2) The displayed tool paths will be deleted upon change of plane.

SCALE change menu

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. (Note) 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.
7	→	Moves the scale frame toward the right. The same function can also be achieved by pressing → key.
8	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 tool paths will be deleted upon change of display scale or position.

ROTATION change menu

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

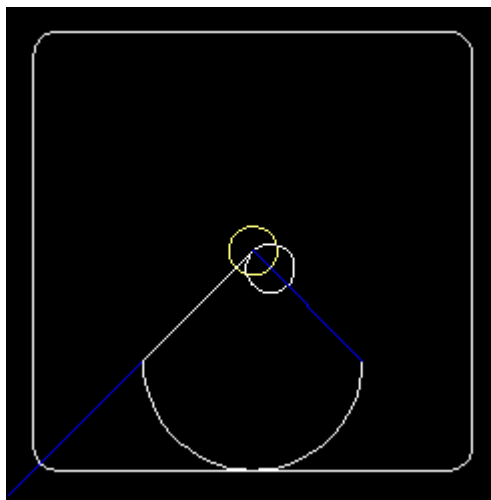
No.	Display item	Details
1	CANCEL	Cancels the ROTATION change and returns to the main menu.
2	↑	Rotates the viewpoint angle in a vertical-forward direction.
3	↓	Rotates the viewpoint angle in a vertical-backward direction.
4	←	Rotates the viewpoint angle in a horizontal-clockwise direction.
5	→	Rotates the viewpoint angle in a horizontal-counter-clockwise direction.
6	SET	Determines the viewpoint angle and returns to the main menu. The same result can also be achieved by pressing [Input].

(Note 1) The displayed tool paths will be deleted upon change of display angle.

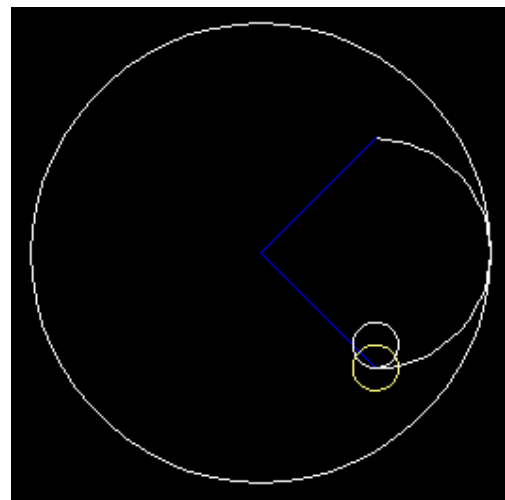
Restrictions on the graphic display function

- Graphic display is not available for the EIA process.
- Graphic displays for Corner C and Corner R show images for reference. The actual movement of tool is different from the display.
- NAVI MILL on PC is not compatible with Cutter Radius Compensation. Therefore, on this Program Checker screen, program path instead of tool path will be displayed in the Cutter Radius Compensation mode. A circled mark indicating the amount of tool diameter offset is shown at the program path starting point.
- Contour Free machining, Pocket L-pattern machining and Pocket U-pattern machining are performed while changing the amount of tool diameter offset. Therefore, multiple circled marks indicating the amount of tool diameter offset are shown at the program path starting point.
- When there is an error in the specified shape data for the counter free machining, the shape data is displayed up to the error point.
- Cutter Radius Compensation is used for the side surface finishing of the Contour/Pocket machining. Program path will be displayed in the Cutter Radius Compensation mode.

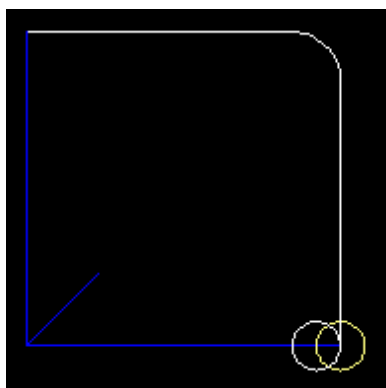
[Contour/Pocket square]



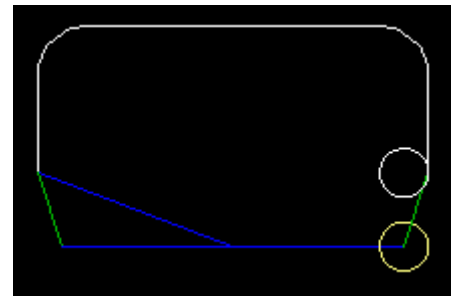
[Contour/Pocket circle]



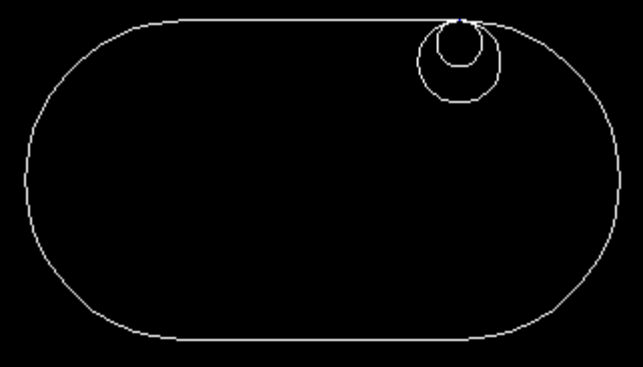
[Pocket L pattern]



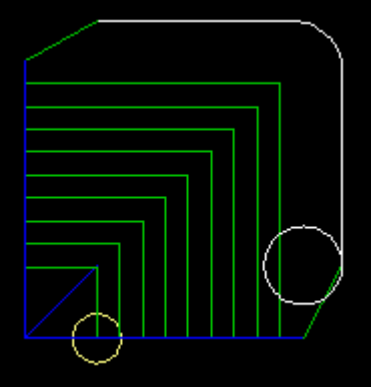
[Pocket U pattern]



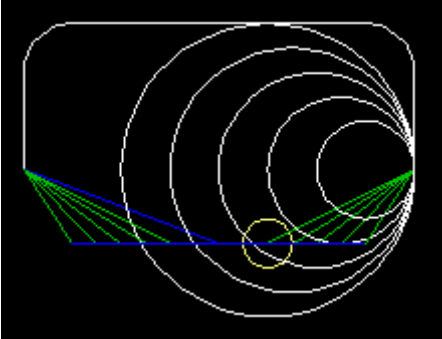
[Contour free]



[Pocket L pattern]



[Pocket U pattern]







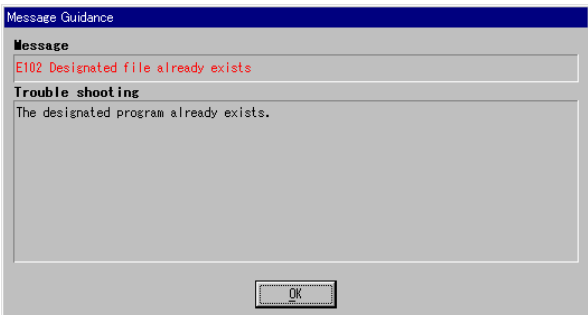


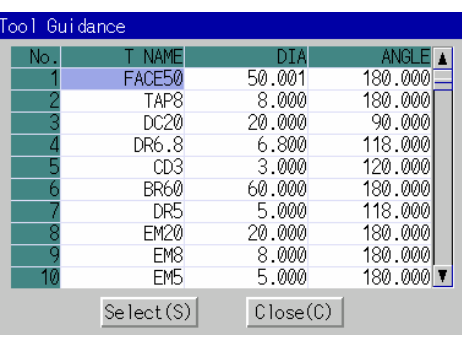
4.8 Guidance Function

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			<p>Details or countermeasures related to the current error and message are displayed.</p> 
Tool Guidance			<p>A segment of tool data registered in the tool file is displayed. Note that no editing is possible.</p> 

