

LNC-T600/T60 Series

Operator's Manual

2008/3 Ver : V04.00.000(4408110007)

Leading Numerical Controller



LNC Technology Co., Ltd.

Table of Contents

1	GENERAL	1
1.1	Hardware Specification	2
1.2	Software Specification	3
1.3	System Setting Unit.....	4
1.4	G Code Table	5
2	CNC OPERATION.....	7
2.1	Operation Installation Type.....	7
2.1.1	LCD Display Device	8
2.1.2	MDI Panel.....	9
2.2	Operation Installation Introduction.....	10
2.3	Screen and Function Description	16
2.3.1	Display Screen Layout	16
2.3.2	Function Group	17
2.3.3	ABSOLUTE COORDINATE SCREEN.....	25
2.3.4	RELATIVE COORDINATE SCREEN.....	26
2.3.5	MACHINERY COORDINATE SCREEN.....	27
2.3.6	HANDLE SCREEN.....	28
2.3.7	EXIT.....	30
2.4	PROG	31
2.4.1	FGPROG	32
2.4.2	BGPROG (Background Program)	35
2.4.3	PROCHK (Program Checking)	36
2.4.4	MDI	37
2.4.5	DIRMNG (File Management).....	38
2.5	PROG Function.....	44
2.5.1	FGPROG (Foregoing Program).....	45
2.5.2	BGPROG (Background Program)	47
2.5.3	PROCHK (Program Checking)	48
2.5.4	DIR MNG (File Management).....	50
2.5.5	COMM (Communication).....	63



2.5.6	DNC Function	69
2.6	OFFSET Functions	70
2.6.1	WEAR (Wear OFFSET)	70
2.6.2	GEOM (Geometry OFFSET)	73
2.6.3	W. SHIFT (Work Coordinate Shift).....	74
2.6.4	MACRO	75
2.6.5	WORK (Coordinate Setting)	77
2.7	Lathe EASY-CAM.....	80
2.7.1	Function Description	80
2.7.2	Operation Procedure of How to Produce Working Path Program	80
2.7.3	Operation Screen and Function Key Instruction	82
2.8	GRAPH (GRAPH)	109
2.8.1	Function Introduction.....	109
2.8.2	SIMU.....	109
2.8.3	Windows Definition.....	112
2.9	DGNOS.....	114
2.9.1	ALARM	114
2.9.2	System Upgrade	116
2.9.3	IOCSA.....	122
2.9.4	MLC2.....	124
2.9.5	SYSTEM.....	129
2.10	SOFTPL	132
2.11	PARAM.....	134
2.11.1	NC. SYS (System Parameter).....	134
2.11.2	Users Parameter.....	137
2.11.3	Authorization.....	138
2.12	RESET	143
3	OP PANEL OPERATION	144
3.1	Operation Panel	144
3.1.1	Power Switch (ON/OFF).....	145
3.1.2	EMG-STOP Key.....	145
3.1.3	CYCLE START.....	146
3.1.4	FEED HOLD	147



3.1.5	LED SIGNAL.....	147
3.1.6	MODE SELECT	148
3.1.7	Feedrate/Rapid Speed Adjusting Button.....	150
3.1.8	SPINDLE SPEED ADJUSTMENT SWITCH.....	151
3.1.9	SPINDLE ROTATION.....	152
3.1.10	COOLANT SUPPLY	153
3.1.11	AXIS SELECTION.....	154
3.1.12	OT RELEASE	155
3.1.13	TOOL MAGAZINE.....	156
3.1.14	Single Block Stop	157
3.1.15	Additional Function Selection	157

1 General

LNC-T600 is a PC-based controller for CNC turning machines. It is a high-tech product that is developed by the **LNC Technology Co., Ltd.** through years of dedicating research and development, involving massive manpower and resources. LNC-600 serial controller is featured in high-speed, high-precision, and high-efficiency. Operation of LNC-T600 will be introduced in the following chapters.

1.1 Hardware Specification

Specification	Basic	Standard
PC	Industrial PC	Industrial PC
Display Interface	VGA interface	VGA interface
Data Transfer Interface	Ethernet、RS-232/485	Ethernet、RS232
Storage Interface	IDE、FDD、CF	IDE、FDD
Output	n/a	5V、12V
SDRAM	32M or above	32M or above
Storage Device	CF card 32M or above	CF card 32M or above
Wiring	front	front and above
Servo System	position loop	position loop / speed loop control
Spindle System	Pulse control / DA output	Pulse control / DA output
Remote I/O	256 In / 256 Out	256 In / 256 Out
DNC	RS232 19200 Baud Rate	RS232 19200 Baud Rate
Max Control Axes	4 axes Pulse	5 Pulse or 6 Vcmd(Ver2.1)
Spindle	1	1
Work Temp.	0~55 °C	0~55 °C
Input	12V(2A)、5V(6A or higher)	AC110/220V 50/60HZ

1.2 Software Specification

LNC T600	Specification
File Types	DOS FAT
Program Specifications	Normal G, M Code Program
	Background Editing Function
	MACRO Program Function
Modes	EDIT Mode
	MEM Mode
	MDI Mode
	JOG Mode
	MPG Mode
	RAPID Mode
	HOME Mode
Group Functions	POS Function
	PROG Function
	OFFSET Function
	CAM Function
	GRAPH Function
	DGNOS Function
	SOFTPL Function
	PARAM Function
PLC (Machine Logic Controller)	I/O/C/S/A BIT
	Timer/Counter/Register
	Immediately Ladder Program Display
Language	Simplify/Traditional Chinese and English

1.3 System Setting Unit

Smallest Input Unit	Smallest Commanding Value	Maximum Travel Setting
0.001 mm	0.001 mm	99999.999 mm
0.0001 inch	0.0001 inch	9999.9999 inch
0.001 deg	0.001 deg	99999.999 deg

1.4 G Code Table

Table 1-1 G Code Table

Function Description	Group	TYPE A	TYPE B	TYPE C
Positioning (Rapid Traverse)	01	G00	G00	G00
Linear Interpolation (Cutting Feed)	01	G01	G01	G01
Circular/Helical Interpolation CW	01	G02	G02	G02
Circular/Helical Interpolation CCW	01	G03	G03	G03
Dwell	00	G04	G04	G04
Exact Stop	00	G09	G09	G09
Programmable Data Input	00	G10	G10	G10
Input in Inch	06	G20	G20	G70
Input in mm	06	G21	G21	G71
Stored Stroke Check Function ON	09	G22	G22	G22
Stored Stroke Check Function OFF	09	G23	G23	G23
Return to Reference Position	00	G27	G27	G27
1 st Reference Position Return	00	G28,G29	G28,G29	G28,G29
2 nd , 3 rd , 4 th Reference Position Return	00	G30	G30	G30
Tool Noise Radius Compensation Cancel	07	G40	G40	G40
Tool Noise Radius Left	07	G41	G41	G41
Tool Noise Radius Right	07	G42	G42	G42
Absolute Programming	03	--	G90	G90
Incremental Programming	03	--	G91	G91
Coordinate System Setting or Max. Spindle Speed Setting	00	G50	G92	G92
Per Minute Feed mm/min.	05	G98	G94	G94
Per Revolution Feed mm/rev.	05	G99	G95	G95
Constant Surface Speed Control	02	G96	G96	G96
Constant Surface Speed Control Cancel	02	G97	G97	G97
Thread Cutting	01	G32	G33	G33

Function Description	Group	TYPE A	TYPE B	TYPE C
Finishing Cycle	00	G70	G70	G72
Stock Removal in Turning	00	G71	G71	G73
Stock Removal in Facing	00	G72	G72	G74
Pattern Repeating	00	G73	G73	G75
End Face Per Drilling	00	G74	G74	G76
Outer/Internal Diameter Drilling	00	G75	G75	G77
Multiple Threading Cycle	00	G76	G76	G78
Outer/Internal Diameter Cutting Cycle	01	G90	G77	G20
Thread Cutting Cycle	01	G92	G78	G21
End face Turning Cycle	01	G94	G79	G24
Cycle for Face Drilling	10	G80-G89	G80-G89	G80-G89
MACRO Calling	00	G65	G65	G65
MACRO Modal Calling	12	G66	G66	G66
MACRO Modal Calling Cancel	12	G67	G67	G67
Returning to Initial Level	11	---	G98	G98
Returning to R Point Level	11	---	G99	G99
Exact Stop Mode	15	G61	G61	G61
Normal Cutting Mode	15	G64	G64	G64
Working Coordinate Systems Selection	14	G54~59	G54~59	G54~59

Note : TYPE A, B or C are decided by Parameter 153, the default value is TYPE B.

2 CNC Operation

2.1 Operation Installation Type

The control panels can be divided into two units: **MDI** and **OP**. The MDI is used to edit part programs as well as to enter relevant working data. The OP (Operation Panel), on the other hand, is used to manipulate operational conditions. There are multiple function keys, keys and pulse generator (hand wheel) and etc. OP will have different designs due to different tool machine designs. But, this system has one standard panel that can be selected by tool machine producers. MDI will be introduced in the following. OP and its functions will be introduced in Chapter Three.

2.1.1 LCD Display Device

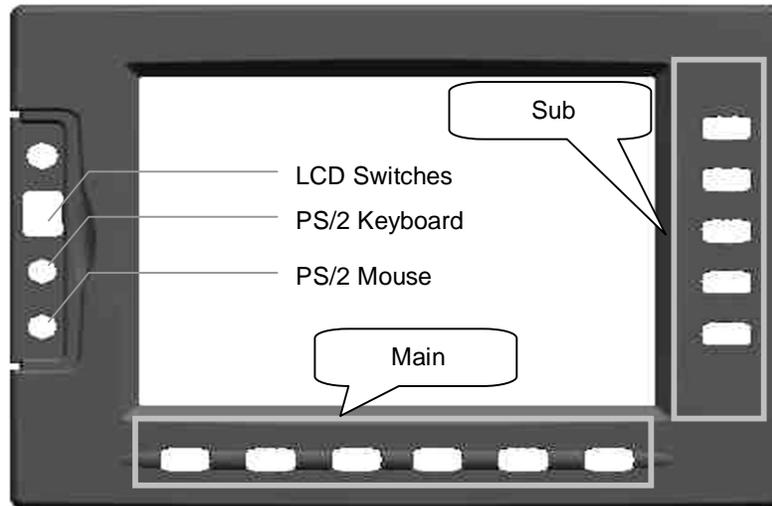


Figure 2.1-1 LCD Display Device

2.1.2 MDI Panel

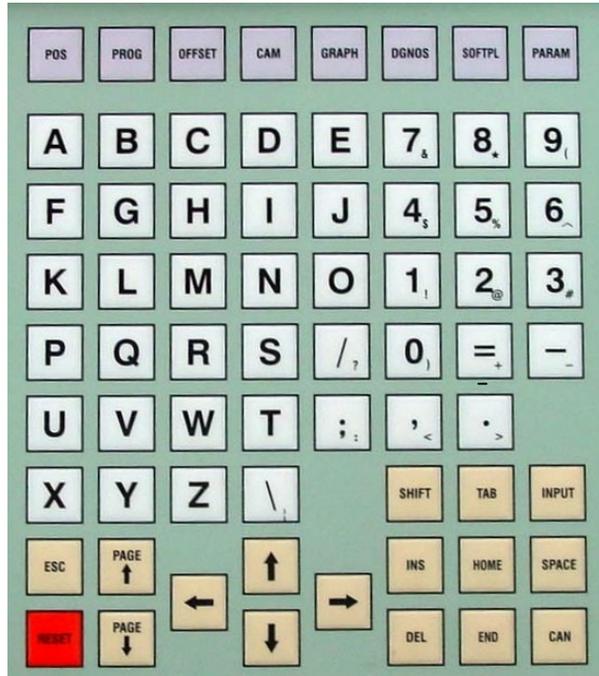


Figure 2.1-2 MDI Panel

2.2 Operation Installation Introduction

The following will introduce 4 types of function keys, based on their function, on MDI panel for LCD display device only:

(1) Main Function Keys :

There are 6 horizontal function buttons at bottom of LCD screen. Users can choose the desired function button corresponding to those function selections at bottom of the display screen by press them.

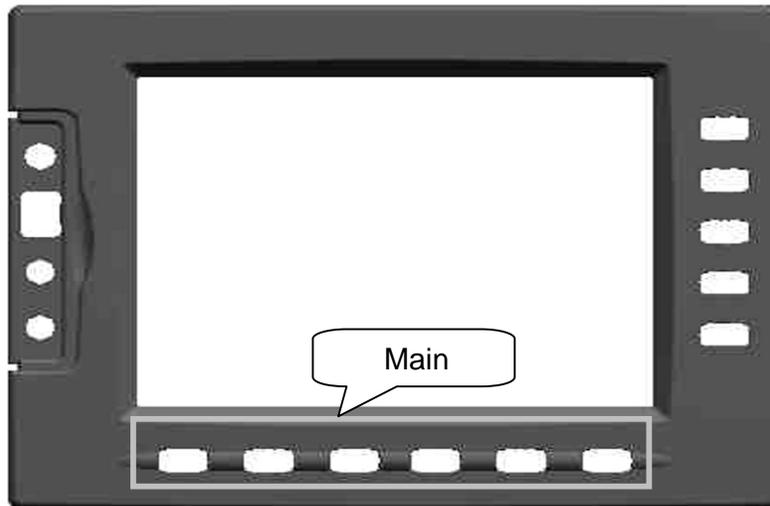


Figure 2.2-1 6 Main Function Keys at Bottom of Screen

(2) Sub-Function Keys :

After choosing the main function button, the content of sub-function will occur at right side of the screen. Pressing the corresponding function button to select the wanted function. Left diagram shows the sub-function content of the corresponding coordinate (main function buttons). Selecting any one function, the screen will display the last chosen screen.

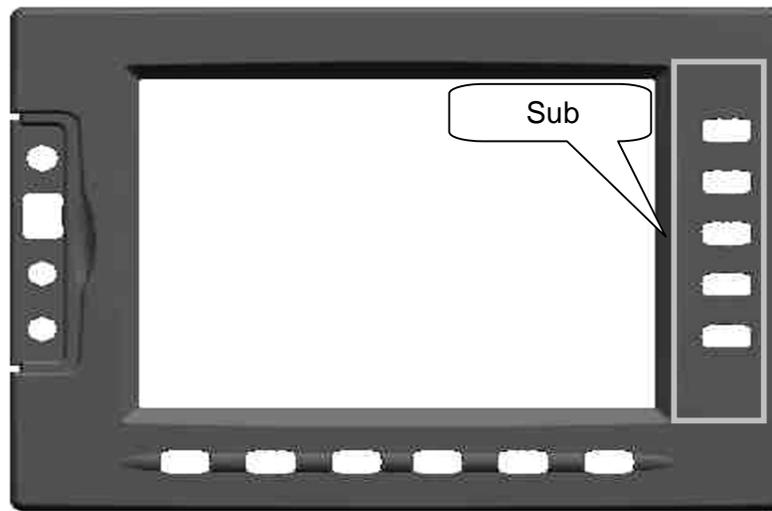


Figure 2.2-2 5 Sub-Function Keys on the Right of the Screen

(3) **Function Group Selection Keys :**

To select 8 functions such are **POS, PROG, OFFSET, CAM, GRAPH, DGNOS, SOFTPL** and **PARAM**.

<POS> : to display positions

<PROG> : to edit and to display program

<OFFSET> : to set and to display tool offsest

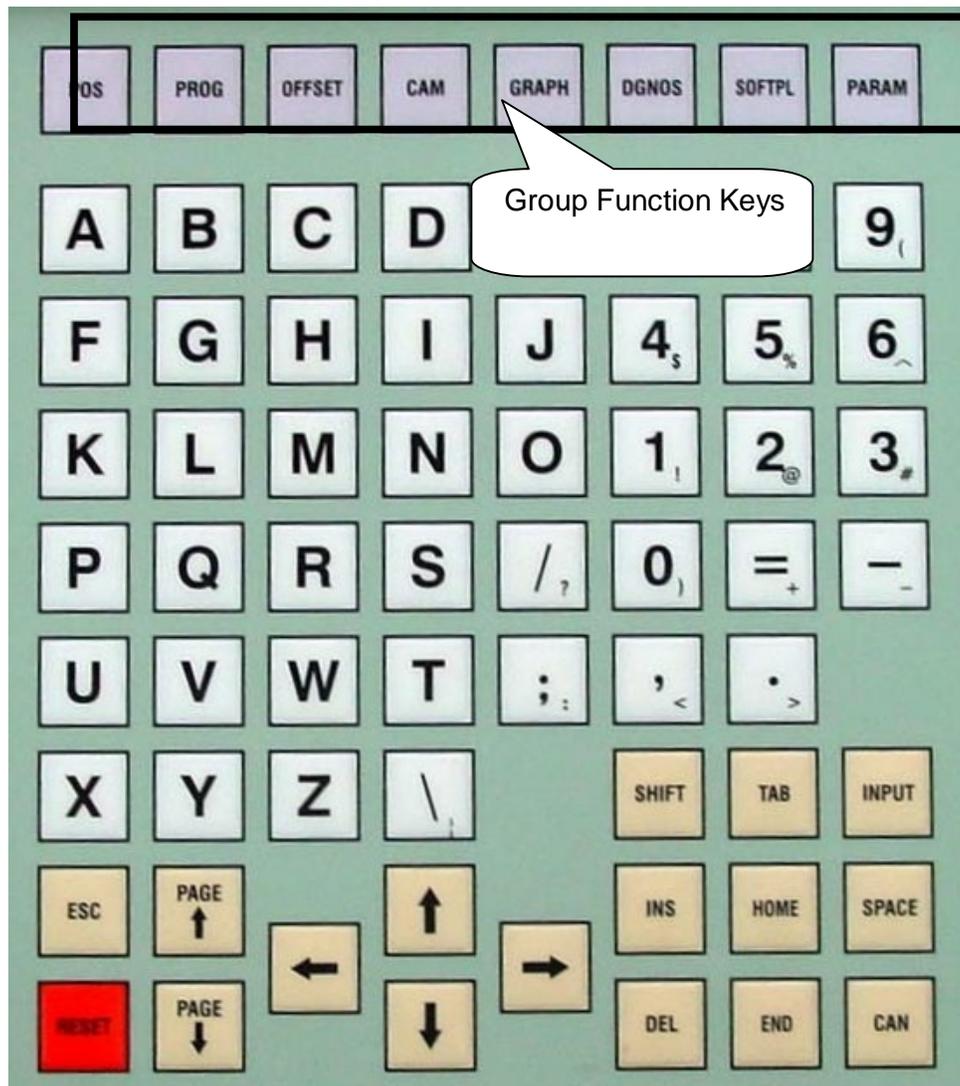
<CAM> : to edit working program by Figure method

<GRAPH> : to draw tool path

<DGNOS> : to display instance messge at DGNOS page

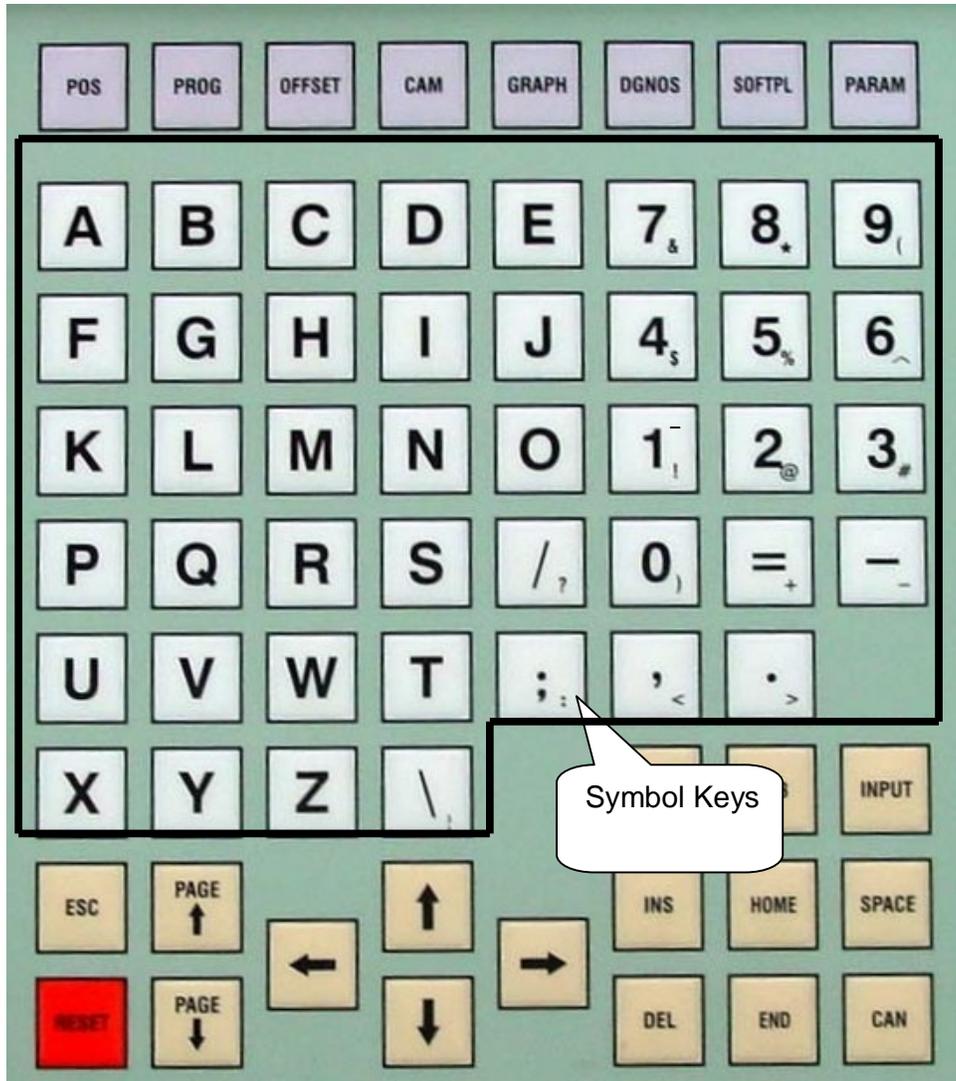
<SOFTPL> : to select software panel switches

<PARAM> : to display parameter screen



(4) **Character & Symbol Keys:**

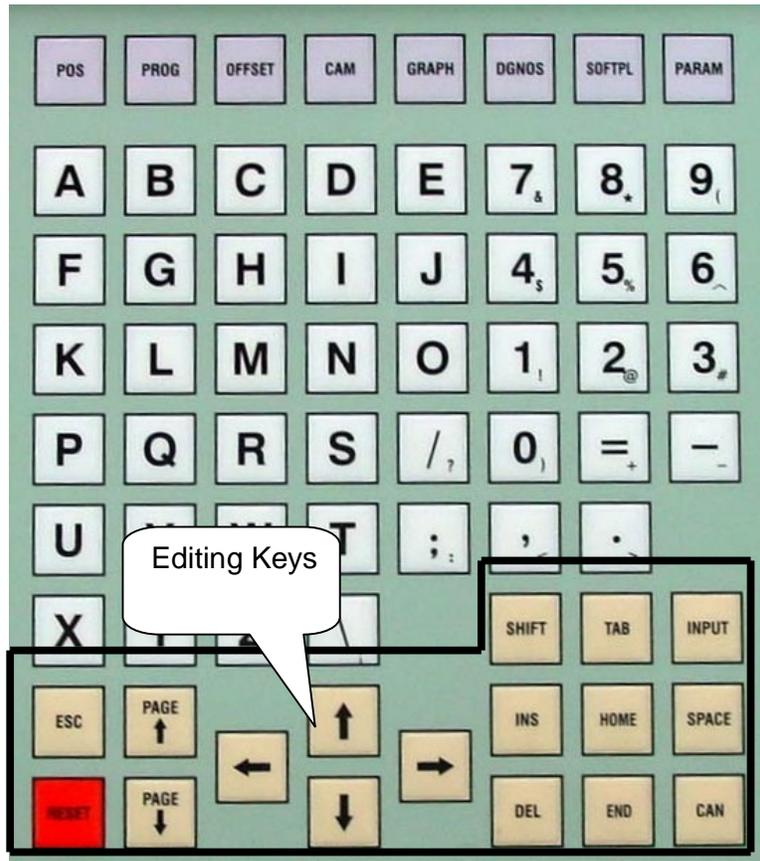
These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.



(5) **Editing Keys :**

Using these keys with cursor on the screen will be able to modify program, to set data and to change page.

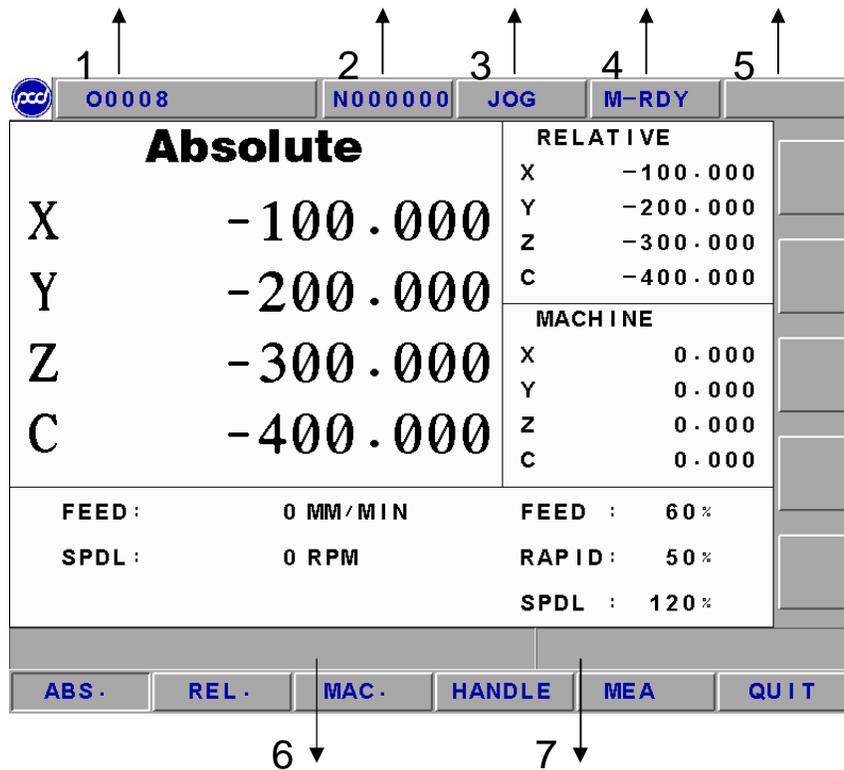
- <SHIFT> : To key-in special symbols with use of symbol number keys.
- <INPUT> : To confirm entered data.
- <INS> : To switch enter/replace characters modes.
- : To delete.
- <HOME> : To return cursor to the beginning of the sentence when editing program.
- <END> : To return cursor to the end of the sentence when editing program.
- <SPACE> : To key-in into empty space.
- <CAN> : To cancel the previous character.
- <PAGE↑> : To turn to the previous page.
- <PAGE↓> : To turn to the next page.
- <→> : To move cursor right.
- <←> : To move cursor left.
- <↑> : To move cursor up.
- <↓> : To move cursor down.
- <RESET> : To reset system.



2.3 Screen and Function Description

8 function groups in this controller: **POS**, **PROG**, **OFFSET**, **CAM**, **GRAPH**, **DGNOS**, **SOFTPL** and **PARAM**. Using **【...】** to indicate function keys at bottom and at right of the screen and to use **<... >** to indicate keys on MDI panel.

2.3.1 Display Screen Layout



- 1 : present designate file name
- 2 : present single block that is executed by controller
- 3 : CNC mode signal
- 4 : machine condition signal
- 5 : wrong alarm/warning message
- 6 : simply message hint area
- 7 : entry area

