

LNC Milling Large Screen

# Operation Manual

2020/07 Ver : V01.00 (LN4408210135)

*Enable intelligent machines*

寶元數控股份有限公司

LNC Technology Co., Ltd.

---

## MENU

<b>1</b>	<b>CNC OPERATION .....</b>	<b>1</b>
1.1	Operation device type .....	1
1.1.1	M650 Series / M600 Series / M602 Series / M615 Series .....	1
1.1.2	M520 Series / M522 Series .....	15
1.1.3	M515i Series / M516i Series .....	20
1.1.4	XPCNC Series .....	26
1.2	Screen and Function Description .....	32
1.2.1	Display Screen Layout .....	32
1.3	POS Function ( POS ) .....	34
1.3.1	ABS .....	36
1.3.2	REL .....	39
1.3.3	MAC .....	40
1.3.4	HANDLE .....	41
1.3.5	MEA .....	42
1.3.6	EXIT .....	43
1.4	PROG .....	44
1.4.1	FGPROG .....	44
1.4.2	DIRMNG .....	53
1.4.3	PROCHK .....	69
1.4.4	MDI .....	71
1.4.5	PROGRAM RESTART .....	73
1.4.6	RS232 .....	77
1.5	OFFSET .....	79
1.5.1	TL. OFF .....	79
1.5.2	TDC .....	80
1.5.3	MACRO .....	81
1.5.4	WORK .....	83
1.5.5	TOOL .....	88
1.6	EASY-CAM .....	89
1.6.1	Preface .....	89
1.6.2	Screen Operation Description .....	90
1.7	MILLING FIGURE CONVERSATION .....	111
1.7.1	REFACE .....	111
1.7.2	Function Specification .....	111
1.7.3	Process type .....	111
1.7.4	Cutting Parameter Setting .....	113

1.7.5	Operation Screen Description .....	113
1.8	GRAPH .....	119
1.8.1	Function .....	119
1.8.2	GRAPH .....	119
1.8.3	SET .....	120
1.9	DGNOS.....	122
1.9.1	ALARM.....	122
1.9.2	IOCSA .....	126
1.9.3	MLC2.....	127
1.9.4	SYSTEM .....	130
1.9.5	Parameter .....	132
1.9.6	SYSUPD .....	133
1.9.7	CIRCUL.....	143
1.9.8	Servo Info.....	145
1.10	SOFTPL.....	147
1.11	PARAM .....	149
1.11.1	System .....	149
1.11.2	USROPT .....	151
1.11.3	Switch user.....	152
1.11.4	TLIM .....	152
1.11.5	Net.....	154
1.11.6	Servo parameter .....	159
1.11.7	AIC-II.....	162
<b>2</b>	<b>OP.....</b>	<b>163</b>
2.1	OP.....	163
2.2	LED SIGNAL.....	164
2.3	AXIS SELECTION .....	164
2.4	Additional Function Selection .....	165
2.5	EMG-STOP .....	168
2.6	CYCLE START&FEED HOLD .....	169
2.7	Feedrate/Rapid Rate Adjusting Button .....	170
2.8	PROGRAM PROTECTION LOCK.....	170
2.9	POWER ON/OFF.....	170
<b>3</b>	<b>PC Keyboard.....</b>	<b>171</b>
3.1	OP.....	171
3.2	AXIS SELECTION .....	172
3.3	MODE SELECT .....	172

3.4	SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH.....	174
3.5	PROGRAM AUXILIARY.....	175
3.6	AUXILIARY .....	176
3.7	CYCLE START & FEED HOLD .....	177
3.8	Feedrate.....	178
<b>4</b>	<b>RS232 COMMUNICATION SOFTWARE .....</b>	<b>179</b>
4.1	Transmission line preparation.....	179
4.2	PC end installation (use LNC ReCON 232 software) .....	180
4.3	NC end.....	182
4.4	Communication (RS232) - 【 DNC file upload and download functions 】 .....	197
4.4.1	Path Program Upload & save (using ReCON software) .....	197
4.4.2	Path Program Upload & save (using ReCON software) .....	200
4.4.3	Path Program Upload & save (using general software).....	203
4.4.4	Path Program Upload & save (using general software).....	207
4.5	Setting & Attention for USB to RS232 port .....	212
<b>5</b>	<b>Alarm &amp; Warning .....</b>	<b>215</b>
5.1	OP OPERATION ALARM.....	215
5.2	OP.....	217
5.3	INT .....	218
5.4	MOT MOTION RELATED ALARM.....	229
5.5	MOT .....	247