4.8.1 Tool Guidance Screen

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

Screen layout

No.	T NAME	DIA	ANGLE
1	FACE50	50.001	180.000
2	TAP8	8.000	180.000
3	DC20	20.000	90.000
4	DR6.8	6.800	118.000
5	CD3	3.000	120.000
6	BR60	60.000	180.000
7	DR5	5.000	118.000
8	EM20	20.000	180.000
9	EM8	8.000	180.000
10	EM5	5.000	180.000

Select(S) Close(C)

Screen display items

No.	Display item	Details
1	No.	This is the tool registration No. set with the machining condition. (1 to 99)
2	T NAME	Displays the tool name.
3	DIA	Displays the tool diameter.
4	ANGLE	Displays the tool nose angle.

Button

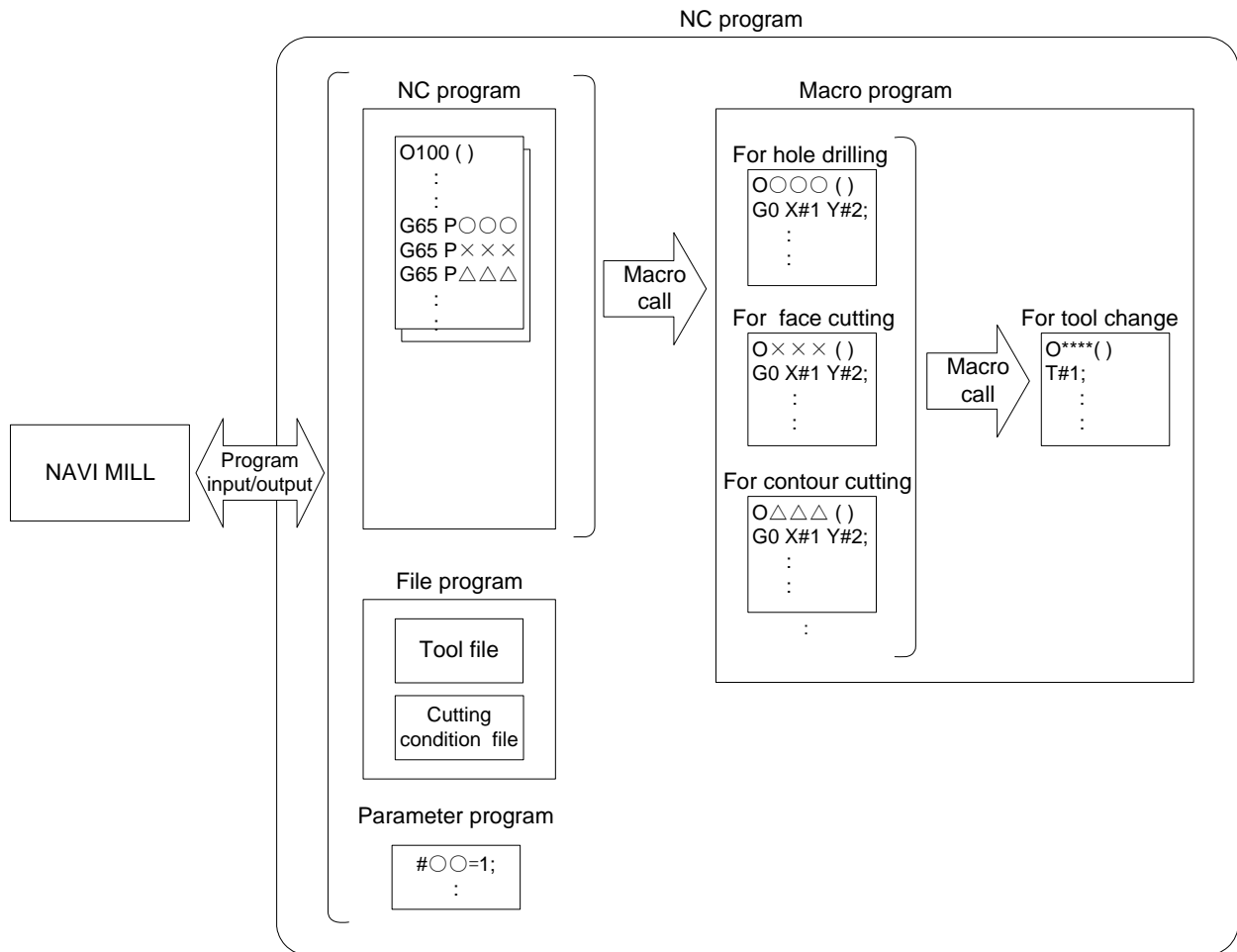
No.	Button	Application
1	Select	The tool registration No. at the cursor position is set to "TOOL REG No." in each process screen. This button is valid only when the tool guidance screen is opened while the cursor is at the "tool registration No." in each process screen.
2	Close	This closes the tool guidance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

5. PROGRAM SPECIFICATIONS

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

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

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



5.1 NC Program

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

5.1.1 Output Method for NC Program

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

Process	Machining program
Face cutting (Square)	(NAVI-FACE-SQR); ••• (/NAVI);
Hole drilling (Deep hole Line)	(NAVI-HOLE-PECK-LINE); ••• (/NAVI);
Hole drilling (Drill Line)	(NAVI-HOLE-DRILL-LINE); ••• (/NAVI);
Hole drilling (Drill Line)	(NAVI-HOLE-TAP-LINE); ••• (/NAVI);
Pocket (Square)	(NAVI-POCKET-SQR); ••• (/NAVI); •••

Process start comment

Process	Comment	Remarks
Initial setting	(NAVI-INIT);	
Hole drilling	Drilling (NAVI-HOLE-DRILL-****);	The symbol which indicates the machining pattern of the hole drilling is set in the "****" part. <Pattern symbol> Random (RNDM), Linear (LINE), Arc (ARC), Circle (CIR), Square (SQR), Grid (GRID)
	Pecking (NAVI-HOLE-PECK-****);	
	Step (NAVI-HOLE-STEP-****);	
	Boring (NAVI-HOLE-BORE-****);	
	Tapping (NAVI-HOLE-TAP-****);	
	Helical boring (NAVI-HOLE-HELIX);	
Face cutting	Square (NAVI-FACE-SQR);	
	Circle (NAVI-FACE-CIR);	
Contour cutting	Square (NAVI-CONTOUR-SQR);	
	Circle (NAVI-CONTOUR-CIR);	
	Free (NAVI-CONTOUR-FREE);	
Pocket	Square (NAVI-POCKET-SQR);	
	Circle (NAVI-POCKET-CIR);	
	L pattern (NAVI-POCKET-UPT);	
	U pattern (NAVI-POCKET-LPT);	
	Track (NAVI-POCKET-TRK);	
EIA	(NAVI-EIA);	
End process	(NAVI-FIN);	