2.3.2 Function Group

POS (Position Coordinate) Function Keys

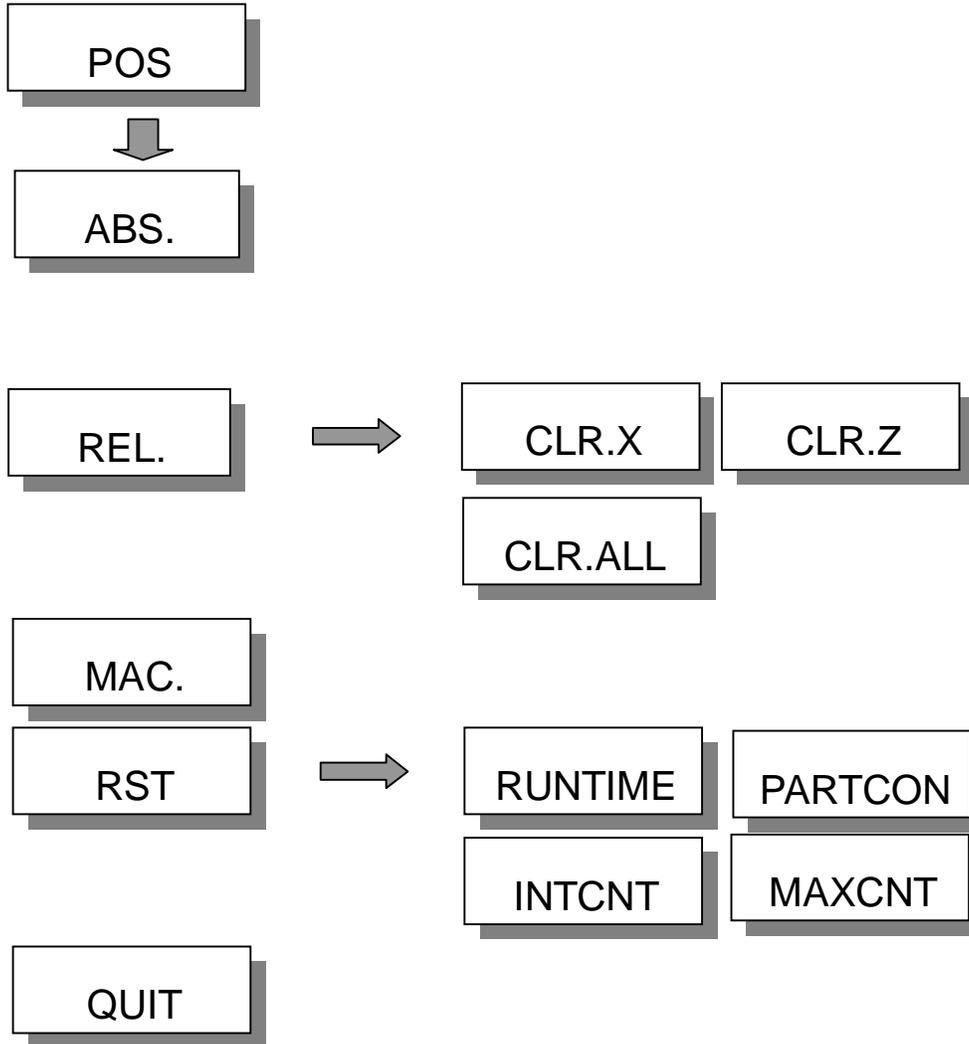


Figure 2.3-1 POS Structure Figure

Program Editing Function Keys

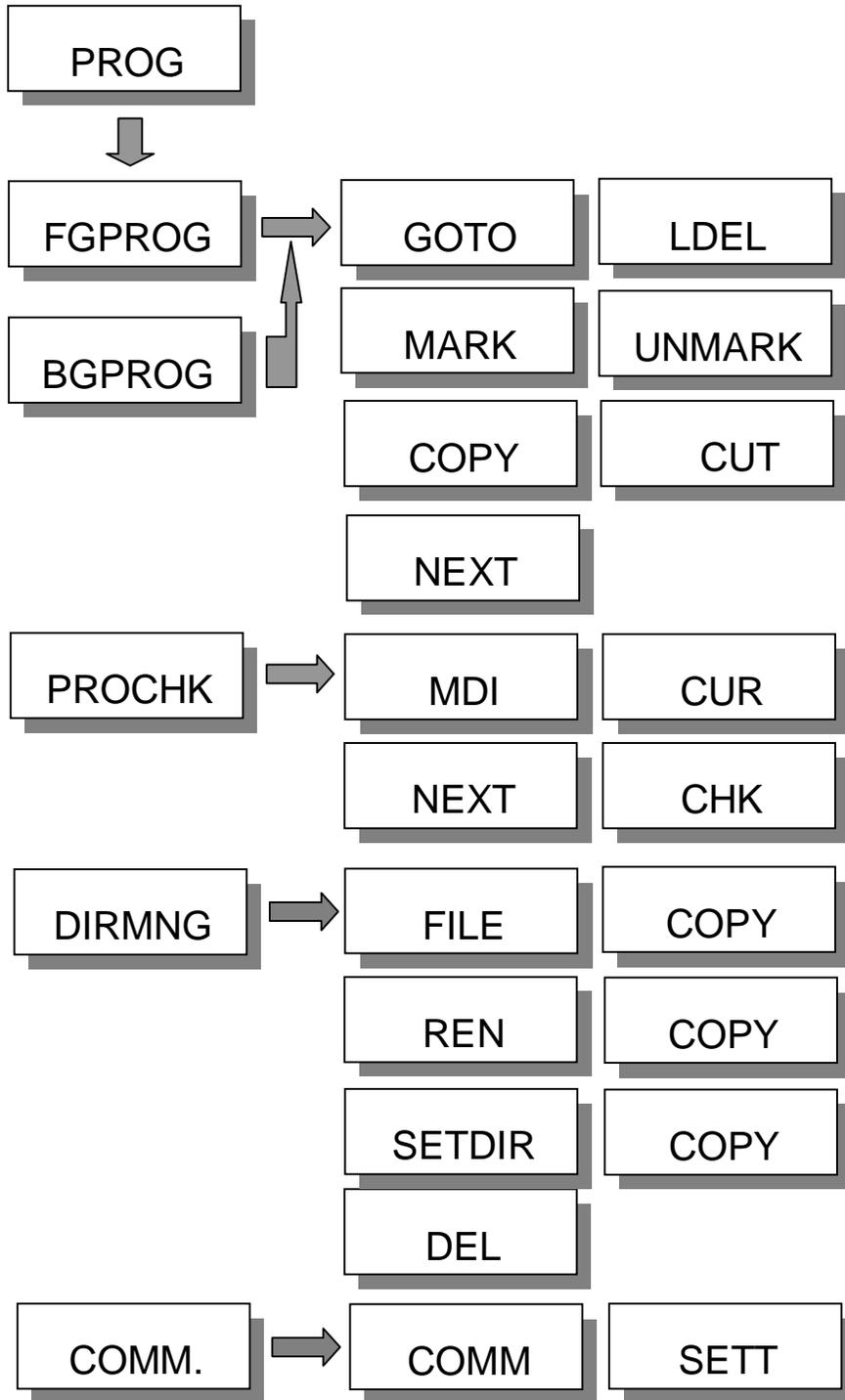


Figure 2.3-2 Program Editing Function Keys Figure

Offset Screen Function Keys

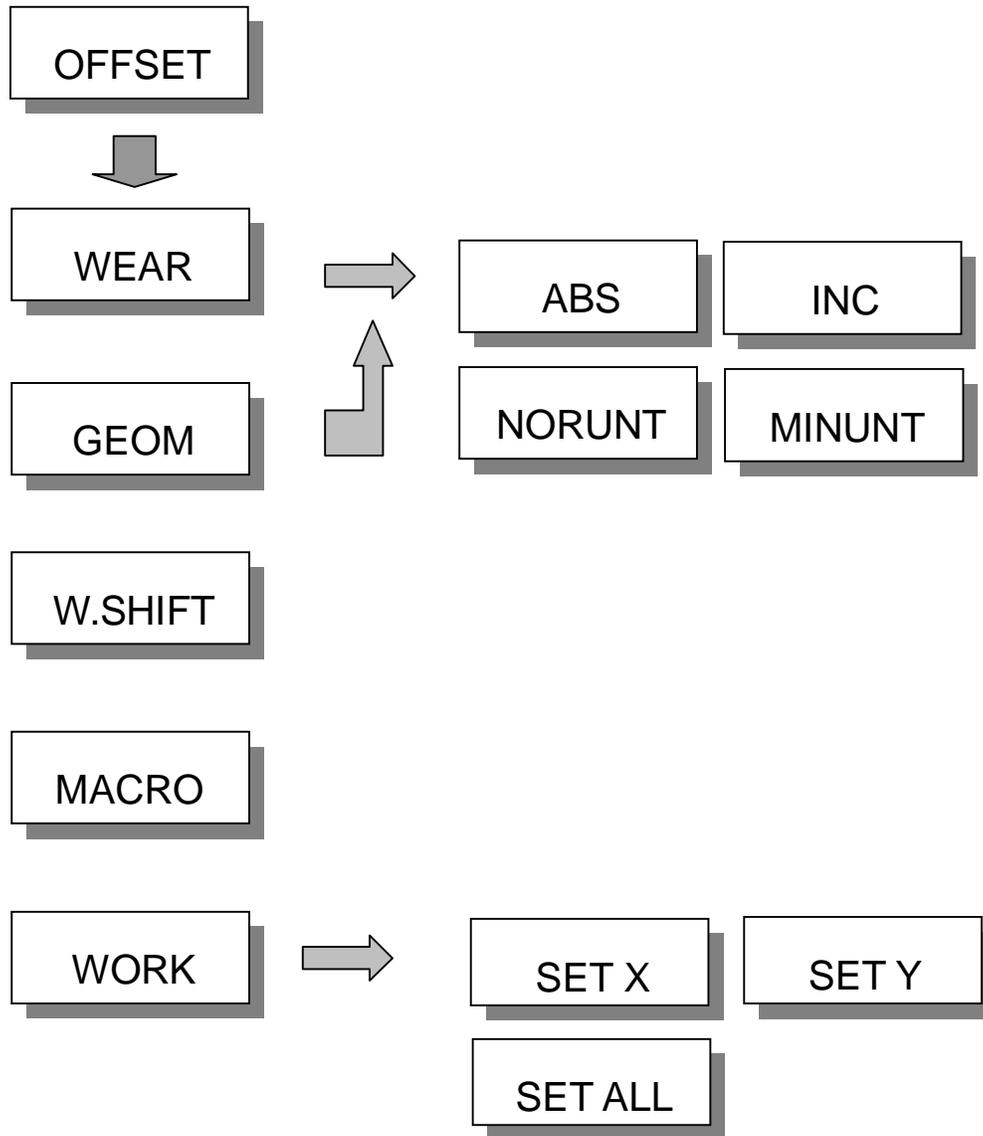


Figure 2.3-3 OFFSET Structure Figure

CAM Screen Function Keys

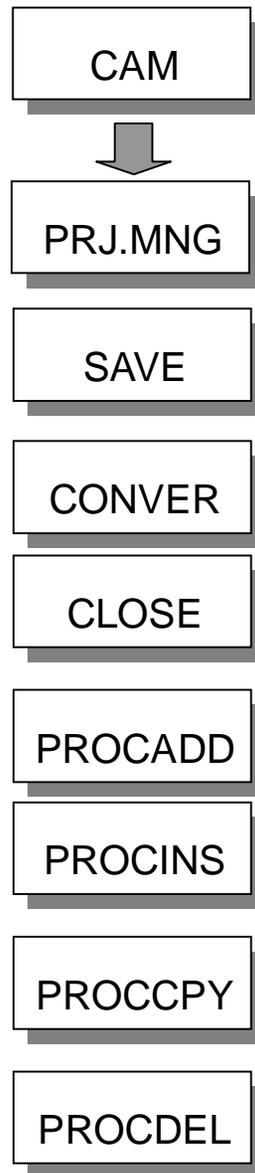


Figure 2.3-4 CAM Structure Figure

Preview Screen Function Keys

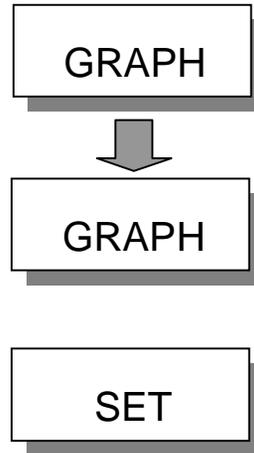


Figure 2.3-5 Preview Screen Structure Figure

DGNOS Screen Function Keys

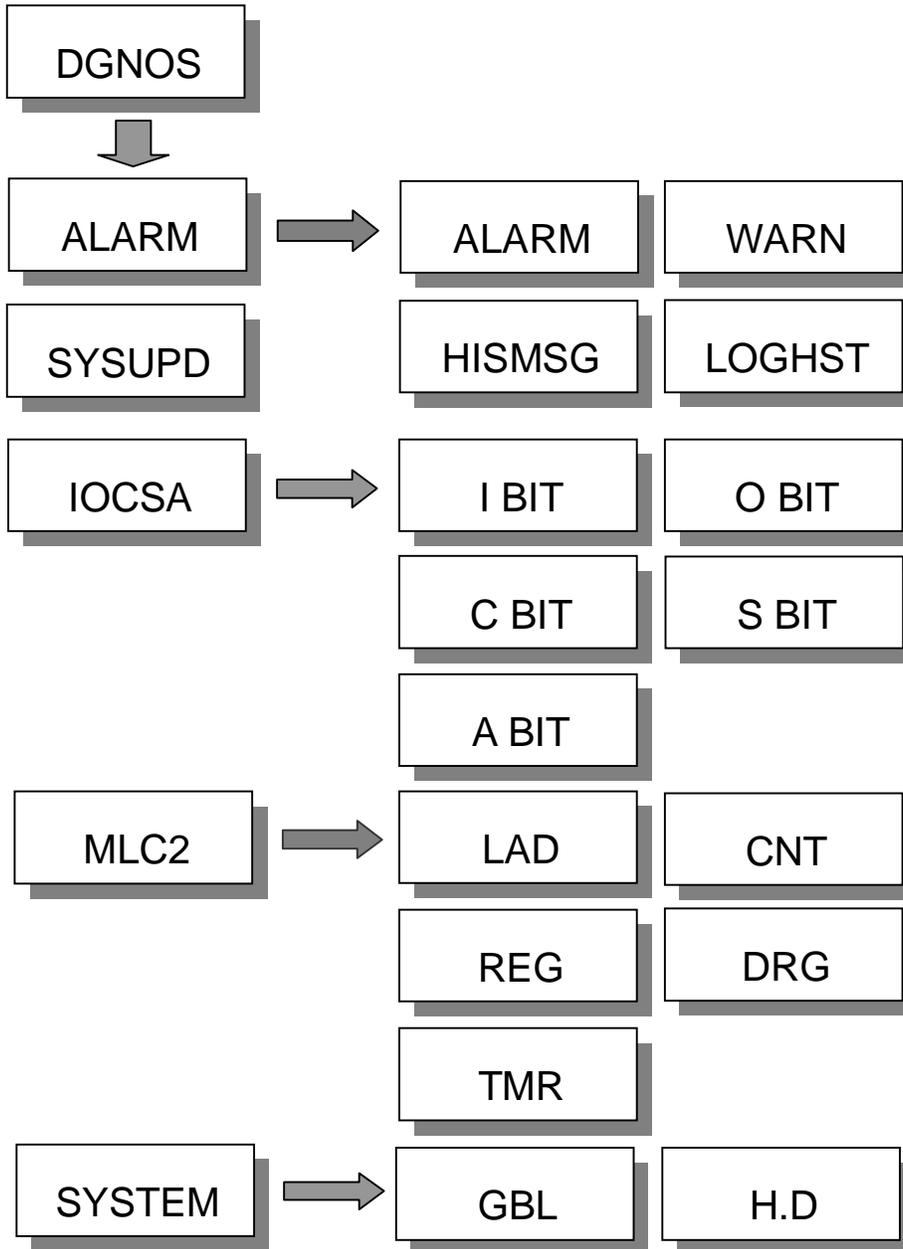


Figure 2.3-6 DGNOS Structure Figure

SOFTPL Function Keys

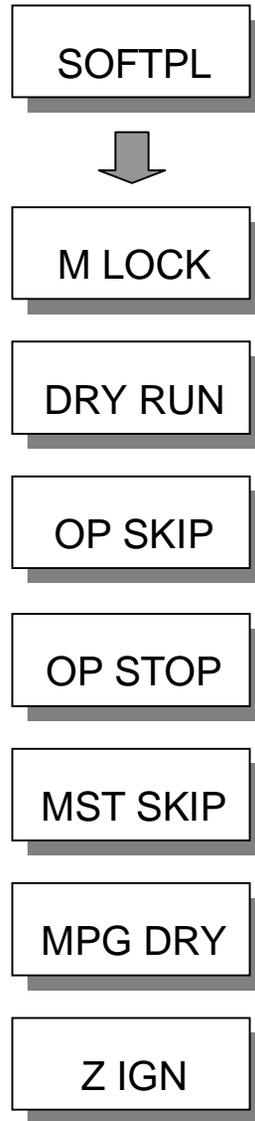


Figure 2.3-7 SOFTPL Structure Figure

PARAM Function Keys

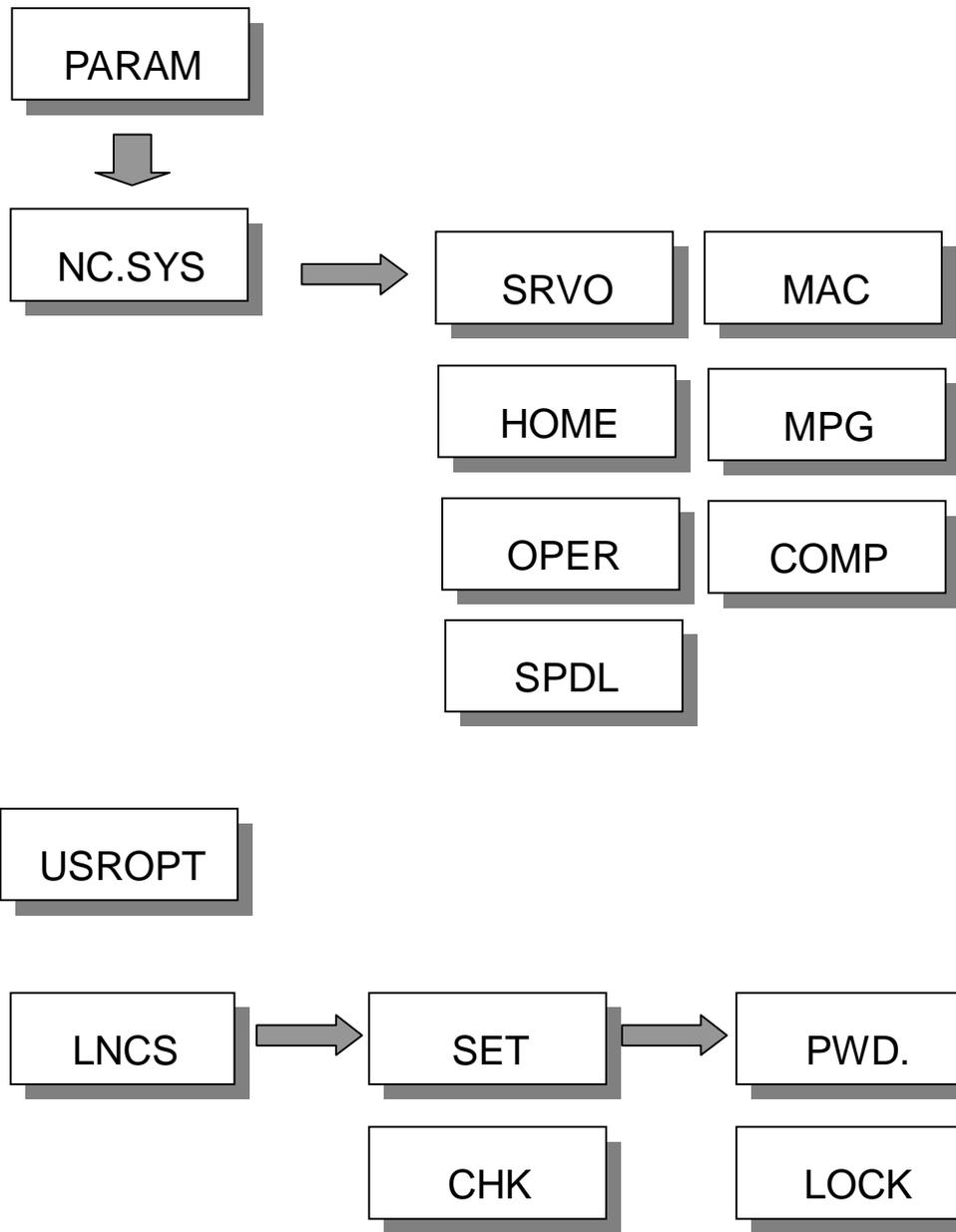


Figure 2.3-8 Parameter Structure Figure

2.3.3 ABSOLUTE COORDINATE SCREEN

Pressing **【ABS.】** to enter into absolute coordinate screen. Beside the absolute coordinate screen are sub-screens for **【REL】**, and **【MAC】**.

O0008		N000000		JOG	M-RDY
Absolute			RELATIVE		
X	-100.000	X	-100.000		
Y	-200.000	Y	-200.000		
Z	-300.000	Z	-300.000		
C	-400.000	C	-400.000		
			MACHINE		
			X	0.000	
			Y	0.000	
			Z	0.000	
			C	0.000	
FEED :		0 MM/MIN	FEED :		60%
SPDL :		0 RPM	RAPID :		50%
			SPDL :		120%
ABS.		REL.		MAC.	
HANDLE		MEA		QUIT	

Figure 2.3-9 **【ABS.】** Screen

Absolute coordinate is the program coordinate, which is [present value – tool offset = program value.] This will display the present executing position of each axis minus each axis offset.

2.3.4 RELATIVE COORDINATE SCREEN

Pressing **【REL.】** to enter into the relative coordinate screen. Beside the relative coordinate screen are sub-screens for **【ABS.】** and **【MAC.】**.

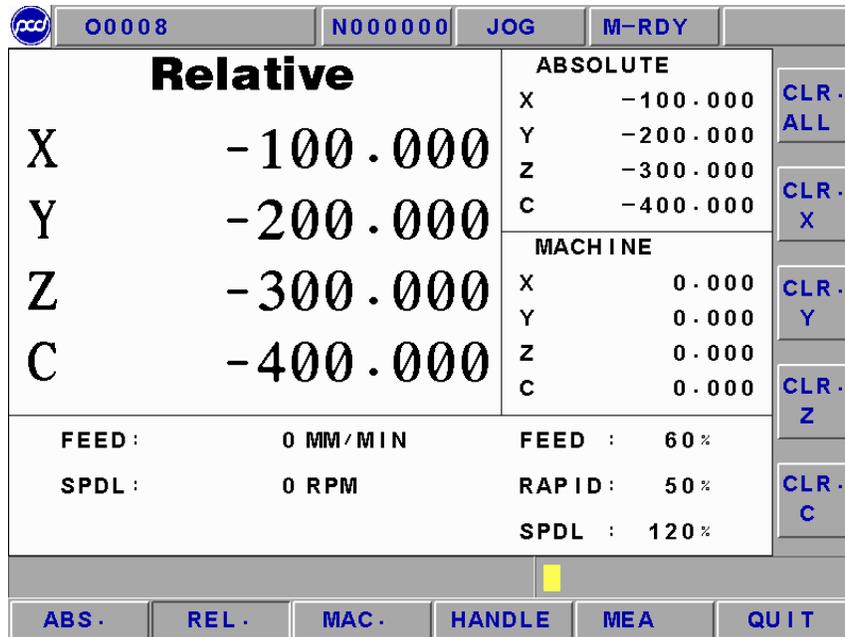


Figure 2.3-10 **【REL.】** Screen

The relative coordinate system means the distance between the present position and any point that is decided by users. So users can set the relative coordinate value to zero anytime, or enter the non-zero value directly.

If want to reset the coordinate value and to let X coordinate to 100.00, Y coordinate to 200.000 and Z coordinate to 300.000, only need to set X100, Y200, Z300. Then, pressing <INPUT> to reset the coordinate value immediately.

If want to set the 3 axes relative coordinate values to zero separately or at the same time, users only need to press the corresponding sub-function buttons **【CLR. X】**, **【CLR. Y】**, **【CLR. Z】**, **【CLR. ALL】** to execute the corresponding clean.

2.3.5 MACHINERY COORDINATE SCREEN

Pressing **【MAC.】** to enter into the relative coordinate screen. Beside the machine coordinate screen are sub-screens for **【ABS.】** and **【REL.】**.

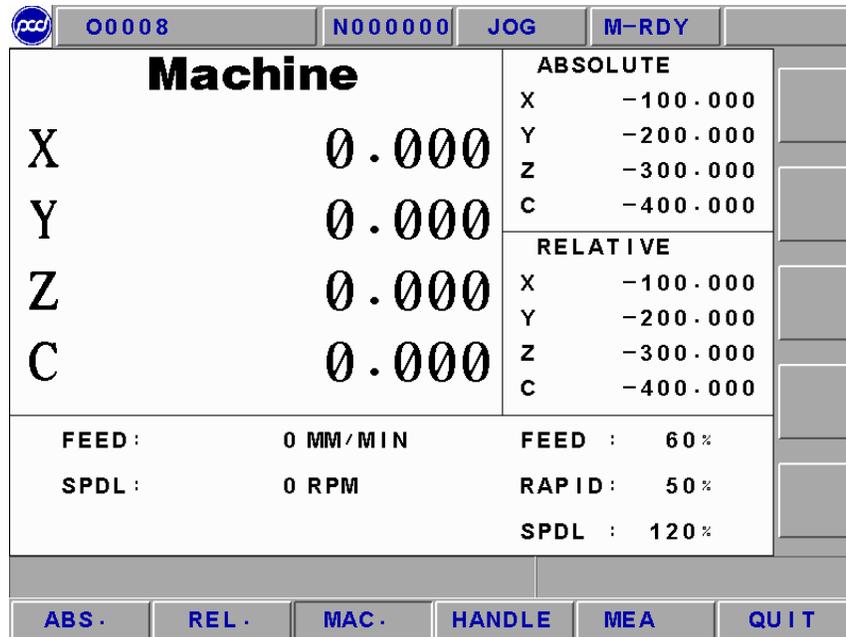


Figure 2.3-11 **【MAC.】** Screen

Machine coordinate is the distance of the present position corresponding to the reference point. Each machine has its own reference point. For safety concern, please looking for the reference point whenever reboot machine before executing work.

2.3.6 HANDLE SCREEN

Pressing **【HANDEL.】** to enter into the handle screen. Beside the handle screen are sub-screens for **【ABS.】** and **【REL.】**.

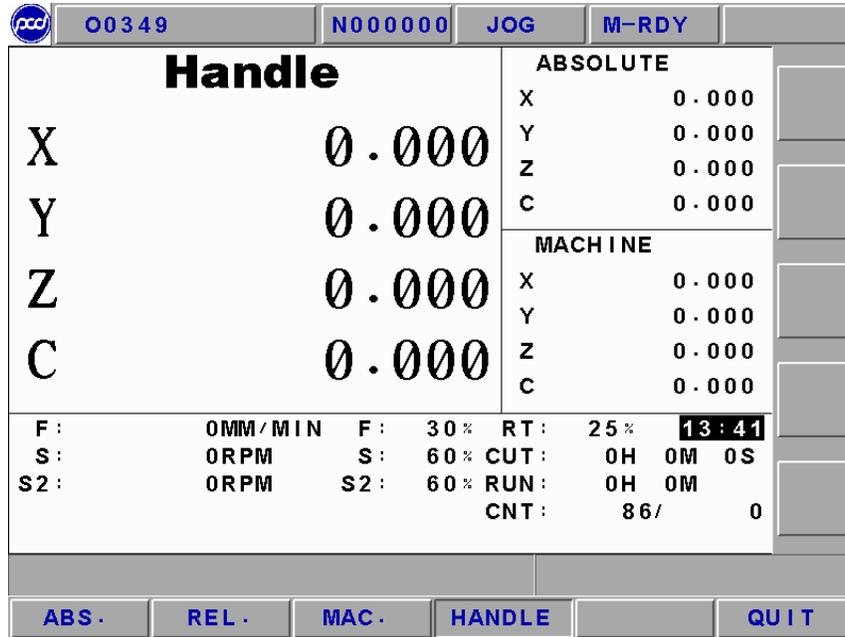


Figure 2.3-12 **【HANDLE.】** Screen

Under the AUTO mode, the user can increase or reducing the movement amount of tool with the handwheel, and then change the tool path.

Absolute Coordinate	Will not be influenced by Handle Interrupt
Relative Coordinate	Will not be influenced by Handle Interrupt
Machine Coordinate	Will be influenced by Handle Interrupt. Show the real position of the machine.

Because the Absolute Coordinate is not changed by Handle Interrupt, the actual tool path have a offset with the original tool path when using this function. This mount of offset must be reset to zero by Manual Zero Return.

About this function, reference to the Maintenance Manual, Handle Interrupt Single(C Bit 066~071) and the MPG input multiple(REGISTER 014).

2.3.7 EXIT

Pressing **【EXIT】**function button to enter into code entry windows and key-in correct codes.
 Pressing **【OK】** key to exit CNC system and returning back to DOS system. Pressing
【CANCEL】 key to return back to program and to continue executing.

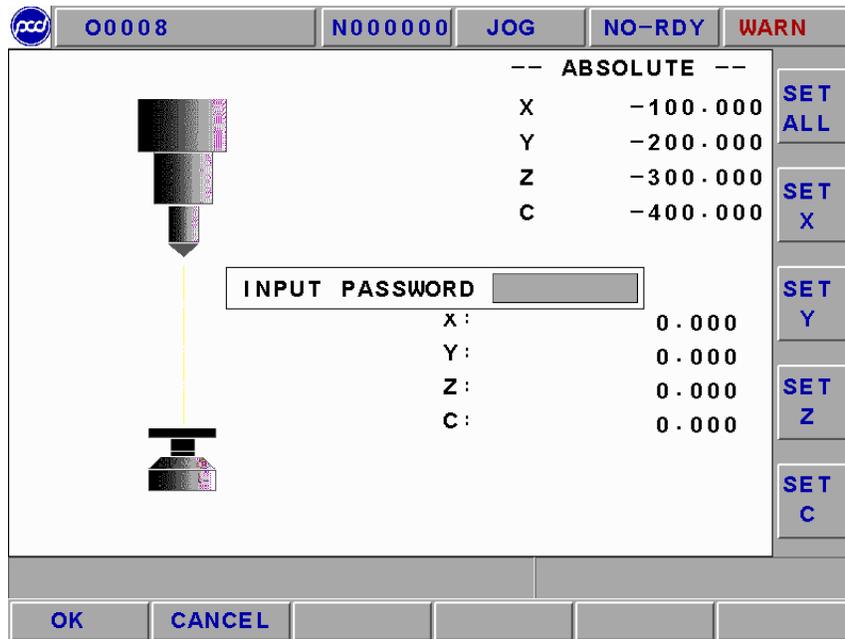


Figure 2.3-13 EXIT

2.4 PROG

Pressing <PROG> to enter into program function group screen. This function group provides part program editing, checking, file managing, and other related functions.

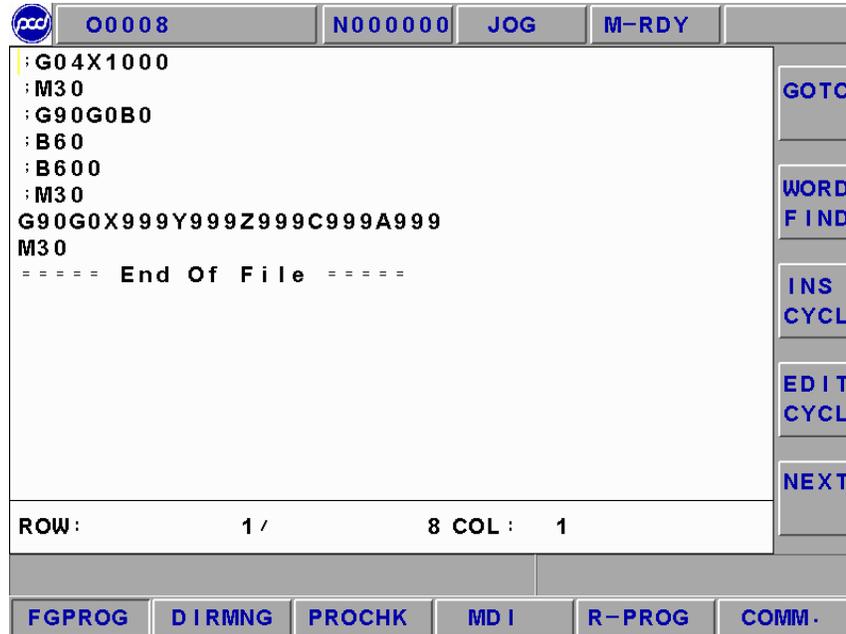


Figure 2.4-1 PROG Screen

2.4.1 FGPROG

Under editing condition, pressing **【FGPROG】** to display the program content of the current opened controller. At this time, using the sub-function button at the right side and the entering rows at the bottom of the screen to do the program editing.

【GO TO】 (Row Orientation)

Pressing **【GO TO】** function button at the right side to key-in “ROW” number of the program in the dialog box. Pressing **【INPUT】** to move the cursor to the assigned row orientation. Please refer to the below figure.

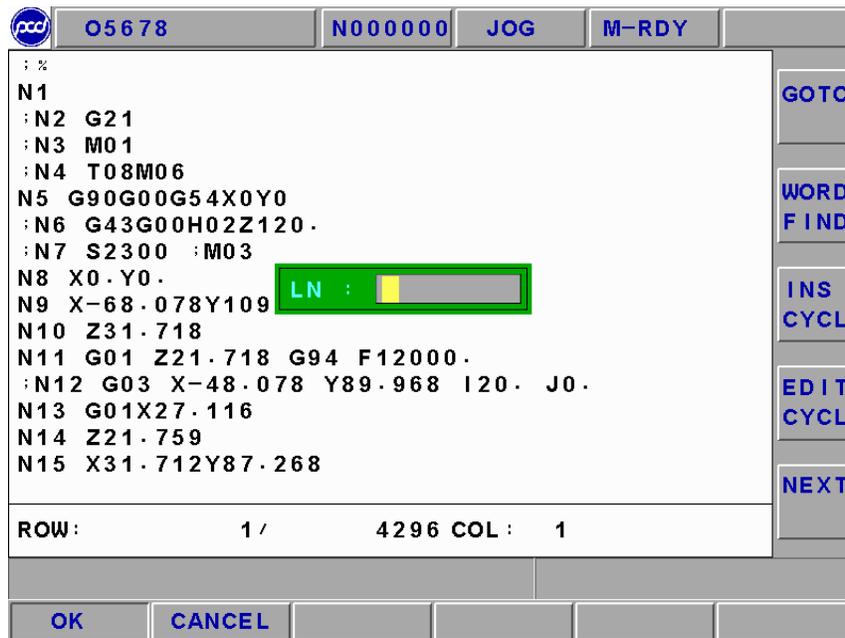


Figure 2.4-2 Row Orientation Page

【LDEL】 (Row Delete)

Able to delete a row of program codes at the cursor position.

【MARK】

Pressing 【MARK】 at the right side of the screen to move the cursor to the wanted marking row's starting/ending point. Moving the cursor to the starting/ending position of the wanted marking row and pressing the 【MARK】 function key again to mark the designated marking rows. Please refer to the below figure.

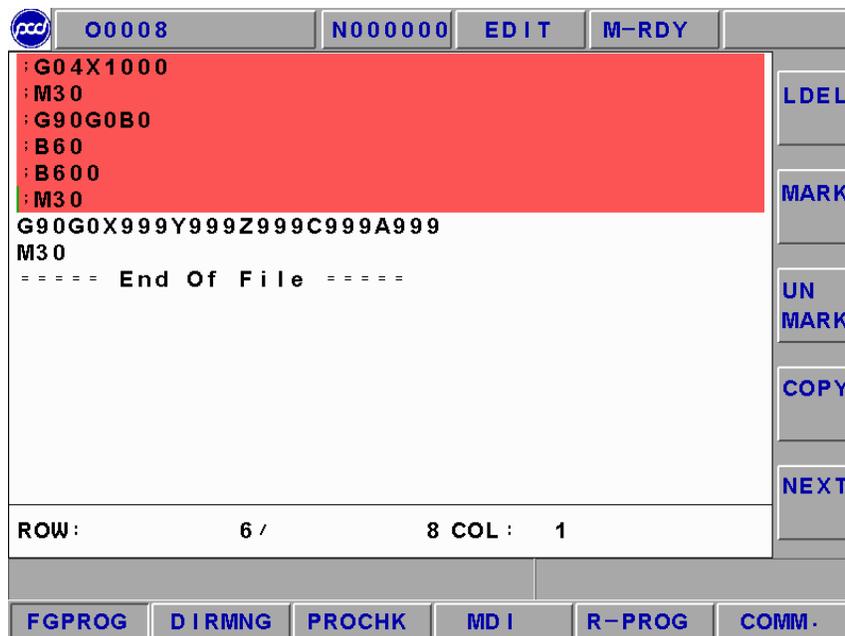


Figure 2.4-3 Marking Function Page

【UNMARK】

To cancel previous marking sign.

【COPY】

To copy the marked program in local range.

【CUT】

To cut down the marked program in local range.

【BIND】

To paste on the program codes that are copied or cut at previous time.

Note: Not able to executing program enabling under editing mode.

