Changes for the Better



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

### Details described in this manual

# 

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

Refer to the following documents.

MITSUBISHI CNC 700 Series	Instruction Manual	IB-1500042
MITSUBISHI CNC 700 Series	Setup Manual	IB-1500124
MITSUBISHI CNC 700 Series	Programming Manual (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".



When the user may be subject to imminent fatalities or major injuries if handling is mistaken.



When the user may be subject to fatalities or major injuries if handling is mistaken.



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.

# 

Not applicable in this manual.

# 

Not applicable in this manual.

# 

### 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.
- $\triangle$  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)

	🛆 CAU	TION
2. Iten	ns related to installation and assembly	
9	•	e system operation. Also ground the NC uni I machine to one point, so they all have the
3. Iten	ns related to preparation before use	
0	Always set the stored stroke limit. Failure machine end.	e to set this could result in collision with the
9	Always turn the power OFF before con Failure to do so could damage the I/O de	necting/disconnecting the I/O device cable. vice and NC unit.
4. Iten	ns related to screen operation	
	the feedrate and spindle speed are autom file screen and the cutting condition file sc No." is input in the face cutting screen, the screen, the feedrate and spindle speed ar	re automatically determined. Note that the ss determined once will not be changed by nd the cutting condition file screen.
	NC program mode	Variables used by NAVI MILL
	User macro mode	#150 to #179
	MTB macro mode	#450 to #479

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.

#### 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.
- O 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)

# 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. O Do not connect or disconnect the connection cables between each unit while the power is ON. O Do not connect or disconnect the PCBs while the power is ON. $\bigcirc$ Do not connect the cable by pulling on the cable wire. Do not short circuit, charge, overheat, incinerate or disassemble the battery. $\triangle$ Dispose the spent battery according to local laws. $\triangle$ Dispose the spent cooling fan according to local laws. $\triangle$ Do not replace the control unit while the power is ON. $\triangle$ 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. $\triangle$ Do not replace the expansion PCB while the power is ON. $\triangle$ Do not replace the memory cassette while the power is ON. $\triangle$ Do not replace the cooling fan while the power is ON. $\triangle$ Do not replace the battery while the power is ON. A Be careful that metal cutting chips, etc., do not come into contact with the connector

- contacts of the memory cassette.
- $\underline{\wedge}$  Do not replace the high-speed program server unit while the power is ON.

1. OUTLIN	Ε	1
1.1	System Outline	
1.2	Input Procedures	
1.3	Screen Configuration	
1.4	Starting NAVI MILL	
1.5	Setting up NAVI MILL	
1.0		
2. FUNCTI	ONS 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	
2.5	Menu Display Area	
	DPERATIONS	
3.1	Changing Active View	
3.2	Changing Screen	
3.3	Setting Data	.14
3.4	Switching Windows	
3.5	Switching Selection Tags	.17
3.6	Inputting Operations	.18
	N SPECIFICATIONS	10
4.1	Starting NAVI MILL	
4.2	Screen Related to the Program	
1.0	4.2.1 Program Edit Screen	
4.3	Screens Related to the Process Edit Functions	
	4.3.1 Process List Screen	
	4.3.2 Multiple Parts Screen	
	4.3.3 Operating Process	
	4.3.4 Process Mode Selection Screen	
	<ul><li>4.3.5 Initial Condition Setting</li><li>4.3.6 Hole Drilling</li></ul>	
	4.3.7 Face Cutting	
	4.3.7 Face Cutting	
	4.3.9 Pocket	
	4.3.10 EIA Screen	
4.4	Screens Related to File Editing	
	4.4.1 Tool File Screen	
	4.4.2 Cutting Condition File Screen	
4.5	Screen Related to the Parameters	
4.5	4.5.1 Parameter Screen	
	4.5.2 PREFERENCE Screen	
4.6	Screen Related to the Version	
ч.0	4.6.1 Version Screen	
4.7	Program Checker Screen	
4.8	Guidance Function	
<del>.</del> .0	4.8.1 Tool Guidance Screen	

5. PROGR	RAM SPECIFICATIONS	93
5.1	NC Program	
	5.1.1 Output Method for NC Program	94
	5.1.2 Restrictions	
5.2	File Program	
5.3	Parameter Program	
5.4	Macro Program	
6. TOOL F	FUNCTIONS	
6.1	Tool Change Command (M6) and T-Command	
6.2	Editing Tool Number and Next Tool Number	100
7. RESTR	ICTIONS FOR CNC FUNCTION SPECIFICATIONS	101
8. ALARM	I MESSAGE	
8.1		
8.2	-	
8.3		
APPENDI	X 1. VARIABLES USED IN NAVI MILL	106
APPENDI	X 2. PROGRAMMING EXAMPLE	
	pendix 2.1 Machining Drawing	
App	pendix 2.2 Process Table	
App	pendix 2.3 Condition Setting	
Арр	pendix 2.4 Creating Program	110

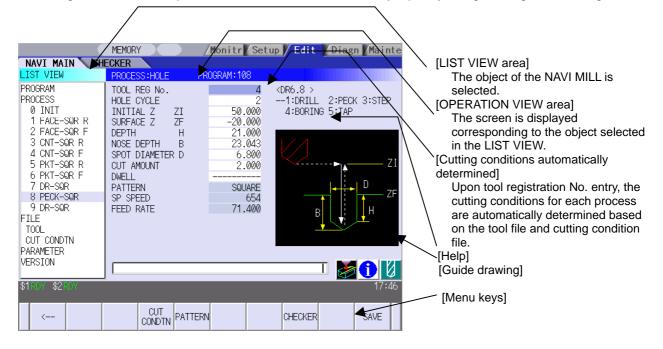
# 1. OUTLINE

### 1.1 System Outline

This manual is an instruction manual for NAVI MILL for 700 (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)
- Face cutting (Circle, square)
- Contour cutting (Circle, square, free)
- Pocket machining (Circle, square, L pattern, U pattern)
- 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.

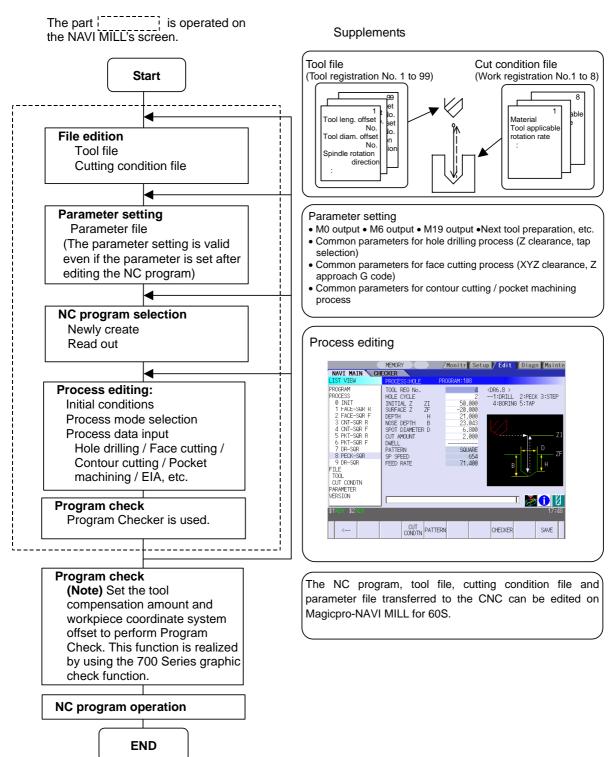


\*The operation screen size is fixed to 800(width) x 600(length).

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

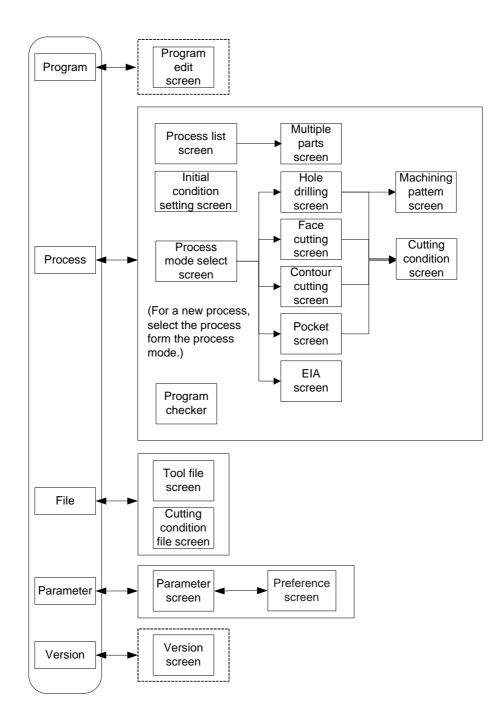
### 1.2 Input Procedures

The input procedure for the NAVI MILL is shown below.



# 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	The process mode (hole drilling, etc.) is selected.
screen	
Initial conditions setting	The initial conditions for the process program are set.
screen	
Hole drilling screen	The parameters for the hole drilling process are input.
Hole drilling machining	The parameters related to the machining pattern of the
pattern screen	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	The parameters related to the machining pattern of the
screen	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	The cutting conditions (speed rate) by each process are
screen	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 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

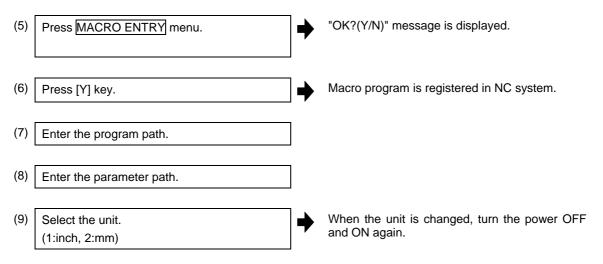
Part program output from NAVI MILL is a macro-program-based NC program. Thus, macro programs have to be registered in the NC system in advance. Also, the destinations where NC programs or NAVI MILL's reference files are saved, as well as the unit for data input, have to be specified prior to NAVI MILL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

#### NAVI MILL setup items

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

NAVI MILL setup procedures

- (1) Open PARAMETER screen. Set "999 MAINTE" to 1. (2) PREFERENCE menu is displayed. (3) Press PREFERENCE menu. PREFERENCE screen is displayed. MEMORY NAVI MAIN CHECKER Monitr Setup / Edit / Diagn Mainte LIST VIEW FROGRAM PROCESS 0 INIT 1 FACE-SGR R 2 FACE-SGR R 3 ONT-SGR R 4 ONT-SGR R 5 PKT-SGR R 7 DR-SGR 9 DR-SGR FILE TOOL CUT CONDTN PARAMETER PATH PARAMETERD:///FILE ARAMETERD:///FILE AND MEMORY -- NEM:/ > ADTA SERVER -- DS:/DIR... > ADTHER -- DRIVE:/DIR... > ADTHER -- DRIVE:/DIR... > 2--1:USER 2:MTB 2--1:inch 2:mm PARAMETER VERSION MEM:/1 🛃 🕦 🚺 MACRO ENTRY RETURN SAVE Select the macro type. (4)
  - Select the macro type. (1:Uer macro 2:MTB macro)



#### Addendum)

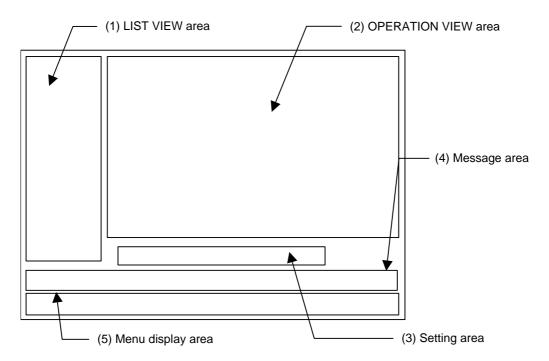
- Always carry out a macro program registration when setting up NAVI MILL or switching "MACRO" types.
- Change "PROGRAM PATH" and "PARAMETER PATH" when necessary.
- When "UNIT" is changed, turn the power OFF and ON again.
- If the tool file, cutting condition file and parameter file do not exist in "PARAMETER 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
- (2) OPERATION VIEW area
- (3) Setting area
- (4) Message area
- (5) Menu display area



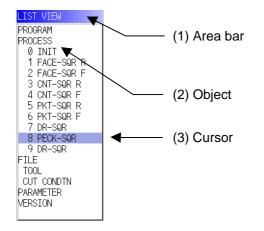
#### < Screen example>

	MEMORY	Monitr Setup / Edit / Diagn / Mainte
NAVI MAIN CH	ECKER	
LIST VIEW	PROCESS:FACE PR	OGRAM:108
PROGRAM PROCESS Ø INIT 1 FACE-SQR R 2 FACE-SQR R 3 CNT-SQR R 4 CNT-SQR R 4 CNT-SQR R 5 PKT-SQR R 6 PKT-SQR F 7 DR-SQR 8 PECK-SQR 9 DR-SQR FILE TOOL CUT CONDTN PARAMETER	TOOL REG No. FACE CYCLE PROCESS BASE POS X WIDTH X I Y J SURFACE Z ZF ALLOWANCE H FINISH ALLOW Q NUM OF CUTS T START POINT(1-4) ANGLE A CUT DIRECTION	<pre></pre>
VERSION		
\$1 RDY \$2 RDY		17:52
<	CUT CONDTN	CHECKER SAVE

#### 2. FUNCTIONS OF DISPLAY AREA

### 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	Displays the currently edited process list.
	1 DR-LINE	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.