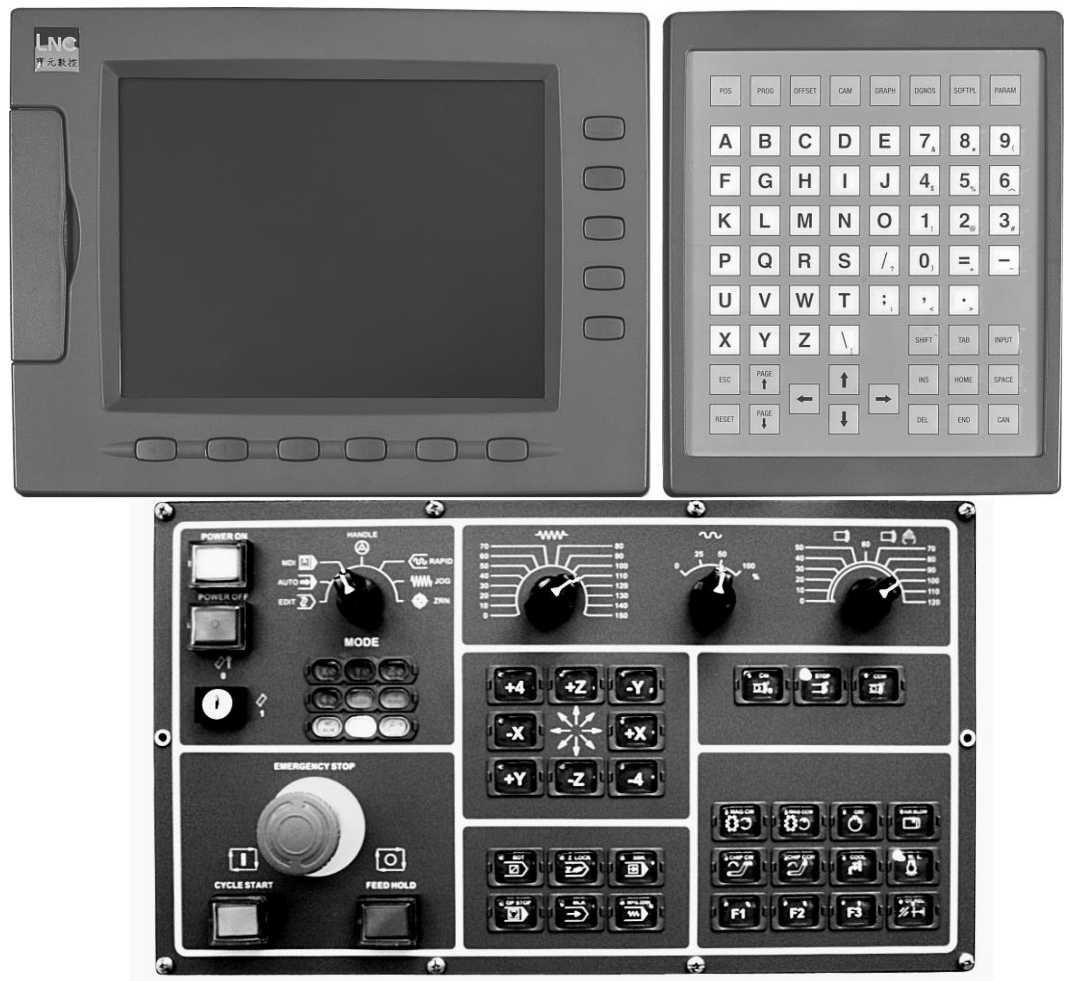
# 1 CNC OPERATION

## 1.1 Operation device type

Below is LNC large monitor models, you can choose model to select. There are MDI and OP. MDI is for you to edit word by word or change program and setting value. OP is for you to control different functions to process work piece. There are many switches, buttons, and MPG. OP may be different from machine design. We offer a standard OP for you to choose, details are as below :

### 1.1.1 M650 Series / M600 Series / M602 Series / M615 Series...

#### 1.1.1.1 TYPE 1



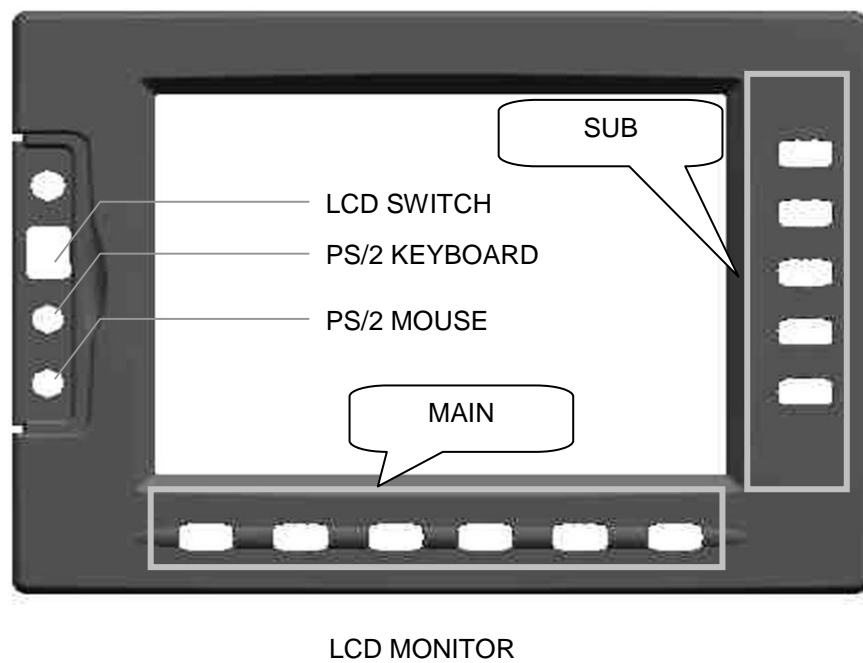
There are two kinds of key at the LCD part :

(1). **Main key :**

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.

(2). **Sub-Function Keys :**

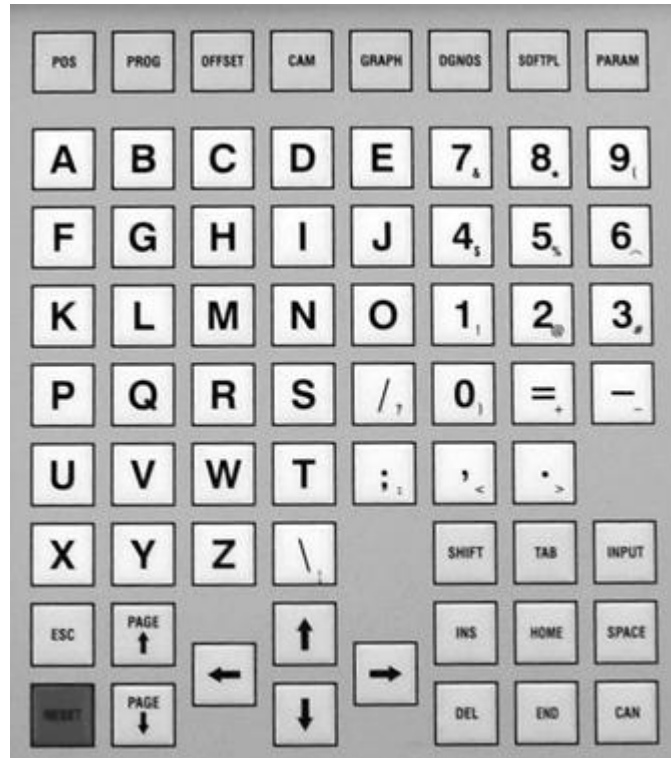
After choosing main function, you can choose sub function key to continue next step.