Process data

Process		Program block	Remarks
Initial setting		G65 P9110 A B C D E F...Z;	
Hole drilling	Random	G65 P9120 A B C D E F...Z; X10. Y10.; X20. Y20.; ... ; G80; G00 Z#159; G49; M5 M9;	Common in drilling, pecking, step, boring and tapping. ← Hole position data
	Line	G65 P9120 A B C D E F...Z;	
	Arc		
	Circle		
	Square		
	Grid		
	Helical boring	G65 P9123 A B C D E F...Z;	
Face cutting	Square	G65 P9130 A B C D E F...Z;	
	Circle		
Contour cutting	Square	G65 P9140 A B C D E F...Z;	The macro program No. depends on the machining pattern.
	Circle		
	Free	(ROUGH-CENTER) #5=3; #1=0; WHILE[#5 GT 0.] DO1; G65 P9140 A B C D E F...Z; G00 X10. Y10.; ← Start position data G01 Z#180 F160.0000; F320.0000; G1 X20. Y20.; G1 X30. Y30.; ← Shape data ... ; G91 G0 Z#176; G90; #1=1; #5=#5-1; END1; G00 Z#159; G49; M5 M9;	Ex.) Machining type: ROUGH Tool path: CENTER Block depends on the machining type (ROUGH, FIN.) and tool path.

Process		Program block	Remarks
Pocket	Square	G65 P9150 A B C D E F...Z;	The macro program No. depends on the machining pattern.
	Circle		
	L pattern		
	U pattern		
	Track		
EIA		...;	
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 MTB macro is selected, refer to the section 5.4.

Process end comment

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

5.1.2 Restrictions

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

(1) Deleting block

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

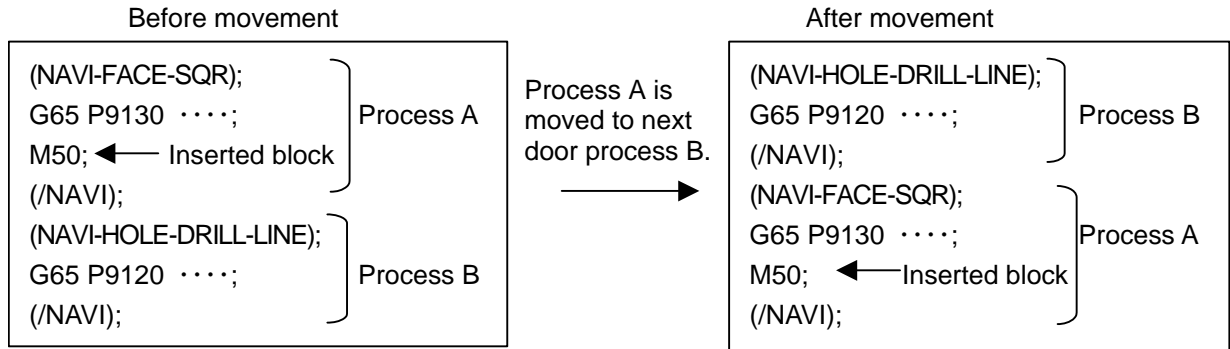
(2) Inserting block

If a block is inserted into the process of the NC program (between the process start comment and process end comment), the inserted block will not be recognized in most cases while NAVI MILL is editing the process. Note that if NAVI MILL edits the process which a block is inserted into, the block may be lost. Inserting a block between the processes of the NC program (between the process end comment and next process start comment) has no problem.

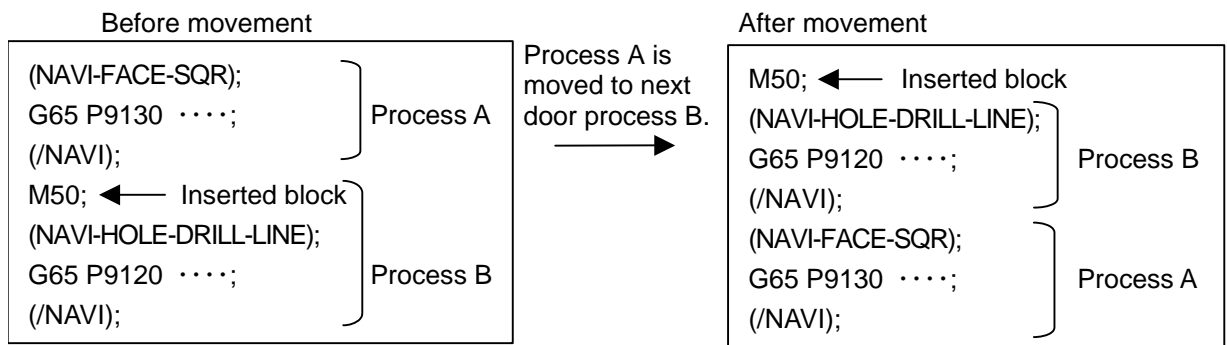
In response to operating process (moving process, deleting process, copying process) with the NAVI MILL, an inserted block is operated as follows.

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 (A inserted block exists in the process.)



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



(3) Changing process data

If the contents of the macro program call block in the process data is changed, editing the program with the NAVI MILL 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 the NAVI MILL's each 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	9112	100019112	CUT CONDITION FILE

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

5.3 Parameter Program

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

<Program No., Comment>

No.	Name	User macro No.	MTB macro No.	Program comment
1	Parameter	9113	100019113	PARAMETER

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

5.4 Macro Program

This program is called from the NC program.

(Macro program will be registered in the NC memory of MELDAS 700/70 Series in which NAVI MILL 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 hole drilling process	9120 to 9123	100019120 to 100019123	HOLE MACRO HOLE-HELIX-MACRO
3	Macro program for face cutting process	9130	100019130	FACE MACRO
4	Macro program for contour cutting process	9140 to 9147	100019140 to 100019147	CNT-SQR-MACRO CNT-CIR-MACRO CNT-FREE-MACRO
5	Macro program for pocket process	9150 to 9155	100019150 to 100019155	PKT-SQR-MACRO PKT-CIR-MACRO PKT-LTYPE-MACRO PKT-UTYPE-MACRO PKT-TRACK-MACRO
6	Macro program for tool change	9102	100019102	TOOL-CHANGE
7	Macro program for end process	9190	100019190	END-MACRO
8	Macro program for parameter setting	-	100019103	PARAM-SET-MACRO
9	Macro program for variable control	-	100019105	VARIABLE-CTRL-MACRO

6. TOOL FUNCTIONS

The tool change methods that are applicable to NAVI MILL are as shown below.

Tool Change Method*		0 (Without Next Tool Stand-by Function)	1 (With Next Tool Stand-by Function)	2 (With Next Tool Stand-by Function)
Machining Program	Initial Sec.		T1;	T1;
	Process Sec.	M6T1 ; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T1</div>	M6T2; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T1</div>	M6; T2; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T1</div>
		M6T2; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T2</div>	M6T3; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T2</div>	M6; T3; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T2</div>
		M6T3; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T3</div>	M6; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T3</div>	M6; <div style="border: 1px dashed black; padding: 2px; display: inline-block;">Process with T3</div>
End Sec.**	M6T□;	T□; M6;	T□; M6;	

* Tool change method is set on the parameter screen (103 NEXT TOOL PREP).

** Tool change command of the end sec. is output when "END TOOL No." on the initial conditions setting screen is filled with specific number other than zero.