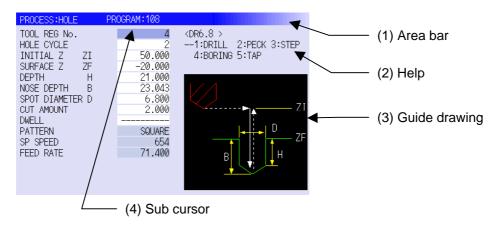
#### <Cursor Movement>

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 $\uparrow$ 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 $\downarrow$ 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.
$[\rightarrow]$ 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
[1] Cursor key	Moves the cursor one field up.
	Note that if the $\uparrow$ 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 $\downarrow$ 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]	Moves the displayed data toward the bottom.
key	

### 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.)

### 3.1 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 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 [ $\leftarrow$ ] menu key or click "LIST VIEW" with a pointing device to turn the LIST VIEW area active.

#### Operation example

1) The program edit screen is displayed.	The OPERATION VIEW area is active.
	MEMORY /Monitr Setup Edit Diagn Main
	NAVI MAIN CHECKER
	LIST VIEW PROGRAM EDIT PROGRAM:108
	PROGRAM PROGRAM LIST
	PROJESS       0       INIT         0       INIT       1         1       FACE-SQR       101       EXAMPLE1         2       FACE-SQR       101       EXAMPLE3         3       CAT-SQR       102       EXAMPLE3         4       CHT-SQR       103       EXAMPLE3         5       PAT-SQR       103       EXAMPLE3         6       PIT-SQR       106       EXAMPLE6         7       DR-SQR       106       EXAMPLE6         9       DR-SQR       106       EXAMPLE6         9       DR-SQR       106       EXAMPLE6         9       DR-SQR       106       EXAMPLE6         9       DR-SQR       106       EXAMPLE7         9       DR-SQR       107       EXAMPLE8         TOOL       CUT CONDTN       PARAMETER       VERSION         VERSION       VERSION       VERSION       VERSION       VERSION
2) Press the [ $\leftarrow$ ] menu key.	→ The LIST VIEW area will turn active.
	MEMORY /Monitr Setup / Edit / Diagn /Main NAVI MAIN CHECKER
	MEWORY /Monitr Setup Edit Diagn Main NAVI MAIN CHECKER LIST VIEW PROGRAM:108
	MEMORY /Monitr Setup / Edit / Diagn /Main NAVI MAIN CHECKER

PARAMETER VERSION

NEW OPEN

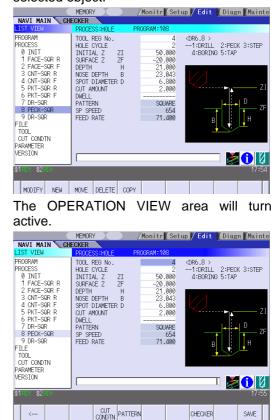
COPY COMMENT RENAME DELETE

🥭 🚹 🛛

3.2 Changing Screen

- Select the object with the cursor key. (3)
- The OPERATION VIEW area will change into the screen corresponding to the selected object.

<---



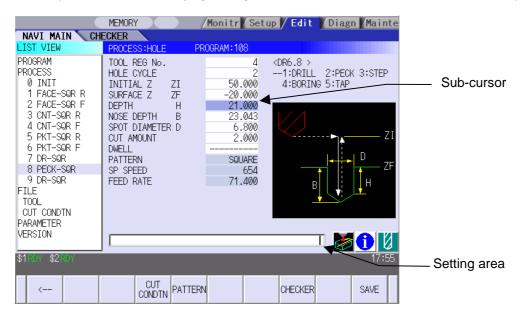
CHECKER

SAVE

Press the [MODIFY] menu key. (4)

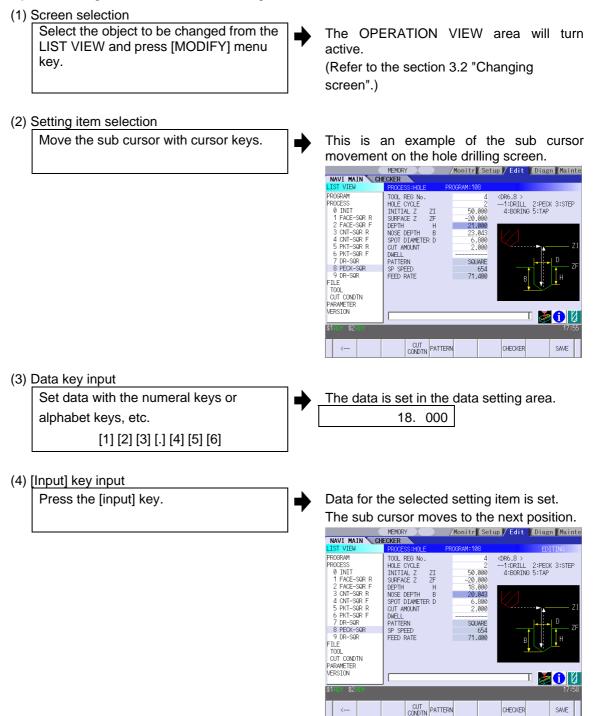
### 3.3 Setting Data

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



#### Operation method

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



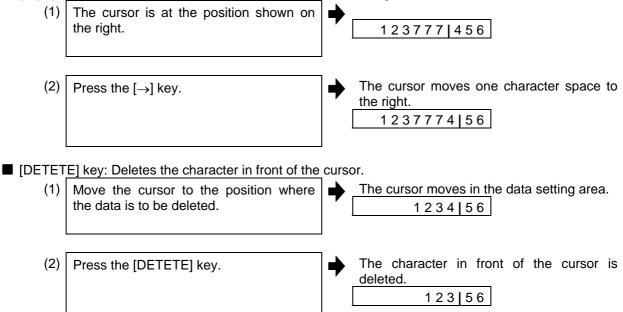
- (Note 1) The contents in the data setting area are only displayed when [INPUT] key is not pressed and will be invalidated if the screen is changed at this time. Data for the currently selected setting item will be set when [INPUT] key is pressed.
- (Note 2) If illegal data is set, an error occurs when [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.

 $\blacksquare$  [ $\rightarrow$ ] / [ $\leftarrow$ ] keys: Moves the cursor one character to the left or right.



# 3.4 Switching Windows

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

	, , , , , , , , , , , , , , , , , , , ,
Button	Application
LIST	Displays the tool guidance window.
?	Displays the message guidance window.
	Displays the checker window.

# 3.5 Switching Selection Tags

	MEMORY	Monitr Setu	up 🖊 Edit	/Diagn /Mainte	
	HECKER	55005 W 400			
LIST VIEW	PROCESSIMI	PROGRAM:108			
PROGRAM	WORK REG No.	1	S45C	<work list=""></work>	
PROCESS	INITIAL POS Z	50.000		1:S45C	
0 INIT	WORK COORDINATE	54	_ G54	2:FC25	
1 FACE-SQR R	ATC PATTERN X	2	2:#1_BEF.	3:SUS4	
2 FACE-SQR F	POSITION X		$\sim$ $\sim$	4:SKD4	
3 CNT-SQR R	PATTERN Y	2	2:#1_REF-	5:	
4 CNT-SQR F	POSITION Y			6.	
5 PKT-SQR R	END TOOL No.	99		7:	
6 PKT-SQR F	PATTERN X	2	2:#1_REF.	8:	Menu tag
7 DR-SQR	POSITION X				
8 PECK-SQR	PATTERN Y	2	2:#1_REF.		
9 DR-SQR	POSITION Y				
FILE	M CODE	1	1:M30		
TOOL					
CUT CONDTN					
PARAMETER					
VERSION					
				] 🋃 🚺 🖉	
\$1RDY \$2RDY	_			17:52	
MODIFY NEW	MOVE DELETE CO	PY			

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.

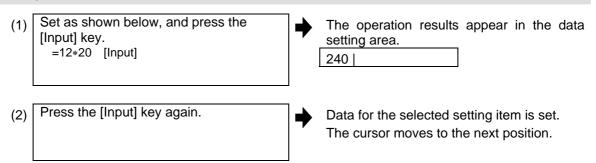
The operation results appear when the [INPUT] key is pressed. Data for the currently selected setting item will be set when [INPUT] key is pressed again.

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**



#### 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 $\theta$ is degree (°).
Atangent	: $-90 < operation results < 90.$

#### Restrictions

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

### 4.1 Starting NAVI MILL

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

#### Screen layout

			MEMORY			Monitr	Setup	/ Edit	Diag	n Mainte
LIS	VI MAI T VIEW	LN CH	ECKER PROGRA	M EDIT						
	GRAM CESS		PROGR	AM LIST						
0	INIT			NAME	Ξ	COMMEN	Т			
FIL				101		EXAMPL			_	
	UL T CONDI	ΓN		102		EXAMPL				
	AMETER			103		EXAMPL			_	
VER:	SION			104		EXAMPL				
				105		EXAMPL			_	
				106 107		EXAMPL EXAMPL			-	
				107		EXAMPL			-	
				100		EARMEL	EO			
									- 🌽	
\$1R	DY \$2F	RDY								18:15
	>	NEW	OPEN	COPY	COMMENT	RENAME	DELETE			

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

			MEMOR	Y		Monitr	Setup	/Edit	Diag	n Mainte
Př Př F ( Pr	NAVI MAJ ST VIEW ROGEAS Ø INIT ILE TOOL CUT COND ARAMETER ERSION		1	AM EDIT AM LIST 101 102 103 104 105 106 107 108		COMMEN EXAMPL EXAMPL EXAMPL EXAMPL EXAMPL EXAMPL	E1 E2 E3 E4 E5 E6 E7			
\$	1RDY \$2F	RDY		_		_		_	] 🎽	<b>1</b> 18:15
	>	NEW	OPEN	COPY	COMMENT	RENAME	DELETE			

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

Proce	Process name		Remarks
Hole drilling	Drilling	DR-***	The symbol (abbrev.) which indicates the machining pattern is applied to the "****" part.
	Pecking	PECK-****	• Random : RNDM
	Step	STEP-***	● Linear :LINE ● Arc :ARC
	Boring	BORE-***	• Circle : CIR
	Tapping	(TAP-****)	<ul> <li>Square : SQR</li> <li>Grid : GRID</li> </ul>
Face cutting	Square	FACE-SQR ?	The symbol which indicates the machining
	Circle	FACE-CIR ?	type (rough / finishing) is applied to the "?"
Contour	Square	CNT-SQR ?	part.
cutting	Circle	CNT-CIR ?	• Rough : R
	Free	CNT-FREE ?	• Finishing : F
Pocket	Square	PKT-SQR ?	
	Circle	PKT-CIR ?	
	L pattern	PKT-LPT ?	
	U pattern	PKT-UPT ?	
EIA		EIA (EIA)	

< Process displays >

## 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	<i>←</i>	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 MAME COMMENT 101 EXAMPLE1 102 EXAMPLE2 103 EXAMPLE3
4	COPY	Copies the existing NC program to another program. (Note 1)
		< Display in the setting area when pressing the menu >
		$O() \rightarrow O()$
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 >
		$O( ) \rightarrow O( )$
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 999999999 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.

4.2 Screen Related to the Program

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.

NAVI MAIN		Acres 1	and the second	188. 10. August 10.
IST VIEW	PROGRAM	EDIT		
PROCESS 9 INLT FILE TOOL CUT CONDTN CARAMETER VERSION	PROGRAM	1 LIST 181 182 183 184 185 186 186 187 188	CONFIGNT EXMPLE1 EXMPLE2 EXMPLE3 EXMPLE5 EXMPLE5 EXMPLE5 EXMPLE5 EXMPLE7 EXMPLE8	
8 8		1		- 2012
				ci-si

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

PROGRAM PROCESS	PROGRAM LIST		
0 INIT	WE	COMMENT	
FILE	181	EXMPLE1	
TOOL CUT CONDIN	182	EXAMPLE2	
PARAMETER	183	EXAMPLE3	
VERSION	184	EXAMPLE4	
	185	EXAMPLE5	
	186	EXAMPLE6	
	187	EXAMPLE7	
	188	EXAMPLE8	
	0 183		
silor \$207			
816° 260			

(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.