There are two kinds of areas in operation panel :

(1).MDI :

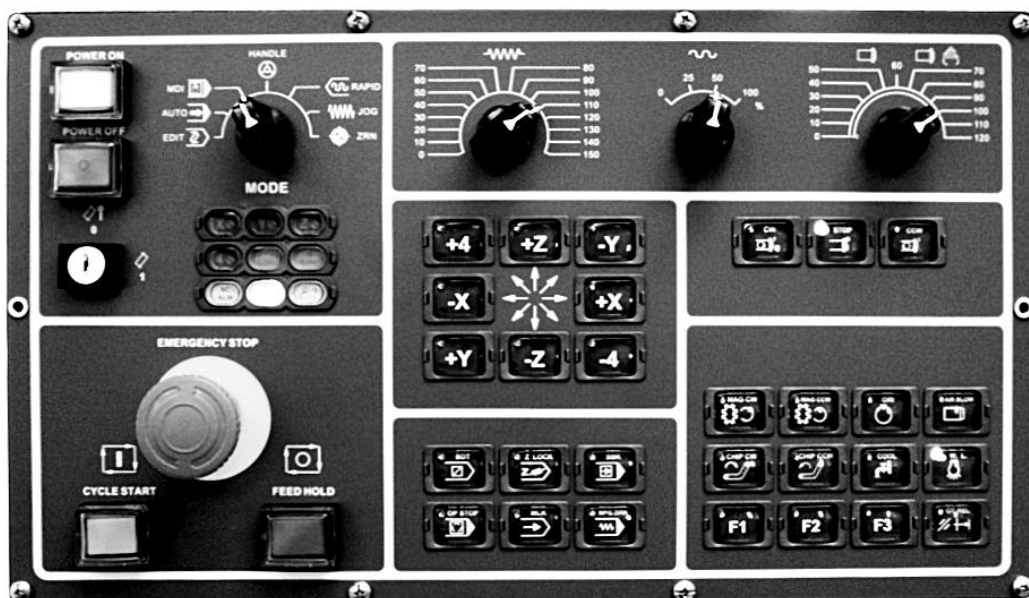
For users to input numbers and letters.



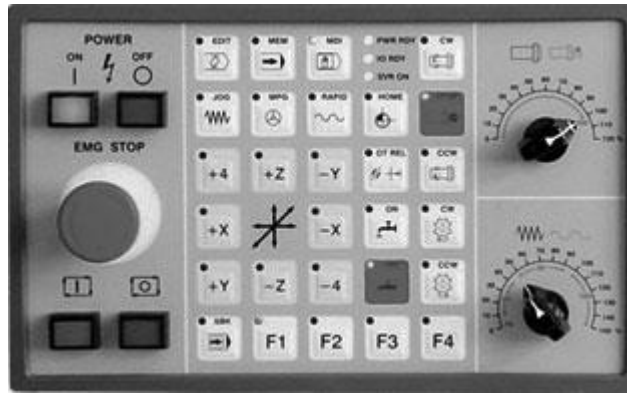
MDI

(2).OP :

For user to operate.



OP2520



OP1540





(3).MDI :

A. CNC :

Name	Group Key	Description
<POS>	POS	To display positions.
<PROG>	PROG	To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ).
<OFFSET>	OFFSET	To set and to display tool offset.
<CAM>	CAM	Show easy cam conversational function.
<GRAPH>	GRAPH	To draw tool path.
<DGNOS>	DGNOS	To display instance message at DGNOS page.
<SOFTPL>	SOFTPL	Show soft panel.
<PARAM>	PARAM	Show parameter.



## B. Character & Symbol Keys :










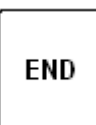

These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.





Name	Help Key	Description
Character	  ~	Character, from A to Z, total 26 buttons for position command or argument command..
Numbers	  ~	Numbers, 0~9, total 10 keys for value inputing..
Symbol	/	For slash skip when editing program.
Symbol	;	Press this key to end program segment when editing program. Put this key before the single block for not executing this single block.
Symbol	.	Decimal point segment value for edit program.
Symbol	( , ) , < , > , , , : , & , * , ? , [ , ] , \$ , % , ^ , ! , @ , # , + , - , = , _ ,	Symbol keys for edit program.

## C. Editing key :

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

Name	Help Key	Description
RESET		To reset system to return default. When alarm removing, cancel alarm. When start working, to cancel working. At edit mode, cursor returns to program head.
PAGE↑		To turn to the previous page.

Name	Help Key	Description
PAGE↓		To turn to the next page.
Enter	 <Enter>	Input at the data input area, press enter to confirm. The column will be filled with the data. At file management, move cursor to the file location, press enter to open the file. At edit mode, press enter to insert space line.
↑		Under edit mode, move cursor upper. To move cursor upper.
↓		Under edit mode, move cursor down. To move cursor down.
←		Under edit mode, move cursor left. To move cursor left.
→		Under edit mode, move cursor right. To move cursor right.
Space		To key-in into empty space.
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
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.
CAN		Cancel before words.

Name	Help Key	Description
TAB		When editing program, press tab to move right 8 grid.
INS		Insert letters or replace letters..
DEL		Delete the later letters..
ESC		Return back Note: this function will be valid under BIOS mode.

[ Note ]

- This controller did not have 5-axis contouring control and cutting function, only suitable for industrial machines.
- This controller did not have 5 axes TCP tool tip compensation functions.
- This controller did not have rotation axis velocity graduation planning.
- This controller did not have 5-axis machining interference anti-collision check function.

### 1.1.1.2 TYPE 2



TYPE 2 OP1

Note : Digital model did not support USB and CF card port.





TYPE 2 OP2

Note : Digital model did not support USB and CF card port.