2.4.2 BGPROG (Background Program)

Pressing **【BGPROG】** to enter into background editing mode under auto mode. Background editing allows users to edit another part program while executing one part program in auto mode. Editing method and the environment of doing present program in editing mode are totally the same.

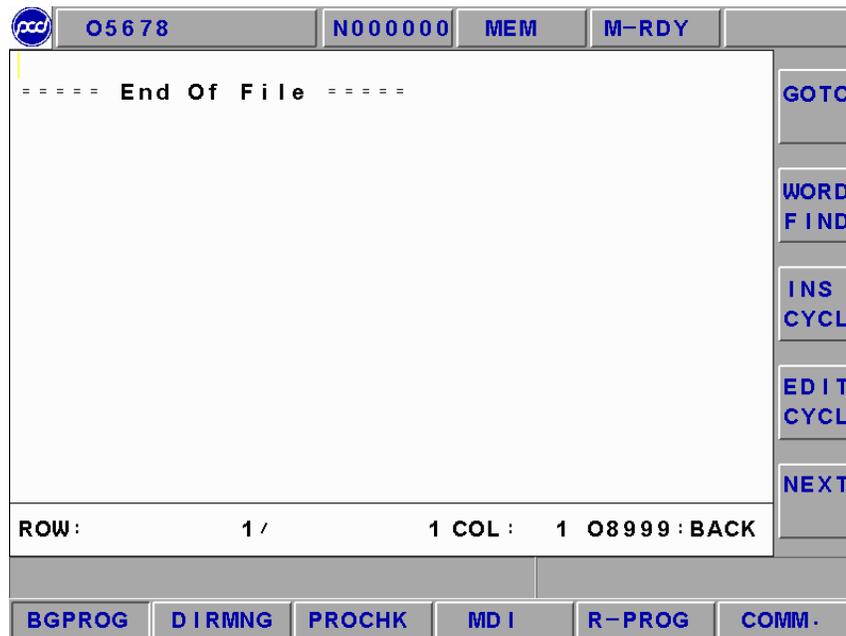


Figure 2.4-4 Background Editing Mode

2.4.3 PROCHK (Program Checking)

Under auto mode, pressing **【PROCHK】** function buttons to enter into the automatic program checking screen as below figure. Sub-function buttons, at right side of the screen, provide checking options.

- 【 C U R 】** To display data information of the present executing single block.
- 【 N E X T 】** To display data information of the next single block
- 【 C H K 】** The upper half of the main screen will display program content and the present executing program will be highlighted. The bottom half has coordinate values, M/S/G/T codes present value, actual speed display and etc.

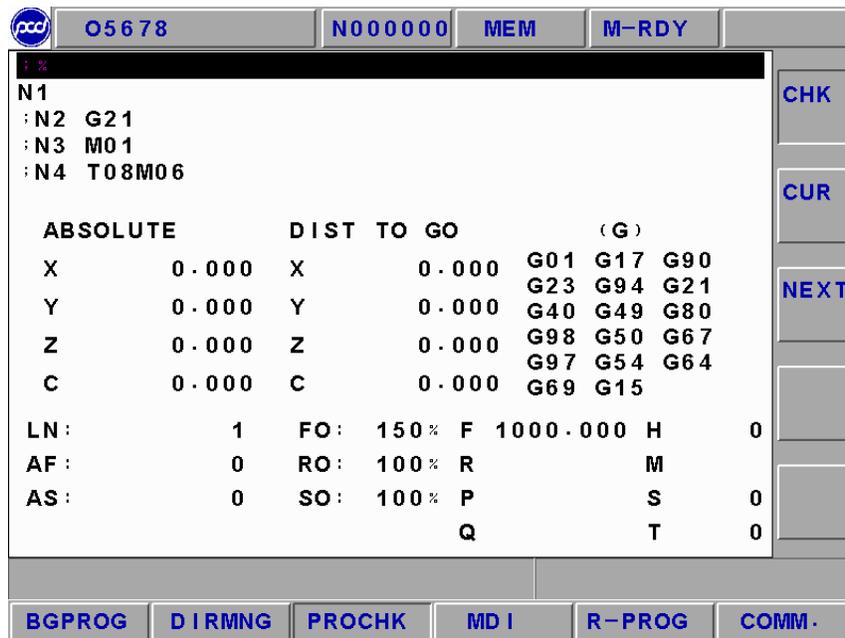


Figure 2.4-5 **【PROCHK】** Screen

2.4.4 MDI

Pressing 【MDI】 key to entry into MDI page under MDI mode. User is to key-in a single command block and then press <INPUT>. Then, the command will occur at the left section of the main page. Then, pressing Cycle Start key and the command will be executed and related data will be displayed. Using this method to do test running is more safe and more time-efficiency.

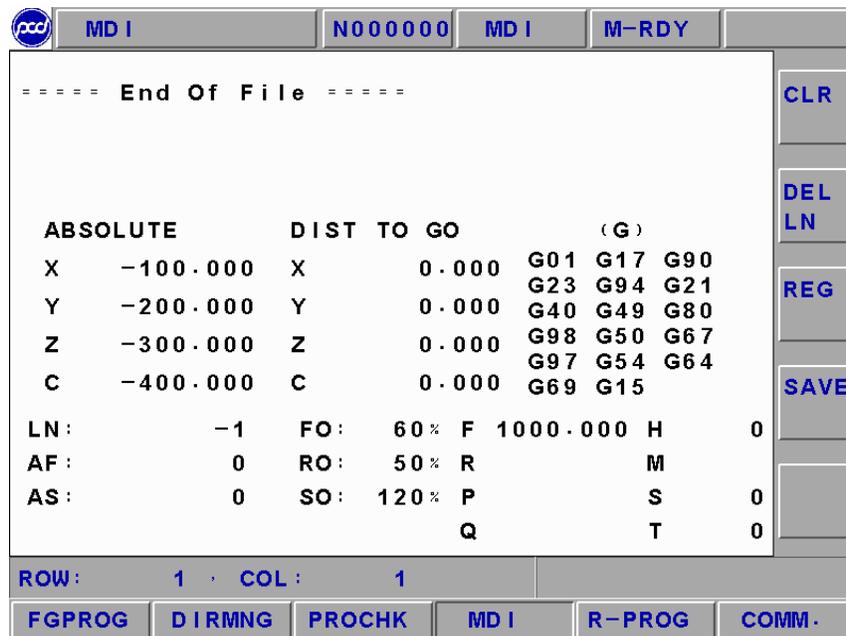


Figure 2.4-6 MDI Page

2.4.5 DIRMNG (File Management)

In file management screen, system provides the related file opening, copying, deleting, renaming, and setting menu functions. A detailed description is as following.

【FILE】

1. Able to use direction key to choose the wanted open file after entering into the file management screen. Also, users are able to open a file after pressing <INPUT>.
2. After pressing open file selection button, a dialog box will occur on the screen. Please choose or key-in the wanted open file name and then pressing <INPUT> to open the file.
3. The opened file is a background program under auto mode. Automatically setting the opened file to the present program under editing mode means NOT able to open the file under other modes.

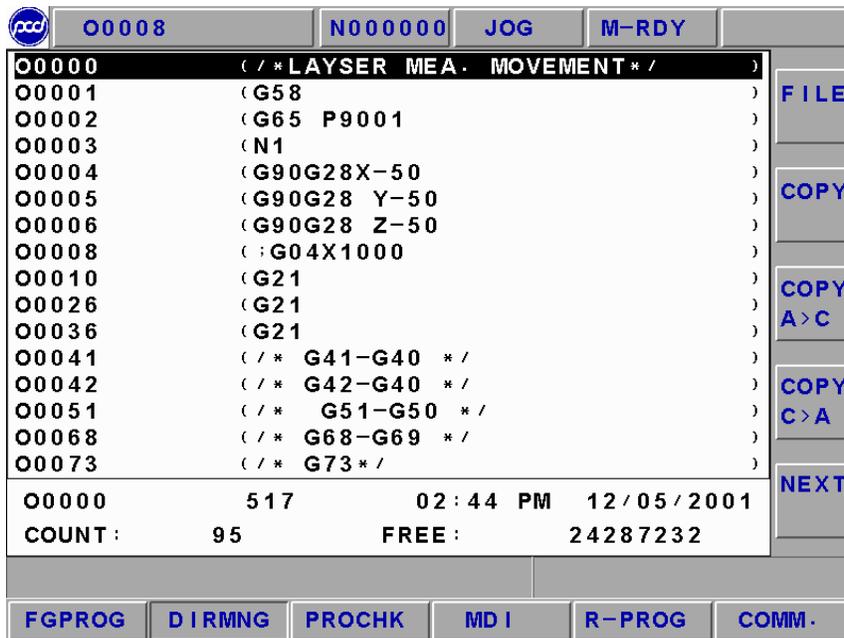


Figure 2.4-7 DIRMNG Screen



Figure 2.4-8 Screen of Pressing **[FILE]** Button

A dialog box of file copying will occur on the screen by pressing copy button. Users are able to choose or enter the file's path or file name as the source file when copy files.

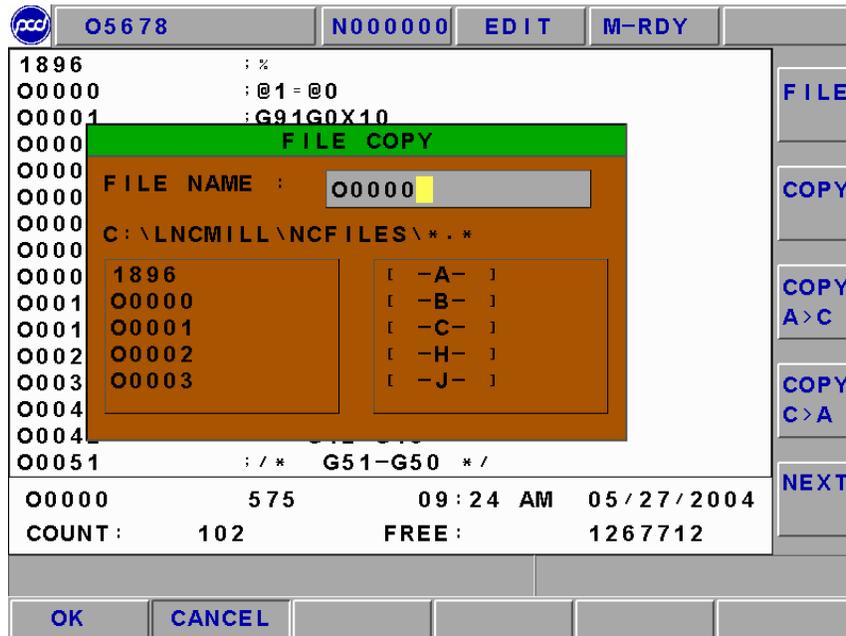


Figure 2.4-9 Screen of Pressing **[COPY]** Function Button

Able to enter the copied destination file name in the destination file column. If only enter file name, then the file path is the system default value.

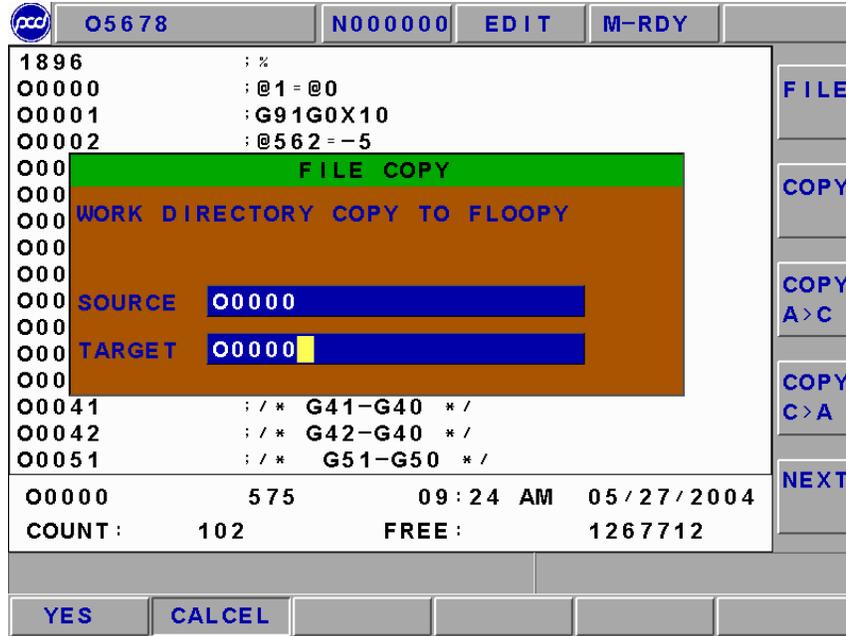


Figure 2.4-10 Screen of Key-In Destination File Name

If the file is already existed, then the reminding windows will occur. Please confirm if want to execute overwriting action.

1. **RUNTIME** : Pressing this button to clear the working time at clear coordinate display page and to return time back to zero.
2. **PARTCONT** : pressing this button to clear working piece number at clear coordinate display page.
3. **INTCNT**: a code entry dialog box appears by pressing this button. Key-in the correct passing codes to set the working piece number at coordinate display page.
4. **MAXCNT** : setting the maximum working piece number. The system will send S134 to inform MLC to be active when the working piece number reaches the maximum. On the other hand, S134 will be disabled when the working piece number is not equal to the maximum working piece setting number.

Exit System Screen

Pressing **[EXIT]** function button to enter into code entry windows and key-in correct codes.
Pressing **[OK]** button to exit CNC system and returning back to DOS system. Pressing **[CANCEL]** button to return back to program and to continue executing.

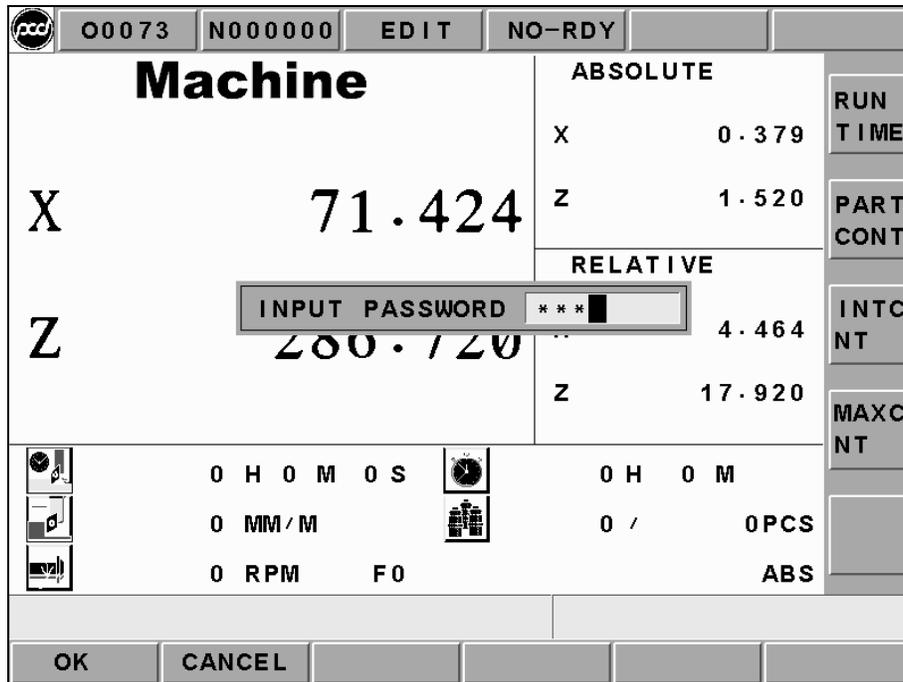


Figure 2.4-12 Exit System Screen

2.5 PROG Function

Pressing **<PROG>** to enter into program function group screen. This function group provides part program editing, checking, file managing, and RS232 transmitting and other related functions.

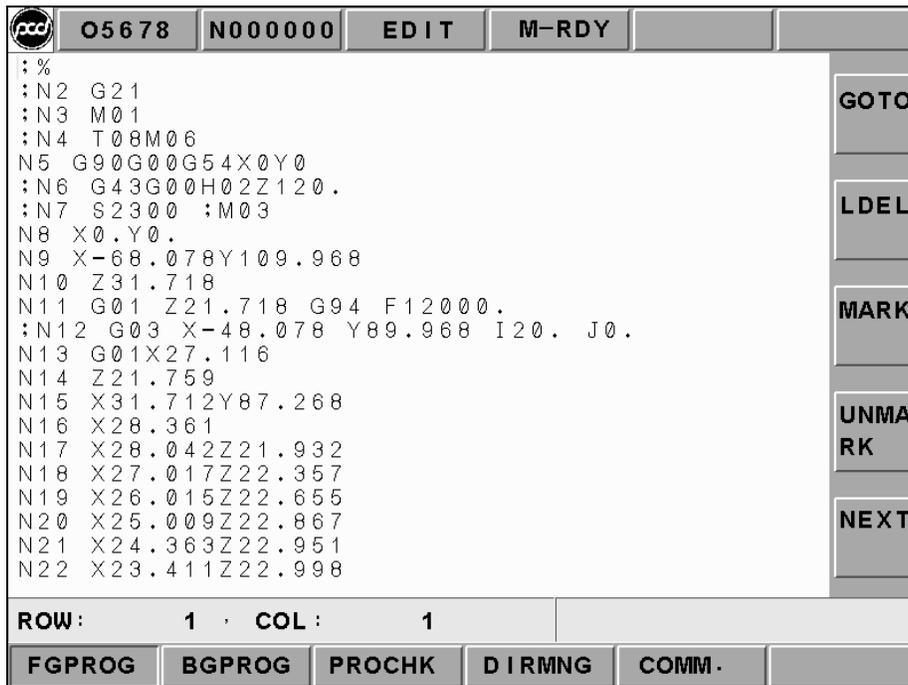


Figure 2.5-1 FGPROG Screen

2.5.1 FGPROG (Foregoing Program)

Under editing condition, pressing **【FGPROG】** to display the program content of the current opened controller. At this time, using the sub-function button at the right side and the entering rows at the bottom of the screen to do the program editing.

1. **GOTO**: pressing **【GOTO】** function button at the right side to key-in "ROW" number of the program in the dialog box. Pressing **【INPUT】** to move the cursor to the assigned row orientation (like Figure 2.5-2).
2. **LDEL (Row Delete)** : deleting a row of program codes at the cursor position.
3. **MARK**: pressing **【MARK】** at the right side of the screen to move the cursor to the wanted marking row's starting/ending point. Moving the cursor to the starting/ending position of the wanted marking row and pressing the **【MARK】** function key again to mark the designated marking rows (like Figure 2.5-3).
4. **UNMARK** : unmarking previous marking
5. **COPY** : copying the marked program in local range.
6. **CUT** : cutting down the marked program in local range.
7. **PASTE**: pasting on the program codes that are copied or cut at previous time.

Note: Not able to execute program enabling in editing mode.

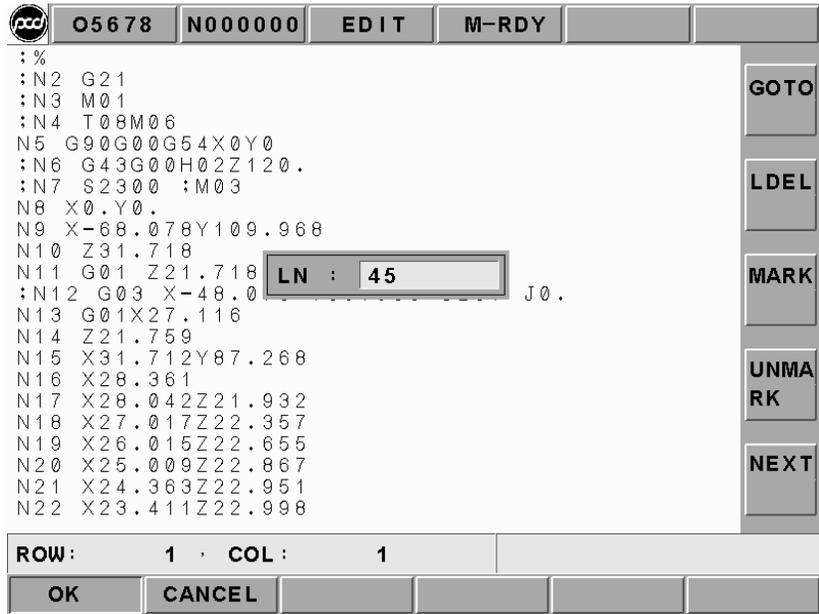


Figure 2.5-2 Screen of Row Position Entry Column

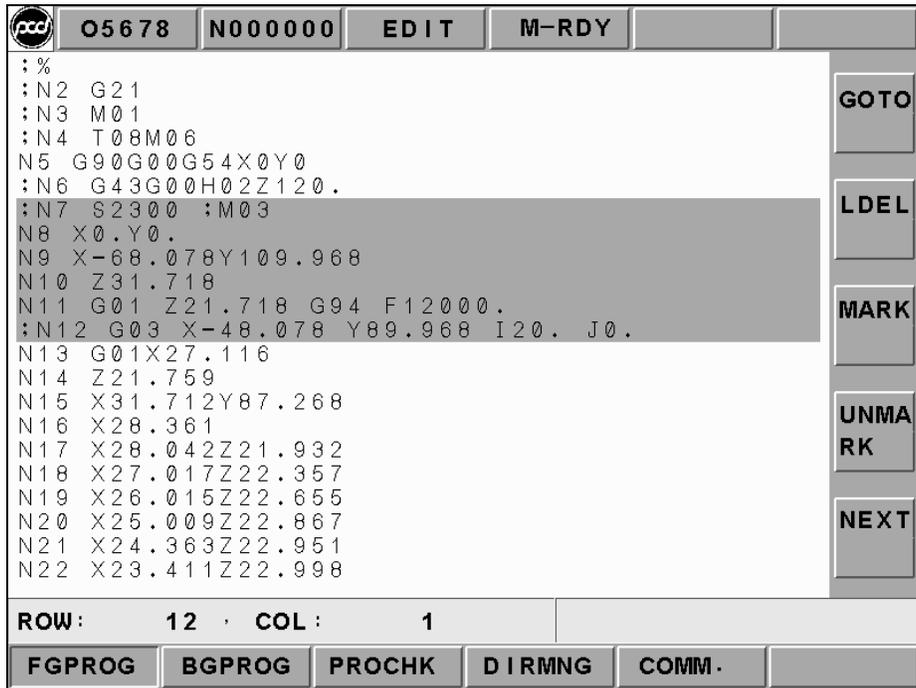


Figure 2.5-3 Screen of Row Position Entry Column

2.5.2 BGPROG (Background Program)

Pressing **【BGPROG】** to enter into background editing mode under auto mode. Background editing allows users to edit another part program while executing one part program in auto mode. Editing method and the environment of doing present program in editing mode are totally the same.



Figure 2.5-4 BGPROG Editing Mode

2.5.3 PROCHK (Program Checking)

Under auto mode, pressing **【PROCHK】** function buttons to enter into the auto program checking screen as below figure. Sub-function buttons, at right side of the screen, provide checking options.

【PROCHK】

To display data information of the present executing single block.

【NEXT】

To display data information of the next single block

【CHK】

The upper half of the main screen will display program content and the currently executing program will be highlighted. The bottom half has coordinate values, M/S/G/T codes present value, actual speed display and etc.

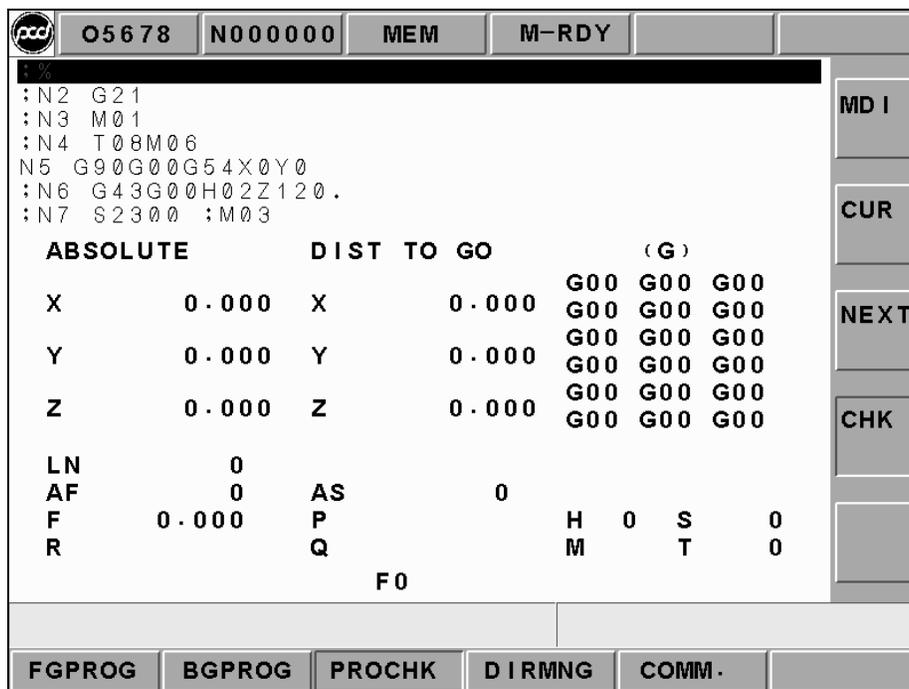


Figure 2.5-5 **【PROCHK】** Screen

Pressing **【PROCHK】** function button under MDI mode to enter into the MDI screen as the below figure. In this screen, users are able to execute a single block program. The operation method is as the following steps:

1. To key-in a single motion command,
2. To press <INPUT> button and this command will occur at left half side of the main screen,
3. To press (Cycle Start) button, and then the system will execute this command immediately and the related information will occur.

It is more safe and time-efficiency when using this method to do testing.

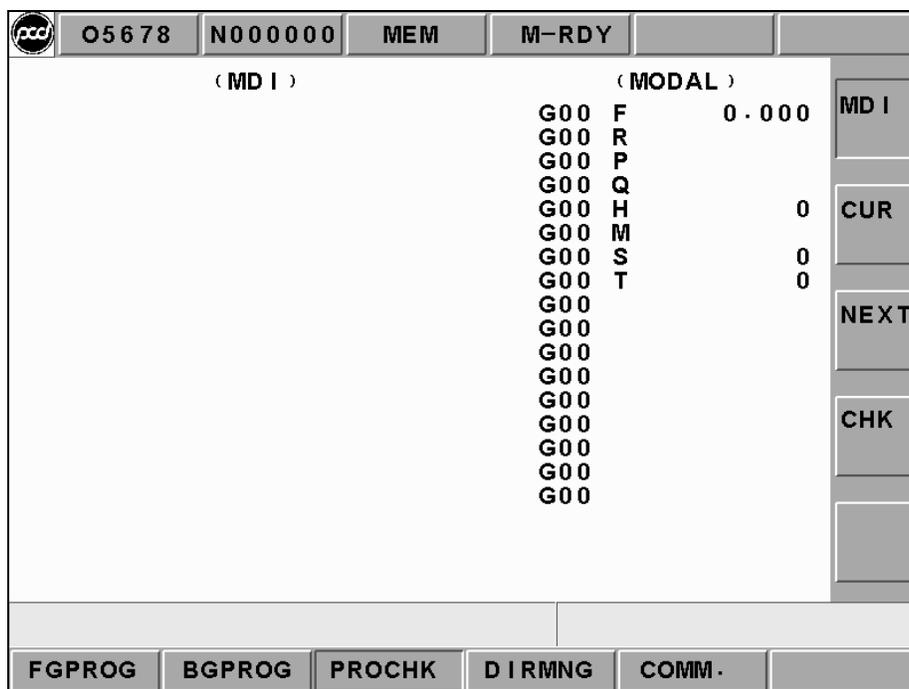


Figure 2.5-6 **【PROCHK】** Screen

2.5.4 DIR MNG (File Management)

In file management screen, system provides the related file opening, copying, deleting, renaming, and setting menu functions. A detailed description is as following.

【FILE】

1. Able to use direction key to choose the wanted open file after entering into the file management screen. Also, users are able to open a file after pressing <INPUT>.
2. After pressing open file selection button, a dialog box will occur on the screen. Please choose or key-in the wanted open file name and then pressing <INPUT> to open the file.
3. The opened file is a background program under auto mode. Automatically setting the opened file to the present program under editing mode means NOT able to open the file under other modes.

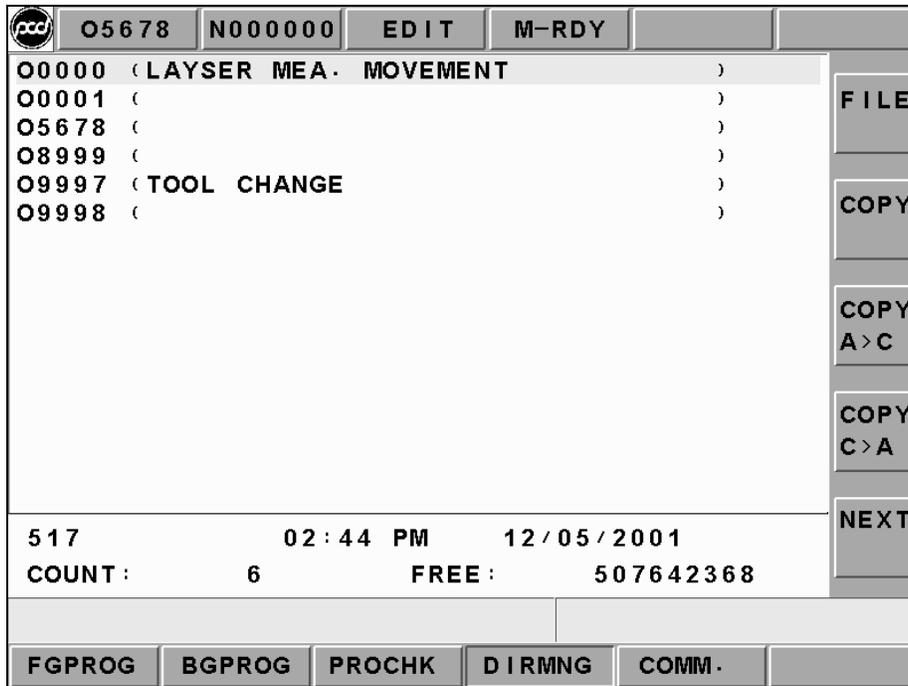


Figure 2.5-7 DIR MNG Screen

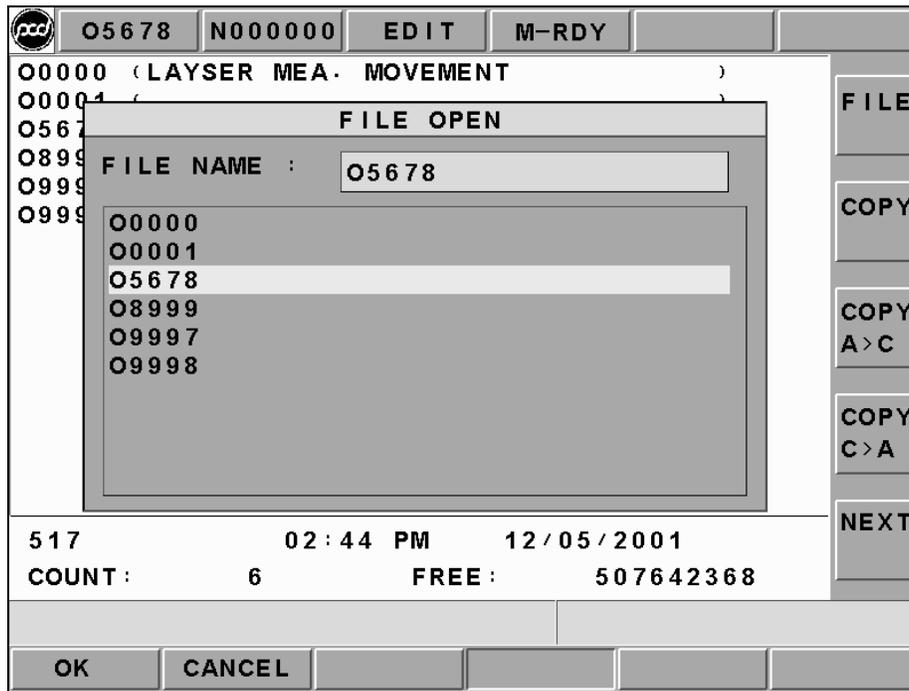


Figure 2.5-8 Screen of Pressing **【FILE】** Button

【COPY】

1. A dialog box of file copying will occur on the screen by pressing copy button. Users are able to choose or key-in the file's path or file name as the source file when copy files. The screen after pressing copy function button is as Figure 2.5-9. After completed entering and pressing confirm button, the screen of entering the destination file is as figure 2.5-10.
2. Able to key-in the copied destination file name in the destination file column. If only key-in file name, then the file path is the system default value.
3. If the file is already existed, then the reminding windows will occur. Please confirm if want to execute overwriting action. The screen is as Figure2.5-11.
4. If the program is copying, the message-reminding dialog box will occur "Copying....". When complete, the message-reminding dialog box will occur "Copy Complete".