IST VIEW	PROOFW	MEDIT	PROGRAM: 188	
ROGRAM ROCESS Ø INIT 1 FACE-SOR R 2 FACE-SOR R 2 FACE-SOR R 4 DOT-SOR R 4 DOT-SOR R 6 ROCT-SOR R 6 ROCT-SOR R 9 IN-SOR 9 IN-SOR 10 FOC 10	100000000000000000000000000000000000000	VI LIST 181 182 183 184 185 186 187 183	COMMENT EXAMPLE1 EXAMPLE2 EXAMPLE3 EXAMPLE5 EXAMPLE5 EXAMPLE5 EXAMPLE5 EXAMPLE5	
AERSION				

## 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

IST VIEW	PRO	ESS LIST	PROGRAM	:108					
PROGRAM	No.	PCS	T NAME	Т	NT	Н	D	S	F
PROCESS Ø INIT	0	INIT			0.				
1 FACE-SQR R	1	FACE-SQR R	FACE50	1	0	1	1	720	360.000
2 FACE-SQR F	2	FACE-SQR F	FACE50	1	0	1	1	720	360.000
3 CNT-SQR R 4 CNT-SQR F	3	CNT-SQR R	EM20	8	0	8	8	640	64.000
5 PKT-SQR R	4	CNT-SQR F	EM20	8	0	8	8	640	64.000
6 PKT-SQR F	5	PKT-SQR R	EM20	8	0	8	8	640	64.000
7 DR-SQR 8 PECK-SQR	6	PKT-SQR F	EM20	8	0	8	8	640	64.000
9 DR-SQR	7	DR-SQR	CD3	5	0	5	5	1050	70.000
ILE	8	PECK-SQR	DR6.8	4	0	4	4	654	71.400
TOOL CUT CONDTN	9	DR-SQR	DC20	3	0	3	3	112	56.000
ARAMETER									
/ERSION								- D	
STRDY \$2RDY									18:2

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	Т	Specify the No. of tool to be used.	0 to 9999	
4	NT	Specify the No. of tool to be used in the next	0 to 9999	
		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.		
5	Н	Specify the tool length offset No.	1 to number of tool	
		The maximum value for the H is changed	sets	
		according to the specifications.		
6	D	Specify tool diameter offset No.	1 to number of tool	
		The maximum value for the D is changed	sets	
		according to the specifications.		
7	S	Input the spindle rotation speed.	1 to 99999 rev/min	
8	F	Input the feedrate.	0.001 to	
		When the hole cycle type is "TAP", input the pitch	60000.000 mm/min	
		(mm/rev).	0.001 to	
			999.999 mm/rev	

#### Menus

No.	Menu	Details
1	$\leftarrow$	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.

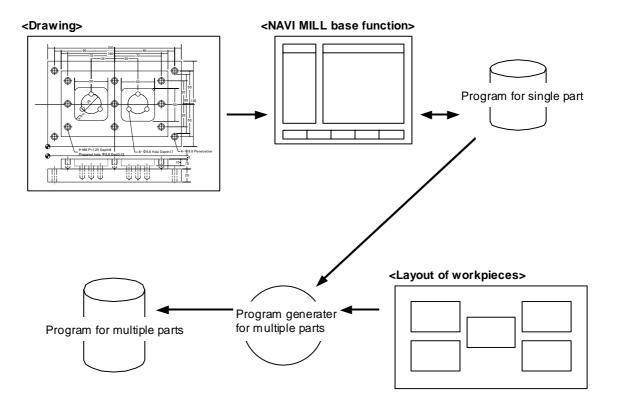
#### 4.3.2 Multiple Parts Screen

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, you may either specify their positions 1) by identifying the values of the work-cordinate system per each workpiece, or 2) by determining each offset amount from one specific work coordinate.

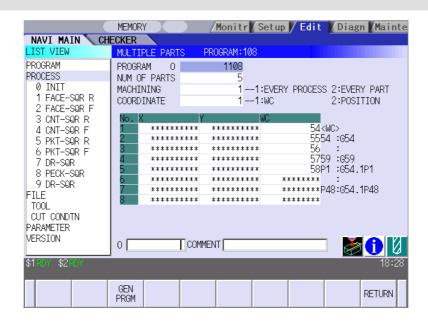
As for machining pattern, there are two options available: 1) to have one machining process completed for all workpieces on the table before moving on to the next machining process, and 2) to have entire machining processes completed per each workpiece. The number of times to execute tool-changes can be reduced by choosing option 1).

(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.



This screen is used to generate a NC program for the Multiple Parts machining. To view the "Multiple Parts Screen", go to the "Process List Screen" and press [MULTI PT] menu.

#### Screen Layout



Screen Details

No.	Display item	Details	Setting range
1	PROGRAM O	Program No. and comments are input.	_
		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.	
2	NUM OF PARTS	Number of parts is input.	2 to 8
3	MACHINING	Machining pattern is selected.	1 to 2
		1: A pattern that has one machining process	
		completed for all workpieces before moving on	
		to the next machining process	
		2: A pattern that has entire machining processes completed per one workpiece before moving	
		on to the next workpiece	
		0 0	

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

(Note) Program No. can be selected from 1 to 7999 or 10000 to 999999999. Comment section allows up to 18 characters/numbers.

### Menu

No.	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.	

Operation example (Selecting the process)

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



(2) Press the [MODIFY] menu key.

The OPERATION VIEW area will turn active.

	MEMORY	/Monitr Setup Edit Diagn Maint
NAVI MAIN C	HECKER	
LIST VIEW	PROCESS/FACE PI	ROGRAM: 108
PROCESS 0 THIT 1 FACE-SGR R 2 FACE-SGR R 2 FACE-SGR R 3 CHT-SGR R 4 CHT-SGR R 5 PHT-SGR R 6 PHT-SGR R 6 PHT-SGR R 9 PED-SGR 9 DE-SGR 9 DE-SGR 100-SGR 9 DE-SGR 100-SGR 9 DE-SGR 100-SGR 9 DE-SGR 100-SGR	TOOL REP No. FACE CYCLE FACESS BASE POS X WIDTH X I SURFACE Z ZF ALLOWANCE H FINISH ALLOW 0 NUM OF CUTS T START POINT(1-4) MALE A CUT DIRECTION	1
\$1107 \$2100		18:25
<	CUT CONDITN	CHECKER SAVE

Operation example (Deleting the process)

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



(2) Press the [DELETE] menu key.

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

	MENORY	/Monitry Setup / Edit / Diagn / Mainte
NAVI MAIN C	HECKER	And the second sec
IST VIEW	PROCESS:HOLE	PROGRAM:108
PROGRAM PROCESS 0 INIT 1 FACE-SQR R 2 FACE-SQR F 3 CNT-SQR R 4 CNT-SQR F 5 PKT-SQR F 6 PKT-SQR F	TOOL REG No. HOLE CYOLE INITIAL Z ZI SURFACE Z ZF DEPTH H NOSE DEPTH B SPOT DIAVETER D CUT AVIDUNT DVELL	4 006.8 > 2
7 DR-SQR	PATTERN	SOLARE D
8 PEDK-SOR 9 DR-SOR FILE	SP SPEED FEED RATE	654 71.490 B H
TOOL CUT CONDTN PARAMETER VERSION	ſ	
\$1107 \$250	0K? (Y/N)	18:31
MODIFY NEW	NOVE DELETE COP	24

(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.

	MEMORY	Monitr Setup Edit Diagn Mainte
	HECKER	
LIST VIEW	PROCESS:HOLE	PROGRAM:108
PROGRAM PROCESS Ø INIT 1 FACE-SOR R 2 FACE-SOR F 3 CNT-SOR R	TOOL REG No. HOLE CYOLE INITIAL Z ZI SURFACE Z ZF DEPTH H NOSE DEPTH B	3 (DC20) → 11:0R1LL 2:PECK 3:STEP 50.000 -20.000 -5.100 3.900
4 CNT-SQR F 5 PKT-SQR R 6 PKT-SQR F 7 DR-SQR	SPOT DIAMETER D CUT AMOUNT DWELL PATTERN	7.000 0.000 SOLUPE
8 DR-SOR FILE TOOL CUT CONDTN PARAMETER VERSION	SP SPEED FEED RATE	56.000 B H
\$1907 \$280		
MODIFY NEW	MOVE DELETE COP	

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.

IST VIEW	PROCESS HOLE PR	ROGRAM: 108
PROGRAM PROCESS 0 INIT 1 FACE-SQR R 2 FACE-SQR F 3 CNT-SQR R 4 CNT-SQR R 5 PKT-SQR R	TOOL REG No. HOLE CYCLE INITIAL Z ZI SURFACE Z ZF DEPTH H NOSE DEPTH B SPOT DLAMETER D CUT AMDUNT	5 0.000 −1:DRILL 2:PECK 3:STEF 50.000 4:50RING 5:TAP 3.000 3.000 3.000
6 PKT-SQR F 7 DR-SQR 8 DR-SQR 9 PEDK-SQR 10 DR-SQR FILE TOOL CUT CONDTN PARAMETER VERSION	DRELL PATTERN SP SPEED FEED RATE	

Operation exam	nple (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.	MODIFY         NEW         MOVE         DELETE         OOPY           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.	NAVI MAIN     CHECKER     /Monite     Setup     Edit     Diugn     Mainte       IST VIEU     PROCESS     PROCESS     0

MODIFY NEW

NOVE DELETE COPY

### 4.3 Screens Related to the Process Edit Functions

<EM20 > --1:ROUGH 2:FIN

<EM20 > --1:ROUGH 2:FIN

R)

10

10

Press the [INPUT] key. (4) The message to confirm a movement is displayed. MEMORY NAVI MAIN CHECKER Monitr Setup Edit Diagn Mainte If the [MOVE] menu key is pressed again 
 PROCESS:CONTOUR
 PRO

 TOOL REG No.
 PROCESS

 SURFACE Z
 ZF

 ALLOWANCE Z
 H

 XY
 E

 FIN ALLOW Z
 FH

 NUM OF CUTS Z
 ZT

 CUT TYPE

 <1:DOWN 2:UP>

 PATTERN
 during the movement operation, the PROGRAM PROGRAM PROCESS Ø INIT 1 FACE-SOR R 2 FACE-SOR R 3 ONT-SOR R 4 ONT-SOR R 5 PKT-SOR R 6 PKT-SOR R 6 PKT-SOR R 6 PKT-SOR R 8 PEOK-SOR 9 DR-SOR FILE TOOL OUT CONDTN PARMETER 2 -5.000 20.000 3.000 3.000 3.000 movement operation will be canceled. <1:DO PATTERN SP SPEED FEED RATE XY SQUARE 640 64.000 PARAMETER VERSION 0K? (Y/N) MODIFY NEW DELETE COPY (5) Press the [Y] key. The process of the movement source will be moved to the cursor position. The highlight of the [MOVE] menu will turn When not moving the process, press the OFF. [N] key. Monitr Setup Edit Diagn Mainte MEMORY NAVI MAIN CHECKER PROGRAM TOOL REG No. PROGRAM PROCESS 0 INIT 1 FACE-SQR R 2 FACE-SQR F 3 CNT-SQR R 4 CNT-SQR R 5 DR-SQR 6 PKT-SQR F 8 PECK-SQR 9 DR-SQR FILE TOOL REG No. PROCESS SURFACE Z ZF ALLOWANCE Z H XY E FIN ALLOW Z FH XY FE NUM OF CUTS Z ZT CUT TYPE (1:DOWN 2:UP> PATTERN FEED RATE XY -5.000 20.000 20.000 3.000 3.000 SQUARE 640 64.000

FILE TOOL CUT CONDTN PARAMETER VERSION

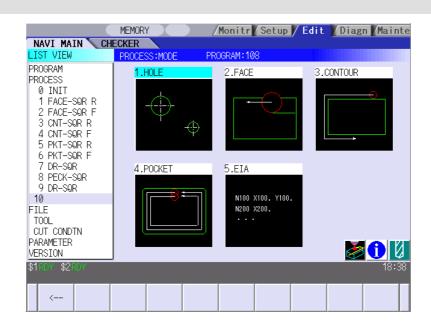
MODIFY NEW MOVE DELETE COPY

(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. When the [NEW] menu key is pressed at the position of the [PROCESS] in the LIST VIEW area, this screen is displayed.

#### Screen layout



#### Screen display item

No.	Display item	Details	Setting range
1	Process mode	Displays the process mode that can be selected.	1 to 5
		Select the process mode by moving the sub-cursor	
		or inputting numerical values.	

#### Menu

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

Operation example(Adding a new process)

(2)

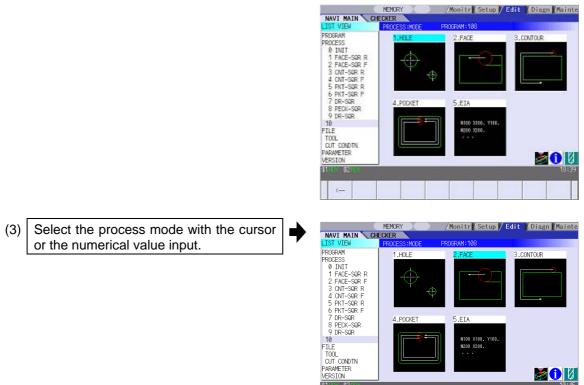
(1) Validate the LIST VIEW area, and select the position where the process is added with the cursor key.

Press the [NEW] menu key.

	HECKER	and a second second second second	
LIST VIEW	PROCESSIHOLE	PROGRAM: 108	
PROGRAM PROCESS Ø INIT 1 FACE-SQR R 2 FACE-SQR F	TOOL REG No. HOLE CYCLE INITIAL Z ZI SURFACE Z ZF DEPTH H	3 1 50,000 -20,000 -6,100	<pre></pre>
3 CNT-SQR R 4 CNT-SQR F 5 PKT-SQR R 6 PKT-SQR F 7 DR-SQR 8 PEDK-SQR 9 DR-SQR	NOSE DEPTH B SPOT DIAMETER D CUT ANDUNT DWELL PATTERN SP SPEED FEED RATE	3,900 7,900 9,000 SQLARE 112 56,000	
FILE TOOL CUT CONDTN PARAMETER VERSION		30.000	
\$1907 \$290			18:3

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.



<---

Press the [INPUT] key.

(4)

## 4.3 Screens Related to the Process Edit Functions

➡ 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.