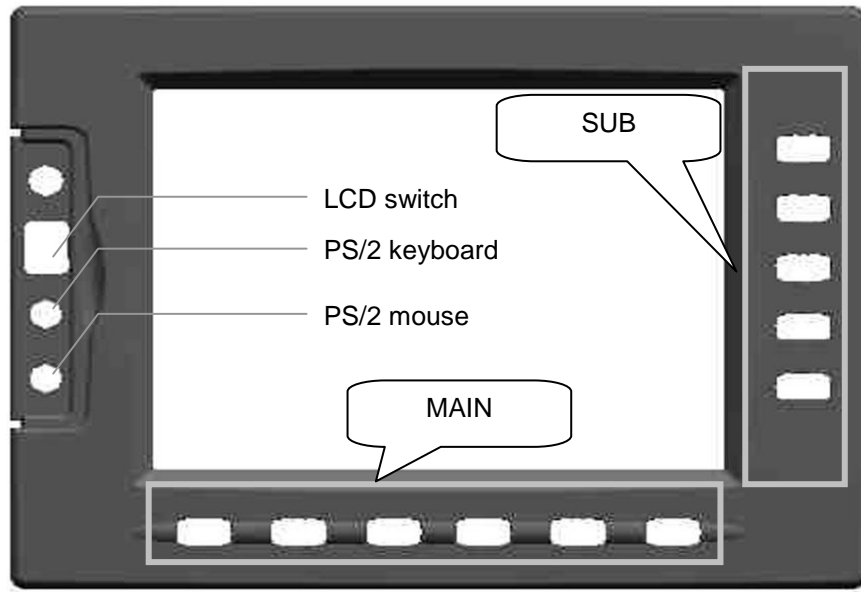
There are two kinds of key at the LCD part :

(1). **Main key :**

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.

(2). **Sub-Function Keys :**

After choosing main function, you can choose sub function key to continue next step.

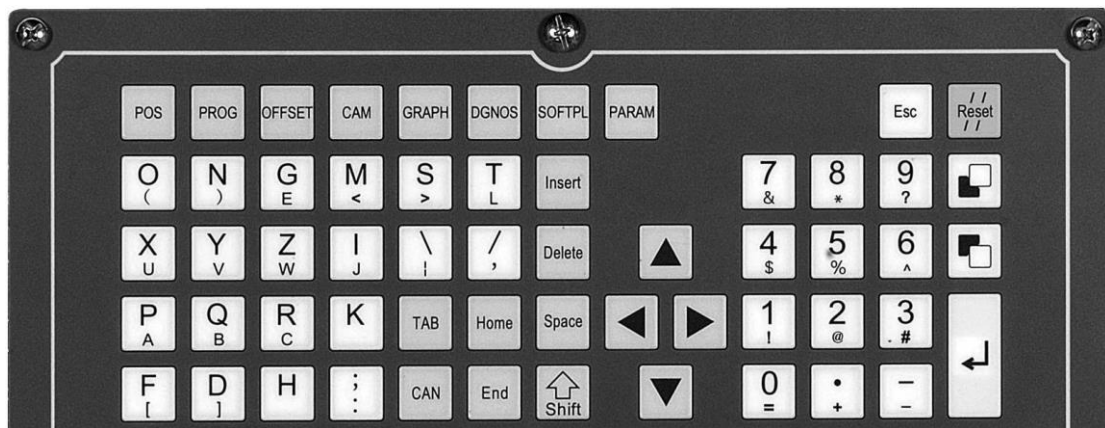


LCD monitor

There are two kinds of areas in operation panel :

(1). **MDI :**

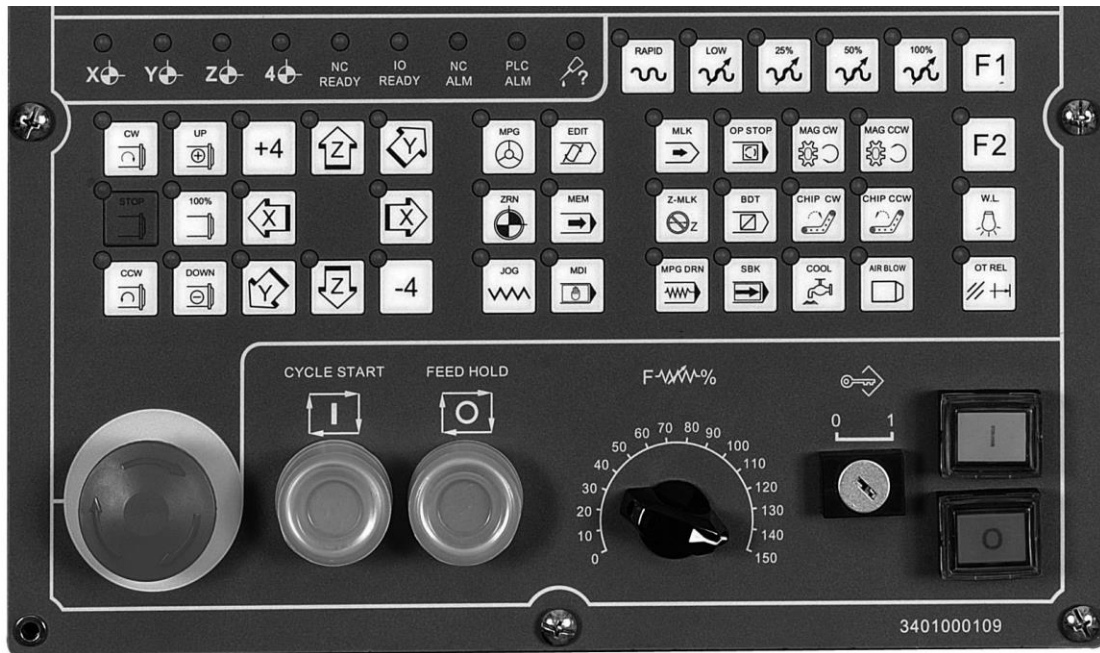
For users to input numbers and letters.



MDI

(2).OP :

For users to operate.