6.1 Tool Change Command (M6) and T-Command

In this system, tool number is assigned per processing.

If there is next tool stand-by function, have the machining program ready so that the T-command for the next process will be output one process ahead. When using the same tool for the multiple consecutive processes, program so that the tool change command and T-command will not be output.

In such cases mentioned above, this system allows tool number and next tool number to be edited on the process list screen. If there is no next tool stand-by function, next tool number is remained invalid.

6.2 Editing Tool Number and Next Tool Number

The chart below shows an example of tool number / next tool number input on the process list screen. According to the chart, the tool used for the first process (FACE-SQR-R) is FACE50 with tool number 1. The tool for the second process is FACE50 with tool number 1, which is the same as the one used for the first process. The tools for the third through sixth processes are the ones with tool number 5, 4, 3, and 2. Refer to the chart below for tool numbers, next tool numbers and correspondences of the output commands.

PROCESS LIST				PROGRAM:100				1/1	Output Command	Spindle	Stand-by
No	PCS	T NAME	T	NT	H	D	S	F			
0	<INIT>	<----->	---	1	--	--	----	-----	T1		→1
1	<FACE-SQR-R>	<FACE50>	1	0	1	1	800	400.000	M6	1	
2	<FACE-SQR-F>	<FACE50>	0	5	1	1	800	400.000	T5	1	→5
3	<DR-GRID>	<CD30 >	5	4	5	5	1500	100.000	M6T4	5	1→4
4	<PECK-GRID>	<DR68 >	4	3	4	4	935	102.000	M6T3	4	5→3
5	<DR-GRID>	<DC20 >	3	2	3	3	160	80.000	M6T2	3	4→2
6	<TAP-GRID>	<TAP80 >	2	0	2	2	317	1.250	M6	2	3

【Example of tool number / next tool number input】

【Tool numbers, next tool numbers and correspondences of the output commands】

T	NT	Output Command	Remarks
=0	=0	No output command	
=0	≠0	T□	
≠0	=0	M6	
≠0	≠0	M6T□	When "2" is set to the parameter "103 NEXT TOOL PREP", the output command is commanded in two blocks.

When the tool for the first process is attached to the spindle in advance, the tool number and the next tool number are to be edited as shown below.

No	PCS	T NAME	T	NT	H	D	S	F
0	<INIT>	<----->	---	0	--	--	----	-----
1	<FACE-SQR-R>	<FACE50>	0	0	1	1	800	400.000
2	<FACE-SQR-F>	<FACE50>	0	5	1	1	800	400.000
3	<DR-GRID>	<CD30 >	5	4	5	5	1500	100.000

7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

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

Required specifications

Division	Specifications	Remarks
Additional specifications	Helical interpolation	
	Synchronous tapping cycle	
	Expansion workpiece coordinate system selection (48 sets)	This is necessary when specifying G54.1Pn (n=1 to 48) in the workpiece coordinate system.
	User macro	
	MTB macro	This is necessary when the macro program mode is MTB macro.
	Variable command 200 sets or more	
	Special fixed cycle (G34 to G37)	
	Program coordinate rotation (G68/G69)	
	Conner chamfering / Corner R	
Selection specification	S code output 8 digits BIN	As for the machining program created with the NAVI MILL, the S code is maximum 5 digits.
	Japanese/English/Chinese (traditional)	
Control parameter	#8102 interference avoidance	Validate when the machining program created with the NAVI MILL is executed.
	#8112 DECIMAL PNT-P	

Recommended specifications

Division	Specifications	Remarks
Additional specifications	Graphic trace	
	Tool path check	

8. ALARM MESSAGE

8.1 Error Message

Division	Message	Details
Common	E001 No Data setting	The data with no setting exists.
	E002 Data range over	The data exceeded a set range was input.
	E003 Setting data error	The setting data is illegal.
	E004 System error	An unexpected error exists.
	E005 No data setting on pattern screen	Incomplete data exists on the pattern screen.
	E005 No data setting on pattern screen 1st PosX,Y	The position X and Y of the first hole has not been set.
	E006 No data setting on cut condition screen	Incomplete data exists on the cutting condition screen.
	E007 Data range over on pattern screen	The data exceeded a set range was input on the pattern screen.
Program editing	E008 Data range over on cut condition screen	The data exceeded a set range was input on the cutting condition screen.
	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	E211 Process number over	The number of processes exceeded 100.
Hole machining	E221 $ZI < ZF$	"Initial Z < Workpiece top position" is applied.
	E222 $B < H$	"Tool nose depth < Hole depth " is applied.
	E223 $D > \text{Tool diameter}$	"Spot diameter > Tool diameter" is applied.
	E244 CUT AMOUNT illegal	Cut amount is illegal.
	E225 Omit number illegal	Omit No. is illegal.
	E226 Maximum hole number over	The maximum number (50 points) of holes is exceeded.
	E227 $FE > E$ (Note 2)	When HELIX is specified: "Finishing allowance $XY > \text{Allowance } XY$ " is applied.
	E228 $E > D / 2$ (Note 2)	When HELIX is specified: "Allowance $XY > \text{Hole diameter}/2$ " is applied.
	E229 Tool diameter > D (Note 2)	When HELIX is specified: "Tool diameter > Hole diameter" is applied.
	E230 Tool diameter > $D - FE * 2$ (Note 2)	When HELIX is specified: "Tool diameter > Hole diameter reduced by Finishing allowance $XY*2$ " is applied.
Face cutting machining	E231 $Q > H$	"Finishing allowance > Allowance" is applied.
	E232 CUT WIDTH > WIDTH/2	"Cutting width > Width/2" is applied.
	E233 CUT WIDTH > R	"Cutting width > Radius" is applied.
Contour/ Pocket	E241 $FH > H$	"Finishing allowance Z > Allowance Z" is applied.
	E242 $FE > E$	"Finishing allowance $XY > \text{Allowance } XY$ " is applied.
	E243 $FE > R$	"Finishing allowance $XY > \text{Radius}$ " is applied.
	E244 Tool diameter > $R * 2$	When CIRCLE and INSIDE are specified: "Tool diameter > Radius*2" is applied.
	E245 Tool diameter > WIDTH	When SQUARE and INSIDE are specified: "Tool diameter > Width" is applied.
	E247 Tool diameter * 2 > WIDTH	(When HELIX is specified:) "Tool diameter*2 > Width" is applied.
	E248 $ \text{Corner Size} > \text{WIDTH} / 2$	"Corner size > Width/2" is applied.
	E249 $\text{WIDTH } K > \text{WIDTH } I$	"Width K > Width I" is applied.
	E250 Geometry record number error	For Contour - Free: Illegal record No. is applied.
	E251 Geometry maximum record number over	For Contour - Free: The maximum number (35 points) of records is exceeded.
	E252 Geometry record number error	For Contour - Free: Illegal record No. is applied.
	E253 I,J agreement with angle L** (Note 1)	Data error in Contour - Free Linear I,J and angle are contradictory.
	E254 No end point on circumference L** (Note 1)	Data error in Contour - Free There is no end point on the circumference.
	E255 No continuity with previous line L** (Note 1)	Data error in Contour - Free Patterns are not connected.