LIST VIEW	PROCESS/FACE PROGRA	4N:108
FROGRAM FROCESS 0 INIT 1 FACE-SGR F 2 FACE-SGR F 3 INT-SGR F 4 INT-SGR F 5 INT-SGR F 6 INT-SGR F 6 INT-SGR F 7 INT-SGR F 8 FEDI-SGR 9 INT-SGR R FILE TOOL CUT CONDTN PARAWETER VERSION	TOOL REG No. FACE CYOLE FACE CYOLE BASE POS X WIDTH X I SURFACE Z ZF ALLOWMOE H FINISH ALLOW Q NUM OF CUTS T START POINT(1-4) ANGLE A CUT DIRECTION	1 a CFACES(s) -1:SQUARE 2:CIPALE -1:SQUARE 2:CIPALE -1:ROUSH 2:FIN CUT DIRECTION 1:X 2:Y 0 t (1) 1 t (2) X,Y ↓ XA (3) (3)
31110/1 \$2100		1814
<	CUT CONDIN	CHECKER SAVE

(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

	MEMORY		Monitr	Setup	🖌 Edit	Diag	n 🛛 Mai	nt
NAVI MAIN CH	ECKER		-		=*	_		
LIST VIEW	PROCESS: INIT	PR	DGRAM:10	8				
PROGRAM PROCESS Ø INIT 1 FACE-SOR R 2 FACE-SOR F 3 CNT-SOR R 4 CNT-SOR F 5 PKT-SOR R 6 PKT-SOR F 7 DR-SOR 8 PECK-SOR 9 DR-SOR FILE TOOL CUT CONDTN PARAMETER	WORK REG No. INITIAL POS Z WORK COORDINAT ATC PATTERN > POSITION > POSITION > POSITION > PATTERN > POSITION > POSITION > POSITION > M CODE		50. 	000 54 G 2 2  2 2 99 2 2  2 2	45C 54 2:#1_REF. 2:#1_REF. 2:#1_REF. 2:#1_REF. 30	1:54 2:FC 3:SU 4:SM 5: 6: 7: 8:	25 JS4	~
VERSION						1 🌽	0	Ø
\$1RDY \$2RDY	-						18	:47
<		WORK SHAPE			CHECKER		SAVE	T

### Screen display items

No.	Display item	Details	Setting range
1	WORK REG No.	Input the registration No. of the workpiece	1 to 8
		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.)	
2	INITIAL POS Z	Input the initial position Z.	-99999.999 to
		In the workpiece coordinate system, input the Z	99999.999mm
		axis position where the workpiece or jig does not	
		interfere with the tool even if the table is moved.	

(Continued to the next page)

(Continued from the previous page)

No.	Display item	Details	Setting range
3	WORK	Specify the workpiece coordinate system to be	54 to 59
	COORDINATE	used.	P1 to P48
		54 : G54	
		:	
		59 : G59	
		P1 : G54.1 P1	
		· P48 : G54.1 P48	
4	ATC PATTERN X	Select the table position at the tool change (ATC)	1 to 4
4	ATC PATTERN X	with the following No.'s.	1 (0 4
		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.)	
5	ATC POSITION X	In the machine coordinate system, input the table	-99999.999 to
	ATC POSITION Y	position at the tool change.	99999.999mm
		This is valid when "4" is set in the "ACT	
6	END TOOL No.	PATTERN".	0 to 9999
0	END TOOL NO.	At the program end, input the tool No. that you want to call.	0 10 9999
		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.	
7	END PATTERN X	Select the table position at the program end with	1 to 4
	END PATTERN Y	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.)	
8	END POSITION X	In the machine coordinate system, input the table	-99999.999 to
	END POSITION Y	position at the program end.	99999.999mm
		This is valid when "4" is set in the "END	
		PATTERN".	
9	END M CODE	At the program end, select the M command to be	1 to 3
		output.	
		1 : M30 2 : M02	
		3 : M99	

Menus

No.	Menu	Details	
1	$\leftarrow$	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

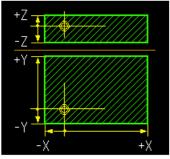
	MEMORY	/Monitr/Setu	ıp 🖊 Edit 🚺 Diagn Mainte
NAVI MAIN 🔪 CH	ECKER		
LIST VIEW	PROCESS:INIT	PROGRAM:108	
PROGRAM PROCESS Ø INIT 1 FACE-SQR F 2 FACE-SQR F 3 CNT-SQR F 4 CNT-SQR F 5 PKT-SQR F 7 DR-SQR 8 PECK-SQR 9 DR-SQR FILE TOOL CUT CONDTN PARAMETER	WORK SHAPE +X -X +Y +Z -Z	1 200.000 0.000 150.000 0.000 5.000 -40.000	1:SQUARE 2:CIRCLE +Z -Z +Y -Y -Y -Y -X +X
VERSION			🎽 🚺
\$1RDY \$2RDY			18:47
			RETURN

Screen display items

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

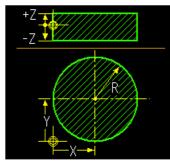
(Note) Parameters to be followed after the initial one are up to the setting of the workpiece shape. The parameters used when working on the following shapes are as shown below.

### • Parameters for SQUARE



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

### • Parameters for CIRCLE



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



Screen display items

No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used.	1 to 99
		Specify it with the No. registered in the tool file.	
2	HOLE CYCLE	<ul> <li>Specify it with the No. registered in the tool file.</li> <li>Input the type of the hole machining cycle.</li> <li>&lt;1: DRILL&gt; (G81, G82)</li> <li>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.</li> <li>&lt;2: PECK&gt; (G83)</li> <li>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.</li> <li>&lt;3: STEP&gt; (G73)</li> <li>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.</li> <li>&lt;4: BORE&gt; (G85, G89)</li> <li>The machining is performed as far as the hole bottom dwell has been executed.</li> <li>&lt;5: TAP&gt; (G84,G74)</li> <li>The tap machining is performed as far as the hole bottom dwell has been executed.</li> </ul>	1 to 5
3	INITIAL Z (ZI)	Input the initial position. The tool is returned to the initial position after the	-99999.999 to 99999.999mm
		machining has been finished.	
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

(Continued to the next page)

(Continued from the previous page)

No.	Display item	Details	Setting range
7	SPOT DIAMETER	Input the tool radius of the workpiece face.	0.001 to
	(D)	When the tool radius of the workpiece face is input,	Tool diameter
		DEPTH and NOSE DEPTH will be automatically	
		updated.	
8	CUT AMOUNT	When the hole cycle type C=2 (PECK) or C=3	0.000 to
		(STEP) is selected, input the cutting amount for one	99999.999mm
		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.	
9	DWELL	When the hole cycle type C=1 (Drilling), C=3 (Step),	0.000 to
		C=4 (Boring), C=5 (Tap) is selected, input the dwell.	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.	1 to
		Perform changing the spindle rotation speed on the	99999rev/min
		cutting condition screen.	
12	FEED RATE	The feedrate is displayed.	0.001 to
		When the hole cycle type is "TAP", the pitch is	60000.000
		displayed.	mm/min
		Perform changing the feedrate on the cutting	0.001 to
		condition screen.	999.999 mm/rev

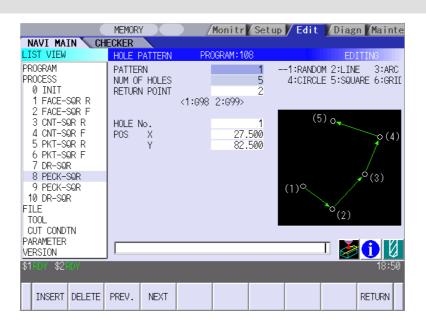
### Menus

No.	Menu	Details
1	$\leftarrow$	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	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.

#### Screen layout

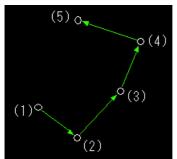


#### Screen display items

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

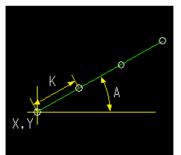
(Note) The parameters displayed at the second step or below are changed according to the setting of pattern. The parameters displayed in each pattern are as follows.

#### • Parameters for RANDOM



No.	Displayed item	Details	Setting range
1	NUM OF HOLES	Displays the number of holes.	1 to 50
2	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
3	HOLE No.	Input the hole No. to set the hole position.	1 to the number of holes+1
4	POS X	Input the hole position (X) specified with the HOLE No.	-99999.999 to 99999.999mm
5	POS Y	Input the hole position (Y) specified with the HOLE No.	-99999.999 to 99999.999mm

#### Parameters for LINE



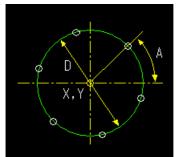
No.	Display item	Details	Setting range
1	ANGLE (A)	Input the angle formed with the machining direction	-359.999 to
		and the positive direction of the X-axis.	360.000°
2	PITCH (K)	Input the space from the machining point to the next	0.000 to
		machining point.	99999.999mm
3	NUM OF HOLES	Input the number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

• Parameters for ARC



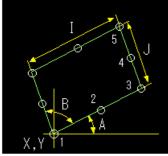
No.	Display item	Details	Setting range
1	RADIUS (R)	Input the arc radius.	0.001 to
			99999.999mm
2	START ANGLE	Input the angle formed with the first machining point	-359.999 to
	(A)	and the X-axis direction.	360.000°
3	PITCH (K)	Input the angle from the previous machining point to	-359.999 to
		the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

### • Parameters for CIRCLE



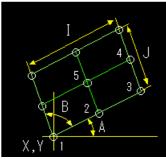
No.	Display item	Details	Setting range
1	DIAMETER (D)	Input the circular diameter.	0.001 to
			99999.999mm
2	START ANGLE	Input the angle formed with the first machining point	-359.999 to
	(A)	and the positive direction of the X-axis.	360.000°
3	NUM OF HOLES	Input the number of holes.	1 to 999
4	BASE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

### • Parameters for SQUARE



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

• Parameters for GRID



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

### Menus

No.	Menu	Details
1	RETURN	Returns the screen to the input screen for the hole drilling process parameter.
2	INSERT	Inserts the hole position data before the current HOLE No. This is available only for the RANDOM pattern.
3	DELETE	Deletes the hole position data of the current HOLE No. This is available only for the RANDOM pattern.
4	PREV.	Changes into the previous hole position data. This is available only for the RANDOM pattern.
5	NEXT	Changes into the next hole position data. This is available only for the RANDOM pattern.

### (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.	1 to 99
		Specify it with the No. registered in the tool file.	
2	TOOL T No. (T)	Specify the tool No. to be used.	0 to 9999
		(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.	
3	TOOL LENG	Specify the tool length offset No.	1 to the number
	OFFSET (H)	The maximum value for the H is changed according	of tool sets
		to the specifications.	
		When the tool registration No. is specified, the tool	
		length offset No. registered in the tool file will be	
		automatically set.	

(Continued to the next page)

(Continued from the previous page)

NI -	Diamlars 'torr		from the previous
No.	Display item	Details	Setting range
4	TOOL DIAM	Specify tool diameter offset No.	1 to the number
	OFFSET (D)	The maximum value for the D is changed according	of tool sets
		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.	
5	TOOL	Input the tool diameter.	0.001 to
	DIAMETER	When the tool registration No. is specified, the tool	99999.999 mm
		diameter registered in the tool file will be	
		automatically set.	
6	SP SPEED (S)	Input the spindle rotation speed.	1 to 99999
		The spindle rotation speed is automatically set based	rev/min
		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	0.004.44
7	FEED RATE	Input the feedrate.	0.001 to
	(Z)	Specify the pitch with mm/rev when tapping is	60000.000
	(XY)	selected in the hole drilling machining mode.	mm/min
		Specify the feedrate with mm/min in the other	0.001 to
		machining mode.	999.999
		The feedrate is automatically set based on the	mm/rev
		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	
		<pre><contour cutting="" pocket="" process="" process,=""></contour></pre>	
		FEED RATE Z	
		FEED RATE XY	
8	SP DIRECTION	Input the spindle rotation direction.	1, 2
Ŭ		When the tool registration No. is specified, the tool	.,_
		rotation direction registered in the tool file will be	
		automatically set.	
		1: CW	
		2: CCW	
L	1		1

### Menu

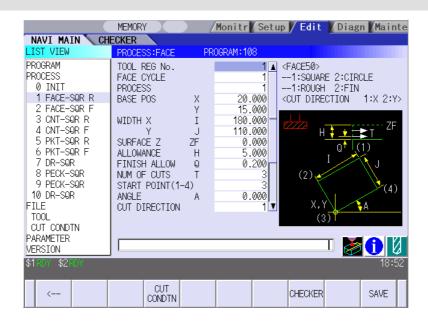
No.	Menu	Details
1	RETURN	Returns the screen to the input screen for the hole process parameter.

### 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

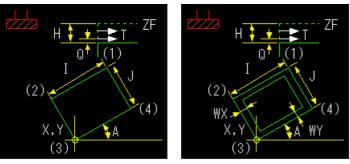
No.	Display item	Details	Setting range
1	TOOL REG No.	Input the tool registration No. to be used. Specify it	1 to 99
		with the No. registered in the tool file.	
2	FACE CYCLE	Input the type of the face cutting cycle.	1, 2
		<1: SQUARE>	
		The face cutting machining in the square area is	
		performed.	
		<2: CIRCLE>	
		The face cutting machining in circular area is	
		performed.	
3	PROCESS	Input the type of the machining.	1, 2
		<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.	