OP

(3).MDI :





A. CNC :

Name	Group Key	Description
<POS>	POS	To display positions.
<PROG>	PROG	To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ).
<OFFSET>	OFFSET	To set and to display tool offset.
<CAM>	CAM	Show easy cam conversational function.
<GRAPH>	GRAPH	To draw tool path.

Name	Group Key	Description
<DGNOS>		To display instance message at DGNOS page.
<SOFTPL>		Show soft panel.
<PARAM>		Show parameter.


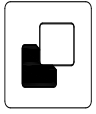

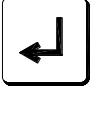



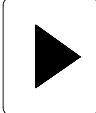

### B. Character & Symbol Keys :

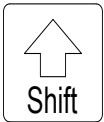






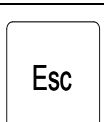
These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.

Name	Help Key	Description
Character	 ~ 	Character, from A to Z, total 26 buttons for position command or argument command..
Numbers	 ~ 	Numbers, 0~9, total 10 keys for value inputing..
Symbol	/	For slash skip when editing program.
Symbol	;	Press this key to end program segment when editing program. Put this key before the single block for not executing this single block.
Symbol	.	Decimal point segment value for edit program.
Symbol	( ) , < > , ' , : , & , * , ? , [ ] , \$ , % , ^ , ! , @ , # , + , - , _ ,	Symbol keys for edit program.

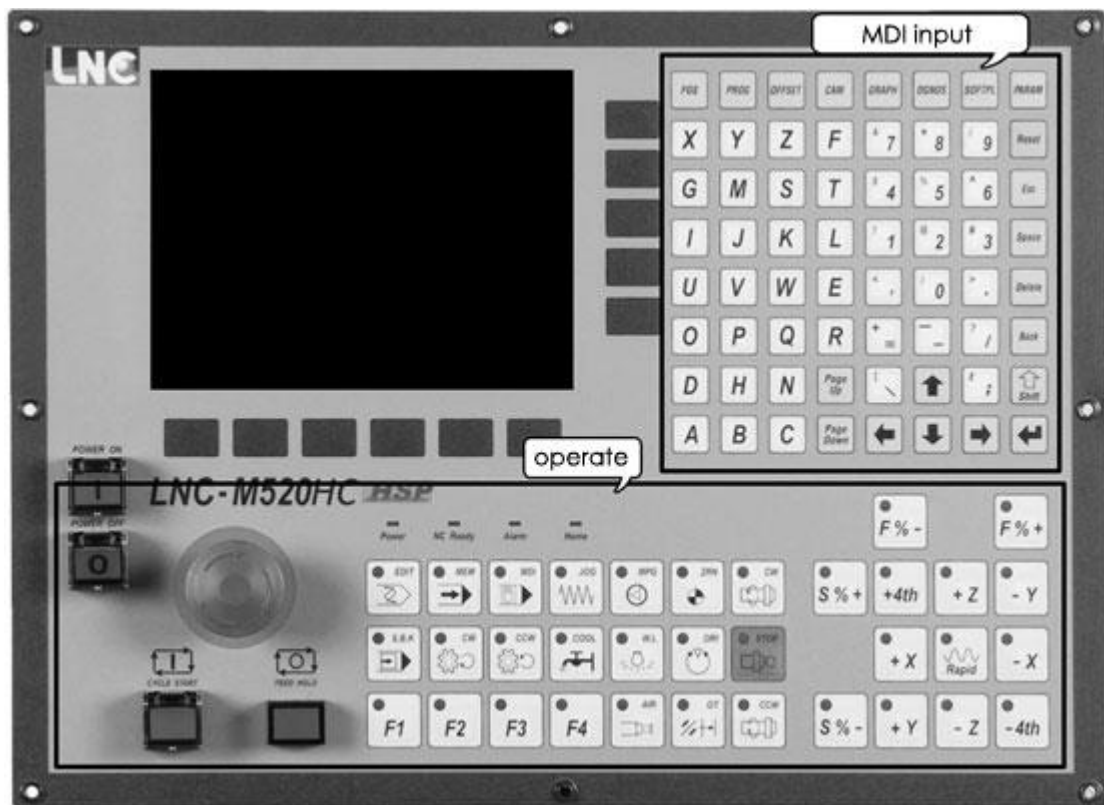
**C. Editing key :**

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

Name	Help Key	Description
RESET		To reset system to return default. When alarm removing, cancel alarm. When start working, to cancel working. At edit mode, cursor returns to program head.
PAGE↑	 <Page Up>	To turn to the previous page.
PAGE↓	 <Page Down>	To turn to the next page.
Enter	 <Enter>	Input at the data input area, press enter to confirm. The column will be filled with the data. At file management, move cursor to the file location, press enter to open the file. At edit mode, press enter to insert space line.
↑		Under edit mode, move cursor upper. To move cursor upper.
↓		Under edit mode, move cursor down. To move cursor down.
←		Under edit mode, move cursor left. To move cursor left.
→		Under edit mode, move cursor right. To move cursor right.
Space		To key-in into empty space.

Name	Help Key	Description
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
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.
CAN		Cancel before words.
TAB		When editing program, press tab to move right 8 grid.
INS		Insert letters or replace letters..
DEL		Delete the later letters.
ESC		Return back Note: this function will be valid under BIOS mode.

### 1.1.2 M520 Series / M522 Series...



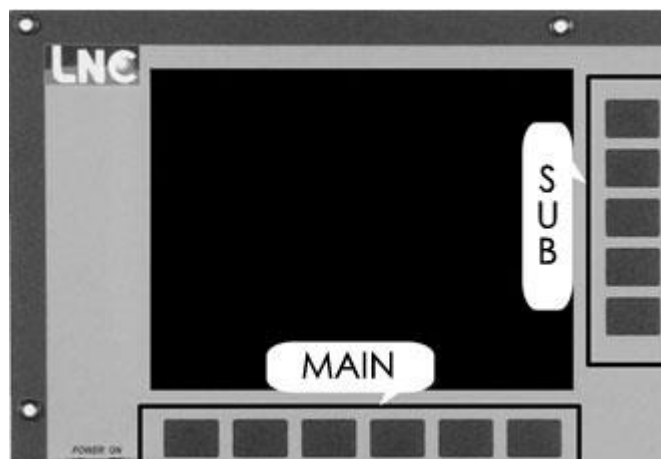
There are 2 kinds of key at the MDI :

(1) **Main key :**

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.