(Continued to the next page)

(Continued from the previous page)

Division	Message	Details
Contour/ Pocket	E256 No circle L** (Note 1)	Data error in Contour - Free Circle cannot be determined from set data.
	E257 Corner C error L** (Note 1)	Data error in Contour - Free Corner C cannot be determined.
	E258 Corner R error L** (Note 1)	Data error in Contour - Free Corner R cannot be determined.
	E259 Free shape input error L** (Note 1)	Data error in Contour - Free An error is found when inputting a free shape.
	E260 Last line has corner R/C L** (Note 1)	Data error in Contour - Free Corner R/C was set in the last line.
	E261 Start point error L** (Note 1)	Data error in Contour - Free An error is found in the start point.
	E262 Corner no move L** (Note 1)	Data error in Contour - Free The block following corner C/R is not a movement command.
	E263 Corner short L** (Note 1)	Data error in Contour - Free When corner C/R is commanded, the movement amount is smaller than the corner C/R amount.
	E264 Corner short L** (Note 1)	Data error in Contour - Free When corner C/R is commanded, the movement amount of the next block is smaller than the corner C/R amount.
	E266 WIDTH I < WIDTH J (Note 2)	When PKT-TRK is specified: "Width I < Width J" is applied.
	E267 Tool diameter > WIDTH J (Note 2)	When PKT-TRK is specified: "Tool diameter > Width J" is applied.
	E268 FE > WIDTH J / 2 (Note 2)	When PKT-TRK is specified: "Finishing allowance XY > Width J / 2" is applied.
	EIA	E271 Block number over
E272 No EOB		The EOB does not exist in the EIA block.
INIT	E281 -X >= +X	-X position is greater than +X position.
	E282 -Y >= +Y	-Y position is greater than +Y position.
	E283 -Z >= +Z	-Z position is greater than +Z position.
Others	E291 Program No. duplicated	Program No.'s for the Multiple Parts NC program and the currently editing program are duplicated.
	E292 Memory over	The number of program memory characters was exceeded during macro transfer.
	E293 Program entry over	The number of program registrations was exceeded during macro transfer.
	E294 Macro transporting error	An error occurred during macro transfer.
	E295 Program running	The program is running.

(Note 1) When data error in Contour – Free occurs, line No. of the shape data is displayed following "L".

(Note 2) The following tables show the conditions for the display of messages (details of the consistency check).

(1) Hole drilling - Helical

Message	Conditions for the display of messages	
	In rough machining	In finishing machining
E227 $FE > E$	$FE \geq E$	-
E228 $E > D / 2$	$E \geq D/2$	-
E229 Tool diameter $> D$	-	Tool diameter $\geq D$
E230 Tool diameter $> D - FE * 2$	Tool diameter $\geq D$ reduced by $FE*2$	-

(2) Pocket machining - Track

Message	Conditions for the display of messages	
	In rough machining	In finishing machining
E266 $WIDTH I < WIDTH J$	Width $I \leq Width J$	
E267 Tool diameter $> WIDTH J$	Tool diameter $\geq Width J$ reduced by $FE*2$	Tool diameter $\geq Width J$
E268 $FE > WIDTH J / 2$	$FE \geq Width J / 2$	

8.2 Warning Message

Division	Message	Details
Common	W001 Illegal data exists	An illegal data exists in the program.

8.3 Operation Message

Division	Message	Details
Common	OK? (Y/N)	Message to confirm the operation. Y: Execute the operation. N: Do not execute the operation.
	Save data?(Y/N)	Message to confirm saving data Y: Save data. N: Do not save data.
	Clear the pattern data? (Y/N)	Message to confirm clearing the pattern data. Y: Clear the pattern data. N: Do not change the pattern.
	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 MILL. Edited the program that was generated with Multiple Parts function. 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...
	Designated file already exists, overwrite OK?(Y/N)	Message to confirm overwriting the program on Multiple Parts Screen Y: Overwrite the program. N: Cancel the program generation
	Generating program	Generating a program on Multiple Parts Screen
	Program generation completed	The program generation has been completed on Multiple Parts Screen
	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 MILL

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

(1) Operation variables during program operation

Variable No.		Code	Data name	Setting range	Remarks
User macro mode	MTB macro mode				
#150	#450	I/D	ATC PATTERN X (Integral part)	1 to 4	Variable for operation
			ATC PATTERN Y (Decimal part)	1 to 4	
#151	#451		ATC POSITION X	-99999.999 to 99999.999mm	Variable for operation
#152	#452		ATC POSITION Y	-99999.999 to 99999.999mm	Variable for operation
#153	#453	I/D	END TOOL No.	0 to 9999	Variable for operation
#154	#454	I/D	END PATTERN X (Integral part)	1 to 4	Variable for operation
			END PATTERN Y (Decimal part)	1 to 4	
#155	#455		END POSITION X	-99999.999 to 99999.999mm	Variable for operation
#156	#456		END POSITION Y	-99999.999 to 99999.999mm	Variable for operation
#157	#457		WORK COORDINATE	54 to 59, 101 to 148	Variable for operation
#158	#458		END M CODE	1 to 3	Variable for operation
#159	#459		INITIAL POS Z	-99999.999 to 99999.999mm	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	M0 OUTPUT	0: INVALID, 1: VALID	0	Common
#161	#461	102	M6 OUTPUT	0: INVALID, 1: VALID	0	Common
#162	#462	103	NEXT TOOL PREP	0: INVALID, 1: VALID	0	Common
#163	#463	104	M19 OUTPUT	0: INVALID, 1: VALID	0	Common
#164	#464	105	ATC POS Z	1: G28, 2: G30	1	Common
#165	#465	201	HOLE Z CLEARANCE	0.001 to 99999.999mm	10.000(mm), 0.3937(inch)	HOLE
#166	#466					HOLE
#167	#467					HOLE
#168	#468	202	SYNC TAP	0, 1	1	HOLE
#169	#469	205				HOLE
#170	#470	301	FACE WIDTH PCT (%)	1 to 100%	70(%)	FACE
#171	#471	302	FACE Z CLEARANCE	0.001 to 99999.999mm	10.000(mm), 0.3937(inch)	FACE
#172	#472	303	FACE XY CLEARANCE	0.001 to 99999.999mm	10.000(mm), 0.3937(inch)	FACE
#173	#473	304	FACE Z APPROACH	1:RAPID, 2:CUT	1	FACE
#174	#474	406	E-ML Z APPROACH	1:CUT, 2:RAPID	1	CONTOUR
#175	#475	401	E-ML CUT WIDTH PCT(%)	1 to 100%	50(%)	CONTOUR, POCKET
#176	#476	402	E-ML Z CLEARANCE	0.001 to 99999.999mm	10.000(mm), 0.3937(inch)	CONTOUR, POCKET
#177	#477	403	E-ML XY CLEARANCE	0.001 to 99999.999mm	10.000(mm), 0.3937(inch)	CONTOUR, POCKET
#178	#478	404	E-ML SHIFT FEED PCT(%)	1 to 100%	50(%)	CONTOUR, POCKET
#179	#479	405	E-ML EMPTY D OFS NUM	0 to 999	0	CONTOUR, POCKET