(Continued to the next 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	0.001 to
		the machining top surface.	99999.999mm
8	FINISH ALLOW	Input the size of the finishing allowance form the	0.000 to
	(Q)	finishing surface.	99999.999mm
		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.	
9	NUM OF CUTS	Input how many times the allowance is cut in the	1 to 99999
	(T)	Z-axis direction when performing the rough cutting.	times
		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.	
10	START POINT	Input form which corner ((1) to (4)) in the four	1 to 4
		screen guide drawings the machining is started.	
11	CUT DIRECTION	Input the direction (X-direction or Y-direction) in	1, 2
		which the machining is performed.	
		1: X	
		2: Y	
12	PATH	Input the machining path (reciprocation machining,	1 to 3
		single-direction machining, shape machining)	
		1: RECIP	
		2: SINGLE	
		3: SHAPE	
13	SP SPEED	The spindle rotation speed is displayed.	1 to
		Perform changing the spindle rotation speed on the	99999rev/min
		cutting condition screen.	
14	FEED RATE	The feedrate is displayed.	0.001 to
		Perform changing the feedrate on the cutting	60000.000
		condition screen.	mm/min

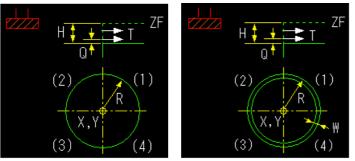
(Continued from the previous page)

• 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	0.001 to
		in the X-direction (horizontal direction).	99999.999mm
		This is valid when the face cutting cycle is	
		SQUARE.	
16	WIDTH Y (J)	Input the machining width of the machining surface	0.001 to
		in the Y-direction (vertical direction).	99999.999mm
		This is valid when the face cutting cycle is	
		SQUARE.	
17	ANGLE (A)	Input the inclination angle of the machining surface.	-359.999 to
		This is valid when the face cutting cycle is	360.000°
		SQUARE.	
18	CUT WIDTH X	Input the width of machining part in the X-direction	0.000 to
	(WX)	during the square machining.	99999.999mm
		This is valid when the face cutting cycle is	
		SQUARE.	
19	CUT WIDTH Y	Input the width of machining part in the Y-direction	0.000 to
	(WY)	during the square machining.	99999.999mm
		This is valid when the face cutting cycle is	
		SQUARE.	

• Parameter used when the face cutting cycle is CIRCLE



No.	Display ite	em	Details	Setting range
20	WORK RADIL	JS	Input the radius of the workpiece surface.	0.001mm to
		(R)	This is valid when the face cutting cycle is CIRCLE.	99999.999mm
21	CUT WIDTH	(W)	Input the width of machining part during the circular	0.000 to
			machining.	99999.999mm
			This is valid when the face cutting cycle is	
			SQUARE.	

### Menus

No.	Menu	Details
1	$\leftarrow$	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

	MEMORY	Monitr Setup	Edit Diag	n Mainte
NAVI MAIN 🔪 CH	ECKER			
LIST VIEW	PROCESS:CONTOUR PR	OGRAM:108		
PROGRAM PROCESS Ø INIT	TOOL REG No. PROCESS SURFACE Z ZF		<em20> 1:ROUGH 2:FIN</em20>	
1 FACE-SQR R 2 FACE-SQR F	ALLOWANCE Z H XY E	20.000	FF 🔸	
3 CNT-SQR R 4 CNT-SQR F 5 PKT-SQR R	FIN ALLOW Z FH XY FE NUM OF CUTS Z ZT	3.000 3.000 3	H + + E-	ZF
6 PKT-SQR F 7 DR-SQR 8 PECK-SQR	CUT TYPE <1:DOWN 2:UP PATTERN	1	(2)	
9 PECK-SQR 10 DR-SQR	SP SPEED FEED RATE XY	64.000	X.Y	A (4)
FILE TOOL CUT CONDTN			(3)	<u>• A</u>
PARAMETER VERSION			🎽	
\$1RDY \$2RDY		1	1	18:52
<	CUT CONDTN PATTERN		CHECKER	SAVE

Screen display items

No.	Display item	Details	Setting
			range
1	TOOL REG	Input the tool registration No. to be used. Specify it with the No.	1 to 99
	No.	registered in the tool file.	
2	PROCESS	Input the machining type.	1, 2
		<1: ROUGH>	
		<ul> <li>Z-direction : The machining is performed with cuttings of the number of cuttings, remaining the finishing allowance Z.</li> <li>XY-direction: The machining is performed with shifting, remaining the finishing allowance XY.</li> <li>&lt;2: FIN&gt;</li> <li>First, the finishing machining of the bottom surface is performed and then that of the side surface.</li> <li>Finishing machining of the side surface</li> </ul>	
		<ul> <li>[Finishing machining of the bottom surface]</li> <li>Z-direction : The machining of the finishing allowance Z is performed at a stretch.</li> <li>XY-direction: The machining is performed with the shift of the allowance, remaining the finishing allowance XY.</li> </ul>	
		When the finishing allowance Z is 0, the finishing machining of the bottom surface is not performed.	
		[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	
		When the finishing allowance XY is 0, the finishing machining of the side surface is not performed.	
3	SURFACE Z	Input the workpiece top surface position.	-99999.99
	(ZF)		9 to
	()		99999.999

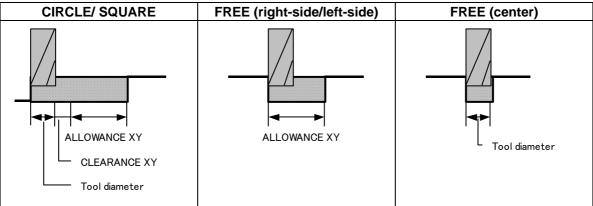
(Continue to the next page)

(Continued from the previous page)

No.	Display item	Details	Setting range
4	ALLOWANC E Z (H)	Input the machining depth from the workpiece top surface with the addition input method.	0.001 to 99999.999 mm
5	ALLOWANC E 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 99999.999 mm
6	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 99999.999 mm
7	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. (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 99999.999 mm
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 feed rate in the XY-direction is displayed. The feed rate is changed on the setting screen for cutting conditions.	0.001 to 60000.000 mm/min

### Note)

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



#### Menus

No.	Menu	Details
1	$\leftarrow$	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

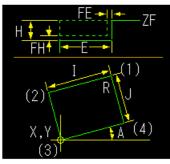


#### Screen display items

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

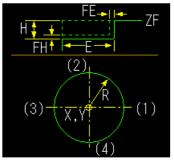
(Note) The parameters displayed at the second step or below are changed according to the setting of the pattern type. The parameters displayed in each pattern are as follows.

### • Parameters for SQUARE



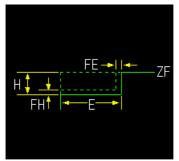
No.	Display item	Details	Setting range
1	MACHINING SIDE	Input the machining position. <1: INSIDE> The machining is performed to the inside of the shape. <2: OUTSIDE> The machining is performed to the outside of the shape.	1, 2
2	BASE POS X (X)	Input the base position X of the square.	-99999.999 to 99999.999mm
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 square in the X-direction.	0.001 to 99999.999mm
5	WIDTH (J)	Input the width of the square in the Y-direction.	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	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 CIRCLE



No.	Display item	Details	Setting range
1	MACHINING	Input the machining position.	1, 2
	SIDE	<1: INSIDE>	
		The machining is performed to the inside of the	
		shape.	
		<2: OUTSIDE>	
		The machining is performed to the outside of the	
		shape.	
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	START POINT	Input the machining start position (1 to 4).	1 to 4
		(Note) This data is valid when the machining	
		position is "OUTSIDE".	

### • Parameters for FREE



No.	Display item	Details	Setting range
1	TOOL PATH	Input the tool path.	1 to 3
		<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).	

No.	0	Display item	Details	Setting range
Shap	e			1 to 35
2	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. Left turn Tangent Right turn	1,2
	3	X Y	<ul> <li>Input the position of the machining end point.</li> <li>(Note 1) Always input if the line is the last one. If it is not the last one, omission is possible.</li> <li>(Note 2) Always input if the corner shape dimensions are input in the previous line.</li> </ul>	-99999.999mm to 99999.999mm
	4	R/A	<ul> <li>When the shape is arc, the radius of arc is entered.</li> <li>Positive value: Arc command (less than 180°)</li> <li>Negative value: Arc command (more than 180°)</li> <li>When the shape is line, angle is entered.</li> <li>(Note 1) When the shape is arc, always input.</li> <li>(Note 2) When the shape is line, and the position X, Y or vector I, J are input, the data is invalid.</li> </ul>	Radius: -999999.999mm to -0.001mm, 0.001mm to 999999.999mm Angle: -359.999 to 360.000

No.	No. Display item		Details	Setting range
Shape	Э			1 to 35
	5	l J	When the shape is line, the gradient (vector) is entered.	-99999.999mm to 99999.999mm
			Programmed 20 60 X zero point	
			center is entered. Y How the shape is arc, and only I or J is entered, the other is regarded as 0.	
			(Note 2) When the shape is line, and the position X, Y or angle is input, this data will be invalid.	
	6	C	Input corner size. Positive value: Corner R, Negative value: Corner C	-99999.999mm to 99999.999mm
			(Note 1) When corner dimensions are specified, the end points (X,Y) are entered for the following line in principle.	

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

#### Menus

• For CIRCLE or SQUARE

-		
N	o. 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) For No.1 (machining start point), this operation is disabled.
2	LINE DELETE	Deletes the shape data at the cursor position.
		(Note) For No.1 (machining start point), this operation is disabled.
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.

### (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.

### 4.3.9 Pocket

### (1) Pocket Screen

The parameters for the pocket 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	1 to 99
		the No. registered in the tool file.	
2	PROCESS	Input the machining type.	1, 2
		<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.	
		<2: FIN>	
		First, the finishing machining of the bottom surface is	
		performed and then that of the side surface.	
		Finishing allowance XY	
		Finishing	
		machining of the $-$	
		side surface	
		Finishing Finishing machining of	
		allowance Z the bottom surface	
		[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.	
		When the finishing allowance Z is 0, the finishing	
		machining of the bottom surface is not performed.	
		[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.	
3	SURFACE Z	Input the workpiece top surface position.	-99999.999 to
	(ZF)		99999.999mm
4	ALLOWANCE Z	Input the machining depth from the workpiece top	0.001 to
_	(H)	surface with the addition input method.	99999.999mm
5	FIN ALLOW Z	Input the finishing allowance of the bottom surface.	0.000 to
	(FH)	The rough machining is performed to the bottom	99999.999mm
		surface, remaining the finishing allowance.	

(Continued to the next page)

(Continued from the previous page)

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

### Menus

No.	Menu	Details
1	$\leftarrow$	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

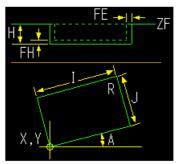
	MEMORY	Monitr Setu	up <b>/ Edit  </b> / Diagn / Mainte
NAVI MAIN CH	ECKER		,
LIST VIEW	POCKET PATTERN PR	OGRAM:108	
PROGRAM PROCESS Ø INIT 1 FACE-SOR R 2 FACE-SOR F 3 CNT-SOR R 4 CNT-SOR F 5 PKT-SOR R 6 PKT-SOR R 7 DR-SOR 8 PECK-SOR 9 PECK-SOR 9 PECK-SOR 9 PECK-SOR FILE TOOL CUT CONDTN PARAMETER VERSION	PATTERN BASE POS X X Y Y WIDTH I J CORNER SIZE R <+:R -:C> ANGLE A Z APPR <1:G01 2:HEL PITCH(HELIX)	1 40.000 50.000 50.000 10.000 0.000 2 IX 3:600> 3	1:SQUARE 2:CIRCLE 3:L PATTERN 4:U PATTERN
\$1 RDY \$2 RDY			18:58
			RETURN

#### Screen display items

No.	Display item	Details	Setting range
1	PATTERN	Input the type of the pocket pattern. <1: SQUARE >	1 to 4
		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	

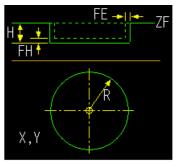
(Note) The parameters displayed at the second step or below are changed according to the setting of the pattern type. The parameters displayed in each pattern are as follows.

• Parameters for SQUARE



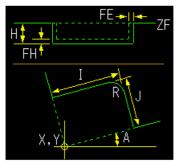
No.	Display item	Details	Setting range
1	BASE POS X (X)	Input the base position X of the square.	-99999.999 to
			99999.999mm
2	BASE POS Y (Y)	Input the base position Y of the square.	-99999.999 to
			99999.999mm
3	WIDTH (I)	Input the width of the X-direction in the square	0.001 to
		shape machining.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

### • Parameters for CIRCLE



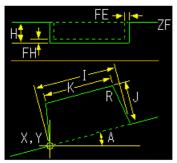
No.	Display item	Details	Setting range
1	BASE POS X (X)	Input the center position X of the circle.	-99999.999 to
			99999.999mm
2	BASE POS Y (Y)	Input the center position Y of the circle.	-99999.999 to
			99999.999mm
3	RADIUS (R)	Input the radius of the circle.	0.001 to
			99999.999mm
4	Z APPR	Select the G code that is commanded in the Z	1 to 3
		approach.	
		1: Linear interpolation (G01)	
		2: Helical (G02/G03)	
		3: Positioning (G00)	
5	PITCH (HELIX)	The helical operations of times specified in the pitch	1 to 99
		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.	