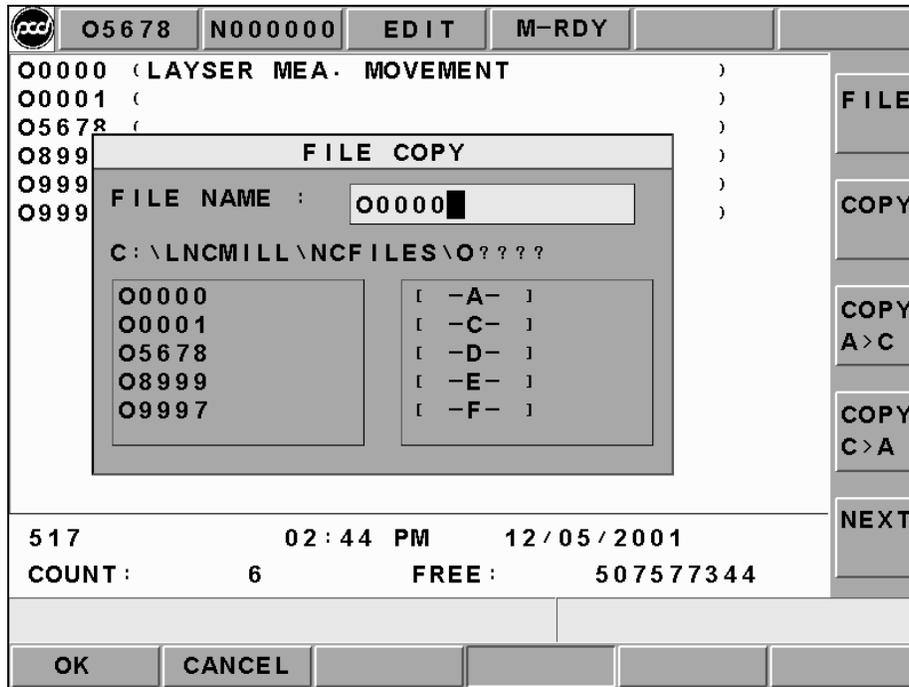


Figure 2.5-9 Screen of Pressing 【COPY】 Function Button

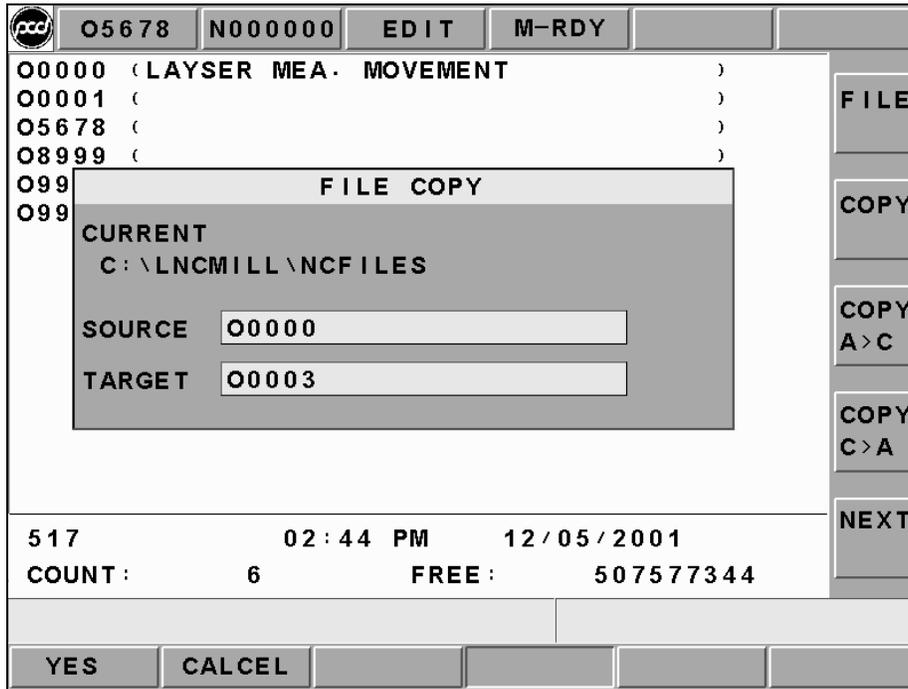


Figure 2.5-10 Screen of Key-In Destination File Name

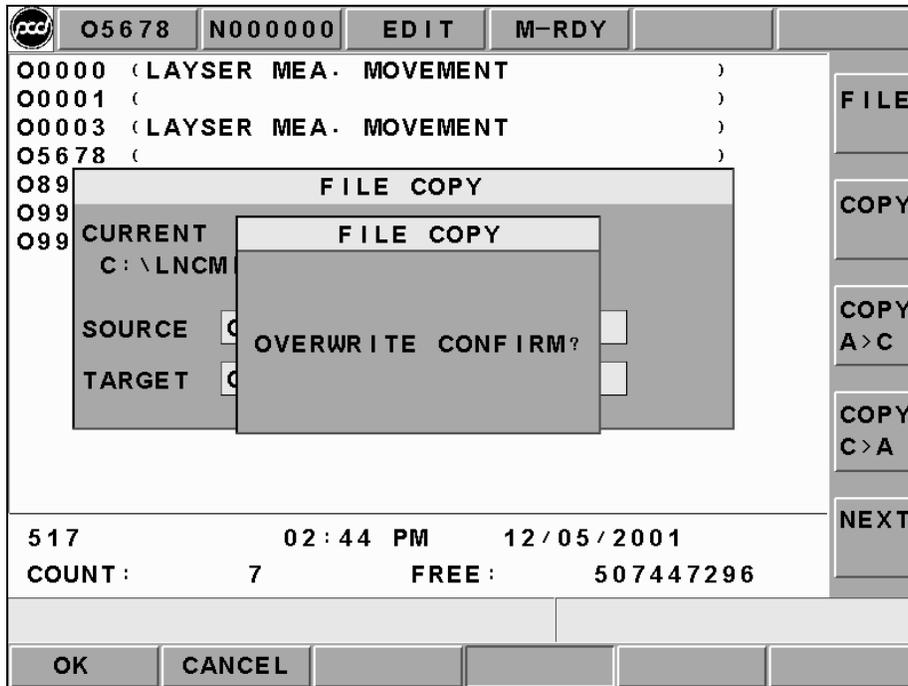


Figure 2.5-11 Screen When the Copied File is Already Existed

【COPY A>C】

1. Pressing (COPY A>C) in Edit mode and a file selection menu like Figure 2.5-12 will appear from the present working path. This will let users to select the wanted copy part program files from A drive.
2. Selecting a wanted copy file and then key-in the source file name and the destination file name from a popup key-in dialog box. See Figure 2.5-13. The destination file name is default as the source file name. If users do NOT want to use the default file name, then only need to key-in the wanted file name directly and then pressing confirm button to copy the source file form A drive to the current working menu.
3. If there is the same file name in the destination drive, the system will ask users to overwrite the already existed file. Pressing confirm button to overwrite the file, like Figure 2.5-14.
4. If the program is executing copy correctly, the message-reminding box will appear “Copy...”. After copy is completed, the message-reminding box will appear “Copy Complete”.

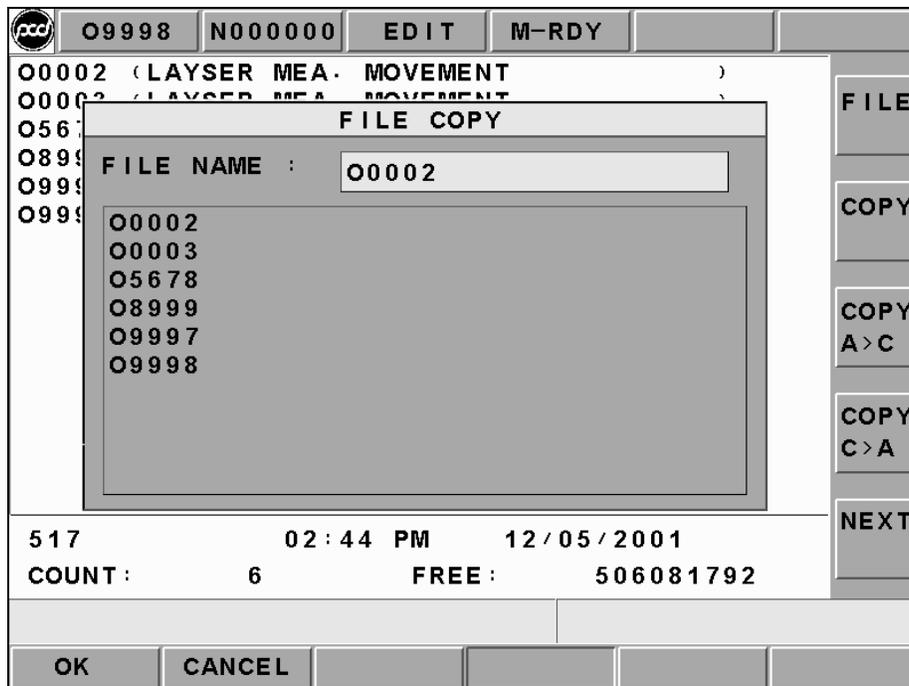


Figure 2.5-12 File Copy Selection Menu

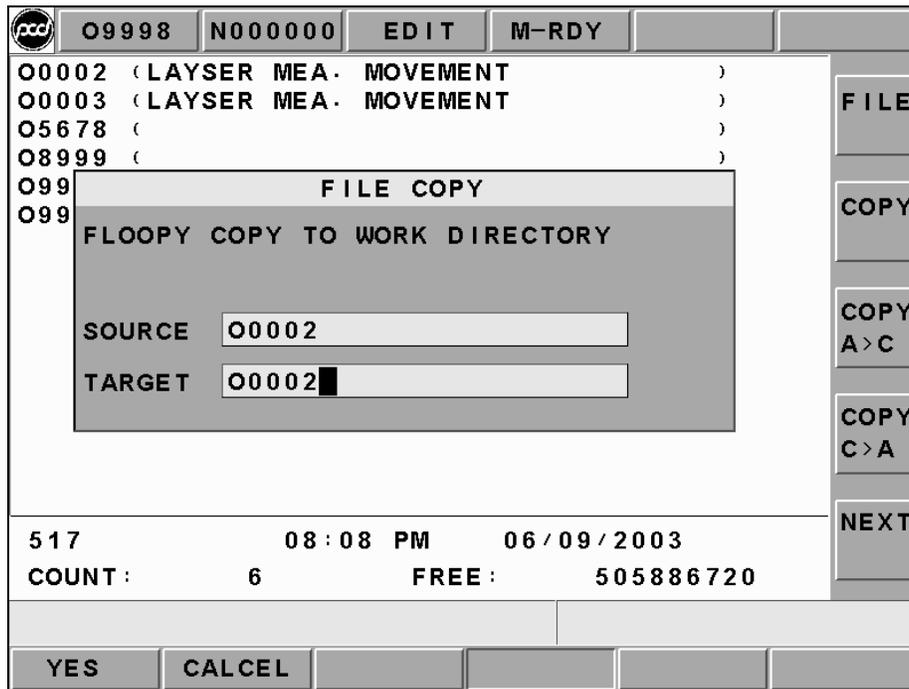


Figure 2.5-13 Target File Name Key-In Screen

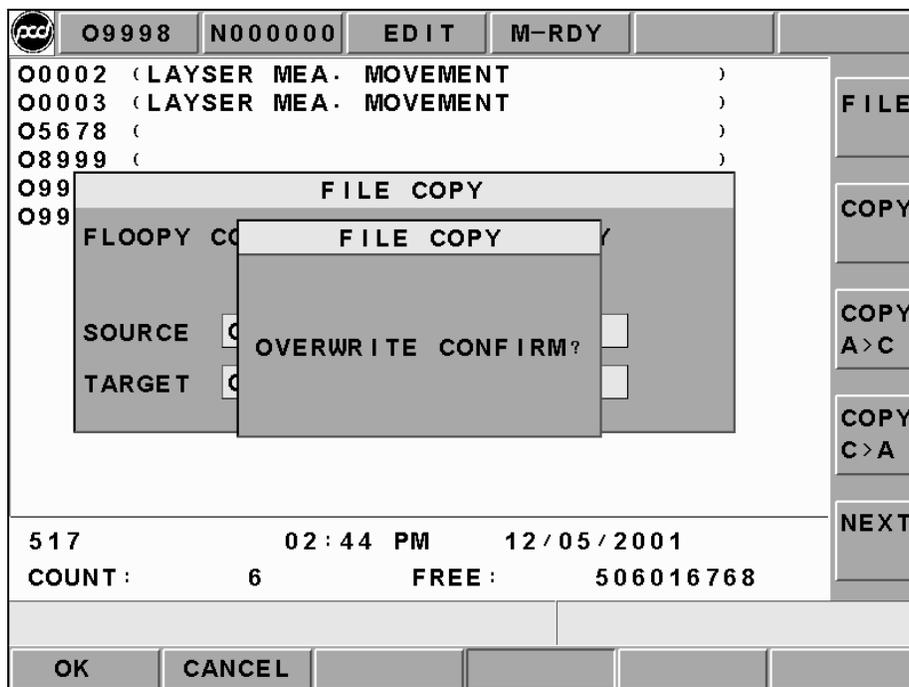


Figure 2.5-14 File Overwrite Confirmation Screen

【COPY C>A】

1. Pressing (COPY C>A) in Edit mode and a file selection menu like Figure 2.5-15 will appear from the present working path. This will let users to select the wanted copy present working program to A drive.
2. Selecting a wanted copy file and then key-in the source file name and the destination file name from a popup key-in dialog box. See Figure 2.5-16. The destination file name is defaulted as the source file name. If users do not want to use the default file name, then only need to key-in the wanted file name directly and then pressing confirm button to copy the source file form A drive to the current working menu.
3. If there is the same file name in the destination drive, the system will ask users to overwrite the already existed file. Pressing confirm button to overwrite the file, like Figure 2.5-17.
4. If the program is executing copy correctly, the message hint box will appear “Copy...”. After copy is completed, the message hint box will appear “Copy Complete”.

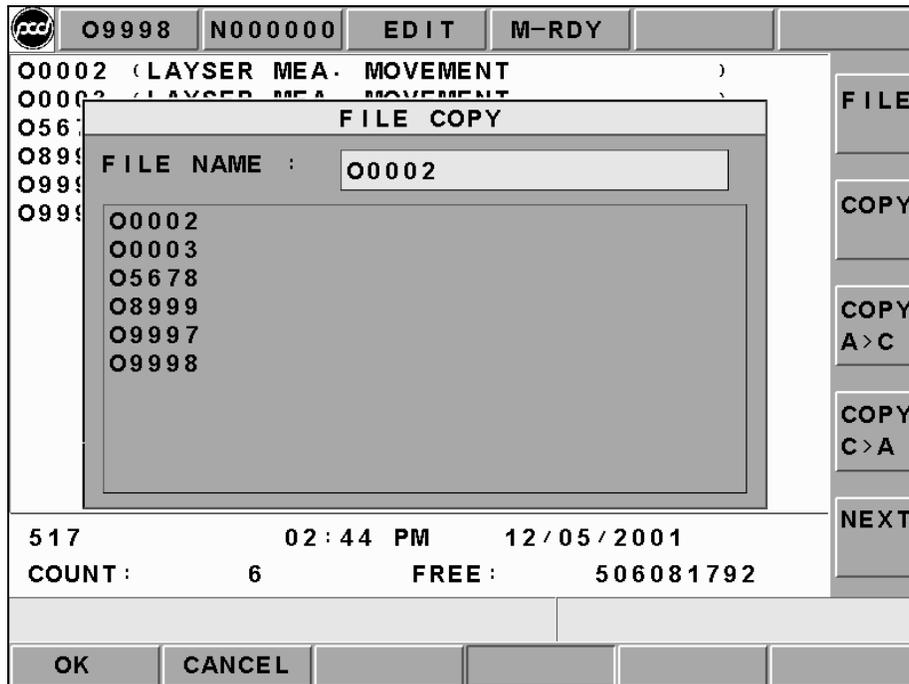


Figure 2.5-15 File Copy Selection Menu

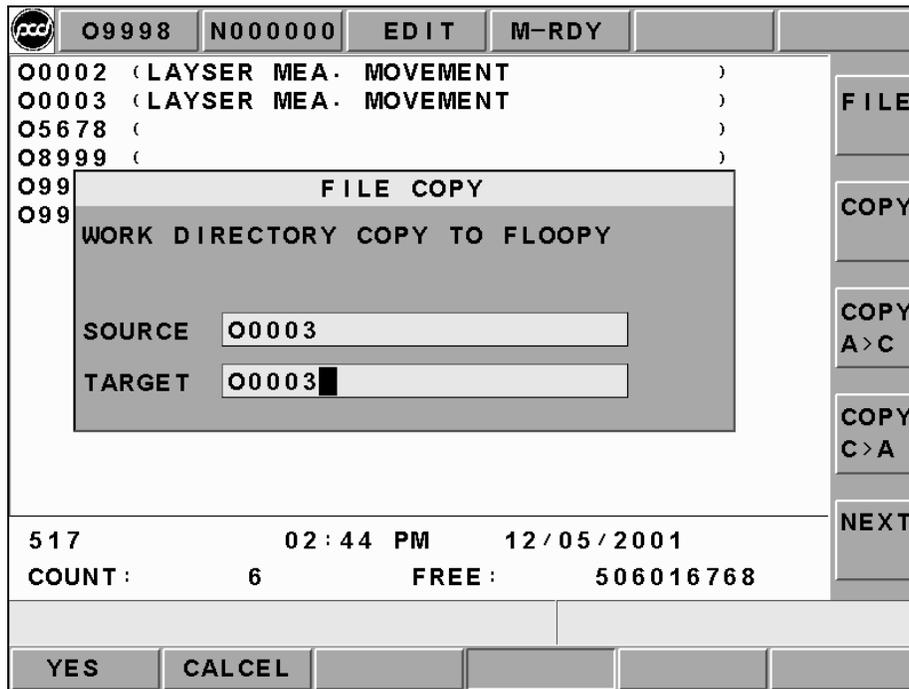


Figure 2.5-16 Target File Name Key-In Screen

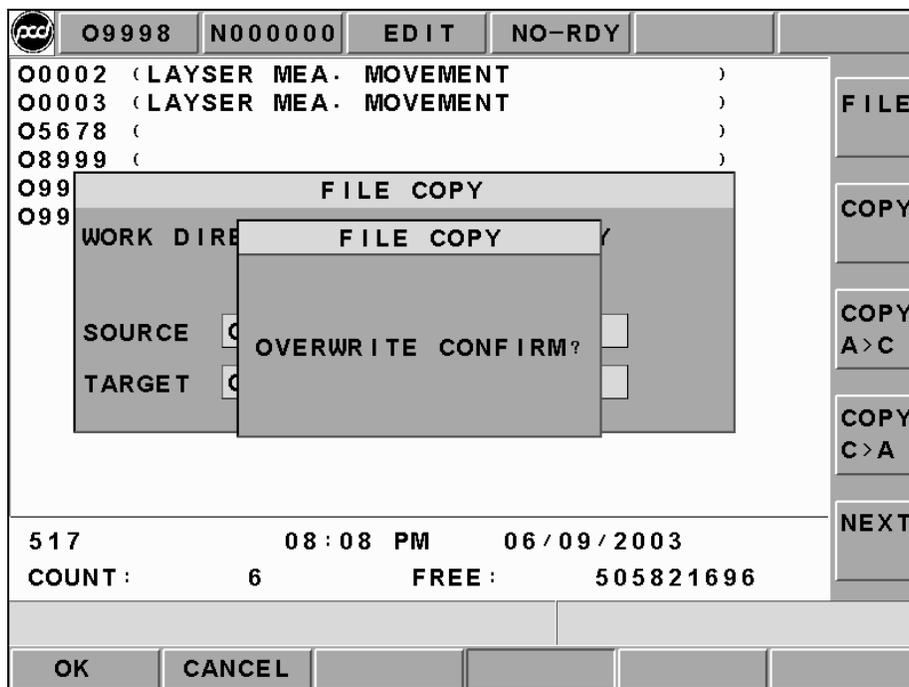


Figure 2.5-17 File Overwrite Confirmation Screen

【REN】

1. After pressing rename function button, a rename file source windows will occur on the screen. Users are able to choose the wanted change's file name. Pressing confirm button to enter into the file rename windows.
- 2 To enter the destination file name in the rename windows and then to press confirm button. After rename complete, users will be able to use program list to confirm.

Note: please key-in “O0000” as a default file name into the source dialog box in order for the rename dialog box to occur. Please see Figures 2.5-18 and 2.5-19 as examples.

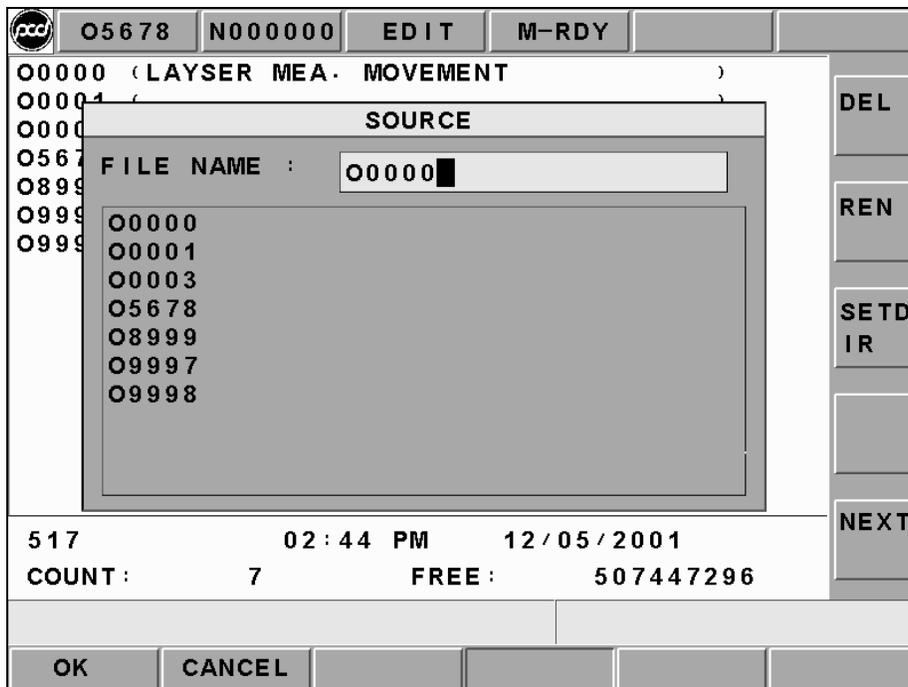


Figure 2.5-18 【REN】 Screen

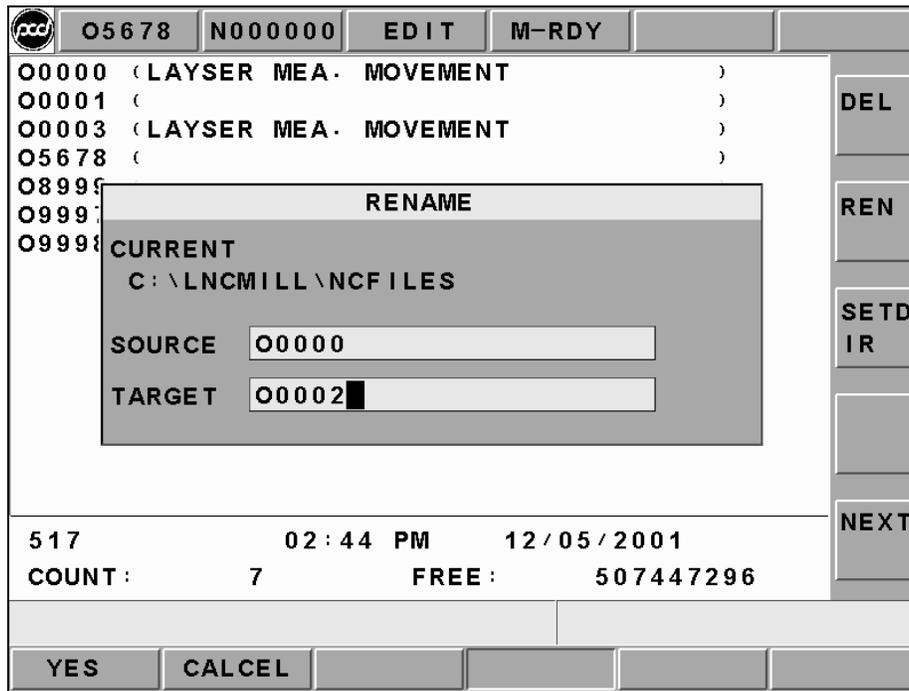


Figure 2.5-19 Key-In Target File Name Screen

【DEL】

Pressing on this sub-function button, a file delete dialog box will appear on the screen. Please choose the wanted delete file in that dialog box. After complete deleting, users can use “Program Table” to confirm the result.

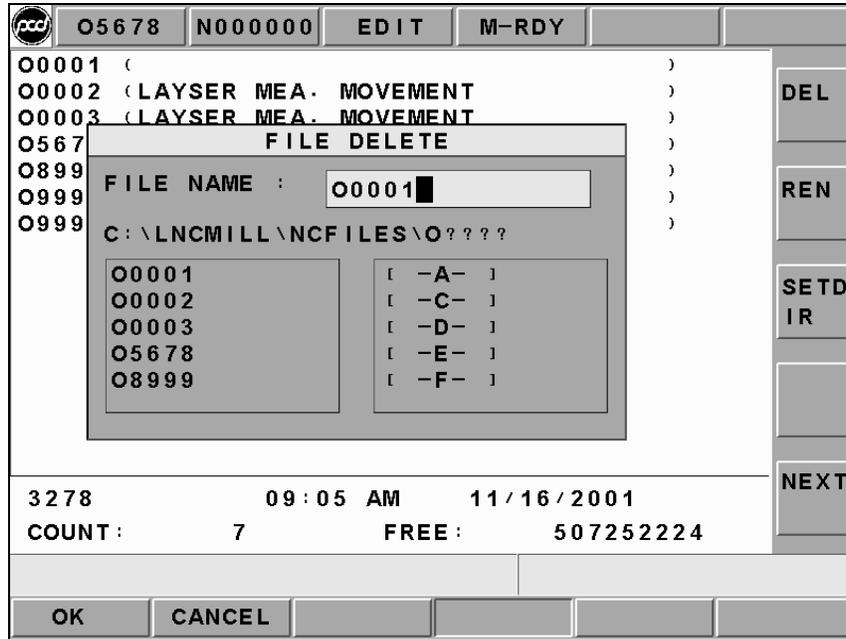


Figure 2.5-20 【DEL】 Screen

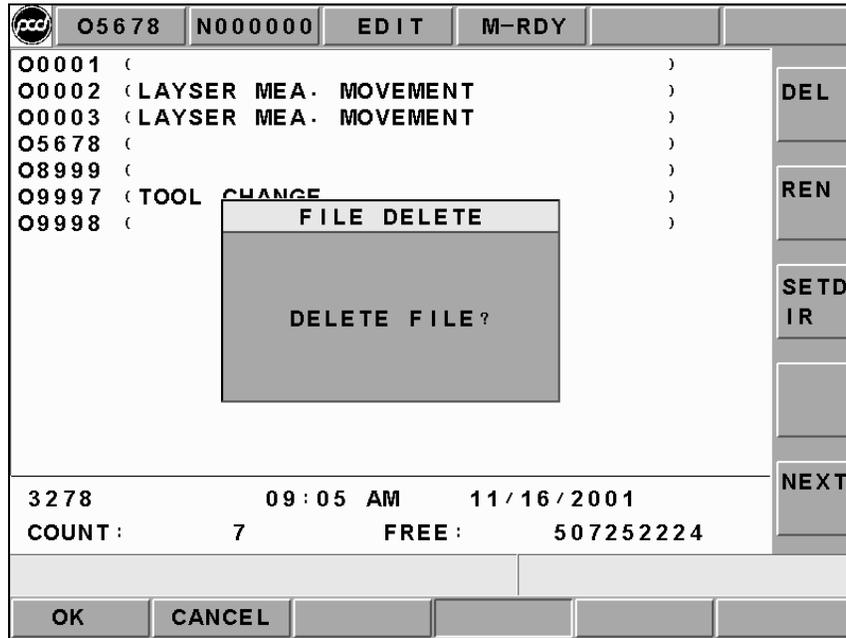


Figure 2.5-21 Double Confirmation Screen

【SETDIR】

A dialog box of part program menu setting will occur on the screen by pressing this sub-function button. Users are able to set the part program menu by using this dialog box or by entering work path directly.

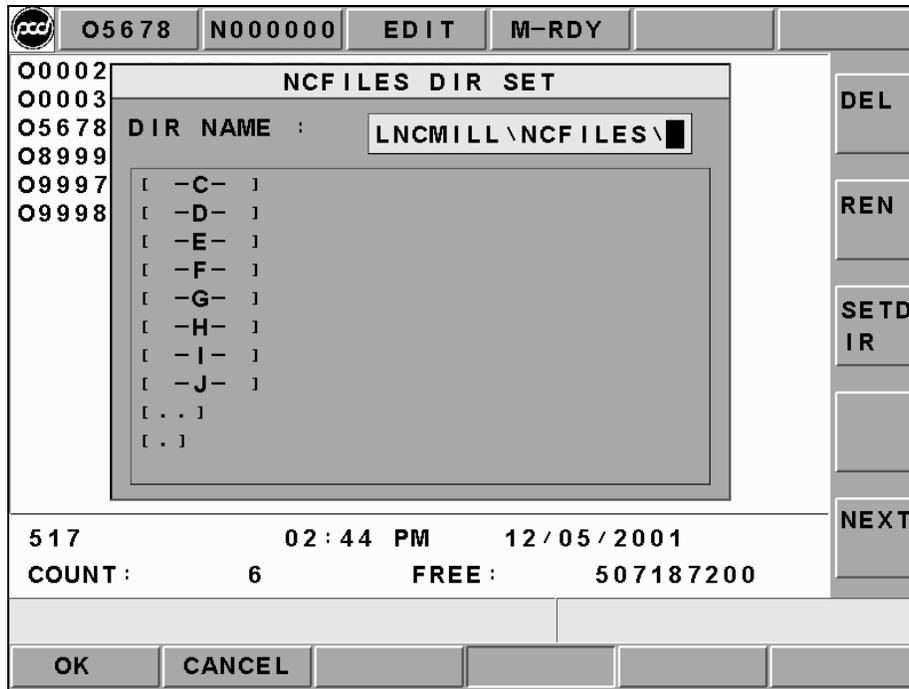


Figure 2.5-22 Setting Part Program Menu Screen

2.5.5 COMM (Communication)

Clicking the **【COMM.】** function button to receive and transmit programs between the controller and other PCs. Descriptions of how to operate sub-function buttons such as **【COMM】** and **【SETT】** are as following.