(2) **Sub-Function Keys :**

After choosing main function, you can choose sub function key to continue next step.



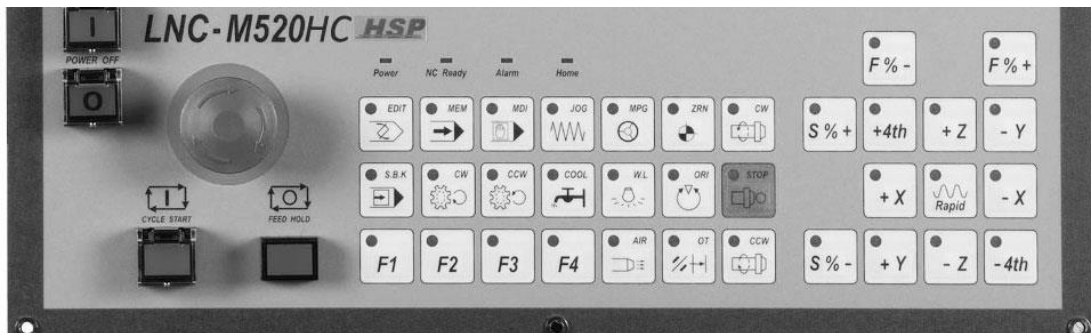
LCD 液晶顯示器

(1).MDI



MDI

(2).OP :








OP

(3).MDI :

A. CNC :





Name	Group Key	Description
<POS>	POS	To display positions.
<PROG>	PROG	To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ).
<OFFSET>	OFFSET	To set and to display tool offset.



<CAM>		Show easy cam conversational function.
<GRAPH>		To draw tool path.
<DGNOS>		To display instance message at DGNOS page.
<SOFTPL>		Show soft panel.
<PARAM>		Show parameter.


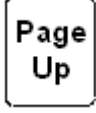
## B. Character & Symbol Keys :

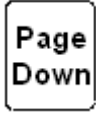







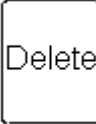

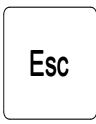
These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.

Name	Help Key	Description
Character	  ~	Character, from A to Z, total 26 buttons for position command or argument command..
Numbers	  ~	Numbers, 0~9, total 10 keys for value input...
Symbol	/	For slash skip when editing program.
Symbol	;	Press this key to end program segment when editing program. Put this key before the single block for not executing this single block.
Symbol	.	Decimal point segment value for edit program.
Symbol	( , ) , < , > , ' , ` , : , & , * , ? , [ , ] , \$ , % , ^ , ! , @ , # , + , - , = , _ , ~	Symbol keys for edit program.

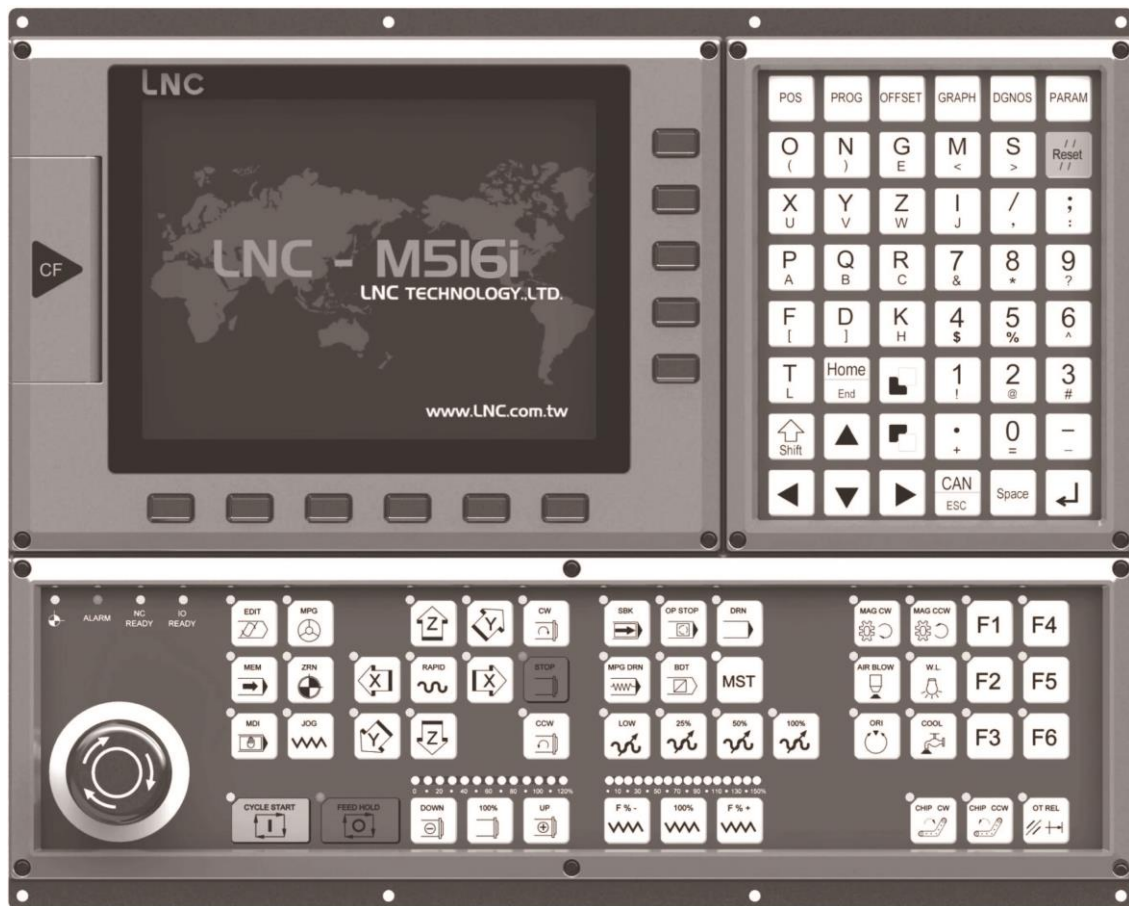
## C. Editing key :

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

Name	Help Key	Description
RESET		To reset system to return default. When alarm removing, cancel alarm. When start working, to cancel working. At edit mode, cursor returns to program head.
PAGE↑		To turn to the previous page.