• Parameters for L PATTERN



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

• Parameters for U PATTERN



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

#### 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

	MEMORY			Monitr	Setup	/ Edit	Diag	n Mainte
	ECKER				~			
LIST VIEW	PROCES	S:EIA	PR	DGRAM:10	18		EDI	TING
PROGRAM PROCESS								
0 INIT	1	G90G540	0X-20.Y	20.;				
1 FACE-SQR R	2	G43Z100	).H4;					
2 FACE-SQR F	3 4	S244M3N	18F48;					
3 CNT-SQR R		Z-20.;						
4 CNT-SQR F 5 PKT-SQR R	5	M98P101	H30;					
6 EIA	6	G90G0Z100.M5M9;						
7 PKT-SQR F	7	G91G28X0Y0Z0;						
8 DR-SQR	8							
9 PECK-SQR	9							
10 PECK-SQR	10							
FILE								
CUT CONDTN						691628	3X0Y0Z0;	1
PARAMETER VERSION							ě	1
\$1 RDY \$2 RDY	-							18:59
< INSERT	DELETE							SAVE

### Screen display item

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

### Menus

No.	Menu	Details		
1	$\leftarrow$	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.	0 to 9999
		(T function code data output as the NC data)	
4	LEN OFS	Specify the tool length offset No.	1 to number of
		The maximum value for the H is changed according	tool sets
		to the specifications.	
5	DIA OFS	Specify the tool diameter offset No.	1 to number of
		The maximum value for the D is changed according	tool sets
		to the specifications.	
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.	0.001 to
		When tapping, input the pitch (mm/rev).	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	$\leftarrow$	Turns the LIST VIEW area active.		
2	SAVE	Saves the changes in the tool file.		

#### (Note)

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 rotation speed are automatically determined. This means that the feedrate and spindle speed of each machining process created once will not be changed even if the data in the tool file screen and the cutting condition file screen is changed.

### 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

IST VIEW	CUT CONDIT	ION	PROGR	AM:108					
ROGRAM	No.	<1>	<2>	<3>	<4>	<5>	<6>	<7>	<8>
ROCESS	MATL	S45C	FC25	SUS4	SKD4				
0 INIT	DR S%	70	100	50	60	0	0	0	
1 FACE-SQR R	F%	70	100	40	60	0	0	0	
2 FACE-SQR F	TP S%	60	100	30	40	0	0	0	
3 CNT-SQR R	BR S%	80	100	60	50	0	0	0	
4 CNT-SQR F	F%	80	100	50	50	0	0	0	
5 PKT-SQR R	ML S%	90	100	60	50	0	0	0	
6 PKT-SQR F	F%	90	100	50	40	0	0	0	
7 DR-SQR	ED S%	80	100	50	50	0	0	0	
8 PECK-SQR	F(XY)%	80	100	40	40	0	0	0	
9 PECK-SQR	F(Z)%	40	100	20	20	0	0	0	
ILE									
TOOL									
CUT CONDTN									
ARAMETER									
ERSION							-		ΙV
							⊥ 🛃	<b>3 ()</b>	14
1RDY \$2RDY	2		_	_	_	_			0:0
v≥nen									

#### 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	Max. 5
		workpiece.	alphanumeric
			characters
3	S%	Input the rate (%) of the applicable rotation speed to	0 to 200 %
		the standard rotation speed of the tool file by the tool	
		type.	
		<tool type=""></tool>	
		DR: For drilling	
		TP: For tapping	
		BR: For boring	
		ML: For face cutting	
		ED: For contour cutting and pocket	
4	F%	Input the rate (%) of the applicable feedrate to the	0 to 200 %
		standard feedrate of the tool file by the tool type.	
		(The tool type is the same as the S%.)	

Menus

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

## 

▲ 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

	MEMORY Moni	tr / Setup / Edit / Diagn / Mainte
NAVI MAIN CH	ECKER	
LIST VIEW	PARAMETER PROGRAM	:108
PROGRAM PROCESS Ø INIT 1 FACE-SQR R 2 FACE-SQR F 3 CNT-SQR R 4 CNT-SQR F 5 PKT-SQR R 6 PKT-SQR F 7 DR-SQR 8 PECK-SQR 9 PECK-SQR FILE TOOL CUT CONDTN PARAMETER VERSION	101 M0 OUTPUT           102 M6 OUTPUT           103 NEXT TOOL PREP           104 M19 OUTPUT           105 ATC POS Z           106 SEQUENCE No. OUTPUT           201 HOLE Z CLEARANCE           202 SYNC TAP           301 FACE CUT WIDTH PCT(%)           303 XY CLEARANCE           304 Z APPROACH           401 E-ML CUT WIDTH PCT(%)           402 Z CLEARANCE	00:INVALID 1:VALID 00:INVALID 1:VALID 00:INVALID 1:VALID 0:INVALID 1:VALID 0:INVALID 1:VALID 11:G28 2:G30 0:INVALID 1:VALID 10.000 10:INVALID 1:VALID 0:INVALID 1:VALID 10.000 11:G00 2:G01 5.000 ▼
\$1 RDY \$2		19:32
<		SAVE

#### Screen display items

No.	Display item	Details	Setting range
1	M0 OUTPUT	Specify whether to output the M0 code at the	0, 1
(101)		beginning of each process (right before the	
		execution of the tool function).	
		0: INVALID	
		1: VALID	
2	M6 OUTPUT	Specify whether to output the M6 code when	0, 1
(102)		"NEXT TOOL PREP" is invalid.	
		0: INVALID	
		1: VALID	
3	NEXT TOOL PREP	Specify whether to prepare the tool for the	0, 1, 2
(103)		next process program beforehand or not.	
		0: INVALID	
		1: VALID	
		(M6T□ is commanded in one block.)	
		2: VALID	
		(M6T□ is commanded in two blocks.)	
4	M19 OUTPUT	Specify whether to validate the orientation	0, 1
(104)		function after the spindle is stopped.	
		0: INVALID	
		1: VALID	

(Continued to the next page)

(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, 2
		1: 2nd zero point	
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	HOLE Z	The distance from the R-point, where the	0.001 to
(201)	CLEARANCE	cutting feed begins, to the hole top position is set.	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)

(Continued from the previous page)

No.	Display item	Details	Setting range
No. 13 (401)	Display item 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	Setting range 1 to 100%
		the tool of \$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.	
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	$\leftarrow$	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 MAINTE", is pressed.

#### Screen layout

	MEMORY	Monitr Setu	p / Edit / Diagn / Mainte
	ECKER		
LIST VIEW	PREFERENCE	PROGRAM:108	EDITING
PROGRAM PROCESS Ø INIT 1 FACE-SQR R 2 FACE-SQR F 3 CNT-SQR R 4 CNT-SQR F 5 PKT-SQR R 6 PKT-SQR F 7 DR-SQR 8 PECK-SQR 9 PECK-SQR 9 PECK-SQR FILE TOOL CUT CONDTN PARAMETER	PARAMETER	MEM:/ D:/NCFILE <nc <br="" mem:="" memory=""><oata dir.<br="" ds:="" server=""><other d<="" drive:="" td=""><td></td></other></oata></nc>	
VERSION			🛃 🚺 🚺
\$1RDY \$2RDY			19:02
		MACRO	RETURN SAVE

#### Screen display items

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

#### Menu

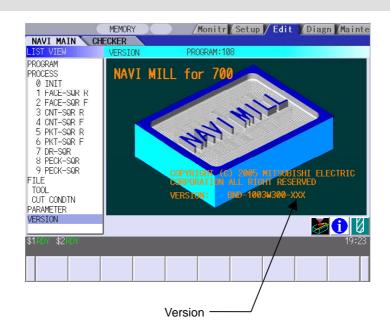
No.	Menu	Details
1	MACRO ENTRY	User macro program or MTB macro program is registered in the NC system.
2	RETURN	Return to the parameter screen.

### 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



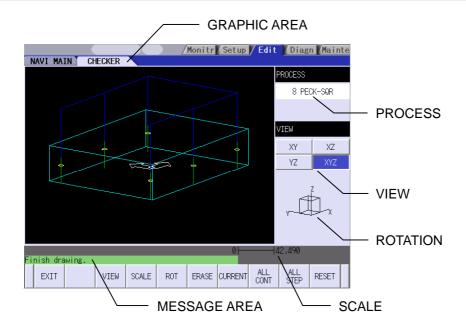
## 4.7 Program Checker Screen

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

Program Checker screen will be appeared by pressing  $\checkmark$  or a  $\square$  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: 1) Tool paths Blue (Rapid traverse), Green (Cutting feed) 2) Program paths White 2) Worknings Light blue
		<ul><li>3) Workpiece Light blue</li><li>4) Tool mark (Cutting start point) Yellow</li></ul>
		<ul> <li>Program paths instead of tool path will be displayed in the Cutter Radius Compensation mode.</li> </ul>
		** 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  $\overline{\text{VIEW}}$  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.
		*The solid scale frame will be drawn in dotted lines when its size
		exceeding 100%.
5	$\uparrow$	Moves up the scale frame. The same function can also be achieved by
		pressing ↑ key.
6	$\downarrow$	Moves down the scale frame. The same function can also be achieved
		by pressing $\downarrow$ key.
7	$\leftarrow$	Moves the scale frame toward the left. The same function can also be
		achieved by pressing←key.
7	$\rightarrow$	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

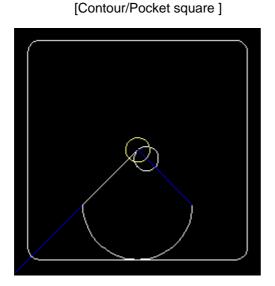
This is the sub menu of the ROTATION menu.

No.	Display item	Details	
1	CANCEL	Cancels the ROTATION change and returns to the main menu.	
2	$\uparrow$	Rotates the viewpoint angle in a vertical-forward direction.	
3	$\downarrow$	Rotates the viewpoint angle in a vertical-backward direction.	
4	$\leftarrow$	Rotates the viewpoint angle in a horizontal-clockwise direction.	
5	$\rightarrow$	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.	

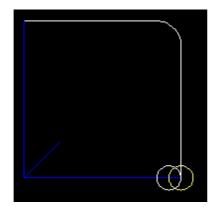
(Note 1) The displayed tool paths will be deleted upon change of display angle.

Restrictions on the graphic display function

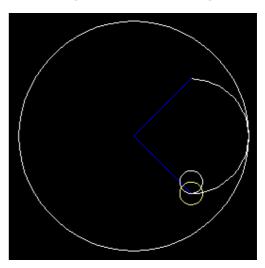
- Graphic display is not available for the EIA process.
- Graphic displays for Corner C and Corner R do not always represent the actual movement of the machining. The display is just a reference.
- NAVI MILL on PC is not compatible with Cutter Radius Compensation. Therefore, on this Program Checker screen, 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.



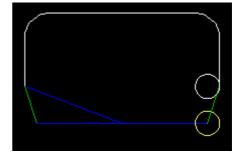
[Pocket L pattern ]

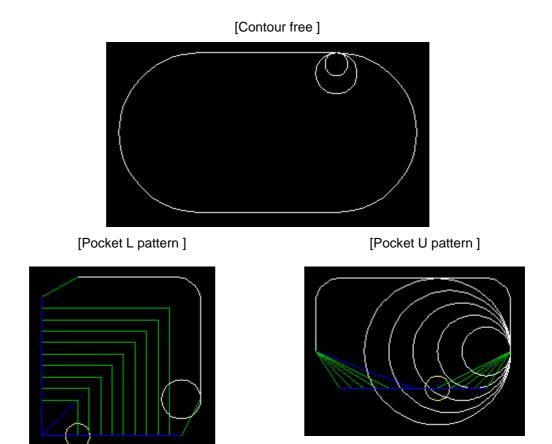


[Contour/Pocket circle ]



[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	Starting	method			
Туре	Key- board	lcon	Details		
Message Guidance	?	1	Details or countermeasures related to the current error and message are displayed. Message Guidance Wessage E102 Designated file already exists The designated program already exists.		
Tool Guidance	LIST		A segment of tool data registered in the tool file is displayed. Note that no editing is possible. Tool Guidance No. T NAME DIA ANGL A 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 9 EM8 8.000 180.000 10 EM5 5.000 180.000 10 EM5 5.000 180.000 10 EM5 5.000 180.000		

### 4.8.1 Tool Guidance Screen

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

### Screen layout

Tool Guida	ince		
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
2	Close	screen. This closes the tool guidance screen.

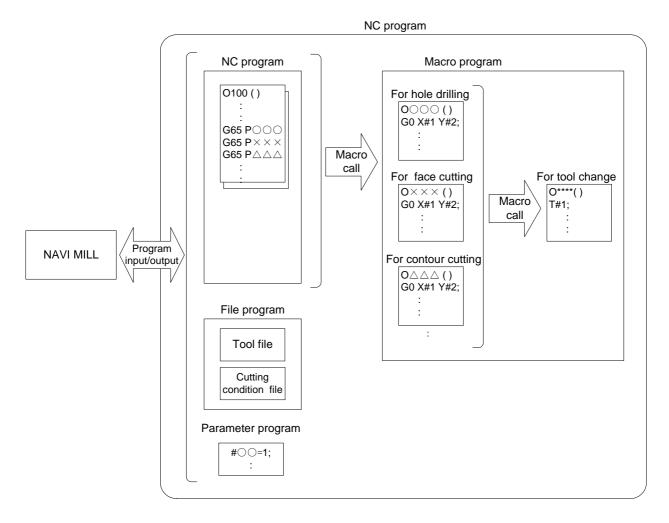
### **5. PROGRAM SPECIFICATIONS**

## **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

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



## 5.1 NC Program

NC program generated by NAVI MILL is limited with its program No. to from 1 to 7999 or from 10000 to 99999999.