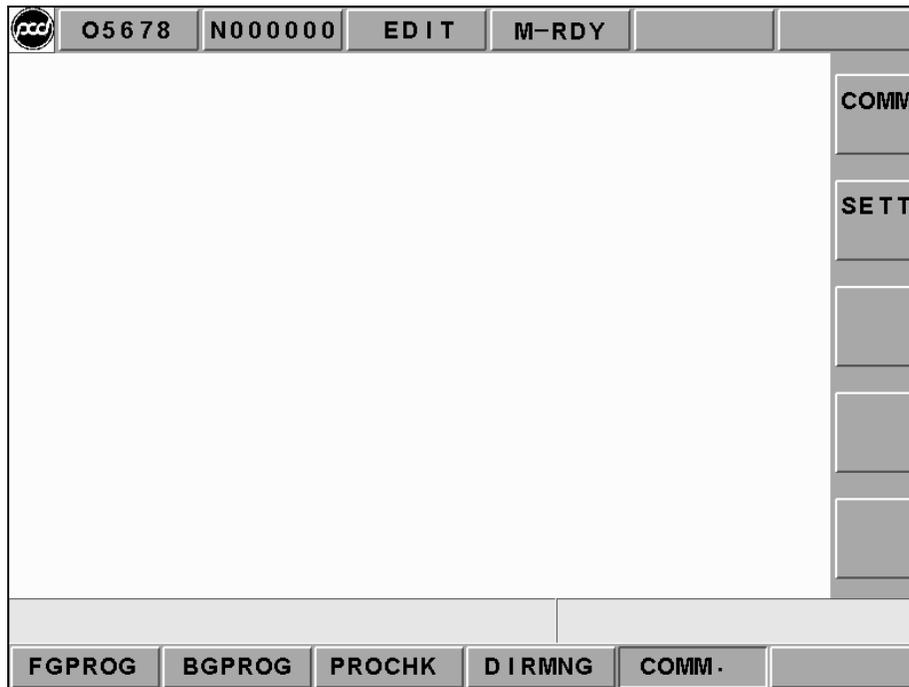


Figure 2.5-23 **【COMM.】** Screen

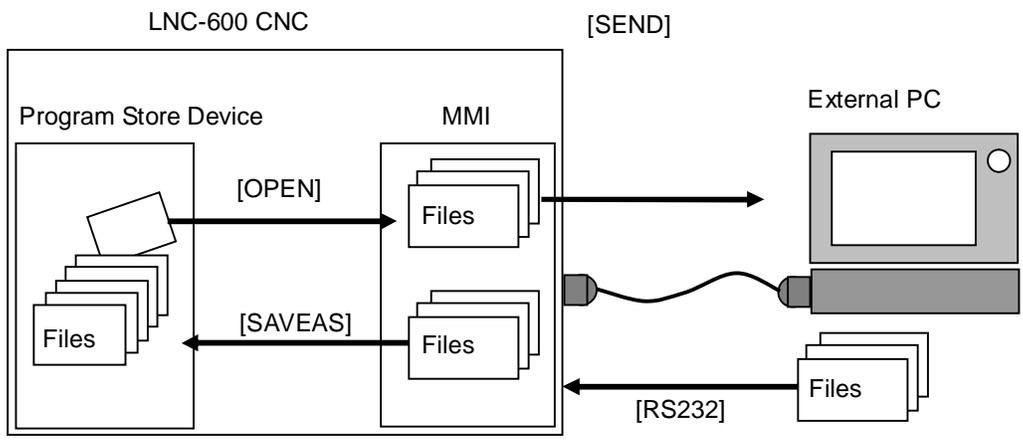


Figure 2.5-24 Files Transmitting/Receiving Figure between Controller and Other PCs

【COMM.】 (Communication)

Users are able to executing RS-232 file transmitting (DNC), reading, saving, re-setting and other operations. Before using RS232 function, please confirm whether or not the hardware connection is correct. Also, the setting of RS232 transmitting protocol and remote device must be the same.

【SEND】

Pressing the transmitting button, the file selection windows will occur. Selecting the wanted transmitting file and pressing confirm button to transmit the file via RS232. (like Figure 2.5-26)

【SAVE AS】

When transmitting one program file to RS232 of the controller's windows from outside, pressing this key to select/key-in the storage path or file name directly.

【CLEAR】

Giving up and clearing program files in the windows.

【RESET】

Giving up file transmitting and resetting communication protocol in order to build up connection with RS232.



Figure 2.5-25 Transmitting Files Screen



Figure 2.5-26 Selecting File Transmitting Function Screen



Figure 2.5-27 Screen of Save File Function

【RS232 SETTING】

This screen provides setting of **RS232**. The setting at each side of the **RS232** transmitting must be correct and the same in order to transmit (or so call DNC function) correctly. Users are able to modify the transmitting protocol for **RS232** on this screen.

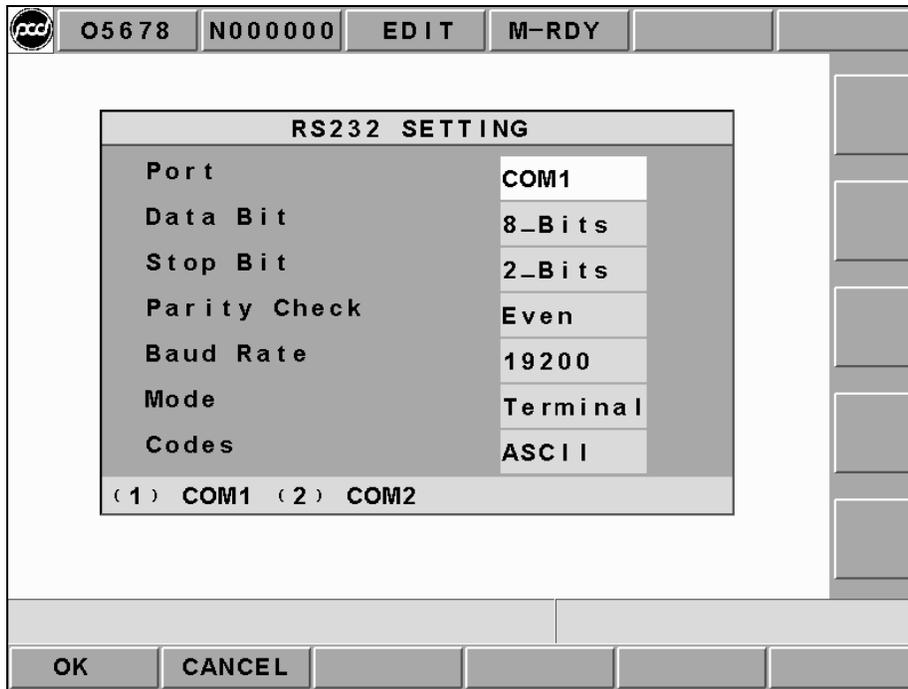


Figure 2.5-28 RS232 Setting Screen

2.5.6 DNC Function

DNC function is a very useful function for CAD/CAM system users, especially the controller is the controller used only (not the controller of PC BASED). Since the storage memory capacity is not very big, so not possible to download all the programs at once. This implies that it is very important for the DNC function to do its function while transmitting. Since DNC program is doing its function while transmitting, so the controller can't edit it, neither using program calling nor using jump commanding.

In order to solve the above problems for users, LNC Technology Co., Ltd. has installed high capacity hard drive. Therefore, users can transmit files into the hard drive and then execute those files (this method is called the internal DNC method.) The DNC function of executing its functions while transmitting is the other DNC method (this method is called the external DNC method.)

The above two methods are very convenience to use. The procedure is as below:

- a. To set Controller and external PC as RS232 modal and to set the communication protocol the same
- b. To switch to EDIT mode
- c. To press function button **【COMM.】** in order to enable file windows and to set file name as RS232
- d. To switch to AUTO mode
- e. To press "CYCLE START" button on OP Panel in order to transmit CNC waiting program
- f. To enable external PC to do RS232 file transmitting
- g. To execute machine work automatically

2.6 OFFSET Functions

Pressing <OFFSET> to enter into the offset function screens such as **[WEAR]**, **[GEOM]**, **[W. SHIFT]**, **[MACRO]**, and **[WORK]**. Users are able to modify these settings under MDI mode or machine ready mode.

2.6.1 WEAR (Wear OFFSET)

Pressing **[WEAR]** button to enter the following screen. Also, users are able to key-in data in MDI Mode.

INPUT: ABS		UNIT: MM		
NUM	TYPE	W_R	W_X	W_Z
001	0	0.000	0.000	0.000
002	0	0.000	0.000	0.000
003	0	0.000	0.000	0.000
004	0	0.000	0.000	0.000
005	0	0.000	0.000	0.000
006	0	0.000	0.000	0.000
007	0	0.000	0.000	0.000
008	0	0.000	0.000	0.000
009	0	0.000	0.000	0.000
010	0	0.000	0.000	0.000
RELATIVE				
		X	3.440	
		Z	13.824	

WEAR
GEOM
W. SHIFT
MACRO
WORK

ABS
INC
NORU
NT
MINU
NT

Figure 2.6-1 WEAR OFFSET Screen-1

Description of Figure and each Column is as below:

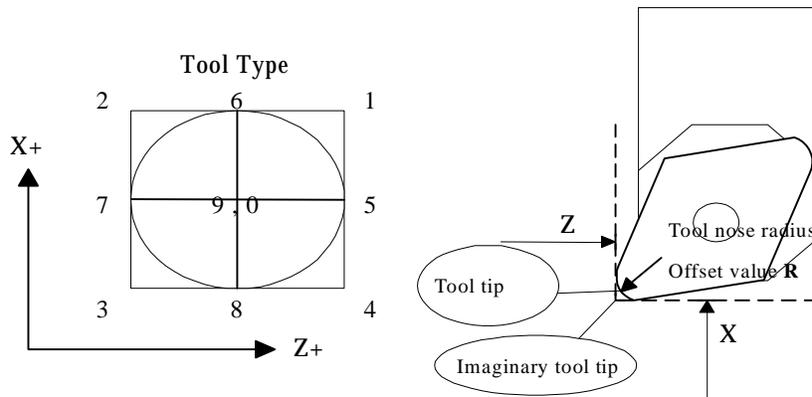


Figure 2.6-2

- Type : Tool type
- W_X : X axis wear offset value
- W_Z : Z axis wear offset value
- W_R : radius wear offset value

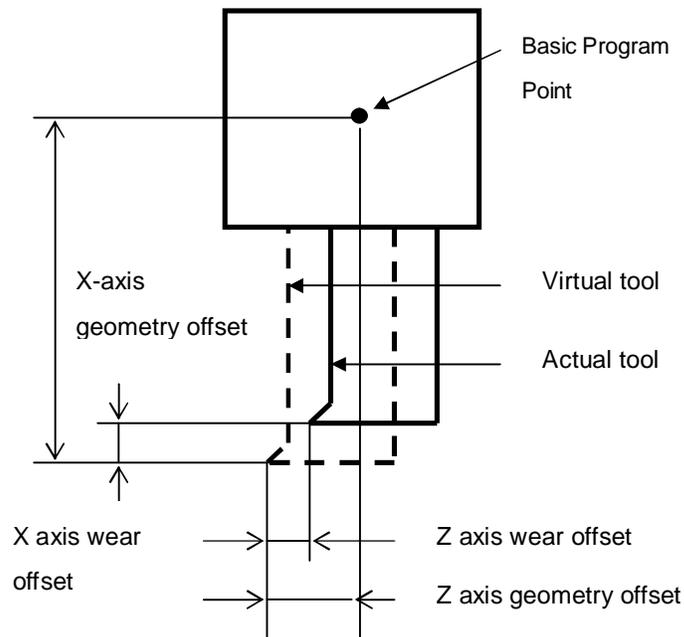


Figure 2.6-3

Everytime when the part program uses one tool offset number, the absolute coordinate will change according to the tool offset value. The value is [absolute coordinate value = machine coordinate value – external work shift - G54~G59 shift - (tool geometry offset value + tool wear offset value)].

Pressing <PAGE ↓ > key to see more setting sets (max. 30 sets)

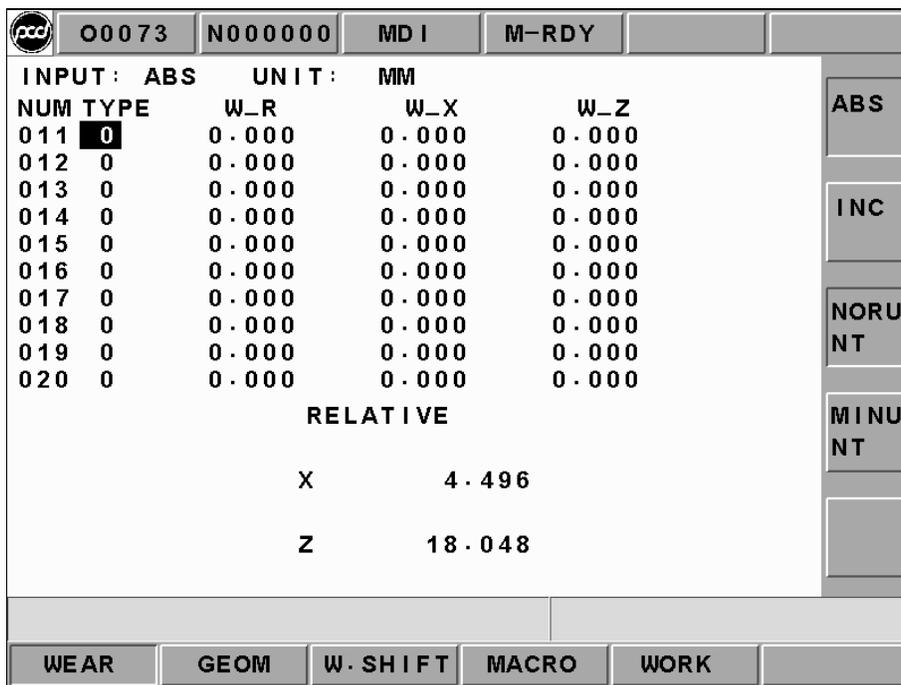


Figure 2.6-4 WEAR OFFSET Screen-2

Moving the highlighter to the wanted setting number. Under MDI mode, key-in the setting value and then pressing <INPUT> to write into a controller.

2.6.2 GEOM (Geometry OFFSET)

Pressing **【GEOM】** button to enter into the below screen so users are able to modify the offset value on that screen.

INPUT: ABS		UNIT: MM			
NUM	TYPE	G_R	G_X	G_Z	ABS
001	0	0.000	0.000	0.000	
002	0	0.000	0.000	0.000	
003	0	0.000	0.000	0.000	
004	0	0.000	0.000	0.000	INC
005	0	0.000	0.000	0.000	
006	0	0.000	0.000	0.000	
007	0	0.000	0.000	0.000	
008	0	0.000	0.000	0.000	NORU
009	0	0.000	0.000	0.000	NT
010	0	0.000	0.000	0.000	
RELATIVE					
		X	3.792		
		Z	15.232		
WEAR		GEOM		W. SHIFT	
MACRO		WORK			

Figure 2.6-5 GEOM OFFSET Screen

- Type : Tool type
- G_X : X axis geometry offset value
- G_Z : Z axis geometry offset value
- G_R : radius geometry offset value

Users are able to key-in the modified value directly under the conditions of MDI mode and CNC ready. But, users are able to use “MXxx” and “MZzz” to set the offset value for G_X and G_Z. The calculation method is as following:

Offset value = Machine coordinate value – external work shift - G54~G59 shift – key-in value.

2.6.3 W. SHIFT (Work Coordinate Shift)

Pressing **【W. SHIFT】** button to enter into the setting screen of the work shift function.

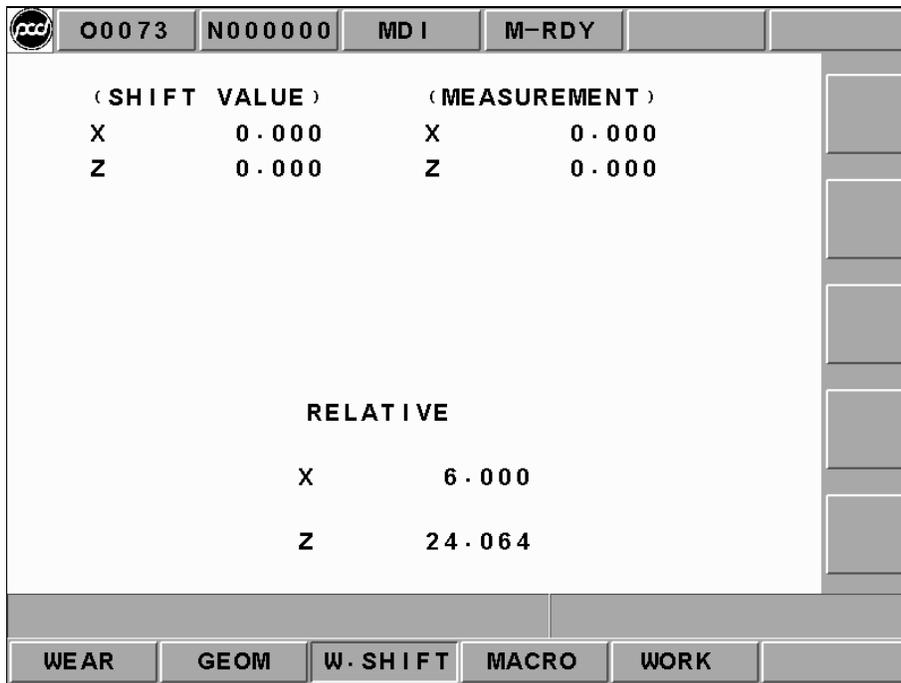


Figure 2.6-6 Setting WORK SHIFT Screen

Two key-in value methods under MDI mode. The description is as below:

Method 1: key-in the shift amount for each axis directly, key-in “X20. Z20.” first. Then, pressing **【ENTER】** to update the new entry value into the shift value. So shift value = entry value.

Method 2: key-in the estimating value. Please key-in “MXxx MZzz” first and then pressing **【ENTER】** to update the new entry value into the estimating value column. So estimating value = entry value.

The calculation of shift value is as below:

$$\text{Shift value} = \text{machine coordinate value} - \text{entry value} - \text{G54~G59 shift} - (\text{wear offset value} + \text{geometry offset value})$$

2.6.4 MACRO

Pressing **【MACRO】** button to enter into the MACRO variable screen. In this function page, variables can be entered or modified. The modification method is to move the highlighter to the wanted modify position and then to press <INPUT> button after entering the wanted value. Variables that start with L are local variables and those start with C are common variables. Please refer to MACRO chapter of program manual for a detailed description.

NUM	VALUE	NUM	VALUE
L001	0.0000	L011	0.0000
L002	0.0000	L012	0.0000
L003	0.0000	L013	0.0000
L004	0.0000	L014	0.0000
L005	0.0000	L015	0.0000
L006	0.0000	L016	0.0000
L007	0.0000	L017	0.0000
L008	0.0000	L018	0.0000
L009	0.0000	L019	0.0000
L010	0.0000	L020	0.0000
RELATIVE			
	X	6.704	
	Z	26.880	

Figure 2.6-7 Local Variables of MACRO Variables

O0073		N000000		MDI	M-RDY
NUM	VALUE	NUM	VALUE		
C001	VACANT	C011	VACANT		
C002	VACANT	C012	VACANT		
C003	VACANT	C013	VACANT		
C004	VACANT	C014	VACANT		
C005	VACANT	C015	VACANT		
C006	VACANT	C016	VACANT		
C007	VACANT	C017	VACANT		
C008	VACANT	C018	VACANT		
C009	VACANT	C019	VACANT		
C010	VACANT	C020	VACANT		
RELATIVE					
	X	7.280			
	Z	29.184			
WEAR		GEOM		W-SHIFT	MACRO
				WORK	

Figure 2.6-8 System Variables of MACRO Variables

2.6.5 WORK (Coordinate Setting)

Pressing **【WORK】** to enter into the setting screen of the work coordinate system. There are 7 sets of work coordinate system for setting, which indicate extending shift for G54 ~ G59. In MDI mode, users are able to use cursor to identify group. Also, after key-in Xxx..Zzz..., users can press <INPUT> to renew the coordinate value in order for part program in different coordinate system.

Extending shift setting will directly affect other 6 coordinate systems (G54 to G59). Please see the following for description:

- # 01: G54 machine coordinate value of the reference point = #01 setting value + #(00) setting value
- # 02: G55 machine coordinate value of the reference point = #02 setting value + #(00) setting value
- # 03: G56 machine coordinate value of the reference point = #03 setting value + #(00) setting value
- # 04: G57 machine coordinate value of the reference point = #04 setting value + #(00) setting value
- # 05: G58 machine coordinate value of the reference point = #08 setting value + #(00) setting value
- # 06: G59 machine coordinate value of the reference point = #06 setting value + #(00) setting value

Additional example: key-in x-256.5, and pressing <INPUT> button will set -256.5 to the x coordinate value on the assigned cursor set.

O0073		N000000		EDIT	M-RDY		
NUM		COORD		NUM		COORD	SET ALL
00				02			
	X	0.000		G55	X	0.000	SET X
	Z	0.000			Z	0.000	
							SET Z
01				03			
G54	X	0.000		G56	X	0.000	
	Z	0.000			Z	0.000	
WEAR		GEOM		W. SHIFT		MACRO	
				WORK			

Figure 2.6-9 Work Coordinate Setting Screen-1

Pressing <PAGE ↓ > to view the next page.

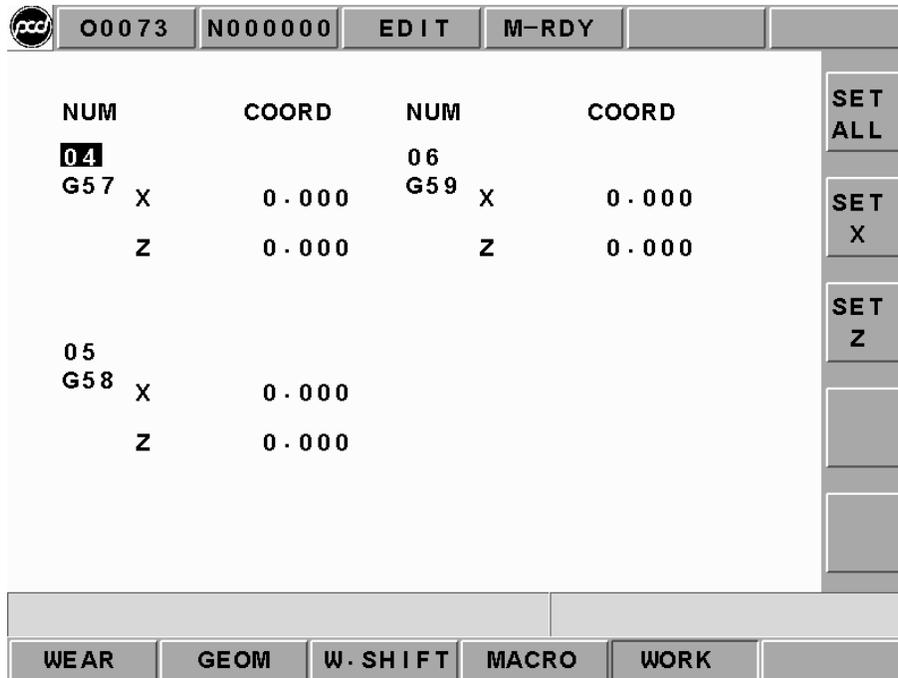


Figure 2.6-10 Work Coordinate Setting Screen-2

Extending shift setting will directly affect other 6 coordinate systems (G54 to G59). When setting, please consider work piece coordinate, tool and program executing relationships.

If want to reset one coordinate system:

- moving the cursor key to the wanted setting number of the coordinate system,
- key-in the axis name and the new coordinate value, users are able to write 3 axes continuously. But, remember to put decimal point in order to prevent confusion. (i.e., X100. Z200.)
- pressing <INPUT> to key-in the value into controller.

2.7 Lathe EASY-CAM

2.7.1 Function Description

Users only need to key-in the needed working information and then to press **【NC Gear Shifting】** button to produce working path program automatically. There are 14 working methods of this function for users to choose. Each working method must be used with different parameter according to its feature in order to produce correct working path program.

2.7.2 Operation Procedure of How to Produce Working Path Program

1. Open a New Project File

Pressing **【DIRMNG】** button on the main function screen to change the screen to the file management screen. Pressing **【New】** button on the project management screen and key-in the project name into the entry column and then pressing **【OK】** button. The screen will change to the main function screen after opening file successfully.

2. New Procedure

Pressing **【PROC ADD】** button on the main function screen.

3. Working Method Selection

Pressing **【ENTER】** button when yellow cursor occurs at the working method column and a working method selection menu will popup. Using **【UP】** or **【DOWN】** key to select the needed working method and then pressing **【ENTER】** or **【OK】** button. Then, the selected working method will appear on the working method column.

4. Tool Number Selection and Tool Information Key-In

When the yellow cursor occurs at the tool number column, pressing **【ENTER】** button to change the screen to the tool setting screen. Key-in the needed information according to different working method and then pressing **【ENTER】** button. After that, pressing **【RTN】** button to change the screen to the main function screen. The chosen tool number will appear on the tool number column.

5. Key-in Head Parameter Info

When the yellow cursor occurs at the head parameter column, pressing **【ENTER】** key to switch the screen to the head parameter screen. After key-in the needed head parameter data, pressing **【ENTER】** key to switch the screen back to the main function screen. Then, a blue “DEFINE” will occur on the head parameter column.

6. Key-in Cutting Parameter Information

When the yellow cursor occurs at the cutting parameter column, pressing **【ENTER】** button to change the screen to the cutting parameter screen. After key-in the needed cutting parameter data, pressing **【RTN】** button to change the screen back to the main function screen. Blue characters, “Define”, will appear on the parameter column.

7. Key-in Geometry Define Information

When the yellow cursor occurs at the geometry define column, pressing **【ENTER】** button to change the screen to the geometry define screen. After key-in the needed geometry define data, pressing **【RTN】** button to change the screen to the geometry define screen. Then, blue characters, “Define”, will appear on the geometry define column.

8. After the above seven steps, users can press **【SAVE】** to save the project. Or, pressing **【NC Gear Shifting】** to produce the working path program.

2.7.3 Operation Screen and Function Key Instruction

◎ Main Function, the screen is as below:

SEQ.	PROC	TOOL	HEAD	CUT.	GEO.
01	OGOV	C01	DEFINE	DEFINE	DEFINE
02	DRILL	A01	DEFINE	DEFINE	DEFINE

PRJ . MNG SAVE CONVER CLOSE

Figure 2.7-1 Main Function Page

【Column Description】

- ※ Project Name : display the currently opened project name.
- ※ Procedure : display procedure order number. The system will auto edit this column.
- ※ Working Method : display the selected working name. A “clear working name” dialog box will be popup when users press **【ENTER】** key at this column.
- ※ Tool # : display the selected tool number. The screen will switch to the tool setting screen when users press **【ENTER】** at this column. 4 types of tool number, which are A type for drilling tool, B type for profile lathe tool, C type for grooving tool, D type for screwing tool. Number means the tool number from the tool list on the tool setting page.

- ※ Head Parameter : red indicates “un-define” and blue indicates “define”. “Un-define” means users select to enter into the head parameter screen that never been in before. On the other hand, “define” means users have been to the head parameter screen. Screen will switch to the head parameter screen when users press【ENTER】key at this column.
- ※ Cutting Parameter : red indicates “un-define” and blue indicates “define”. “Un-define” means the cutting parameter screen that users haven’t been in before. On the other hand, “define” means users have been to the head parameter screen. Screen will switch to the cutting parameter screen when users press 【ENTER】 key at this column.
- ※ Geometry Define : red indicates “un-define” and blue indicates “define”. “Un-define” means the geometry define screen that users haven’t been in before. On the other hand, “define” means users have been to the geometry define screen. Screen will switch to the geometry screen when users press 【ENTER】 key at this column. Screen will switch to different geometry setting screens according to different working methods.

【 Function Keys at Screen Bottom 】

- ※ **PRJ. MNG** : switching screen to the project management windows.
- ※ **SAVE** : saving the currently opened project.
- ※ **CONVER** : switching the currently opened project data to the work path program.
- ※ **CLOSE** : closing the currently opened project.

【 Function Key at Right of Screen 】

- ※ **PROC ADD** : adding a new program to the next row of the row where the yellow cursor occurs.
- ※ **PROC INS** : insert a new program to the previous row of the row where the yellow cursor occurs.
- ※ **PROC CPY** : copy a row of program where the yellow cursor occurs to the next row of the copied row.
- ※ **PROC DEL** : delete all procedures where yellow cursor occurs.

© Screen of Project Management is as below:



Figure 2.7-2 Project Management Page

【Column Description】

- ※ Project # : display the current project number.
 - ※ Display Dialog Box: display all project list
 - ※ Project Date : display the project date where the yellow cursor occurs.
 - ※ File Size : display the file size where the yellow cursor occurs.
 - ※ Procedure # : display the total procedure # of a project where the yellow cursor occurs.
 - ※ File Description : display the project description where the yellow cursor occurs.
- Pressing **【LEFT】** or **【RIGHT】** key to move the yellow cursor to the column. Then, key-in data into that column and press **【ENTER】** key to write “NOTE” for the project.

【Function Keys at Screen Bottom】

- ※ OPEN: downloading a project where the yellow cursor occurs and switching to the main function page.
- ※ NEW : building a new project name and the screen will switch to the main function screen after confirm the key-in project name.
- ※ DELETE : deleting a project where yellow cursor occurs.
- ※ RETURN : switching screen to the main function screen.

【Function Keys at Right of Screen】

- ※ Confirm: pressing **【NEW】** or **【DEL】** key at screen bottom for the confirm key to occur. Pressing **【NEW】** key at the screen bottom. Then pressing **【OK】** to open the new file and to switch the screen to the main function screen. Pressing **【OK】** after pressing **【DEL】** key at the screen bottom, the project where the yellow cursor occurs will be deleted.
- ※ Cancel : pressing **【OPEN】** or **【DEL】** key at the screen bottom first. Then, pressing **【DEL】** key after pressing **【OPEN】** key at screen bottom to cancel opening new file. Pressing **【CANCEL】** key after pressing **【DEL】** key at screen bottom to cancel deleting file.

◎ Screen of Working Method is as below:

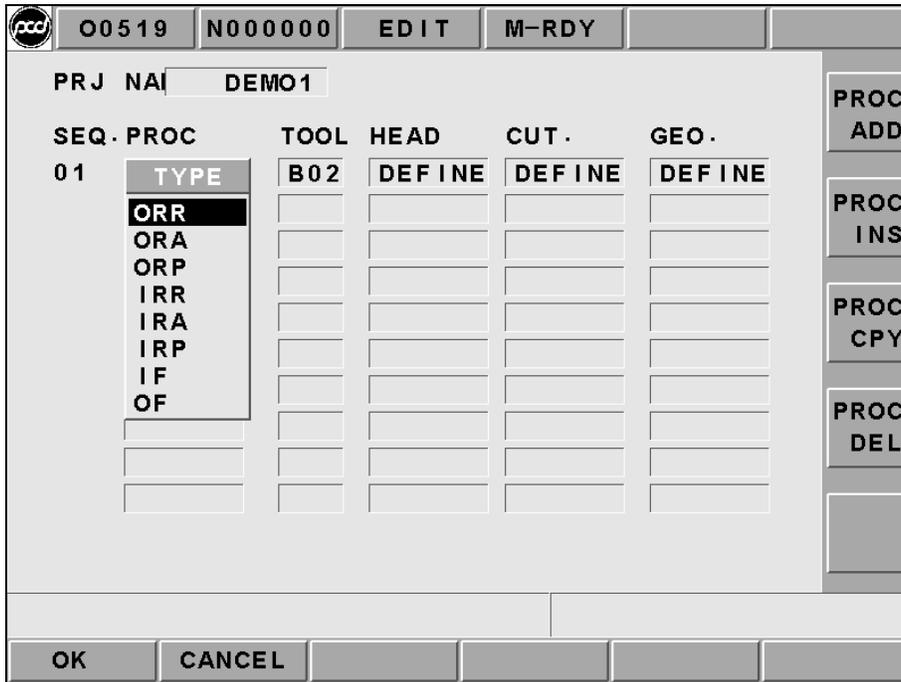


Figure 2.7-3 Working Method Page

【 Column Description 】

※ Working Method List : display a list of 14 types working methods as below show:

1. DRILL: drilling one internal hole on the material
2. ORR: Outer Rough Diameter
3. ORA: Outer Stock Removal in Turning
4. ORP: Outer Pattern Repeating
5. IRR: Internal Rough Diameter
6. IRA: Internal Stock Removal in Turning
7. IRP: Internal Pattern Repeating
8. IF: Internal Finishing Cycle
9. OF Outer Finishing Cycle
10. OGOV: Grooving on Outer Surfaces
11. IGOV: Grooving on Inter Surfaces
12. FGOV: Grooving on End Faces
13. OSEW: External Thread Cutting
14. ISEW: Internal Thread Cutting

※ Description of Each Working Method is as below :

- ⊙ **Drilling** : drilling one internal hole on the material
- ⊙ **ORR**: outer rough diameter. Cutting most of material part that is made by straight line and arch. Tool moving direction is according to the perpendicular Z-axis direction
- ⊙ **ORA** : outer stock removal in turning. Clearing the most material part that is made by line and arch. Tool moving direction is according to the perpendicular X-axis direction.
- ⊙ **ORP**: outer pattern repeating. Clearing the most material of the pattern that is made by straight line and arch. Tool moving direction is according to the curve outline.
- ⊙ **IRR**: internal rough diameter. Cutting most of material part that is made by straight line and arch. Tool moving direction is according to the perpendicular Z-axis direction.
- ⊙ **IRA** : internal stock removal in turning. Clearing the most material part that is made by line and arch. Tool moving direction is according to the perpendicular X-axis direction.
- ⊙ **IRP**: internal pattern repeating. Clearing the most material of the pattern that is made by straight line and arch. Tool moving direction is according to the curve outline.
- ⊙ **IF** : internal finishing cycle. Clearing the rest of the material of the outline that is combined by straight line and arch.
- ⊙ **OF** : outer finishing cycle. Clearing the rest of the material of the outline part that is combined by straight line and arch.
- ⊙ **OGOV** : grooving on outer surfaces,. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.
- ⊙ **IGOV** : grooving on internal surface. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.
- ⊙ **FGOV** : grooving on end faces. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.