Name	Help Key	Description
PAGE↓		To turn to the next page.
Enter	 <Enter>	Input at the data input area, press enter to confirm. The column will be filled with the data. At file management, move cursor to the file location, press enter to open the file. At edit mode, press enter to insert space line.
↑		Under edit mode, move cursor upper. To move cursor upper.
↓		Under edit mode, move cursor down. To move cursor down.
←		Under edit mode, move cursor left. To move cursor left.
→		Under edit mode, move cursor right. To move cursor right.
Space		To key-in into empty space.
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
DEL		Delete the later letters.
BACK		Cancel previous words.
ESC		Return back Note: this function will be valid under BIOS mode.

## 1.1.3 M515i Series / M516i Series...



There are 3 kinds of key at the MDI :

(1).**Main key :**

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.

(2).**Sub-Function Keys :**

After choosing main function, you can choose sub function key to continue next step.

(3).**CF/USB :**

For CF card or USB to use.



Note : M516id did not support CF port.

There are 2 areas :

(1).MDI :

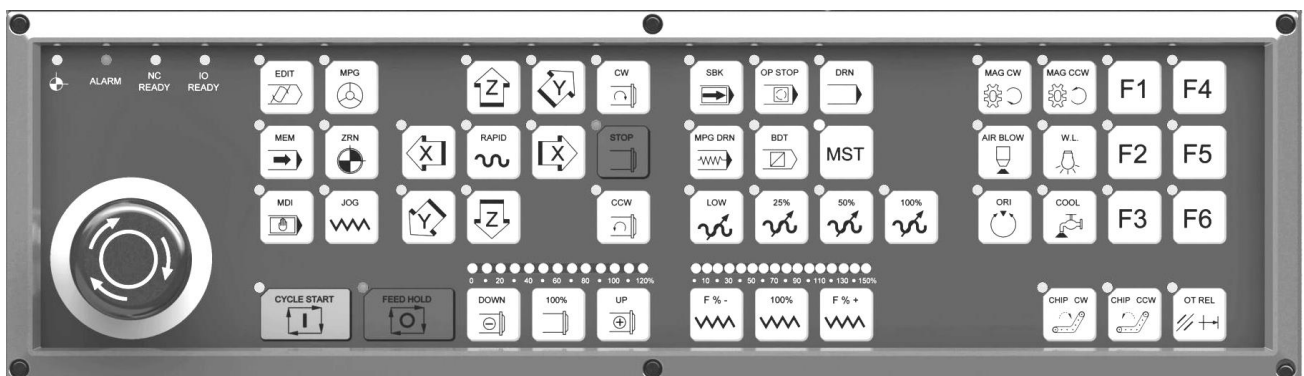
For users to input letter and numbers into controller.



MDI

(2).OP :







For users to select functions to do on processing.



OP





### (3).MDI :

#### A. CNC :

Name	Group Key	Description
<POS>		To display positions.
<PROG>		To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ).
<OFFSET>		To set and to display tool offset.
<GRAPH>		To draw tool path.
<DGNOS>		To display instance message at DGNOS page.
<PARAM>		Show parameter.

#### B. Character & Symbol Keys :



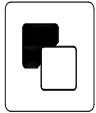


These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.

Name	Help Key	Description
Character	 ~ 	Character, from A to Z, total 26 buttons for position command or argument command..
Numbers	 ~ 	Numbers, 0~9, total 10 keys for value inputing..


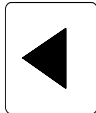




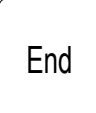

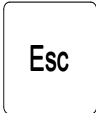
<b>Symbol</b>	/	For slash skip when editing program.
<b>Symbol</b>	;	Press this key to end program segment when editing program. Put this key before the single block for not executing this single block.
<b>Symbol</b>	.	Decimal point segment value for edit program.
<b>Symbol</b>	(、) 、<、>、 、 、:、&、 *、?、[、]、\$、%、^、!、 @、#、+、=、-、_、	Symbol keys for edit program.

### C. Editing key :

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

Name	Help Key	Description
RESET		To reset system to return default. When alarm removing, cancel alarm. When start working, to cancel working. At edit mode, cursor returns to program head.
PAGE↑	 <Page Up>	To turn to the previous page.
PAGE↓	 <Page Down>	To turn to the next page.
Enter	 <Enter>	Input at the data input area, press enter to confirm. The column will be filled with the data. At file management, move cursor to the file location, press enter to open the file. At edit mode, press enter to insert space line.
↑		Under edit mode, move cursor upper. To move cursor upper.



Name	Help Key	Description
↓		Under edit mode, move cursor down. To move cursor down.
←		Under edit mode, move cursor left. To move cursor left.
→		Under edit mode, move cursor right. To move cursor right.
Space		To key-in into empty space.
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
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.
CAN		Cancel before words.
ESC		Return back Note: this function will be valid under BIOS mode.





### (3).MDI :

#### A. CNC :



Name	Group Key	Description
<POS>	POS	To display positions.
<PROG>	PROG	To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ).
<OFFSET>	OFFSET	To set and to display tool offset.
<CAM>	CAM	Show easy cam conversational function.
<GRAPH>	GRAPH	To draw tool path.
<DGNOS>	DGNOS	To display instance message at DGNOS page.
<SOFTPL>	SOFTPL	Show soft panel.
<PARAM>	PARAM	Show parameter.

## B. Character & Symbol Keys :



Upper big words is for program edit and info input. If you want to use down side small words, you will need to press SHIFT together.