⚠ CAUTION

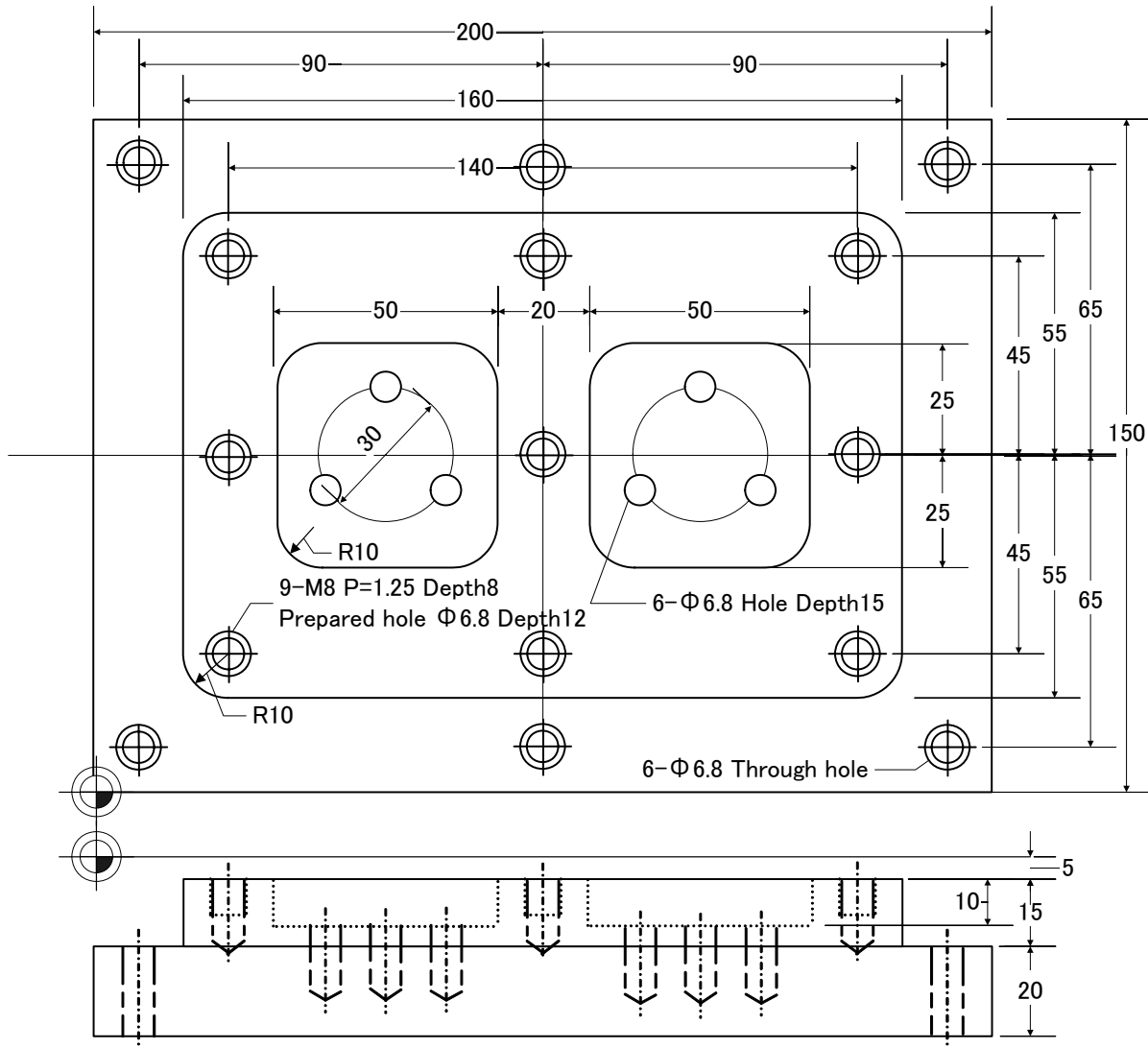
⚠ NAVI MILL uses the following variables in order to operate the NC program.

NC program mode	Variables used by NAVI MILL
User macro mode	#150 to #179
MTB macro mode	#450 to #479

When NC program mode is user macro mode, do not use common variables. 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	Face rough machining	φ50 Face Mill
	Face finishing machining	φ50 Face Mill
2	Contour rough machining	φ20 End Mill
	Contour finishing machining	φ20 End Mill
3	Pocket rough machining	φ20 End Mill
	Pocket finishing machining	φ20 End Mill
4 6-φ6.8	Drilling	φ3 Center Drill
	Drilling	φ6.8 Drill
	Drilling	φ20 Countersink
5 9-M8	Drilling	φ3 Center Drill
	Drilling	φ6.8 Drill
	Drilling	φ20 Countersink
	Tapping	M8 P=1.25 Tap
6 6-φ6.8	Drilling	φ3 Center Drill
	Drilling	φ6.8 Drill
	Drilling	φ20 Countersink

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.

Item	Face Mill	Tap	Countersink	Drill	Center Drill	End Mill
No.	1	2	3	4	5	6
T No.	1	2	3	4	5	6
LEN OFS	1	2	3	4	5	6
DIA OFS	1	2	3	4	5	6
DIA	50	8	20	6.8	3	20
ANGLE	180	180	90	118	120	180
SP SPEED	800	317	160	935	1500	800
FEED RATE	400	1.25	80	102	100	80
SP DIR	1	1	1	1	1	1

(2) Cutting condition file screen

Register the cutting conditions in each workpiece. Input the following values on the cutting condition screen.

Item	1
MATL	S45C
DR S%	70
F%	70
TP S%	60
BR S%	80
F%	80
ML S%	90
F%	90
ED S%	80
F(XY)%	80
F(Z)%	40

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
INITIAL POS Z	50	
WORK COORDINATE	54	G54
ATC PATTERN X	2	1st zero point
POSITION X	-	
PATTERN Y	2	1st zero point
POSITION Y	-	
END TOOL No.	99	
PATTERN X	2	1st zero point
POSITION X	-	
PATTERN Y	2	1st zero point
POSITION Y	-	
M CODE	1	M30

1) Press the [WORK SHAPE] menu and input the following values.

Item	Setting value	Details
WORK SHAPE	1	SQUARE
+X	200.	
-X	0	
+Y	150.	
-Y	0	
+Z	0	
-Z	-40	

- 2) Press the [RETURN] menu.
- 3) Save the initial conditions by pressing the [SAVE] menu.
- 4) 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 Face rough machining (φ50 Face Mill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the face cutting screen and set the following items.

Item	Setting value	
TOOL REG No.	1	
FACE CYCLE	1	SQUARE
PROCESS	1	ROUGH
BASE POS X	20	
Y	20	
WIDTH X I	160	
Y J	110	
SURFACE Z ZF	0	
ALLOWANCE H	5	
FINISH ALLOW Q	0.2	
NUM OF CUTS T	3	
START POINT (1 - 4)	3	
ANGLE A	0	

- 3) Press the next page key [→] and set the following items.

Item	Setting value	
CUT DIRECTION	1	X
PATH	1	RECIP
CUT WIDTH X WX	-	
Y WY	-	
SP SPEED	<720>	
FEED RATE	<360>	

- 4) Save the data of the face rough machining by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

(6) Process 1 Face finishing machining (φ50 Face Mill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following item.

Item	Setting value	
PROCESS	2	FIN.

- 3) Save the data of the face finishing machining by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
2 FACE-SQR
FILE

(7) Process 2 Contour rough machining (φ20 End Mill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the contour cutting screen and set the following items.