- ⊙ **OSEW** : external thread cutting. Cutting thread horizontally and selecting feeding method according to the using method.
- ⊙ **ISEW** : internal thread cutting. Cutting thread horizontally and selecting feeding method according to the using method.

【 Function Key at Screen Bottom 】

- ※ Confirm : selecting the working method where the yellow cursor is at. Switching to the main function screen and then displaying the selected working name on the working method column.
- ※ Cancel : cancel selecting working method action and switch screen to the main function screen.

【 Function Key at Right of Screen 】

None.

◎Screen of Tool Setting is as below screen:

The screenshot shows a CNC control interface for tool setting. At the top, there are status indicators: 'PCC', 'O0519', 'N000000', 'EDIT', and 'M-RDY'. Below this is the 'TOOL SET' section. It includes fields for 'PROC' (01), 'PROCESS' (IRR), and 'TOOL NO' (B02). A table lists 10 tool parameters with columns for NO, DID, DIA, CENTER, WIDTH, ANG1, ANG2, and DIR. The first row (NO 01) has DID 4, DIA 0.400, CENTER 1.000, ANG1 90.0, ANG2 0.00, and DIR 0. The second row (NO 02) has DID 3, DIA 0.400, CENTER 1.000, ANG1 270.0, ANG2 300.0, and DIR 0. The remaining rows (NO 03-10) have DID 2, DIA 0.200, CENTER 1.000, ANG1 90.0, ANG2 0.00, and DIR 0. At the bottom, there is a 'DIR ID · 1~8' field and 'SELECT' and 'RETURN' buttons.

NO	DID	DIA	CENTER	WIDTH	ANG1	ANG2	DIR
01	4	0.400	1.000		90.0	0.00	0
02	3	0.400	1.000		270.0	300.0	0
03	2	0.200	1.000		90.0	0.00	0
04	2	0.200	1.000		90.0	0.00	0
05	2	0.200	1.000		90.0	0.00	0
06	2	0.200	1.000		90.0	0.00	0
07	2	0.200	1.000		90.0	0.00	0
08	2	0.200	1.000		90.0	0.00	0
09	2	0.200	1.000		90.0	0.00	0
10	2	0.200	1.000		90.0	0.00	0

Figure 2.7-4 Tool Setting Page

【 Column Description 】

- ※ Procedure : display the present editing procedure order number.
- ※ Working Method : display the working method name of the present editing procedure.
- ※ Tool NO. : display the selected tool NO. of the present editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.
- ※ Tool Clear Dialog Box at Screen Bottom: able to display and edit 10 sets of data. Users need to key-in different columns according to different working methods. The description of each column is as below:
 - ◎ Tool NO. : tool parameter number, The same type, able to do CAM calculation according to difference parameter. Selecting the NO. 1 tool parameter when one tool is at the first procedure. But, for the 2nd procedure, users can select the 2nd tool parameter for CAM calculation.

- Tool Direction : tool noise direction, which is the imagery tool tip direction. Using 10 numbers (number 0~9) to indicate the corresponding position of the imagery tool tip and tool noise center point. Like Figure 2.6-5, arrowhead position is the tool tip point. Tool tips of imagery Tool # 0 and #9 is overlap with tool noise center point.

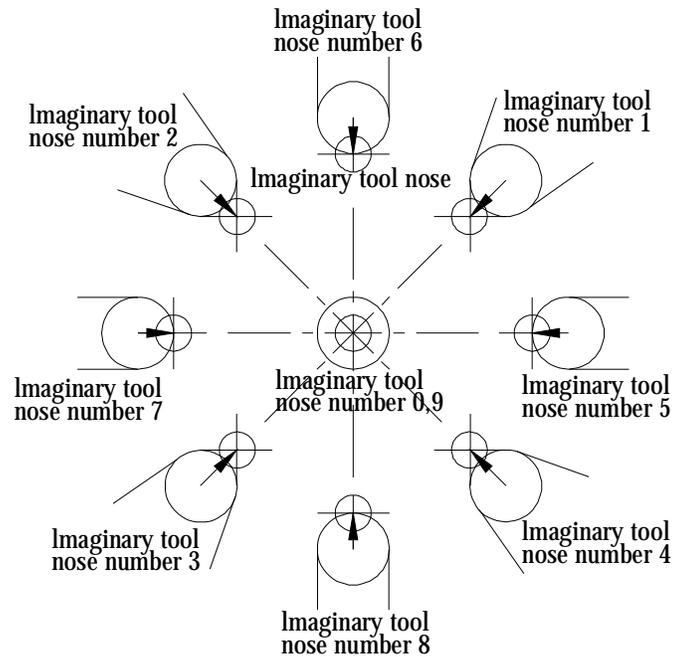


Figure 2.7-5 Corresponding Position Figure of Imagery Tool Tip Point and Tool Noise Center Point

- ⊙ Tool Radius: tool nose radius. The below Figure is the tool tip after scaling in.

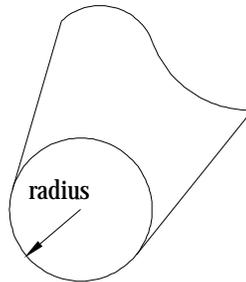


Figure 2.7-6 Scale-In Figure of Tool Tip

- ⊙ Radius: the below Figure is the internal radius of tool

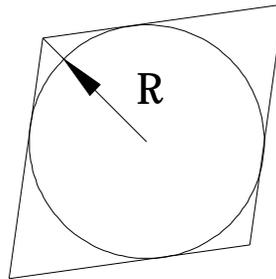


Figure 2.7-7 Internal Tool Radius

- ⊙ Tool Width : Tool width
- ⊙ Feed Angle : Feed angle. To set the needed setting column, external pattern, axis, wheel and fine-lathe to 90 and to set the internal pattern, axis, wheel and fine-lathe to 270.
- ⊙ Cutting Angle : the max. effective cutting angle. To set the needed setting column, external pattern, axis, wheel and fine-lathe within the range of 0~90 when doing profile cutting and to set the internal pattern, axis, wheel and fine-lathe within the range of 270 ~360.
- ⊙ Rotary Direction: Spindle rotary direction, clockwise (CW) or counter clockwise (CCW)

【Function Keys at Bottom at the Screen】

- ※ Selection: select the needed tool parameter data, tool number at the upper screen indicates the selected tool number.
- ※ Return: switch to the main function screen.

【Function Keys at Right of the Screen】

None.

◎ Screen of Head Parameter is as below:

	O0519	N000000	EDIT	M-RDY		
HEAD PARAM						
PROC	01	PROCESS	IRR	TOOL NO	B02	
MAX RPM	3000	CONT	0			
F TYPE	0	T CHG	0			
F V	300.000	PT X	50.000			
F RPM	0.000	PT Z	100.000			
S TYPE	1	H CODE	1			
S V	0	T CODE	1			
S RPM	300					
MAX RPM · 0 < VALUE < 7000						
						RETURN

Figure 2.7-8 Header Parameter Page

【Column Description】

- ※ Procedure : display the currently editing procedure number.
- ※ Working Method : display the working method name of the currently editing procedure.
- ※ Tool NO. : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.

※ Head Parameter Column : users can key-in an appropriate value according to the actual need.

- ⊙ **Max. RPM** : spindle max. rotational speed
- ⊙ **F TYPE** : 0 means using feed/min, 1 means using feed/rotate.
- ⊙ **F V** : when the feed unit is selected as 0, users must key-in any value that is larger than 0 into this column.
- ⊙ **F RPM** : when the feed unit is selected as 1, users must key-in any value that is larger than 0 into this column.
- ⊙ **S TYPE** : 0 means using fixed cutting speed, 1 means using fixed rotating speed.
- ⊙ **S V** : when fixed cutting/rotate is selected as 0, users must key-in any value large than 0 into this column.
- ⊙ **S RPM** : when fixed cutting/rotate is selected as 1, users must key-in any value larger than 0 into this column.
- ⊙ **CONT** : whether or not to use cutting liquid. 0 means no need to use it, 1 means use it.
- ⊙ **T CHG** : 0 means not need to use tool-exchanging point, 1 means use the user default tool-exchanging point.
- ⊙ **PT X** : user default of tool-exchanging X coordinate.
- ⊙ **PT Z** : user default of tool-exchanging Z coordinate.
- ⊙ **H Code** : tool OFFSET number
- ⊙ **T Code** : actual tool number on turret.

【Function Keys at Screen Bottom】

※Return: switch to the main function screen.

【Function Keys at Right of the Screen】

None.

◎ Cutting Parameter is as the below screen:

PCC		O0519	N000000	EDIT	M-RDY
CUT PARAM					
PROC	01	PROCESS	IRR	TOOL NO	B02
START	0	COUNT			
END	2	RES X	1.000		
ORG X		RES Z			
ORG Z		AMOUNT	2.000		
ST X	125.000	TIME			
ST Z	0.000	TYPE			
DEP.X		DRILL	100.000		
WID.Z	5.000				
START X · 0 < VALUE					
					RETURN

Figure 2.7-9 Cutting Parameter Page

【Column Description】

- ※ Procedure : display the currently editing procedure order number.
- ※ Working Method : display the working name of the currently editing procedure.
- ※ Tool NO. : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.
- ※ Cutting Parameter Column : the needed key-in column will be difference due to different working methods.
 - ◎ **START** : 0.
 - ◎ **END** : all linear and arch numbers of profile.
 - ◎ **ORG X** : 0.
 - ◎ **ORG Z** : 0.
 - ◎ **ST X** : material at the upper right point is the diameter coordinate.

- ⊙ **ST Z** : material at the upper right point is the Z coordinate
- ⊙ **DEP. X** : lathe cutting depth
- ⊙ **WID. Z** : lathe cutting width.
- ⊙ **COUNT**: lathe-cutting repetitive times.
- ⊙ **RES X** : lathe-cutting preservation amount.
- ⊙ **RES Z** : 0
- ⊙ **AMOUNT**: lathe-cutting escape amount.
- ⊙ **TIME**: drilling dwell time.
- ⊙ **TYPE**: the lathe-cutting mode selection.
- ⊙ **DRILL**: pre-drilling of internal lathe-cutting.

【 Function keys at Screen Bottom 】

- ※ Return: switch to the main function screen.

【 Function Keys at Right of Screen 】

None.

◎ Geometry Definition: below is the description of geometry drilling, geometry outlining, geometry grooving, and geometry thread.

◇ Screen of Geometry Drilling is as below:

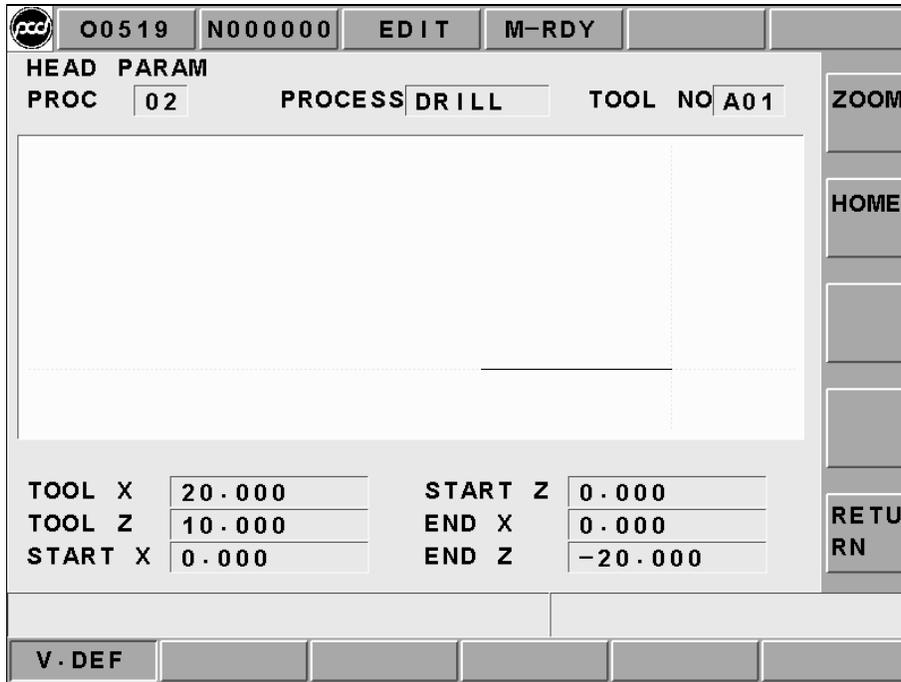


Figure 2.7-10 Geometry Drilling Page

【Column Description】

- ※ Procedure : display the currently editing procedure order number.
- ※ Working Method : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.

※ Geometry Drilling Column :

- ⊙ **TOOL X** : tool point diameter coordinate
- ⊙ **TOOL Z** : tool point Z coordinate
- ⊙ **START X** : starting drilling hole point diameter coordinate, 0.
- ⊙ **START Z** : start drilling hole point Z coordinate.
- ⊙ **END X** : final drilling hole point diameter coordinate, 0.
- ⊙ **END Z** : final drilling hole point Z coordinate

【Function Keys at Screen Bottom】

- ※ Windows Scaling : scale-in or return to original size of figure shape of display box at center of screen.
- ※ Preview : preview the Figure shape of the key-in data
- ※ Return : switching screen to the main function screen.

【Function Keys at Right of Screen】

- ※ SCALE : pressing the【SCALE】key at bottom of the screen for the “SCALE” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use 【UP】 , 【DOWN】 , 【LEFT】 and 【RIGHT】 key to move the dialog box. Moving the dialog box to the wanted scale-in position and pressing the 【ENTER】 key to complete the action.
- ※ Fittest Size: pressing 【WINDOW】 at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.
- ※ Return : pressing the 【SCALE】 key at bottom of the screen to switch to the geometry drilling screen.

◇ Screen of Geometry Profile is as below:

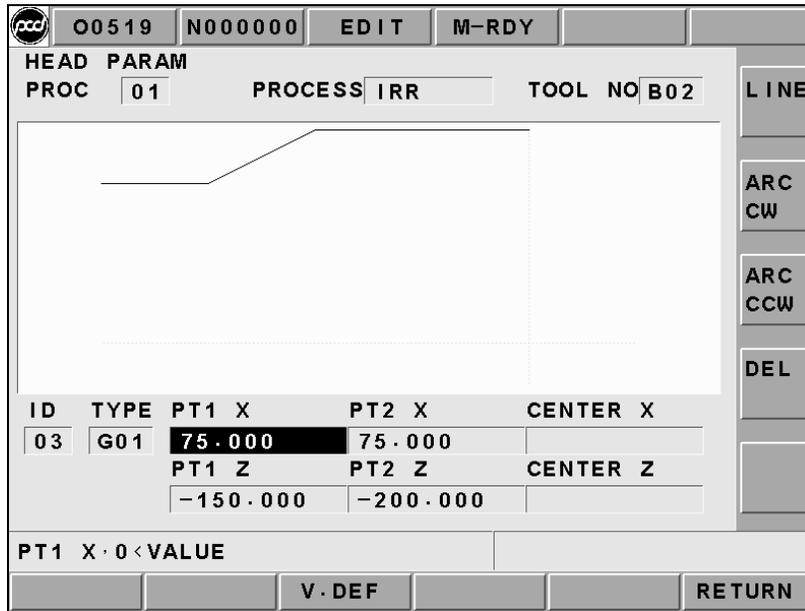


Figure 2.7-11 Geometry Profile Page

【Column Description】

- ※ Procedure : display the currently editing procedure order number
- ※ Working Method : display working name of the currently editing program
- ※ Tool NO. : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.
- ※ Order # : display profile linear order number, the system will auto-edit number.
- ※ Type : display profile linear type, G01 is linear, G02 is clockwise, and G03 is counter-clockwise.

- ※ Geometry Profile Column: key-in the final-point coordinate of linear; cycle needs to key-in multiple center-point coordinate.
 - ⊙ **PT1 X** : starting point diameter coordinate.
 - ⊙ **PT1 Z** : starting point Z coordinate.
 - ⊙ **PT2 X** : final point diameter coordinate.
 - ⊙ **PT2 Z** : final point Z coordinate.
 - ⊙ **CENTER X** : circle center diameter coordinate.
 - ⊙ **CENTER Z** : circle center Z coordinate.

【Function Keys at Screen Bottom】

- ※ Windows scale-in : scale-in or return to original size of the Figure shape of the dialog box at screen center.
- ※ Return: switch to the main function screen.
- ※ Confirm : pressing **【LINE】** or **【CW】** or **【CCW】** key at right of the screen to display key-in data in Figure form and to display the confirm dialog box at the center of the screen.
- ※ Cancel : pressing **【LINE】** or **【CW】** or **【CCW】** key at right of the screen to cancel key-in linear motion.

【Function Keys at Right of Screen】

- ※ Line : adding a new line. Users can key-in data after pressing this key. After pressing this key, users can key-in data. Then, pressing **【OK】** key at screen bottom to finish adding new line.
- ※ CW : drawing a arch by clockwise direction. Users are able to key-in data after pressing this key. Pressing **【OK】** key at screen bottom to complete this new adding action.
- ※ CCW : drawing a arch by counter clock-wise direction. Users are able to key-in data after pressing this key. Pressing **【OK】** key at screen bottom to complete this new adding action.
- ※ Delete : delete the last line of the present profile
- ※ Zoom In : pressing the **【SCALE】** key at bottom of the screen for the “SCALE” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use **【UP】**, **【DOWN】**, **【LEFT】** and **【RIGHT】** key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the **【ENTER】** key to complete the action.
- ※ Fittest Size: pressing **【WINDOW】** at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.
- ※ Return : when pressing**【SCALE】** key at screen bottom to switch to the geometry profile screen.

◇ Screen of Geometry Grooving is as below:

ANC X	200.000	BOT R	2	2.000	FIN	0.200
ANC Z	-30.000	BOT L	0	0.000	CUT	1.000
TOP R	0	WIDTH	40.000			
TOP L	1	DEPTH	30.000			

0 : LINE · 1 : CENTER · 2 : ANGLE

V · DEF VIEW RETURN

Figure 2.7-12 Geometry Grooving Page

【Column Description】

- ※ Procedure : display the currently procedure code number
- ※ Working Method : display the working name of the currently editing program.
- ※ Tool NO. : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.
- ※ Geometry Grooving Column :
 - ⊙ **ANC X** : Reference diameter coordinate.
 - ⊙ **ANC Z** : Reference Z coordinate.
 - ⊙ **TOP R** : upper right chamfer angle condition and amount, condition (0 = right angle, 1 = rounding angle, 2 = chamfer) but, the amount cannot be smaller than the grooving-tool's tool nose radius.
 - ⊙ **TOP L** : upper left chamfer angle condition and amount, condition(0 = right angle, 1= rounding angle, 2 = chamfer) but, the amount cannot be smaller than the grooving-tool's tool nose radius.

- ⊙ **BOT R** : lower right chamfer angle condition and amount, condition (0= right angle, 1= rounding angle, 2= chamfer) but, the amount cannot be smaller than the grooving-tool's tool nose radius.
- ⊙ **BOT L** : lower left chamfer angle condition and amount, condition (0= right angle, 1= rounding angle, 2= chamfer) but, the amount cannot be smaller than the grooving-tool's tool nose radius.
- ⊙ **WIDTH** : Grooving width.
- ⊙ **DEPTH** : Grooving depth.
- ⊙ **FIN** : preservation amount when cutting.
- ⊙ **CUT** : everytime's cutting width.

【Function Keys at Screen Bottom】

- ※ Windows Scale : scale-in or return to original size of figure shape of display box at center of screen.
- ※ Preview: preview the Figure shape of the key-in data
- ※ Return: switch back to the main function screen.

【Function Keys at Right of Screen】

- ※ Scale : pressing the **【SCALE】** key at bottom of the screen for the “Zoom In” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use **【UP】**, **【DOWN】**, **【LEFT】** and **【RIGHT】** key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the **【ENTER】** key to complete the action.
- ※ Fittest Size: pressing **【WINDOW】** at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.
- ※ Return : pressing **【SCALE】** key at screen bottom to switch the screen to the geometry grooving screen.

◇ Screen of Geometry Thread is as below:

HEAD PARAM					
PROC	01	PROCESS	OSEW	TOOL NO	D01
[Empty Diagram Area]					
START	40.000	CREST	1.000	LEAD IN	1.000
START	0.000	DEPTH	1.000	LENGTH	1.000
END X	40.000	ANGLE	60.000	PITCH	1.000
END Z	-20.000	CLEAR	1.100	MINDEP	0.000
MIN DEPTH · 0 < VALUE					
V · DEF				VIEW	RETURN

Figure 2.7-13 Geometry Thread Page

【 Column Description 】

- ※ Procedure: display the currently procedure code number.
- ※ Working Method: display the working name of the currently working programs.
- ※ Tool NO. : display the selected tool NO. of the currently editing procedure. 4 types of tool NO. Type A for drilling tool, Type B for pattern tool, Type C for grooving tool, and Type D for thread tool. The number should be the tool NO. from the tool clear list of the screen.
- ※ Cutting Parameter Column :
 - ⊙ **START** : start tapping diameter coordinate
 - ⊙ **START X** : start tapping Z coordinate
 - ⊙ **END** : start tapping diameter coordinate
 - ⊙ **END Z** : end tapping Z coordinate
 - ⊙ **CREST** : thread height

- ⊙ **DEPTH** : 1st cutting depth
- ⊙ **ANGLE** : tapping angle
- ⊙ **CLEAR** : safety tool-hanging height
- ⊙ **LEADING** : leading cutting amount
- ⊙ **LENGTH** : post-chamfer angle length
- ⊙ **PITCH** : tapping distance
- ⊙ **MINDEP** : smallest lathe-cutting depth

【Function Keys at Screen Bottom】

- ※ Windows Scale : scale-in or return to original size of figure shape of display box at center of screen.
- ※ Preview : preview the Figure shape of the key-in data
- ※ Return: switch to the main function screen.

【Function Keys at Right of Screen】

- ※ Scale : pressing the **【SCALE】** key at bottom of the screen for the “Zoom In” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use **【UP】**, **【DOWN】**, **【LEFT】** and **【RIGHT】** key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the **【ENTER】** key to complete the action.
- ※ Fittest Size : pressing **【WINDOW】** at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.
- ※ Return : pressing **【SCALE】** key at screen bottom to switch the screen to the geometry thread screen.

U NOTE

- ※ Each parameter content will be different due to different working methods.
- ※ When a certain working part has set a geometry parameter, changeable types of working methods of this procedure will be limited into the same working method. Using “grooving on outer surfaces” as an example. Selecting “Procedure 1” as the “grooving on outer surfaces” working method and setting the geometry variable.

At this time, if moving the highlighter to the working method selection, users can only choose “Grooving on Outer Surfaces”, “Grooving on Inter Surfaces” and “Grooving on End Faces”. Please add a new procedure if want to change the working method. Using the same example for other three main types working methods.

- ※ Pattern cutting line must be from right to left and cannot be intersect with each other. The max. line sections are 10.
- ※ Thread lathe-cutting is not able to do facing-thread and variable-lead thread.
- ※ Dwell time of tapping cutting parameter of drilling lathe-cutting cannot be zero.

u Appendix A (Error Code Description)

◎ Table 2-1 Run Time DGNOS Output & Warning Message (R)

MSG Code	Description
0	Complete program executing
1	Not enough memory
2	Curve not block
3	Curve section over 20
4	Lathe-cutting procedure over 10
5	Total tool number over 50
6	Tool parameter setting not reasonable
7	Cutting parameter setting not reasonable
8	Not able to open file
9	Diskette is full

© Table 2-2 Tool Error Code (T)

Error Code	Message (MSG)
102	Tool number setting smaller than 0
102	Spindle CW/CCW setting error
103	Cutting angle setting error
104	Max. effective setting error of external lathe cutting
105	Max. effective setting error of internal lathe cutting
106	Internal lathe-cutting setting error
107	Grooving tool width setting error or smaller than 0
108	Tool type setting error
109	Tool setting over 50 sets
110	Open file failure

© Table 2-3 Exchange File Error Code (E)

Error Code	Message
121	Production number over range
122	G-code output type setting error
123	Diameter/radius program output setting error
124	Production tool number code assign error
125	Max. rotational speed assigning error
126	Assigned G94G95 setting error
127	No fixed value at G94 position
128	No fixed value at G95 position
129	Assigned G96G97setting error
130	No fixed value at G96 position
131	No fixed value at G97 position
132	Tool offset code smaller than 0
133	No assigned tool data in tool magazine
134	Production define error
135	Grooving profile definition (grooving width setting is less than 0)
136	Grooving profile definition (grooving setting value is less then 0)
137	Grooving profile definition (preservation is less than 0)
138	Grooving profile definition (max. cutting width is less than 0)
139	Grooving profile definition (grooving define error)
140	Thread profile definition (tapping height setting is less than 0)
141	1 st time cutting depth setting error
142	Tapping angle setting error
143	Safety height is lower than tapping height
144	Leading cutting value is smaller than 0
145	Tapping distance setting error or smaller than 0
146	Min. cutting depth setting error
147	Thread type setting error
148	Profile lathe-cutting define (G01,G02.G03 assign error)
149	G02G03 define point can't be arch
150	Drilling lathe-cutting parameter (drilling type selection error)
151	Drilling lathe-cutting parameter (feed amount setting error)
152	Drilling lathe-cutting parameter (tool escape amount setting error)
153	Rigid tapping cutting parameter (feel amount setting error)
154	Rigid tapping cutting parameter (tool escape setting error)
155	Rigid tapping cutting parameter (dwell time setting error)

Error Code	Message
156	Thread lathe-cutting parameter (feed type selection error)
157	Profile lathe-cutting parameter (stock removal type selection error)
158	Stock removal in facing (Material starting point setting error)
159	Stock removal in facing (Cutting width setting error)
160	Stock removal in facing (Preservation setting error)
161	Stock removal in facing (Tool escape amount setting error)
162	Stock removal in axis direction (Material starting point setting error)
163	Stock removal in axis direction (Cutting depth setting error)
164	Stock removal in axis direction (Preservation amount setting error)
165	Stock removal in axis direction (Tool escape amount setting error)
166	Stock removal in pattern (Material starting setting error)
167	Stock removal in pattern (Cutting depth setting error)
168	Stock removal in pattern (Preservation setting error)
169	Stock removal in pattern (Tool escape amount setting error)
170	Stock removal in pattern (Cutting times setting error)
171	Profile lathe cutting line and arch numbers over limit
172	Opening file error
173	Tool corresponding turret NO. error
174	Material Z value setting error
175	Use external lathe-cutting tool error
176	Use internal lathe-cutting tool error
177	Use external grooving cutting tool error
178	Use internal grooving cutting tool error
179	Use face-grooving tool error
180	Use internal lathe-cutting tool error
181	Use external lathe-cutting tool error
182	Use drilling lathe-cutting tool error

2.8 GRAPH (GRAPH)

2.8.1 Function Introduction

Pressing function key **【GRAPH】** to enter into the **【PROSIM】** screen. This screen is able to show the present working path or the working path of a preview program. Pressing **【CONSIM】** to set the path display angle and the display range.

2.8.2 SIMU

The path display screen is like the below Figure. Coordinate at upper right displays the absolute coordinate of the present tool and lower right display the coordinate view angle.

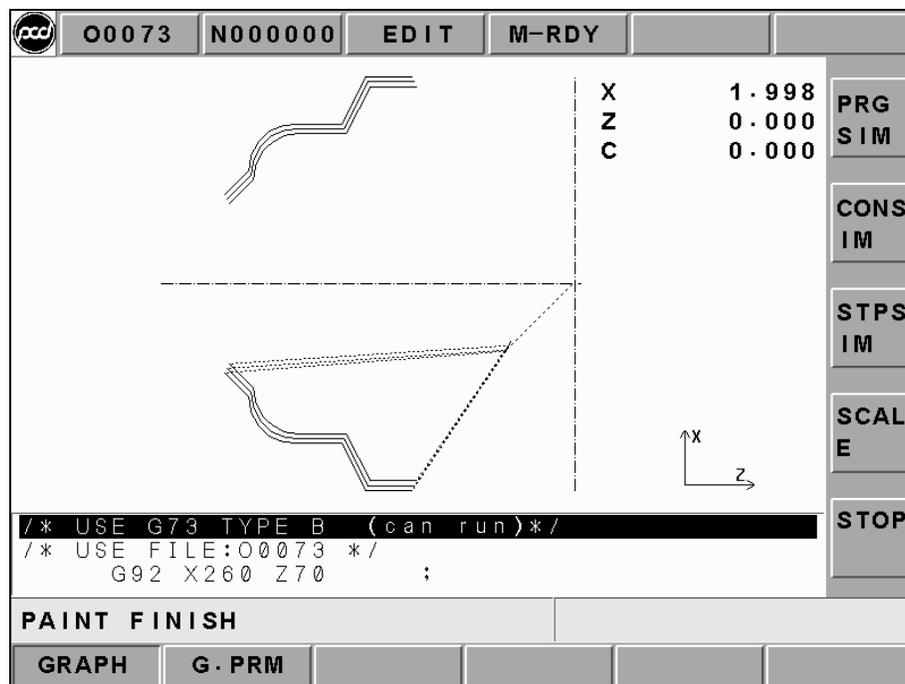


Figure 2.8-1 Path Display Screen

【PRG SIM】 : pressing this function key to draw the program setting's working path in program editing and CNC ready moods.

【CON SIM】 : pressing this function key to draw the already restored program working path again.

【STPSIM】 : pressing this key to execute the single step drawing in order to draw the downloaded program working path.

【STOP】 : pressing this function key to stop the path drawing function when executing **【PRG SIM】** or **【CON SIM】** .

【SCALE】 : a square border line will occur when pressing this function key, just like the below Figure. This borderline indicates the zoom-in range of a Figure. Using Page Up and Page Down keys to adjust the range size of the borderline. Using direction key to move the borderline position. Pressing INPUT key or **【SCALE】** key to renew the working path inside the borderline when the borderline has finished adjustment.

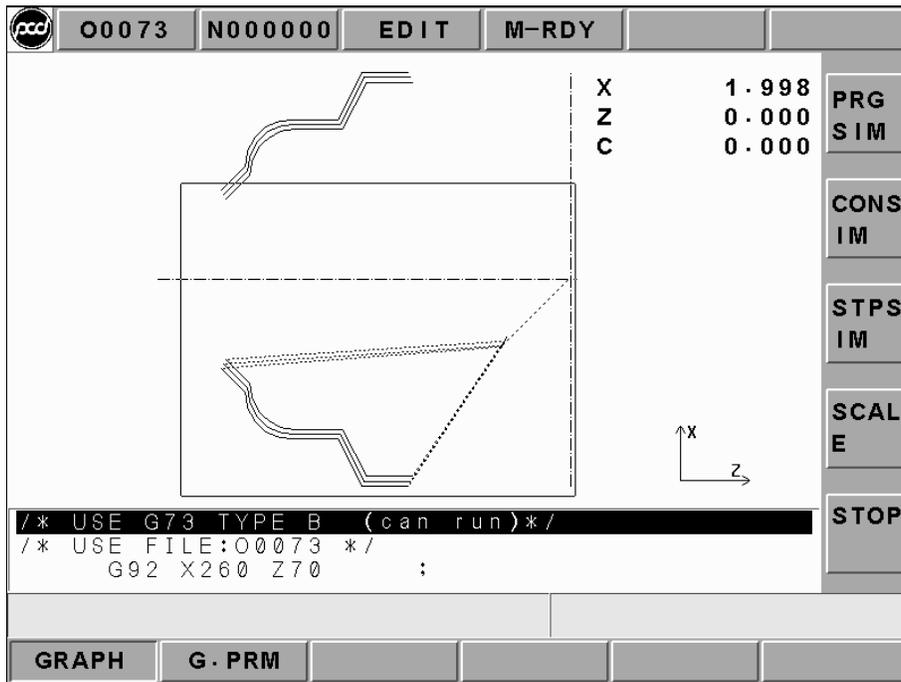


Figure 2.8-2 Before Figure Scale Out

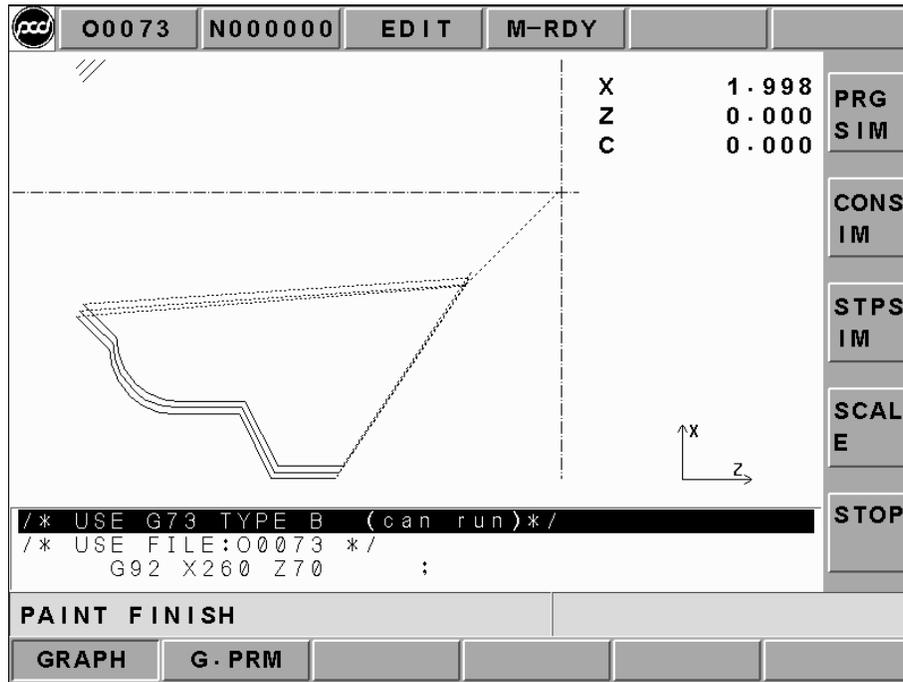


Figure 2.8-3 Before Figure Scale Out

- ※ Cursor is green when executing program, G00 is red, and G01, G02 and G03 are yellow.
- ※ Cursor is green when program preview, G00 is blue, G01, G02 and G03 are pink. The centerline is bright green.