### 5.1.1 Output Method for NC Program

In the NAVI 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);	Machining start comment Process data
Hole drilling (Deep hole Line)	(/NAVI); (NAVI-HOLE-PECK-LINE);	Process end comment
Hole drilling (Drill Line)	(/NAVI); (NAVI-HOLE-DRILL-LINE);	
Hole drilling (Drill Line)	(/NAVI); (NAVI-HOLE-TAP-LINE);	
Pocket (Square)	(/NAVI); (NAVI-POCKET-SQR); ••• (/NAVI);	
	•••	

Process start comment

Proce	ess	Comment	Remarks
Initial setting		(NAVI-INIT);	
Hole drilling	Drilling	(NAVI-HOLE-DRILL-****);	The symbol which indicates the
	Pecking	(NAVI-HOLE-PECK-****);	machining pattern of the hole drilling is
	Step	(NAVI-HOLE-STEP-****);	set in the"****"part.
	Boring	(NAVI-HOLE-BORE-****);	<pattern symbol=""></pattern>
	Tapping	(NAVI-HOLE-TAP-****);	Random (RNDM), Linear (LINE),
			Arc (ARC), Circle (CIR), Square (SQR),
			Grid (GRID)
Face cutting	Square	(NAVI-FACE-SQR);	
	Circle	(NAVI-FACE-CIR);	
Contour	Square	(NAVI-CONTOUR-SQR);	
cutting	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);	
EIA		(NAVI-EIA);	
End process		(NAVI-FIN);	

### **5. PROGRAM SPECIFICATIONS**

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.	
	Line Arc Circle Square Grid	G65 P9120 A B C D E F•••Z;		
Face cutting	Square Circle	G65 P9130 A B C D E F • • • Z;		
Contour cutting	Square Circle	G65 P9140 A B C D E F • • • Z;	The macro program No. depends on the machining pattern.	
	Free	(ROUGH-CENTER) #5=3; #1=0; WHILE[#5 GT 0.] DO1; G65 P9140 A B C D E F···Z; G00 X10. Y10.; $\checkmark$ Start position data G01 Z#180 F160.0000; F320.0000; G1 X20. Y20.; G1 X30. Y30.; ···; 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.	
Pocket	Square Circle L pattern U pattern	G65 P9150 A B C D E F • • • Z;	The macro program No. depends on the machining pattern.	
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 manufacturer 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

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

#### (2) Inserting block

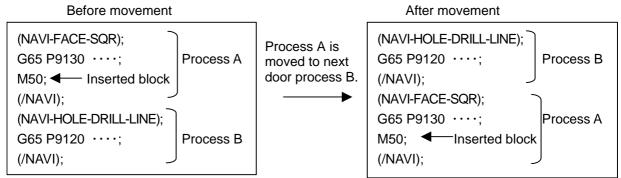
Inserting a block into the processes of the NC program (between the process end comment and next process start comment) has no problem.

When a block is inserted into the process of the NC program (between the process start comment and process end comment), the process can be edited with the NAVI MILL. However, the inserted block is not recognized in most cases. Therefore, if the process which a block is inserted into is edited with the NAVI MILL, the block may be lost.

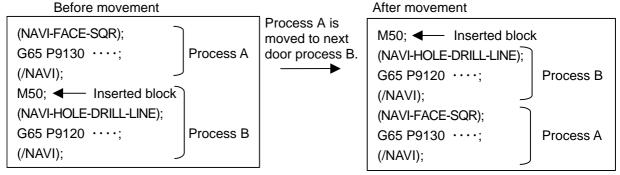
In response to operating process (moving process, deleting process, copying process) with the NAVI MILL, an inserted block is operated as follows.

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. PROGRAM SPECIFICATIONS**

### 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

### 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

### 5.4 Macro Program

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

<Program No., Comment>

No.	Name	User	МТВ	Program comment		
		macro No.	macro No.			
1	Macro program for INIT process	9110	100019110	INIT MACRO		
2	Macro program for hole drilling process	9120	100019120	HOLE MACRO		
3	Macro program for face cutting process	9130	100019130	FACE MACRO		
4	Macro program for	9140 to 9147	100019140 to	CNT-SQR-MACRO		
	contour cutting process		100019147	CNT-CIR-MACRO		
				CNT-FREE-MACRO		
5	Macro program for	9150 to 9155	100019150 to	PKT-SQR-MACRO		
	pocket process		100019155	PKT-CIR-MACRO		
				PKT-LTYPE-MACRO		
				PKT-UTYPE-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**

٦	Fool Change Method*	0 1 (Without Next Tool (With Next Too Stand-by Function) Stand-by Functio		2 (With Next Tool Stand-by Function)		
	Initial Sec.		T1;	T1;		
	Process Sec.	M6T1;	M6T2;	M6; T2; Process with T1		
7		Process with T1	Process with T1			
Machining		M6T2;	M6T3;	M6; T3;		
		Process with T2	Process with T2	Process with T2		
Program		M6T3;	M6;	M6;		
am		Process with T3	Process with T3	Process with T3		
	End Sec.**	M6T□;	T□; M6;	T□; M6;		

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

\* 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. TOOL FUNCTIONS

#### **Editing Tool Number and Next Tool Number** 6.2

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.

PRC	OCESS LIST				PRC	GRA	M:100	1/1	Output		
No	PCS	T NAME	Т	NT	н	D	S	F	Command	Spindle	Stand-by
0	<init></init>	<>		1					T1		<b>→1</b>
1	<face-sqr-r></face-sqr-r>	<face50></face50>	1	0	1	1	800	400.000	M6	1	
2	<face-sqr-f></face-sqr-f>	<face50></face50>	0	5	1	1	800	400.000	T5	1	→5
3	<dr-grid></dr-grid>	<cd30></cd30>	5	4	5	5	1500	100.000	M6T4	5	1→4
4	<peck-grid></peck-grid>	<dr68></dr68>	4	3	4	4	935	102.000	M6T3	4	5→3
5	<dr-grid></dr-grid>	<dc20></dc20>	3	2	3	3	160	80.000	M6T2	3	4→2
6	<tap-grid></tap-grid>	<tap80></tap80>	2	0	2	2	317	1.250	M6	2	3

[Example of tool number / next tool number input]

T	NT	Output Command	Remarks
=0	=0	No output command	
=0	≠ <b>0</b>	TD	
≠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	Т	NT	Н	D	S	F
0	<init></init>	<>	~~~~	(0)				
1	<face-sqr-r></face-sqr-r>	<face50></face50>	(0)	0	1	1	800	400.000
2	<face-sqr-f></face-sqr-f>	<face50></face50>	0	5	1	1	800	400.000
3	<dr-grid></dr-grid>	<cd30></cd30>	5	4	5	5	1500	100.000

# 7. RESTRICTIONS FOR CNC FUNCTION SPECIFICATIONS

In operating the NAVI MILL and the format of the machining program created with the NAVI MILL, the following restrictions are applied to the MELDAS 700 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
	#8112 DECIMAL PNT-P	MILL is executed.

#### **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	Incomplete data exists on the pattern screen.
	screen	
	E006 No data setting on cut condition screen	screen.
	E007 Data range over on pattern screen	The data exceeded a set range was input on the pattern screen.
	E008 Data range over on cut condition screen	The data exceeded a set range was input on the cutting condition screen.
Program	E101 Designated file does not exist	The designated program does not exist.
editing	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	Reading of the cutting condition file was failed.
	error	Check the path(drive/folder) of the file.
	E110 CUT CONDITION file write	Writing to the cutting condition file was failed.
	error	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	Writing to the PREFERENCE data was failed.
	error	
	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	E211 Process number over	The number of processes exceeded 100.
editing		
Hole	E221 ZI < ZF	"Initial Z < Workpiece top position" is applied.
machining	E222 B < H	" Tool nose depth < Hole depth " is applied.
	E223 D > 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.
Face cutting	E231 H < Q	"Allowance < Finishing allowance" is applied.
machining	E232 WIDTH/2 < CUT WIDTH	"Width/2 < Machining width" is applied.
	E233 R < CUT WIDTH	"Radius < Machining width" is applied.
Contour/	E241 FH > H	"Finishing allowance Z > Allowance Z" is applied.
Pocket	E242 FE > E	"Finishing allowance XY > Allowance XY" is applied.
	E243 FE > R	"Finishing allowance XY > Radius" is applied.
	E244 R < Tool diameter / 2	When CIRCLE and INSIDE are specified: "Radius < Tool diameter / 2" is applied.
	E245 WIDTH < Tool diameter	When SQUARE and INSIDE are specified: "Width < Tool diameter" is applied.
	E246 CORNER SIZE > WITDH / 2	When SQUARE and INSIDE are specified: "Corner size > Width / 2" is applied.
	E247 WIDTH < Tool diameter * 2	(When HELIX is specified:) "Width < Tool diameter*2" is applied.
	E248 WIDTH /2 < Corner Size	"Width/2 < Corner size" is applied.
	E249 WIDTH K > WIDTH I	"Width K > Width I" is applied.
	E250 Geometry record number	For Contour - Free:
	error	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	For Contour - Free:
	error	Illegal record No. is applied.
	E253 I,J agreement with angle L**	Data error in Contour - Free
		Linear I,J and angle are contradictory.
	E254 No end point on	Data error in Contour - Free
	circumference L**	There is no end point on the circumference.
	E255 No continuity with previous	Data error in Contour - Free
	line L**	Patterns are not connected.
	E256 No circle L**	Data error in Contour - Free
		Circle cannot be determined from set data.
	E257 Corner C error L**	Data error in Contour - Free
		No corner C.
	E258 Corner R error L**	Data error in Contour - Free
		No corner R.
	E259 Free shape input error L**	Data error in Contour - Free
		An error is found when inputting a free shape.

(Continued to the next page)

(Continued from the previous page)

Division	Message	Details		
Contour/	E260 Last line has corner R/C L**	Data error in Contour - Free		
Pocket		Corner R/C exists in the last line.		
	E261 Start point error L**	Data error in Contour - Free		
		An error is found in the start point.		
	E262 Corner no move L**	Data error in Contour - Free		
		The block following corner C/R is not a movement		
		command.		
	E263 Corner short L**	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**	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.		
EIA	E271 Block number over	The number of EIA blocks was exceeded.		
	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) When data error in Contour – Free occurs, line No. of the shape data is displayed following "L".

# 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.
	The data was changed. Save the	N: Cancel opening the program.
	changes?(Y/N)	Save confirmation for unsaved data
		Y: Save data.
	The page connet be abanged	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

Variat	ole No.				
User macro mode	MTB macro mode	Code	Data name	Setting range	Remarks
#150	#450	I/D	ATC PATTERN X	1 to 4	Variable for operation
			(Integral part)		
			ATC PATTERN Y	1 to 4	
			(Decimal part)		
#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	1 to 4	Variable for operation
			(Integral part)		
			END PATTERN Y	1 to 4	
			(Decimal part)		
#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

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

#### (2) Parameter variables during program operation

# 

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

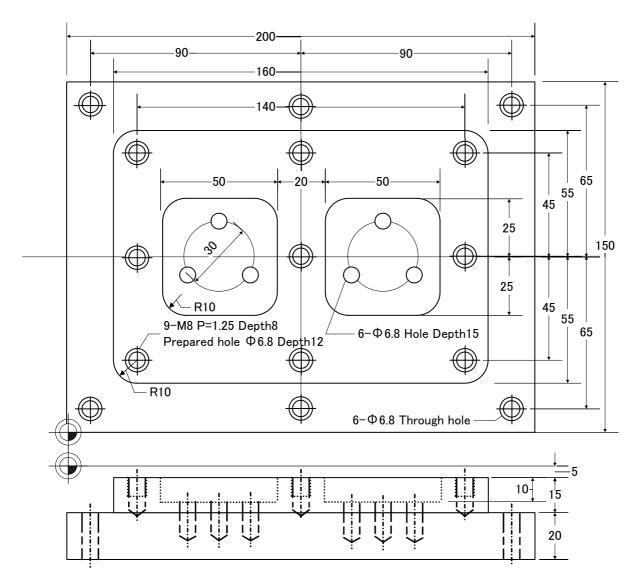
NC program mode	Variables used by NAVI MILL
User macro mode	#150 to #179
MTB macro mode	#450 to #479

When NC program mode is user macro mode, do not use common variables (#150 to #179). 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



Processes are shown below.			
Process	Machining	ΤοοΙ	
1	Face rough machining	φ50 Face Mill	
1	Face finishing machining	φ50 Face Mill	
2	Contour rough machining	φ20 End Mill	
2	Contour finishing machining	φ20 End Mill	
3	Pocket rough machining	φ20 End Mill	
5	Pocket finishing machining	φ20 End Mill	
4	Drilling	φ3 Center Drill	
-	Drilling	φ6.8 Drill	
6-¢6.8	Drilling	φ20 Countersink	
	Drilling	φ3 Center Drill	
5	Drilling	φ6.8 Drill	
9-M8	Drilling	φ20 Countersink	
	Tapping	M8 P=1.25 Tap	
6	Drilling	φ3 Center Drill	
ο 6-φ6.8	Drilling	φ6.8 Drill	
υ-ψυ.υ	Drilling	¢20 Countersink	

# Appendix 2.2 Process Table

### 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.

ltem	Face Mill	Тар	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 wokpiece. 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.