Item	Setting value	
TOOL REG No.	6	
PROCESS	1	ROUGH
SURFACE Z ZF	-5	
ALLOWANCE Z H	20	
ALLOWANCE XY E	20	
FIN ALLOW Z FH	3	
FIN ALLOW XY FE	3	
NUM OF CUTS ZT	3	
CUT TYPE	1	
SP SPEED	<640>	
FEED RATE	<64>	

- 3) Press the [PATTERN] menu and set the following items.

Item	Setting value	
PATTERN	1	
MACHINIG SIDE	2	
BASE POS X X	20	
BASE POS Y Y	20	
WIDTH X I	160	
WIDTH Y J	110	
CORNER SIDE R	10	
ANGLE A	0	
START POINT	3	

- 4) Save the data of the contour rough machining <SQUARE> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

(8) Process 2 Contour finishing machining (φ20 End Mill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following item.

Item	Setting value	
PROCESS	2	FIN.

- 3) Save the data of the contour finishing machining <SQUARE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
2 FACE-SQR
3 CNT-SQR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
2 FACE-SQR
3 CNT-SQR
4 CNT-SQR
FILE

(9) Process 3 Pocket rough machining (φ20 End Mill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the pocket screen and set the following items.

Item	Setting value	
TOOL REG No.	6	
PROCESS	1	ROUGH
SURFACE Z ZF	-5	
ALLOWANCE Z H	10	
FIN ALLOW Z FH	1.5	
FIN ALLOW XY FE	1.5	
NUM OF CUTS Z ZT	2	
CUT TYPE	1	
SP SPEED	<640>	
FEED RATE	<64>	

- 3) Press the [PATTERN] menu and set the following items.

Item	Setting value	
PATTERN	1	
BASE POS X X	40	
BASE POS Y Y	50	
WIDTH I	50	
WIDTH J	50	
CORNER SIDE R	10	
ANGLE A	0	
Z APPR	2	
PITCH	3	

- 4) Save the data of the pocket rough machining <SQUARE> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

(10) Process 3 Pocket finishing machining (φ20 End Mill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following item.

Item	Setting value	
PROCESS	2	FIN.

- 3) Save the data of the pocket finishing machining <SQUARE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
2 FACE-SQR
3 CNT-SQR
4 CNT-SQR
5 PKT-SQR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
2 FACE-SQR
3 CNT-SQR
4 CNT-SQR
5 PKT-SQR
6 PKT-SQR
FILE

(11) Process 3 Pocket rough machining (φ20 End Mill)

1) Move the cursor to the [5 PKT-SQR] and press the [COPY] menu.

2) Move the cursor to the [6 PKT-SQR] and press the [MOVE] menu.

3) Move the cursor to the [7 PKT-SQR] and press the [INPUT] - [Y].

4) Press the [MODIFY] - [PATTERN] menu and set the following item.

Item	Setting value	
BASE POS X X	110	

5) Save the data of the pocket rough machining <SQUARE> by pressing the [SAVE] menu.

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

(12) Process 3 Pocket finishing machining (φ20 End Mill)

1) Move the cursor to the [6 PKT-SQR] and press the [COPY] menu.

2) Move the cursor to the [7 PKT-SQR] and press the [MOVE] menu.

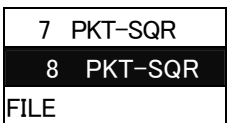
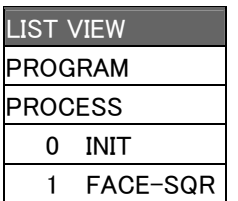
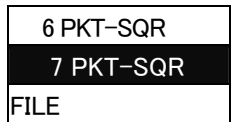
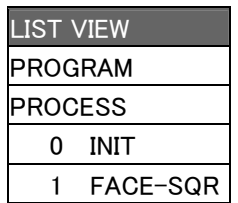
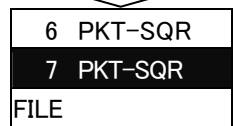
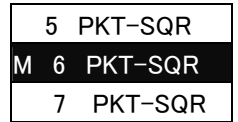
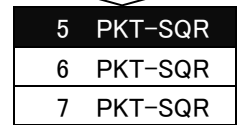
3) Move the cursor to the [8 PKT-SQR] and press the [INPUT]- [Y] menu.

4) Press the [MODIFY] - [PATTERN] menu and set the following item.

Item	Setting value	
BASE POS X X	110	

5) Save the data of the pocket finishing machining <SQUARE> by pressing the [SAVE] menu.

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



(13) Process 4 Drilling (φ3 Center Drill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the hole drilling screen and set the following items.

< Hole drilling screen>

Item	Setting value	
TOOL REG No.	5	
HOLE CYCLE	1	DRILL
INITIAL Z ZI	50	
SURFACE Z ZF	-20	
DEPTH H	3	
NOSE DEPTH B	3.866	
SPOT DIAMETER D	3	
CUT AMOUNT	-	
DWELL	0	
PATTERN	<SQUARE>	
SP SPEED	<1050>	
FEED RATE	<70>	

- 3) Press the [PATTERN] menu and set the following items.

<Hole drilling machining pattern screen (SQUARE)>

Item	Setting value	
X WIDTH	180	
X NUM OF HOLES	3	
Y WIDTH	130	
Y NUM OF HOLES	2	
BASE POS X	10	
BASE POS Y	10	
RETURN POINT	1	G98
ANGLE A	0	
ANGLE B	90	
OMIT 1	0	
OMIT 2	0	
OMIT 3	0	
OMIT 4	0	

- 4) After returning the screen to the hole drilling screen by pressing the [RETURN] menu, save the data of the hole drilling <SQUARE> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
8 PKT-SQR
9 DR-SQR
FILE

(14) Process 4 Drilling (φ6.8 Drill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	4	
HOLE CYCLE	2	PECK
DEPTH H	21	
CUT AMOUNT	2	

- 3) Save the data of the hole drilling <SQUARE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

(15) Process 4 Drilling (φ20 Countersink)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	3	
HOLE CYCLE	1	DRILL
SPOT DIAMETER D	7.8	
DWELL	0	

- 3) Save the data of the hole drilling <SQUARE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
9 DR-SQR
10 PECK-SQR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
10 PECK-SQR
11 DR-SQR
FILE

(16) Process 5 Drilling (φ3 Center Drill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the hole drilling screen and set the following items.

< Hole drilling screen>

Item	Setting value	
TOOL REG No.	5	
HOLE CYCLE	1	
INITIAL Z ZI	50	
SURFACE Z ZF	-5	
DEPTH H	3	
NOSE DEPTH B	3.866	
SPOT DIAMETER D	3	
CUT AMOUNT	-	
DWELL	0	
PATTERN	<GRID>	
SP SPEED	<1050>	
FEED RATE	<70>	

- 3) Press the [PATTERN] menu and set the following items.

<Hole drilling machining pattern screen (GRID)>

Item	Setting value	
X WIDTH	140	
X NUM OF HOLES	3	
Y WIDTH	90	
Y NUM OF HOLES	3	
BASE POS X	30	
BASE POS Y	30	
RETURN POINT	2	G99
ANGLE A	0	
ANGLE B	90	
OMIT 1	0	
OMIT 2	0	
OMIT 3	0	
OMIT 4	0	

- 4) After returning the screen to the hole drilling screen by pressing the [RETURN] menu, save the data of the hole drilling <GRID> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
11 DR-SQR
12 DR-GRID
FILE

(17) Process 5 Drilling (φ6.8 Drill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following item.