2.8.3 Windows Definition

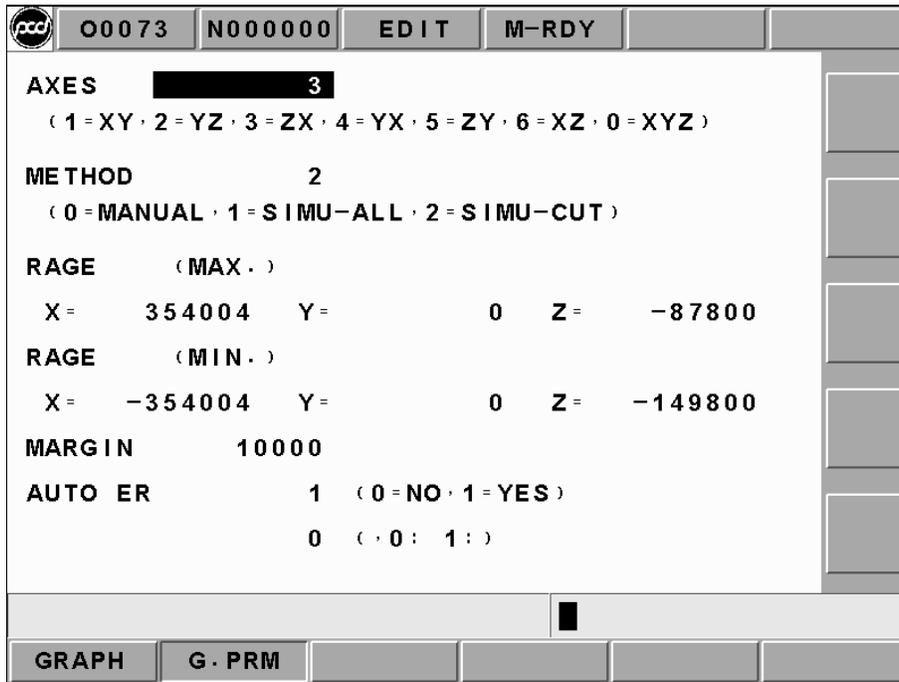


Figure 2.8-4 Screen of Window Definition

- n **Drawing Panel** : key-in the used coordinate view angle on the path display screen (1=XY , 2=YZ , 3=ZX , 4=YZ , 5=ZY , 6=XZ , 0=XYZ).
- n **Setting Method** : key-in drawing range of **[SIMU]** screen (0=MPG , 1=preview result –full , 2=preview result–cutting path) .
 - .. **0 MPG** : preview the max. or min. value, set by manual, of the drawing range.
 - .. **1 preview result–full** : preview the drawing range as the max or min value of the part program.
 - .. **2 preview result–cutting path** : preview the drawing range as the max and min value of the cutting path of the currently reading program.
- n **Drawing Range (max.)** : setting the max range of X, Y or Z axis range by manual drawing.
- n **Drawing Range (min.)** : setting the min. range of X, Y or Z axis range by manual drawing.

- n **Preserve Border** : setting preservation amount for border in 【 SIMU 】 screen.
- Auto-Delete** : under the condition of not executing program preview and doing path display, users are able to use this function to select whether or not to delete the previous path display screen when enable cutting. (0=not delete, 1=auto delete) .

2.9 DGNOS

The following 6 main function screens, **【ALARM】**, **【SYSUPD】**, **【IOCSA】**, **【MLC2】**, **【SYSTEM】** and **【CIRCUL】** will occur by pressing <DGNOS> key. On DGNOS function screen, users are able to know HMI and machine conditions in order to do the maintenance and system testing.

2.9.1 ALARM

Pressing **【ALARM】** to enter into **【ALARM】**, **【WARN】** and **【HISMSG】** sub-functions page. When any alarm or operation warning of MLC occurs, alarm or warning message will occur on the screen. Users can use this screen to prevent any irregular condition of controllers.

【ALARM】

Alarm message will occur when there is any problem in system running. This will make the system stop running and also will display the alarm message on the screen. When the problem is solved, must press <RESET> in order to release the situation.

【WARN】

Warning message is decided by matching with the design of the MLC LADDER. Example is when the safety door is not closed completely, LADDER program will send out “DOOR NOT CLOSE” message. Or, when the Coolant Supply is broken, then the message “COOLANT LAW” message will be sent out.

So when the alarm message occurs, please check the conditions of machine and peripheral equipments according to the LADDER program. The system will stop running when the system sends out the warning message and the warning message will be displayed on the screen. When the problem is solved, must press <RESET> to release the warning.

【HISMSG】

Able to display the version and program debug message after entering into the message screen. Message displaying will not disconnect the working procedure, but it displays the current running condition of the system.

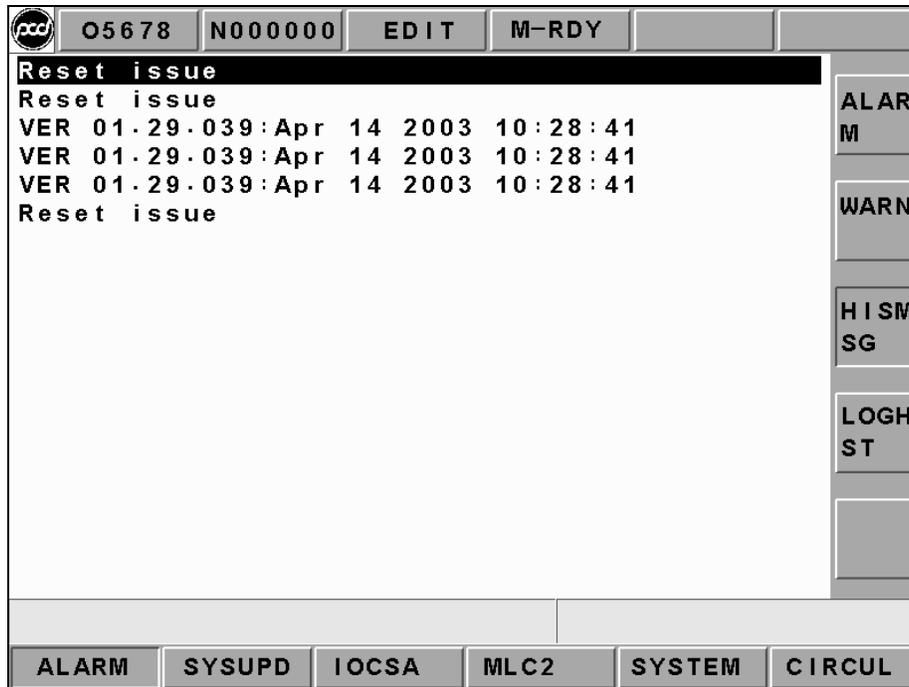


Figure 2.9-1 ALARM\ALARM

2.9.2 System Upgrade

This function can be executed only under the condition that CNC is NOT ready. Please pressing **【UPDT】** button after pressing the EMG-STOP button. At this time, a screen which function is selected by the cursor (like Figure 2.9-2) will occur. This screen allows users to select the wanted working item. Each function is listed as below:

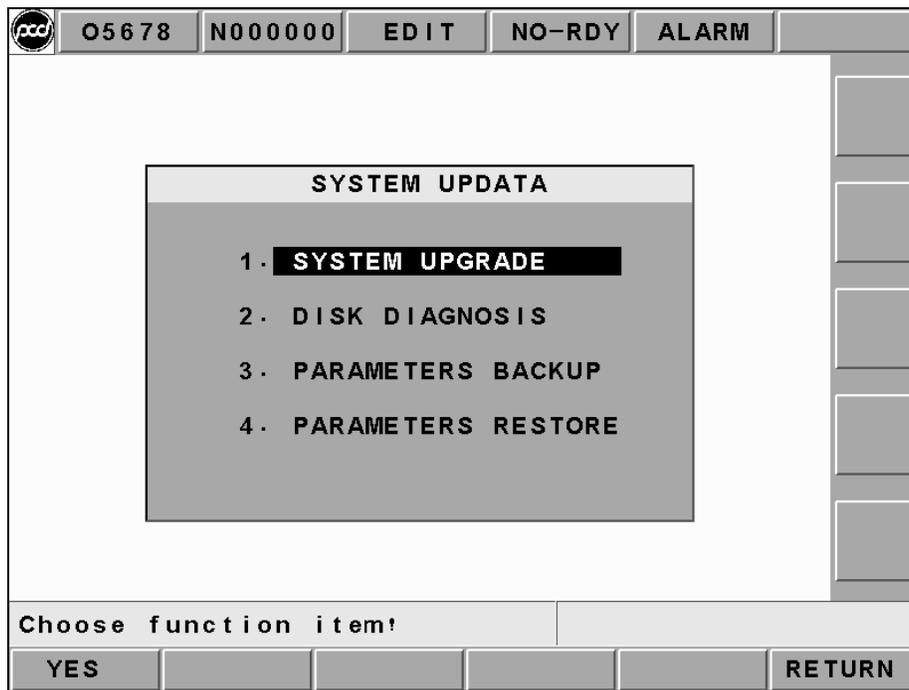


Figure 2.9-2 System Upgrade Screen

1. System Upgrade:

Please install LNC Technology Co., Ltd. newest version software if choosing this function. Pressing **【YES】**, the installation screen will occur. So, users only need to follow the instruction to upgrade the system.

2. Disk Diagnosis :

Selecting this function, a confirmation dialog box will occur (like Figure 2.9-3). Pressing the confirm button to return back to DOS mode (like Figure 2.9-4). Users are able to choose to use A drive or C drive. If users do not choose, the system will use the default C drive. There are 4 types of working items for users to choose:

(1) anti-virus, (2) hard drive scanning, (3) hard drive reset, and (4) exit

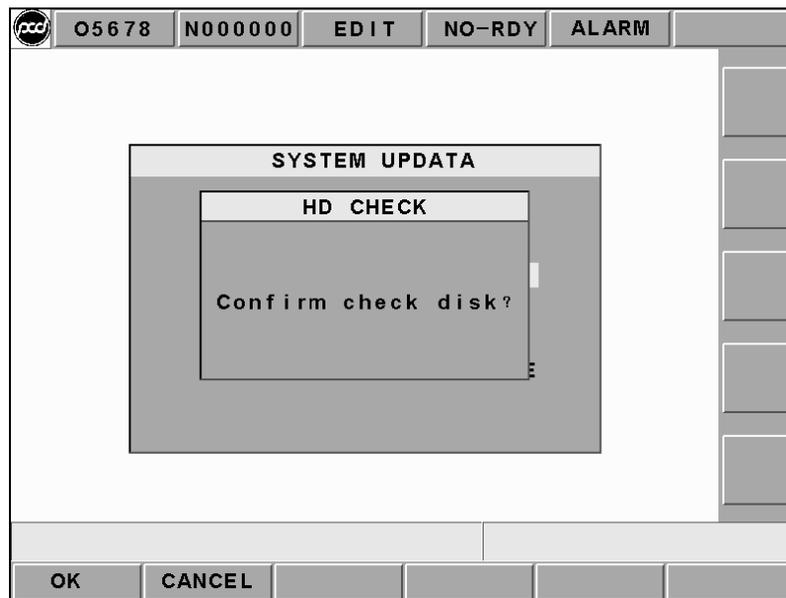


Figure 2.9-3 Confirm Whether or Not to Do the Hardware Checking

```
      Welcome maintenance utility ...  
      What DRIVE to DIAG[A,C]?C  
  
      1.VTSCAN V9.3  
        Virus checking  
      2.DISK doctor  
        disk diagnostic and errer-fix  
      3.DEFRAG  
        Disk access performance enhancement  
      4.Quit  
  
      Choise an Option[1,2,3,4]?
```

Figure 2.9-4 Hardware Checking Function Selection

3. PARAM BACKUP (Parameter Backup)

The below dialog box will occur when choosing this function. Users are able to choose the wanted backup parameter items. Users are able to key-in the dialog box of the backup copy path after pressing the confirm button. Pressing the confirm button after users key-in or select the wanted backup parameter to complete the parameter backup function.

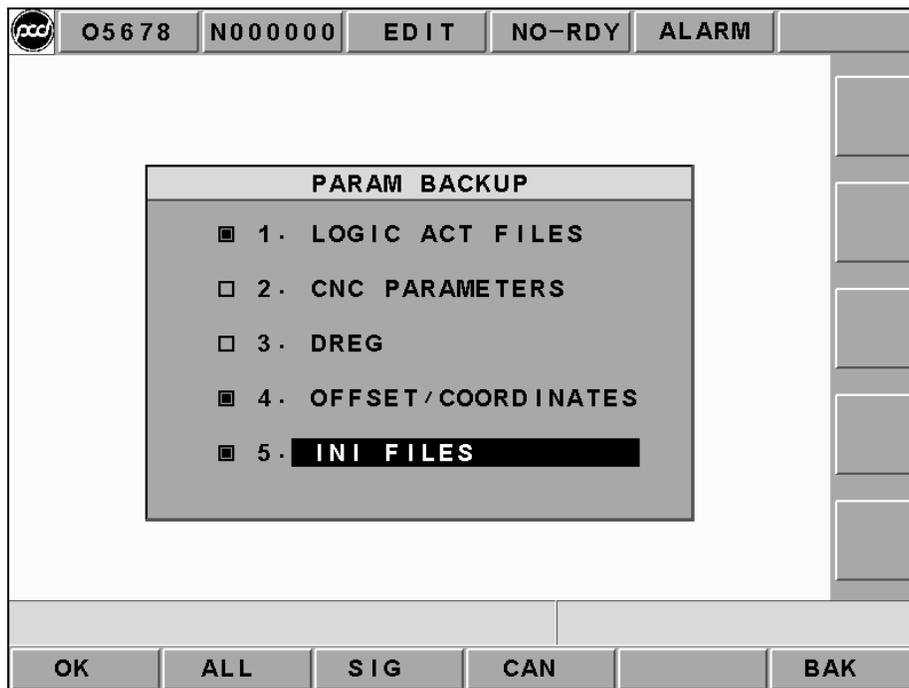


Figure 2.9-5 Parameter Backup Selection Screen

4. Parameter Restore :

Selecting this function, a menu will popup to remind users that the program window will be restart after executing this function (Figure2.9-6). Selecting the “OFFSET/COORDINATE” item from the popup dialog box after pressing confirm button (Figure 2.9-7). A dialog box (Figure 2.9-8) of key-in parameter renew source path occurs after pressing confirm button. Key-in or select the source path and file name after pressing confirm button to key-in parameter file.



Figure 2.9-6 Reminding Users to Reboot after Key-In Parameter



Figure 2.9-7 Screen of PARAM RESTORE

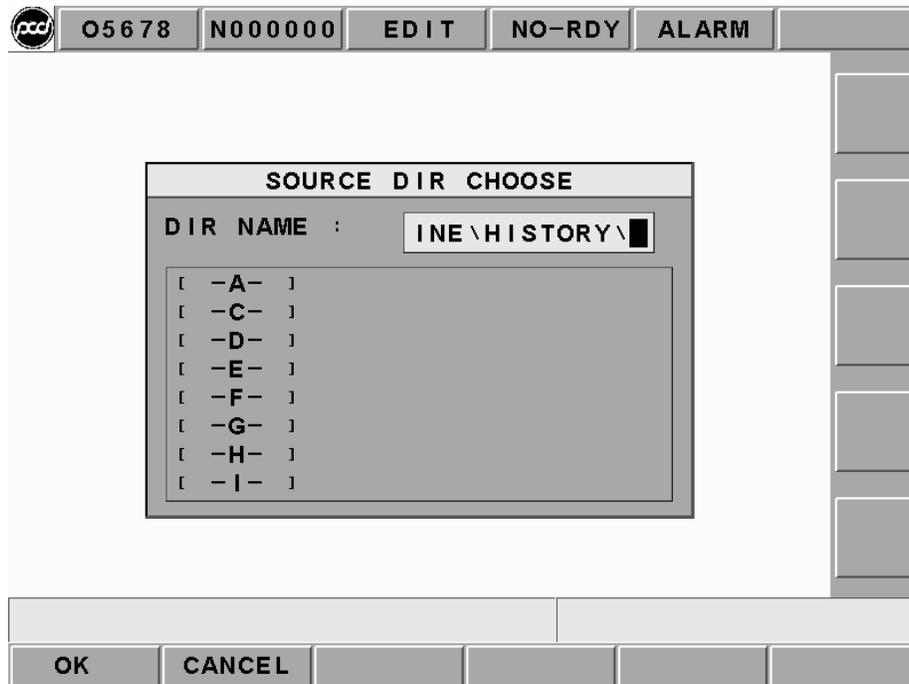


Figure 2.9-8 Screen of Source DIR Choose

2.9.3 IOCSA

【IOCSA】 is to check the screen of the I/O or system internal condition which including I, O, C, S, A.

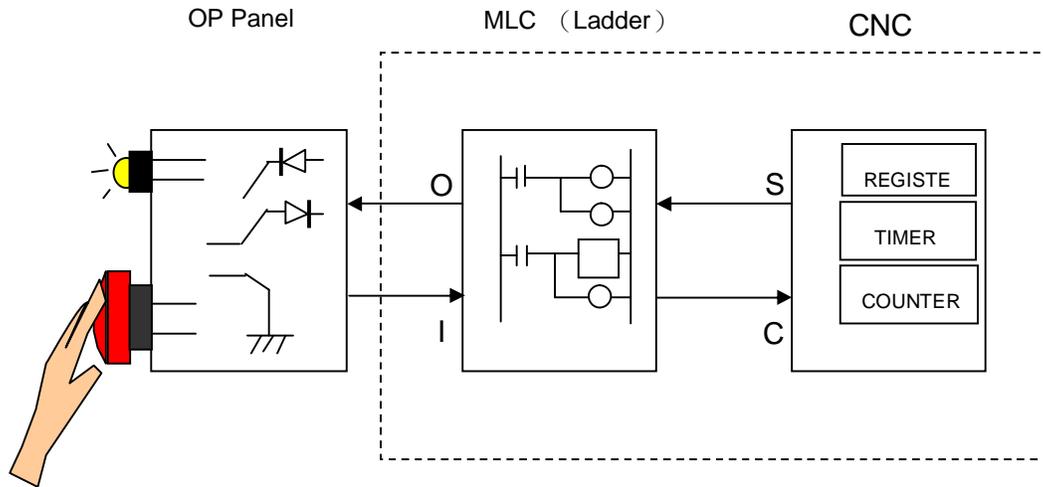


Figure 2.9-9 I/O or Internal Condition

O5678		N000000		JOG	M-RDY						
I	000	1	1	1	1	1	1	1	1	I BIT	
	010	1	1	1	1	1	1	1	1		
	020	1	1	1	1	1	1	1	1		
	030	1	1	1	1	1	1	1	1		
	040	1	1	1	1	1	1	1	1	O BIT	
	050	1	1	1	1	1	1	1	1		
	060	1	1	1	1	1	1	1	1		
	070	1	1	1	1	1	1	1	1		
O	000	0	0	0	0	0	0	0	1	C BIT	
	010	0	0	0	0	0	0	0	0		
	020	0	1	1	1	1	0	1	0		0
	030	0	1	0	0	0	0	0	0		0
	040	0	0	0	0	0	1	0	0	1	S BIT
	050	0	0	0	0	0	1	0	0	1	
	060	0	0	0	0	0	0	0	0	0	
	070	0	0	0	0	0	0	0	0	0	
C	000	1	1	0	0	0	0	0	0	A BIT	
	010	0	0	0	0	0	0	0	0		
	020	0	0	0	0	0	0	0	0		0
	030	0	0	0	0	0	0	0	0		0
	040	1	0	0	0	0	0	0	0	0	
	000	0	0	0	0	0	1	0	0	0	S BIT
	010	1	0	0	0	0	0	0	0	0	
	020	0	0	0	0	0	0	0	0	0	
030	1	0	0	1	0	0	0	0	0		
040	1	0	0	0	0	0	0	0	0		
000	1	0	0	0	0	1	0	0	0	A BIT	
010	0	0	0	0	0	0	0	0	0		
020	0	0	0	0	0	0	0	0	0		
030	0	0	0	0	0	0	0	0	0		
040	0	0	0	0	0	0	0	0	0		
ALARM		SYSUPD		IOCSA	MLC2	SYSTEM	CIRCUL				

Figure 2.9-10 IOCSA Screen

This screen is divided into 5 parts. The assigned method of the focus point is to assign via sub-function button. The assigned type is displayed on the upper left column of the screen (Using the above Figure as an example. Pressing the sub-function button **I BIT**, the type will be displayed at the upper left column of the screen.); < PAGE ↑ > and < PAGE ↓ > keys are used to control the page changing of this **I BIT** sub-function.

2.9.4 MLC2

Pressing **【MLC2】** and the main function key screen is as below. Users will see **【LAD】**, **【CNT】**, **【REG】**, **【DRG】**, and **【TMR】** functions.

Pressing **【LAD】** to see the below figure.

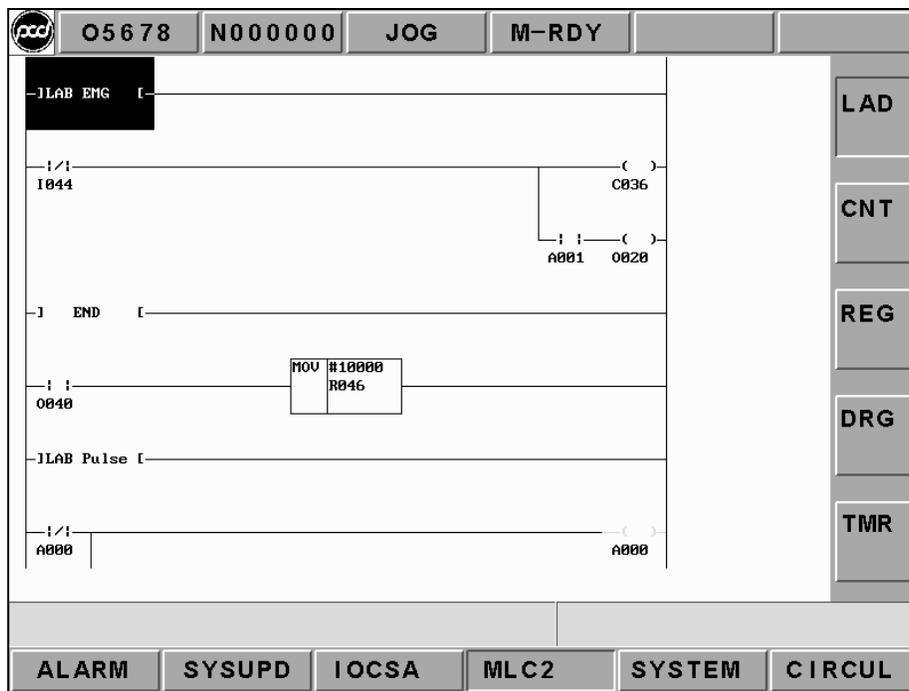


Figure 2.9-11 MLC2\LAD

Users are able to add number on the wanted searching characters, such as I, O, C, S, A, R, TM and est. in **【LAD】** entering row. Pressing **【INPUT】** to start searching the position of those characters. For example: key-in TM001 or TM1 to search for the position.

Pressing **【CNT】** to see the below screen:

O5678		N000000		JOG	M-RDY	WARN						
NUM	CUR	·	VASET	·	VAL	NUM	CUR	·	VASET	·	VAL	LAD
000	0	0	0	016	0	0						
001	0	0	0	017	0	0						
002	0	0	0	018	0	0						CNT
003	0	0	0	019	0	0						
004	0	0	0	020	0	0						
005	0	0	0	021	0	0						
006	0	0	0	022	0	0						REG
007	0	0	0	023	0	0						
008	0	0	0	024	0	0						
009	0	0	0	025	0	0						
010	0	0	0	026	0	0						DRG
011	0	0	0	027	0	0						
012	0	0	0	028	0	0						
013	0	0	0	029	0	0						
014	0	0	0	030	0	0						TMR
015	0	0	0	031	0	0						

ALARM	SYSUPD	IOCSA	MLC2	SYSTEM	CIRCUL
-------	--------	-------	------	--------	--------

Figure 2.9-12 MLC2\CNT

Pressing **[REG]** to see the below screen:

NUM VALUE		NUM VALUE		NUM VALUE		
000	0	016	0	032	0	LAD
001	0	017	0	033	0	CNT
002	0	018	1	034	0	
003	0	019	0	035	0	REG
004	0	020	0	036	0	
005	0	021	0	037	0	DRG
006	0	022	0	038	0	
007	0	023	0	039	0	TMR
008	0	024	0	040	676	
009	0	025	0	041	0	
010	0	026	0	042	0	
011	0	027	0	043	0	
012	0	028	0	044	0	
013	4	029	0	045	0	
014	0	030	0	046	0	
015	0	031	0	047	0	

Figure 2.9-13 MLC2\REG

Pressing **[DRG]** to see the below screen:

O5678		N000000		JOG	M-RDY		
NUM	VALUE	NUM	VALUE	NUM	VALUE		LAD
000	20021	016	0	032	16		
001	0	017	0	033	1		CNT
002	0	018	0	034	1		
003	0	019	0	035	10		
004	0	020	0	036	100		
005	3	021	0	037	2		REG
006	0	022	0	038	1		
007	0	023	0	039	1		
008	0	024	0	040	1		
009	0	025	0	041	10		DRG
010	0	026	0	042	1		
011	0	027	0	043	0		
012	0	028	0	044	0		TMR
013	0	029	0	045	0		
014	0	030	0	046	0		
015	0	031	1	047	0		

ALARM	SYSUPD	IOCSA	MLC2	SYSTEM	CIRCUL
-------	--------	-------	------	--------	--------

Figure 2.9-14 MLC2\DRG

Pressing **【TMR】** to see the below screen:

O5678		N000000		JOG	M-RDY		
NUM	CUR	VASET	VAL	NUM	CUR	VASET	VAL
000	0	0	0	016	0	0	
001	0	0	0	017	0	0	LAD
002	0	0	0	018	0	0	
003	0	5	5	019	0	5	CNT
004	0	5	5	020	0	0	
005	0	5	5	021	0	0	
006	0	2	2	022	0	0	REG
007	0	2	2	023	0	0	
008	0	60	60	024	0	0	
009	0	1	1	025	0	0	DRG
010	0	0	0	026	0	0	
011	0	0	0	027	0	0	
012	0	2	2	028	0	0	
013	0	0	0	029	0	0	TMR
014	0	10	10	030	0	0	
015	0	0	0	031	0	0	

Figure 2.9-15 MLC2\TMR

2.9.5 SYSTEM

Pressing **[GBL]** to see the below screen. (System Data is the system protecting variable screen. This function is provided for designers and/or technical person to use.)

O5678		N000000		JOG	M-RDY		
NUM	VALUE	NUM	VALUE	NUM	VALUE	GBL	
000	0	016	0	032	0	GBL	
001	0	017	0	033	0	H·D	
002	0	018	0	034	0	H·D	
003	0	019	0	035	0	H·D	
004	0	020	0	036	0	H·D	
005	0	021	0	037	0	H·D	
006	0	022	0	038	0	H·D	
007	0	023	0	039	0	H·D	
008	0	024	0	040	0	H·D	
009	0	025	0	041	0	H·D	
010	0	026	0	042	0	H·D	
011		027		043		H·D	
012		028		044		H·D	
013		029		045		H·D	
014		030		046		H·D	
015		031		047		H·D	

Figure 2.9-16 System\GBL

Table 2-4

Code	Definition	Code	Definition
0	X-axis servo lag counting value	16	X-axis hand wheel command counting value
1	Y-axis servo lag counting value	17	Y-axis hand wheel command counting value
2	Z-axis servo lag counting value	18	Z-axis hand wheel command counting value
3	4th-axis servo lag counting value	19	C-axis hand wheel command counting value
4	Waiting MST to ready signal	20	BLOCK times
5	In Position Checking	32	Actual X-axis machine coordinate value
6	Actual spindle rotational speed	33	Actual Y-axis machine coordinate value
7	Spindle servo lag counting value	34	Actual Z-axis machine coordinate value
8	Spindle commanding value	35	Actual 4th-axis machine coordinate value
9	Angle of spindle orientation	36	Index ISR Counter
		38	ISR Counter
10	Spindle orientation arrival range PLUSE	39	Distance between present Spindle position to index

【H.D】

There are 9 items in the diagnosis function list. These function items are to check whether or not the connection cable from motion card to I/O card is connected. Or, checking whether or not the Jump position is correct. If the question mark (?) changes to cross (X) after diagnosis on the right windows side, it indicates this item has error.

The HOME DOG condition at the top of the windows indicates that if the value of each axis is 1, the present position of each axis is on the HOME DOG.

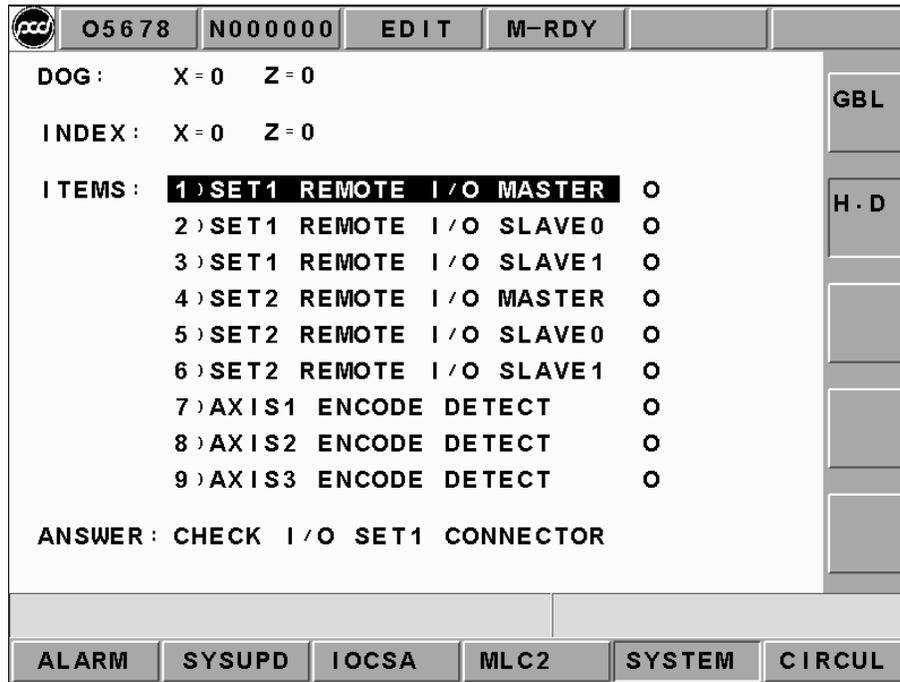


Figure 2.9-17 MLC2\H.D

2.10 SOFTPL

Pressing <SOFTPL> to enter into the Figure Display Screen. This screen provides a configurable SoftKey definition. There are 7 function switches like the below Figure. This design purpose is to extend buttons for hardware OP panel.