Setting value	Details
1	S45C
50	
54	G54
2	1st zero point
-	
2	1st zero point
-	
99	
2	1st zero point
-	
2	1st zero point
-	
1	M30
	1 50 54 2 - 2 - 99 2 -

Setting value

Details

SQUARE

LIST	∕IEW	
PROG	iRAM	
PROC	ESS	
0	INIT	
FILE		

LIST	∕IEW	
PROG	iRAM	
PROC	ESS	
0	INIT	
FILE		

2) Press the [RETURN] menu.

Item

WORK SHAPE

+X -X

+Y

-Y

+Z

-Z

3) Save the initial conditions by pressing the [SAVE] menu.

1 200.

0

0

0 -40

150.

4) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

#### (5) Process 1 Face rough machining ( $\phi$ 50 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	Х	20	
	Y	20	
WIDTH X	Ι	160	
Y	J	110	
SURFACE Z	ZF	0	
ALLOWANCE	Н	5	
FINISH ALLOW	Q	0.2	
NUM OF CUTS	Т	3	
START POINT (1 -	4)	3	
ANGLE	А	0	

3) Press the next page key  $[\rightarrow]$  and set the following items.

Item	Setting value	
CUT DIRECTION	1	Х
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  $[\leftarrow]$  key.

#### (6) Process 1 Face finishing machining ( $\phi$ 50 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.

ltem	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 [ $\leftarrow$ ] key.

LIST	∕IEW
PROG	RAM
PROC	ESS
0	INIT
1	FACE-SQR
FILE	

LIST	LIST VIEW			
PROG	RAM			
PROC	ESS			
0	INIT			
1	FACE-SQR			
2	FACE-SQR			
FILE				

#### (7) Process 2 Contour rough machining ( $\phi$ 20 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.

ltem		Setting value	
TOOL REG No.		6	
PROCESS		1	ROUGH
SURFACE Z	ZF	-5	
ALLOWANCE Z	Н	20	
ALLOWANCE XY	Е	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	Х	20	
BASE POS Y	Y	20	
WIDTH X	I	160	
WIDTH Y	J	110	
CORNER SIDE	R	10	
ANGLE	Α	0	
START POINT		3	

- LIST VIEW PROGRAM PROCESS 0 INIT 1 FACE-SQR 2 FACE-SQR 3 CNT-SQR FILE
- 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  $[\leftarrow]$  key.

#### (8) Process 2 Contour finishing machining ( $\phi$ 20 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.

ltem	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 [ $\leftarrow$ ] key.

LIST \	/IEW
PROG	iRAM
PROC	ESS
0	INIT
1	FACE-SQR
2	FACE-SQR
3	CNT-SQR
4	CNT-SQR
FILE	

#### (9) Process 3 Pocket rough machining (\u00f620 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	Н	10	
FIN ALLOW Z	FH	1.5	
FIN ALLOW XY	FE	1.5	
NUM OF CUTS Z	ΖT	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	Х	40	
BASE POS Y	Y	50	
WIDTH	I	50	
WIDTH	J	50	
CORNER SIDE	R	10	
ANGLE	А	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 [ $\leftarrow$ ] key.

#### (10) Process 3 Pocket finishing machining (\$20 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.

ltem	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 [ $\leftarrow$ ] key.

LIST	/IEW			
PROG	RAM			
PROCESS				
0	INIT			
1	FACE-SQR			
2	FACE-SQR			
3	CNT-SQR			
4	CNT-SQR			
5	PKT-SQR			
FILE				

LIST	VIEW			
PROG	<b>iRAM</b>			
PROCESS				
0	INIT			
1	FACE-SQR			
2	FACE-SQR			
3	CNT-SQR			
4	CNT-SQR			
5	PKT-SQR			
6	PKT-SQR			
FILE				

#### **APPENDIX 2. PROGRAMMING EXAMPLE**

#### Appendix 2.4 Creating Program

## 

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.

ltem		Setting value	
BASE POS X	Х	110	

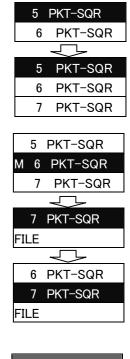
- 5) Save the data of the pocket rough machining <SQUARE> by pressing the [SAVE] menu.
- 6) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

#### (12) Process 3 Pocket finishing machining (\u00f620 End Mill)

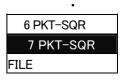
- 1) Move the cursor to the [6 PKT-SQR] and press the [COPY] menu.
- 2) Move the cursor to the [7 PKT-SQR] and press the [MOVE] menu.
- 3) Move the cursor to the [8 PKT-SQR] and press the [INPUT]- [Y] menu.
- 4) Press the [MODIFY] [PATTERN] menu and set the following item.

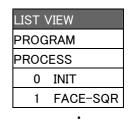
ltem		Setting value	
BASE POS X	Х	110	

- 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 [ $\leftarrow$ ] key.



PROGRAM PROCESS 0 INIT 1 FACE-SOR	LIST \	√IEW			
0 INIT	PROGRAM				
	PROC	ESS			
1 FACE-SQR	0	INIT			
	1	FACE-SQR			





	•
7	PKT-SQR
8	PKT-SQR
FILE	

#### (13) Process 4 Drilling (\phi3 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	Н	3		
NOSE DEPTH	В	3.866		
SPOT DIAMETER	D	3		
CUT AMOUNT		-		
DWELL		0		
PATTERN		<square></square>		
SP SPEED		<1050>		
FEED RATE		<70>		

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

<hole drilling<="" th=""><th>machining</th><th>pattern screen</th><th>(SQUARE)&gt;</th></hole>	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	

LIST \	/IEW		
PROGRAM			
PROC	ESS		
0	INIT		
1	FACE-SQR		
	•		
	•		
	•		
8	PKT-SQR		
9	DR-SQR		

FILE

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  $[\leftarrow]$  key.

### Appendix 2.4 Creating Program

### (14) Process 4 Drilling (\phi 6.8 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.

ltem		Setting value	
TOOL REG No.		4	
HOLE CYCLE		2	PECK
DEPTH	Н	21	
CUT AMOUNT		2	

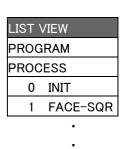
- 3) Save the data of the hole drilling <SQUARE> by pressing the [SAVE] menu.
- 4) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

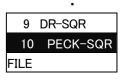
# (15) Process 4 Drilling (\u00f620 Countersink)

- 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.

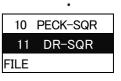
Item		Setting value	
TOOL REG No.		3	
HOLE CYCLE		1	DRILL
SPOT DIAMETER	D	7.8	
DWELL		0	

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





LIST	√IEW
PROG	iRAM
PROC	ESS
0	INIT
1	FACE-SQR



#### (16) Process 5 Drilling (\u00f63 Center Drill)

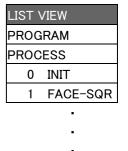
- 1) Open the process mode selection screen by pressing the [NEW] menu.
- 2) Open the hole drilling screen and set the following items.
  - < Hole drilling screen>

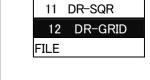
Item		Setting value	
TOOL REG No.		5	
HOLE CYCLE		1	
INITIAL Z	ZI	50	
SURFACE Z	ZF	-5	
DEPTH	Н	3	
NOSE DEPTH	В	3.866	
SPOT DIAMETER	D	3	
CUT AMOUNT		-	
DWELL		0	
PATTERN		<grid></grid>	
SP SPEED		<1050>	
FEED RATE		<70>	

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 [ $\leftarrow$ ] key.

#### Appendix 2.4 Creating Program

#### (17) Process 5 Drilling (\phi 6.8 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	Н	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 [ $\leftarrow$ ] key.

#### (18) Process 5 Drilling (\u00f620 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.

ltem		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 [ $\leftarrow$ ] 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.

ltem		Setting value	
TOOL REG No.		2	
HOLE CYCLE		5	ТАР
DEPTH	Н	8	
NOSE DEPTH	В	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 [ $\leftarrow$ ] key.

LIST	√IEW
PROG	iRAM
PROC	ESS
0	INIT
1	FACE-SQR
	•
	•
	•
12	DR-GRID
13	PECK-GRID

FILE

LIST \	/IEW
PROG	RAM
PROC	ESS
0	INIT
1	FACE-SQR

13	PECK-GRID
14	DR-GRID
FILE	

LIST	√IEW			
PROG	iRAM			
PROCESS				
0	INIT			
1 FACE-SQR				

14	DR-GRID
15	TAP-GRID
FILE	

#### (20) Process 6 Drilling (\u00f63 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	Н	3	
NOSE DEPTH	В	3.866	
SPOT DIAMETER	D	3	
CUT AMOUNT		-	
DWELL		0	
PATTERN		<circle></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	

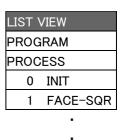
- 4) After returning the screen to the hole drilling screen by pressing the [RETURN] menu, save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 5) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

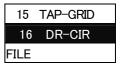
#### (21) Process 6 Drilling ( $\phi$ 6.8 Drill)

- 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.

ltem		Setting value	
TOOL REG No.		4	
HOLE CYCLE		2	PECK
DEPTH	Н	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  $[\leftarrow]$  key.





LIST \	∕IEW			
PROG	iRAM			
PROCESS				
0 INIT				
1	FACE-SQR			

	•
16	DR-CIR
17	PECK-CIR
FILE	

#### (22) Process 6 Drilling (\u00f620 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 [ $\leftarrow$ ] key.

#### (23) Process 6 Drilling (\u03c63 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	

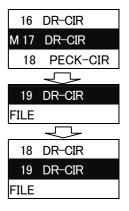
5) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.

6) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

LIST	∕IEW			
PROG	iRAM			
PROCESS				
0	INIT			
1	FACE-SQR			

	•	
17	PECK-CIR	
18	DR-CIR	
FILE		

16	DR-CIR
17	PECK-CIR
	$\downarrow$
16	DR-CIR
17	DR-CIR
18	PECK-CIR



LIST VIEW		
PROGRAM		
PROCESS		
0	INIT	
1	FACE-SQR	
	•	

	•	
18	DR-CIR	
19	DR-CIR	
FILE		

#### Appendix 2.4 Creating Program

#### (24) Process 6 Drilling (\u00f36.8 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.

ltem	Setting value	
POS X	135	

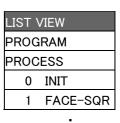
- 5) Save the data of the hole drilling <CIRCLE> by pressing the [SAVE] menu.
- 6) Turn the LIST VIEW area active by pressing the [ $\leftarrow$ ] key.

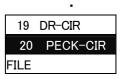
#### (25) Process 6 Drilling (φ20 Countersink)

- 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 [ $\leftarrow$ ] key.





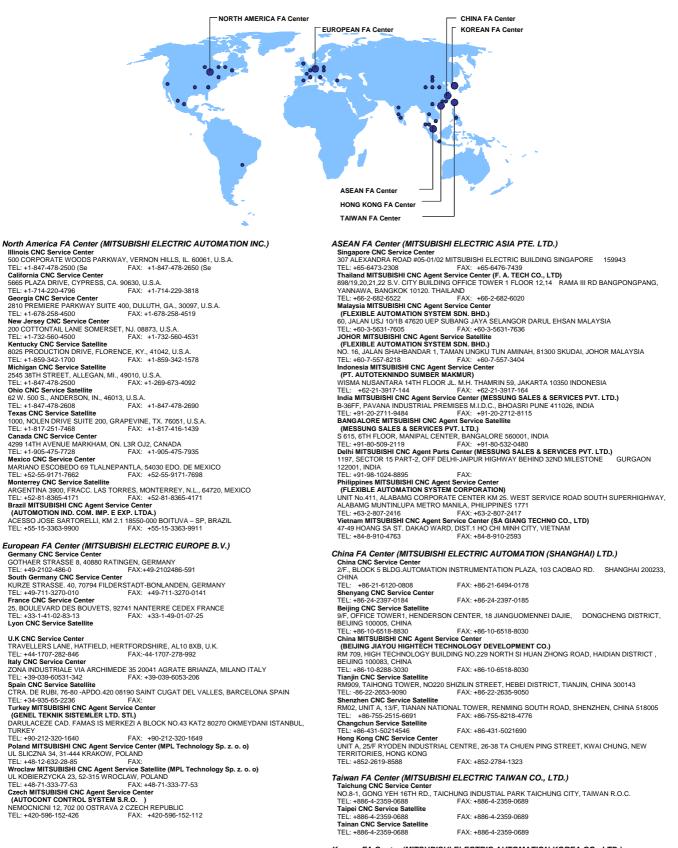
LIST	√IEW	
PROGRAM		
PROC	ESS	
0	INIT	
1	FACE-SQR	
	•	

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><li> "4.8.1 Tool Guidance Screen" was added.</li><li>Mistakes were corrected.</li></ul>

### **Global service network**



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

 Korea CNC Service Center
 Construction

 DONGSEO GAME CHANNEL BLDG. 2F. 660-11, DEUNGCHON-DONG KANGSEO-KU SEOUL, 157-030
 KOREA

 TEL: +82-2-3660-9607
 FAX: +82-2-3663-0475

# Notice

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

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

# **Duplication Prohibited**

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

© 2005 MITSUBISHI ELECTRIC CORPORATION ALL RIGHTS RESERVED.

# **MITSUBISHI CNC**

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

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

Specifications subject to change without notice. Printed in Japan on recycled paper.