Item	Setting value	
TOOL REG No.	4	
HOLE CYCLE	2	PECK
DEPTH H	12	
CUT AMOUNT	2	

- 3) Save the data of the hole drilling <GRID> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

(18) Process 5 Drilling (φ20 Countersink)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	3	
HOLE CYCLE	1	DRILL
SPOT DIAMETER D	9	
DWELL	0	

- 3) Save the data the hole drilling <GRID> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

(19) Process 5 Tapping (M=8 P=1.25 Tap)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	2	
HOLE CYCLE	5	TAP
DEPTH H	8	
NOSE DEPTH B	8	
CUT AMOUNT	2	

- 3) Save the data the hole drilling <GRID> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR

.

.

.

12 DR-GRID
13 PECK-GRID
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR

.

.

.

13 PECK-GRID
14 DR-GRID
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR

.

.

.

14 DR-GRID
15 TAP-GRID
FILE

(20) Process 6 Drilling (φ3 Center Drill)

- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the hole drilling screen and set the following items.

< Hole drilling screen>

Item	Setting value	
TOOL REG No.	5	
HOLE CYCLE	1	
INITIAL Z ZI	50	
SURFACE Z ZF	-5	
DEPTH H	3	
NOSE DEPTH B	3.866	
SPOT DIAMETER D	3	
CUT AMOUNT	-	
DWELL	0	
PATTERN	<CIRCLE>	
SP SPEED	<1050>	
FEED RATE	<70>	

- 3) Press the [PATTERN] menu and set the following items.

<Hole drilling machining pattern screen (CIRCLE)>

Item	Setting value	
DIAMETER	30	
START ANGLE	90	
NUM OF HOLES	3	
POS X	65	
POS Y	75	
RETURN POINT	2	

- 4) After returning the screen to the hole drilling screen by pressing the [RETURN] menu, save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [←] key.

(21) Process 6 Drilling (φ6.8 Drill)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	4	
HOLE CYCLE	2	PECK
DEPTH H	15	
CUT AMOUNT	2	

- 3) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
15 TAP-GRID
16 DR-CIR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
16 DR-CIR
17 PECK-CIR
FILE

(22) Process 6 Drilling (φ20 Countersink)

- 1) Press the [COPY] menu and move down the cursor in the LIST VIEW area.
- 2) Press the [MODIFY] menu and set the following items.

Item	Setting value	
TOOL REG No.	3	
HOLE CYCLE	1	DRILL
SPOT DIAMETER D	7.8	
DWELL	0	

- 3) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [←] key.

(23) Process 6 Drilling (φ3 Center Drill)

- 1) Move the cursor to the [16 DR-CIR] and press the [COPY] menu.

- 2) Move the cursor to the [17 DR-CIR] and press the [MOVE] menu.

- 3) Move the cursor to the [19 DR-CIR] and press the [INPUT] - [Y].

- 4) Press the [MODIFY] - [PATTERN] menu and set the following item.

Item	Setting value	
POS X	135	

- 5) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 6) Turn the LIST VIEW area active by pressing the [←] key.

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
17 PECK-CIR
18 DR-CIR
FILE

16 DR-CIR
17 PECK-CIR
↓
16 DR-CIR
17 DR-CIR
18 PECK-CIR

16 DR-CIR
M 17 DR-CIR
18 PECK-CIR
↓
19 DR-CIR
FILE
↓
18 DR-CIR
19 DR-CIR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
18 DR-CIR
19 DR-CIR
FILE

(24) Process 6 Drilling (φ6.8 Drill)

- 1) Move the cursor to the [17 PECK-CIR] and press the [COPY] menu.
- 2) Move the cursor to the [18 PECK-CIR] and press the [MOVE] menu.
- 3) Move the cursor to the [20PECK-CIR] and press the [INPUT] - [Y].
- 4) Press the [MODIFY] - [PATTERN] menu and set the following item.

Item	Setting value
POS X	135

- 5) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 6) Turn the LIST VIEW area active by pressing the [←] key.

(25) Process 6 Drilling (φ20 Countersink)

- 1) Move the cursor to the [18 DR-CIR] and press the [COPY] menu.
- 2) Move the cursor to the [19 DR-CIR] and press the [MOVE] menu.
- 3) Move the cursor to the [21 DR-CIR] and press the [INPUT] - [Y].
- 4) Press the [MODIFY] - [PATTERN] menu and set the following item.

Item	Setting value
POS X	135

- 5) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 6) Turn the LIST VIEW area active by pressing the [←] key.

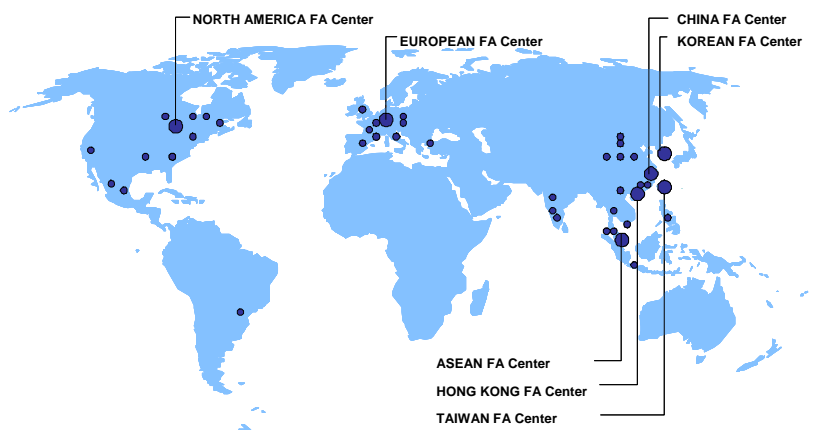
LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
19 DR-CIR
20 PECK-CIR
FILE

LIST VIEW
PROGRAM
PROCESS
0 INIT
1 FACE-SQR
.
.
.
20 PECK-CIR
21 DR-CIR
FILE

Revision History

Date of revision	Manual No.	Revision details
Jun. 2005	IB(NA)1500144-A	First edition created.
Dec. 2005	IB(NA)1500144-B	<ul style="list-style-type: none">• "4.8.1 Tool Guidance Screen" was added.• Mistakes were corrected.
Nov. 2006	IB(NA)1500144-C	Mistakes were corrected.
Mar. 2007	IB(NA)1500144-D	<ul style="list-style-type: none">• "Helical boring" was added to the hole drilling process.• "Track" was added to the pocket machining.• Explanations for 70 Series were added.• Mistakes were corrected.

Global service network



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

Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500 FAX: +1-817-416-1439

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 0J2, 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 FILDERSSTADT-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 TEKNİK SİSTEMLER LTD. STI.)
DARULACEZE CAD, FAMAŞ IS MERKEZİ 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 KRAKÓW, POLAND
TEL: +48-12-632-28-85 FAX:

Wrocław 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 CAOBANG 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 INDUSTRIAL 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
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9631 FAX: +82-2-3664-8668

Notice

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

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

Duplication Prohibited

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

© 2005-2007 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED.



MODEL	700/70 Series
MODEL CODE	008—364
Manual No.	IB-1500144 (ENG)