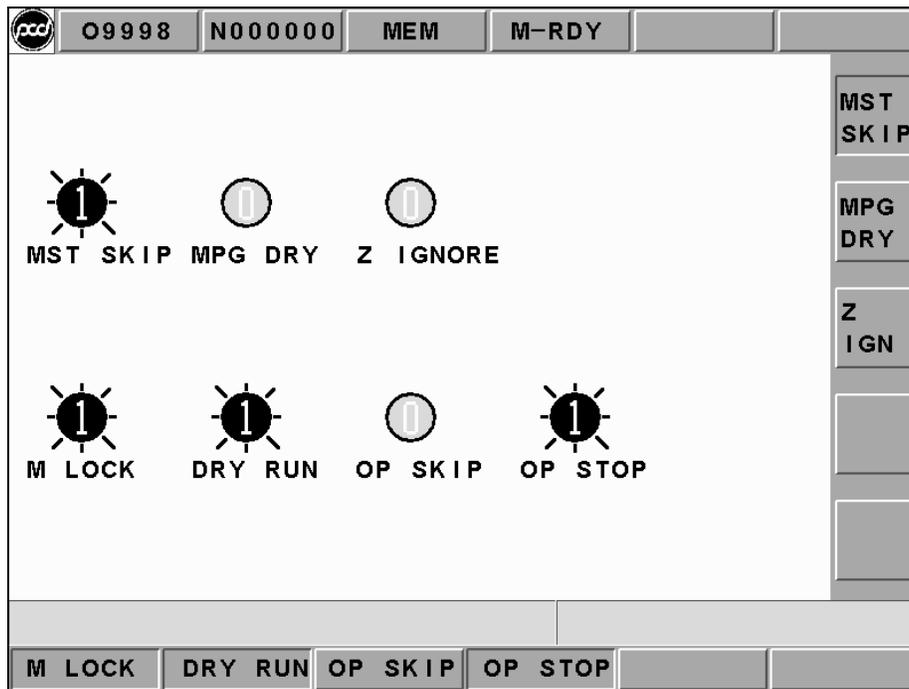


Figure 2.10-1 SOFTPL Screen

Pressing the corresponding function keys separately, the function Figures on the screen will change from red to green. When the button changes to green, it means the function is enabled. Pressing the button again to disable the function.

The below table will describe each function selection switches on the screen:

Table 2-5

Function Switches	ON	OFF
MPG Dry Run	Hand wheel can modify the program coordinate, but the servo axis will not move actually.	Hand wheel can modify the program coordinate and let the servo axis moves according to it.
Optional Jump	Program begins with "/" symbol will be skipped and not be executed.	Single block that begins with "/" symbol will be executed normally.
Optional Stop	The program will stop when the program executes up to M01. If want to continue executing, users need to press "CYCLE START".	Controller will ignore M01 and execute the next single block directly.
Dry Run	While executing the program, the servo feed is according to the dry run speed parameter (G00), not according to the assigned speed of the program.	The assigned speed of the servo feed program.
MST Ignore	Ignore M, S, T codes commanding	Consider M, S, T codes commanding
Machine Lock	The controller will continue executing program while the program is executing. But, the motion command of the servo axis will NOT be outputted. So actually, the servo axis is completed stop (no movement).	The motion command of the servo axis will be outputted with the program while the program is executing.

2.11 PARAM

Pressing **<PARAM>** and the parameter setting screen occurs. There are three main function menus which are **【NC. SYS】** , **【SUROPT】** or **【LNCS】** .

2.11.1 NC. SYS (System Parameter)

Pressing **【NC. SYS】** , screen of each parameter setting will occur. There are eight sub-function menus which are **【SERVO】** , **【MAC】** , **【SPDL】** , **【MPG】** , **【COMP】** , **【HOME】** , **【OPER】** and **【OTR】** .

O9998		N000000		MEM	M-RDY		
0001	30	0013	230	0053	4	SRVO	
0002	30000	0014	100	0054	4		
0003	30000	0022	1	0055	4	MAC	
0004	30000	0024	1	0056	4		
0005	30000	0025	2	0845	0		
0006	50	0026	3	0846	0	SPDL	
0007	50	0027	0	0847	0		
0008	50	0040	10	0066	0		
0009	50	1112	2500	0085	0	MPG	
0010	230	1113	2500	0108	200		
0011	230	1114	2500	0109	200		
0012	230	1115	2500	0110	200	PGDN	
-- PAGE : 1 / 2 --							
SYSTEM LOOP GAINS FOR V CMD							
NC. SYS	USROPT	LNCS					

Figure 2.11-1 System Parameter--1

	O9998	N000000	MEM	M-RDY		
0038	16	1018	30000	0304	0	COMP
0044	0	1019	30000	0305	0	
0045	0	1020	30000	0306	0	HOME
0046	0	1021	30000	0307	0	
0047	0	1046	0	0308	0	
0112	20	1047	0	0309	0	OPER
0113	20	1048	0	0310	0	
0114	20	1049	0	0311	0	
0115	20	0300	0	0312	0	OTR
0117	0	0301	0	0313	0	
0118	0	0302	0	0314	0	
0119	0	0303	0	0315	0	PGUP
-- PAGE : 1 / 7 --						
BACKLASH / PITCH COMP UNITS						
NC-SYS	USROPT	LNCS				

Figure 2.11-2 System Parameter-2

In parameter function group, only able to modify parameters under MDI mode. Users must reboot after complete modification in order for the modified parameter to be effective. The modification parameter method is to key-in Pxxx first. Then, after pressing <INPUT>, it will switch to the page of the parameter # xxxx.

At this time, the description of the cursor parameter will occur at lower bottom of the message reminding area. Then key-in the wanted modifying parameter value directly in the key-in area and pressing <INPUT> to modify the parameter. After all modifications are completed, please reboot again in order for the new parameter setting to be effective.

In parameter code function, since CNC parameter will decide operation of the CNC, so must be extra careful of parameter adjustment and modification and also each time's calculation. Users must be fully understanding the parameter's definition first, then start executing modification. In order to prevent the situation that parameter is modified by accident, users need to key-in the pass codes during parameter setting.

The new setting will be accepted by CNC only if users key-in the correct passing codes. Once the passing codes are entered correctly, the system will not remind users to give the passing codes for later parameter entering. So users will be able to do the parameter modification continuously. CNC system will remind users to give the passing codes when rebooting and when modifying parameters next time.

2.11.2 Users Parameter

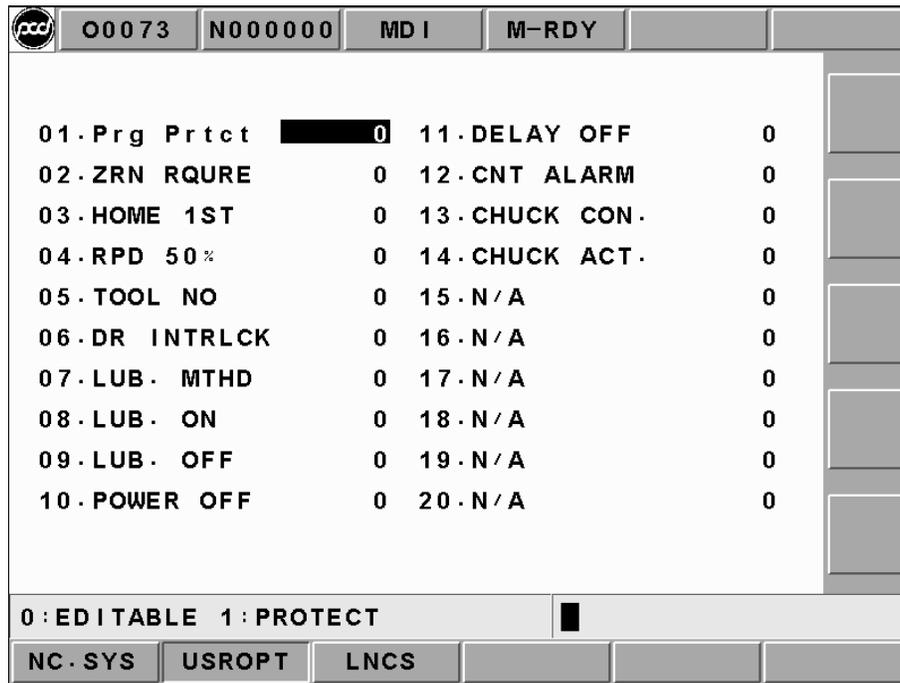


Figure 2.11-3 Users Parameter

In parameter function group, users only able to modify parameter under MDI mode. Must rebooting after modifying in order for the modified parameter to be effective. The method of modifying parameter is the same as system parameter. Please refer to the previous chapter if there is any question.

	O9998	N000000	MEM	M-RDY		
DESC.: POU-YUEN TECH CORP.LTD						
DATE:		2003/06/09	TIME:		19:15:18	
MKR.:		2003/01/06	ACC.:		000065:43	
LOTS 0						
NO.	DEADLINECHK	NO.	DEADLINECHK			
1	----/--/-- -	6	----/--/-- -			
2	----/--/-- -	7	----/--/-- -			
3	----/--/-- -	8	----/--/-- -			
4	----/--/-- -	9	----/--/-- -			
5	----/--/-- -	10	----/--/-- -			
PRIVILEGE MODE				<input checked="" type="checkbox"/>		
		PWD.	LOCK		RETURN	

Figure 2.11-5 Authorization Setting Mode

This page provides a machine structure that machine makers/dealers can trace and restrict customer's use for this machine.

There are two modes: one is [Lock Mode] which is to let END-USER to release term restriction. The other one is [Authorization Setting Mode] which is to provide the term authorization requirements for machine makers and/or dealers. The descriptions for these two functions are introduced and described as below:

n Lock Mode

System will check the expiry date of each term everytime rebooting. The checking method is as below:

- whether or not the present date of the system is greater than the expiry date.
- whether or not the sum of the out-of-factory date + using days is greater than the expiry day.

If one of the above conditions is met, then the system will disable the CYCLE START function due to the expiry date is created. At this time, users need to move the cursor to the term column. Also, pressing the [Unlock Code] key and key-in this terms' passing codes (permit from the agent).

If the passing codes are correct, the CNC system will unlock this term's lock. At the same time, 'Y' will occur on the [CHK] column (**Note:** please enable the system again after unlocking in order to release "CYCLE START" restriction.)

Also, the system allows users to modify [System Date] and [System Time]. The modification method is to move the cursor to the date column and to key-in the correction date format. For example: using [2002/7/11] to do the date setting. By using the same principle, users also need to move the cursor to the time column if want to set the current system time. Using hour:min:sec: format, i.e., 11:40:50 (like Figure 2.11-6 as show), in order to complete time setting.

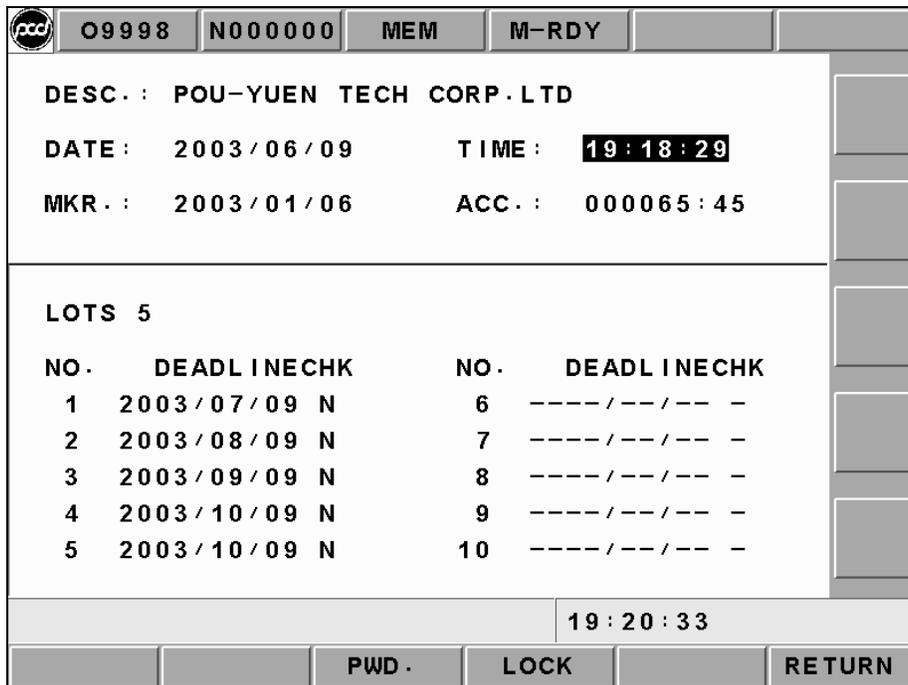


Figure 2.11-6 Screen of Locking Mode

Since this system allows users to set the using system date setting, so there is a need to calculate the actual [Using Numbers] in order to get the number of already using days of this system. (Please refer the above Figure 000000:00 column, the six digits at the left of the colon means hour and the two digits at the right of the colon means minutes.) The calculation number will be used to determine whether or not the system time limit is reached.

n Authorization Setting Mode

Different from [Lock Mode]; [Authorization Setting Mode] is use to set any requirement under locking mode. The method is to press [Setting Complete] key and then to key-in special passing codes. If the passing codes are correct, users are able to enter into the authorization setting mode. At the same time, the system will allow the data item in the setting (Please refer to Figure2.11-7) .

- .. Machine Description
- .. Expire days
- .. Expire date of each term
- .. Passing codes of each term

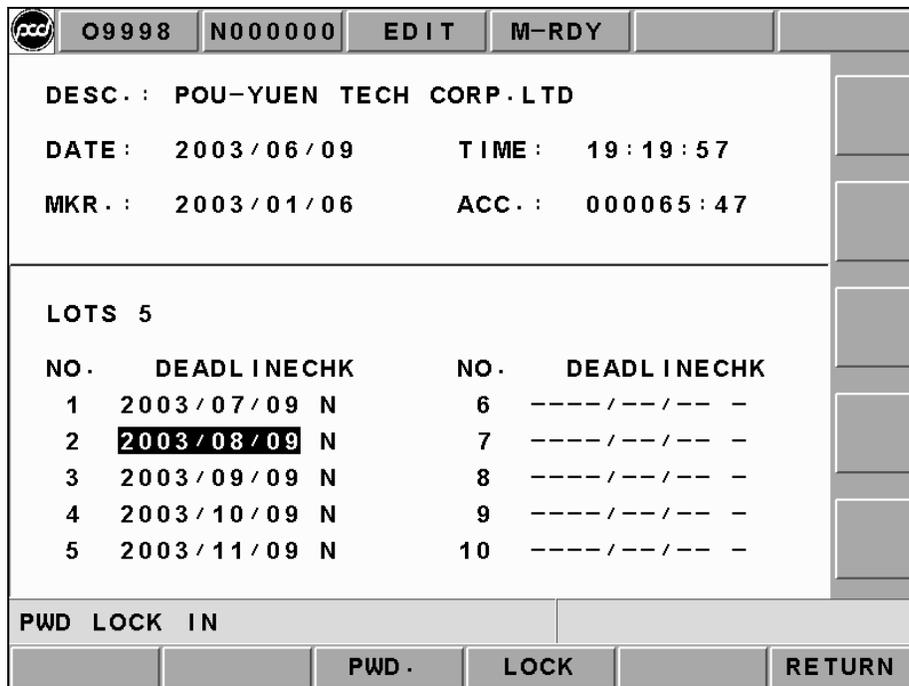


Figure 2.11-7 Screen of Authorization Setting Mode

Machine Description: this column provides the description characters for machines (max: 30 English/number characters). Only need to move the cursor to the column and key-in machine description.

Term #: to assign the term # of the term checking (max term number is 10).

Note: If this value has any irregular moving, the original checking condition will set to “N” (not passing).

Expiry Date of Each Term: to assign the expiry date of each term. Only need to move the cursor to the column to key-in the characters for setting.

Passing Codes of Each Term: to assign passing codes when the system has noticed a certain expiry date. Users need to press [Unlock] key and key-in the passing codes. Only need to move the cursor to the column and to press [Codes] to key-in the passing codes in order to complete setting. When finish setting, must press the [Setting Complete] key (like the above Figure) and enter the locking codes in order to do ‘Key-in Setting’ confirmation. At the same time, returning back to [Lock Mode]

Note: Assumed this set of passing codes is keyed-in. So only need to key-in the passing codes again when entering into this mode next time. On the other hand, if users do not want to change the previous passing codes, please press ENTER.)

2.12 RESET

Using the <RESET> button in the following 3 conditions:

- a. When alarm system is activated, must solve the problem according to it. After the condition is release, pressing <RESET> key in order to let the system to work normally.
- b. After editing programs, the cursor stops at somewhere of the program. Using <RESET> in order to move the cursor back to the beginning of the program. Otherwise, the program will start executing at the position where the cursor occurs.
- c. Cancel the executing motion. If pressing <RESET> during executing program, the machine will stop its motion. Also, the cursor will return back to the beginning of the program and returning back to “CNC Ready” condition.

3 OP Panel Operation

The design of the operation panel and the key arrangements are maker-specific, each with its own features. Only the commonly used function keys are introduced here.

3.1 Operation Panel

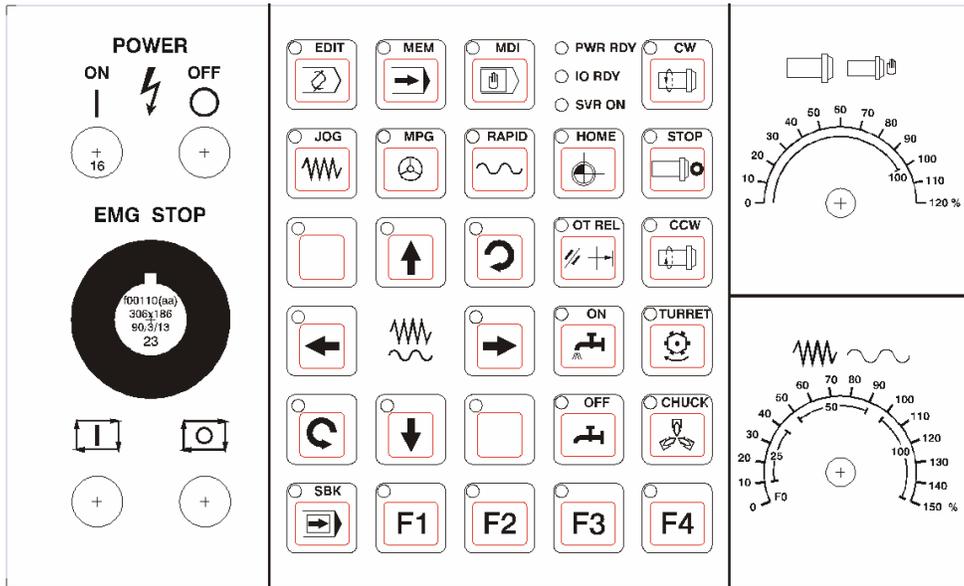


Figure 3.1-1 Operation Panel

3.1.1 Power Switch (ON/OFF)

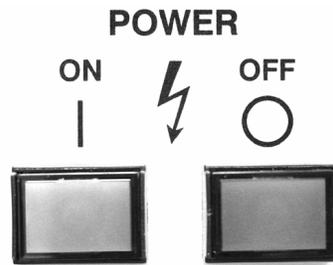


Figure 3.1-2

When users press “ON” button on power switch, the CNC controller’s power will be enable. On the other hand, when users press “OFF” button, must wait for a while until the servo is completely release electricity and then the CNC controller’s power will be disable.

3.1.2 EMG-STOP Key

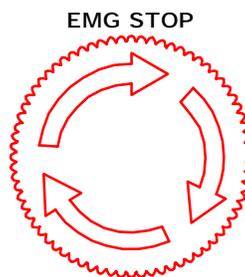


Figure 3.1-3

Pressing this button under the condition of danger or emergency and all motions will stop. To cancel is to turn the knob by following the arrow direction. When the knob jumps up automatically, the emergency stop is released. When the knob is pressed, the system is in “Not Ready” condition (condition column will occur Not Ready). In order to reach completed safety, the feed driver power in the power cabinet will be disconnected. Before releasing emergency stop, please ensure whether or not the broken source is excluded. Please executing home return procedure after the emergency stop is released in order to ensure the accuracy of the coordinate position.

3.1.3 CYCLE START

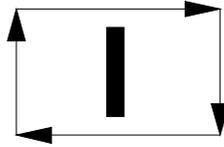


Figure 3.1-4

After key-in program, switching the operation mode to MEM or MDI mode and pressing “CYCLE START” key to executing programs. While the program is executed, its light will be ON. The timings of using “CYCLE START” key are the following:

a. Auto-Executing in MEM Mode:

When a program is selected in MEM mode, pressing “CYCLE START” key to execute program. The light will remain ON during program executing time until the program execution is finished.

Before executing program, the three axes must return to the reference point. If not returning back to the reference point, users can switch the Reference Need to OFF on the Users Define screen. By doing this, program will be able to execute without returning back to the reference point.

b. Auto-Executing in MDI Mode:

Users can key in single block program commanding such as G91 G01 X100. Z100 in MDI mode. Then pressing “CYCLE START” key to execute this single block command. The intension of this executing mode is different than that of MEM mode, which is always used in testing some motion. The light will remain ON during executing time until the executing is finished.

3.1.4 FEED HOLD

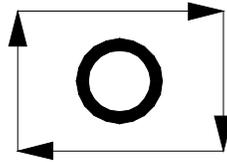


Figure 3.1-5

Pressing this key will stop program executing temporarily. During the pause time, FEED HOLD light will be ON. M, S and T will remain the current condition. Although the feed is stopped, auxiliary function (M), spindle function (S), and tool function (T) will remain in the current status. Please press the “CYCLE START” key in order to continue executing the unfinished program.

3.1.5 LED SIGNAL

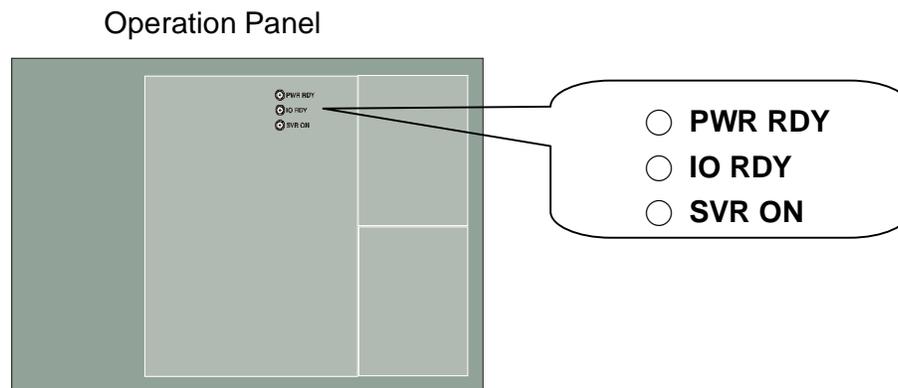


Figure 3.1-6

PWR RDY : this light will be ON to ensure the power is ON.

IO RDY : this light will be ON after self-hardware connection testing is confirmed.

SVR ON : this light will be ON automatically after confirm servo system works normally.

3.1.6 MODE SELECT

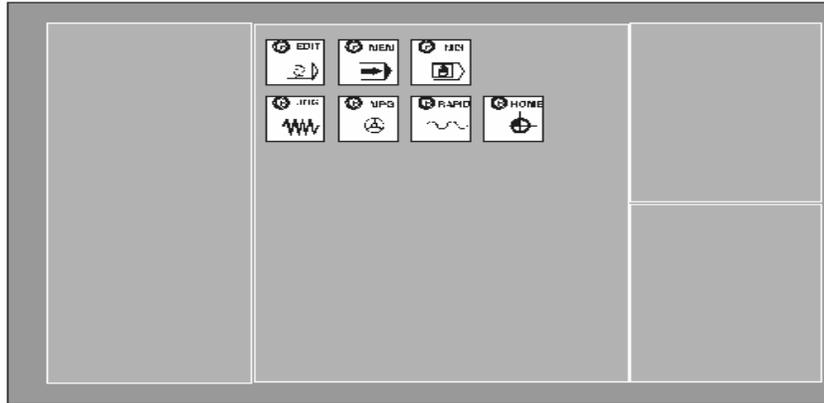


Figure 3.1-7

There are 7 modes on this operation panel, which are: MDI, MEM, EDIT, JOG, RAPID, HOME and MPG.

(1) **EDIT**

Users can edit new program or modify old one.

(2) **MEM**

Users can do program auto executing.

(3) **MDI**

Users can execute single block program, modify parameters and setting data.

(4) **JOG**

Users can move axis by choosing moving direction, but the moving speed is decided by feed speed.

(5) **MPG**

Users can use MPG to control the feed of servo axis. All hand wheel control panels provide ratio selection switches, such as 1 ratio, 10 ratio, 100 ratio, and unit means the smallest commanding unit (0.001 mm or 0.0001 inch). Axis direction selection keys are used on control panel.

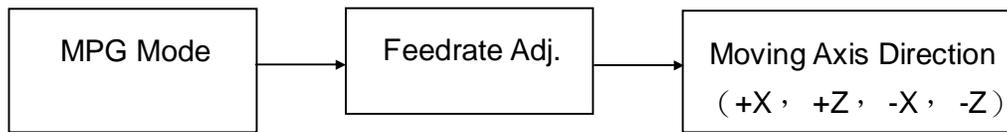


Figure 3.1-8

(6) RAPID

Users can choose axis-moving direction to move axis under this mode. The rapid speed button decides moving speed.

(7) HOME

Operating home return of each axis. When changing to this mode, pressing home return direction key of each axis (the same key as JOG). This axis will start home return procedure according to the setting speed of the parameter until reaching DOG.

Also, servo axis starts searching for the home position and it will stop when reaching home. At this time, this axis +ive direction light will be ON. Also, when users switches to HOME, this light will be ON in order to remind the machine to complete HOME RETURN motion. Please doing the Home Return procedure before doing other part program everytime reboot the machine in order to ensure each coordinate's accuracy.

3.1.7 Feedrate/Rapid Speed Adjusting Button

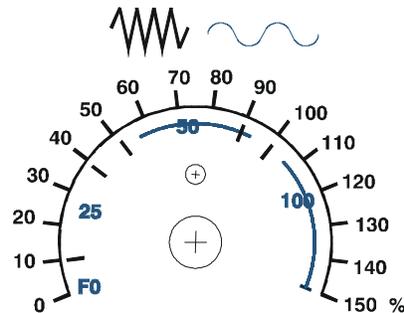


Figure 3.1-9

This turning knob is used in both feedrate/rapid speed adjustments:

(1) **FEEDRATE :**

When making servo axis feeds by G01...F...command in MEM or MDI mode, the actual feed speed can be adjusted period by period, between 0% to 150%, by this switch. For example, the designated F100 means the feed speed is 100mm/min. But, if adjust this rotary button to 50%, the actual feed speed is only 50mm/min. This rotary button works for servo feed speed under JOG mode on multiple machines. When Dry Run mode is effective, the feed speed is adjusted by this button too.

(2) **RAPID SPEED :**

Rapid speed will be effective under the following three conditions that are G00 in executing program, manual RAPID mode and first-half section speed of home return. There are 4 sections of Rapid Speed: F0, 25%, 50% and 100%. F0 is speed set by Parameter 40.

3.1.8 SPINDLE SPEED ADJUSTMENT SWITCH

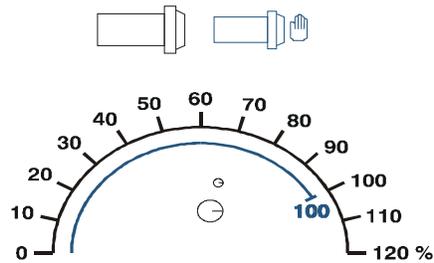


Figure 3.1-10

When the spindle is under MEM or MDI mode, using commands M3 (or M4) Sxx...to enable spindle rotation. The actual rotational speed of Spindle can be adjusted via this button, such as from 0% to120%. Example: if commanding M3 S1000 and setting the switch to 120%, then the actual rotational speed is 1200RPM.

3.1.9 SPINDLE ROTATION

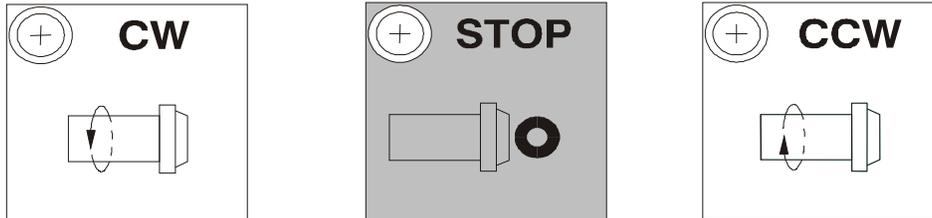


Figure 3.1-11

In JOG or RAPID mode, the following three buttons can control the rotation of spindle.

CW : spindle close-wise rotates

STOP : spindle stops rotating

CCW : spindle counter close-wise rotates

The rotational command is adjusted between 0% ~ 120% no mater the spindle is rotating (CW) or reverse rotating (CCW) in MPG mode. But, users must pay attention when direction changing. Must rotating via STOP procedure, otherwise it will not be effective.

3.1.10 COOLANT SUPPLY

Two Coolant Supply Buttons:

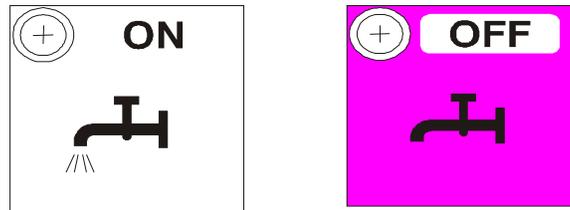
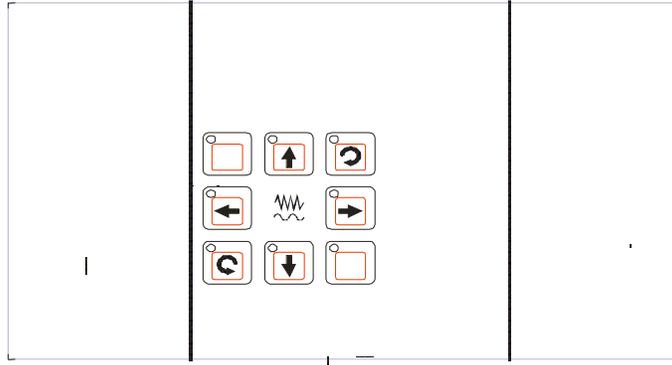


Figure 3.1-12

- (1) ON : force to turn on coolant supply. Under any condition, when pressing this button, coolant supply will be forced to ON until someone press OFF key.
- (2) OFF : force to turn off coolant supply. Coolant supply will be forced on OFF when someone presses this button under any condition.

3.1.11 AXIS SELECTION



These buttons are used to command moving axis direction under JOG and RAPID modes. For example, pressing +X button under JOG mode, X-axis will move toward +ive direction. Release this button will stop X-axis moving.

3.1.12 OT RELEASE

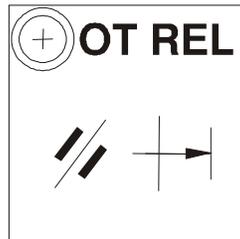


Figure 3.1-13

OT REL is the short name of Over Travel RELEase. There is one limit switch at end of each traveling side of servo axis in order to prevent to damage servo structure by colluding. When servo structure reaches traveling limit, over traveling will occur, which implies emergency stop. When the screen has “EMERGENCY STOP OR OVER TRAVEL” and this light is ON, then users need to check whether or not the servo structure is over traveling.

If it is over traveling, change the mode to MPG or JOG mode first. Then pressing this light (let this light ON), so the controller will neglect this over traveling emergency situation temporarily, which means users can use hand wheel or axis-direction key to move servo axis back to the travel range. Releasing (OT REL) key at this time in order to let the system continue travel checking.

If everything is working normally, which means “CNC Ready” will replace “CNC Not Ready”, then continue operating. If other warning messages occur, users need to press [RESET] key before returning back to normal. Please be careful about moving direction, and moving speed while moving the servo structure in order to prevent collusion.

3.1.13 TOOL MAGAZINE



Figure 3.1-14

TURRET

The turret rotates in the clockwise direction. In MDI mode, when pressing this button (TURRET light ON), the turret will rotate in the clockwise direction. Until the figure releases the button, the turret will stop at the next position. This button will not remain self-contain, which means that when releasing the button, the condition will be canceled (TURRET light OFF).

CHUCK

Clamping chuck button. Pressing this button to control loosing of tool. The motion method is the same as the TURRET button.

3.1.14 Single Block Stop

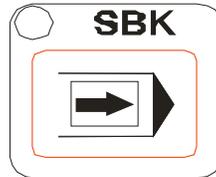


Figure 3.1-15

When this single block switch is ON, program operates single block one by one without continuously motion. Every signal block needs to press CYCLE START circular enable switch to operate.

3.1.15 Additional Function Selection

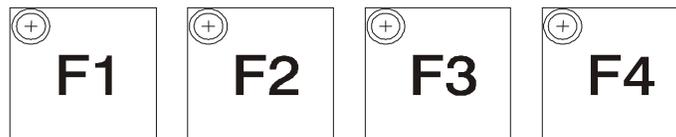


Figure 3.1-16

F1 : The button is reserved for other application.

F2 : The button is reserved for other application.

F3 : The button is reserved for other application.

F4 : The button is reserved for other application.