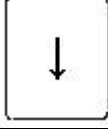
Name	Help Key	Description
Character	<div>X</div> <div>A</div> ~ <div>Z</div> <div>B</div>	Character, from A to Z, total 26 buttons for position command or argument command..
Numbers	<div>0</div> <div>=</div> ~ <div>9</div> <div>?</div>	Numbers, 0~9, total 10 keys for value inputing..
Symbol	/	For slash skip when editing program.
Symbol	;	Press this key to end program segment when editing program. Put this key before the single block for not executing this single block.
Symbol	.	Decimal point segment value for edit program.
Symbol	( , ) , < , > , , , : , & , * , ? , [ , ] , \$ , % , ^ , ! , @ , # , + , - , = , _ , ` , ~	Symbol keys for edit program.




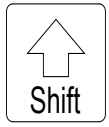



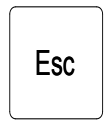


### C. Editing key :



Use this key to move cursor to change program or setting.

Name	Help Key	Description
RESET		To reset system to return default. When alarm removing, cancel alarm. When start working, to cancel working. At edit mode, cursor returns to program head.
PAGE↑		To turn to the previous page.
PAGE↓		To turn to the next page.
Enter		Input at the data input area, press enter to confirm. The column will be filled with the data. At file management, move cursor to the file location, press enter to open the file. At edit mode, press enter to insert space line.
↑		Under edit mode, move cursor upper. To move cursor upper.
↓		Under edit mode, move cursor down. To move cursor down.

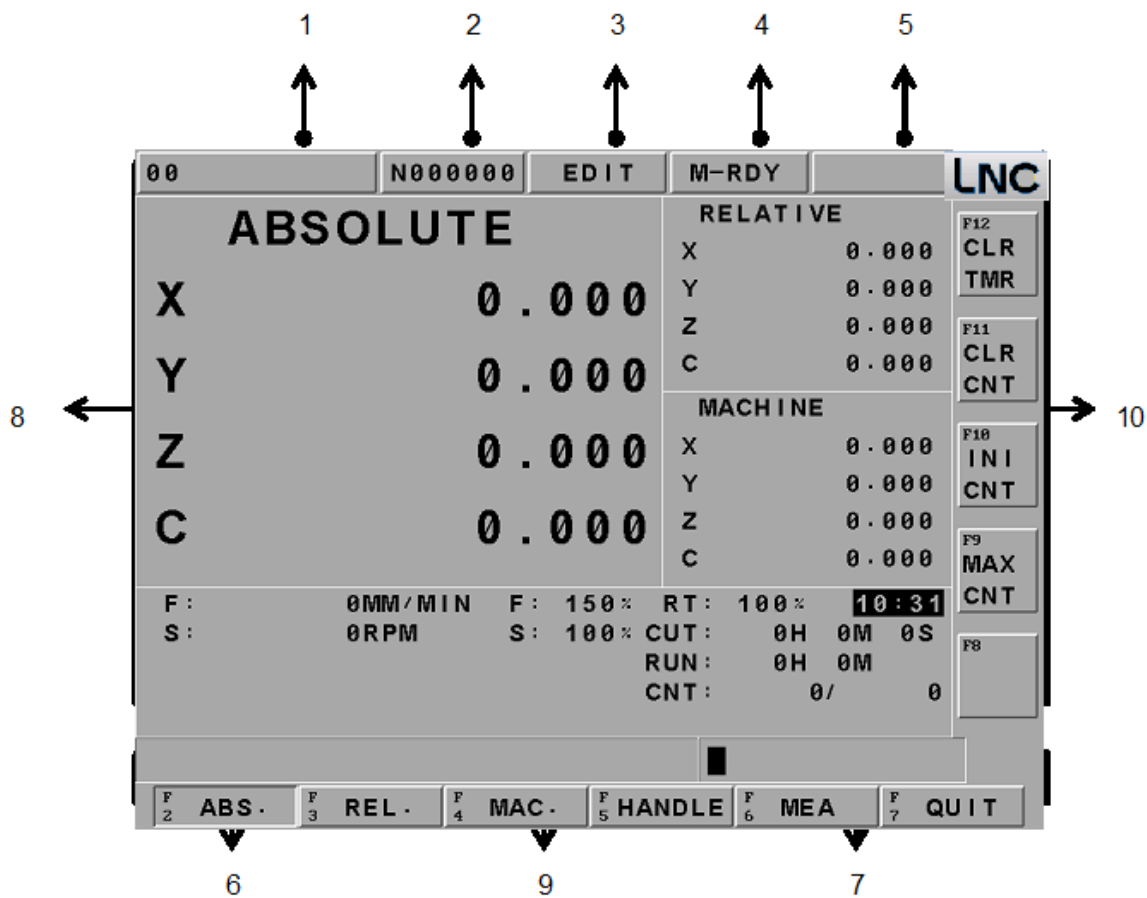
Name	Help Key	Description
←		Under edit mode, move cursor left. To move cursor left.
→		Under edit mode, move cursor right. To move cursor right.
Space		To key-in into empty space.
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
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.
Back space		Cancel before words.
ESC		Return back Note: this function will be valid under BIOS mode.

## 1.2 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 using <....> to indicate keys on MDI panel ( Note : M515/M516i did not support 「CAM」 & 「SOFTPL」 )

### 1.2.1 Display Screen Layout





**1 : Present designate file name**

Note: Means the file name that controller is using now.

**2 : Present single block that is executed by controller**

Note: Means the controller is using program to execute N file.

**3 : CNC mode signal**

Note: 1.Edit 2.MEM 3.MDI 4.MPG 5.RAPID 6.JOG 7.ZRN

**4 : Machine condition signal**

Note: 1.NO-RDY 2.M-RDY 3.CSTART 4.F-HOLD 5.B-STOP

**5 : Wrong alarm/warning message**

Note: 1.Alarm 2.Waring

**6 : Simply message hint area**

Note: Controller offers a simple message to remind users.

**7 : Input area**

Note: Controller offer users to input data.

**8 : Display range**

Note: Each function groups to display changes

**9 : Main function button display**

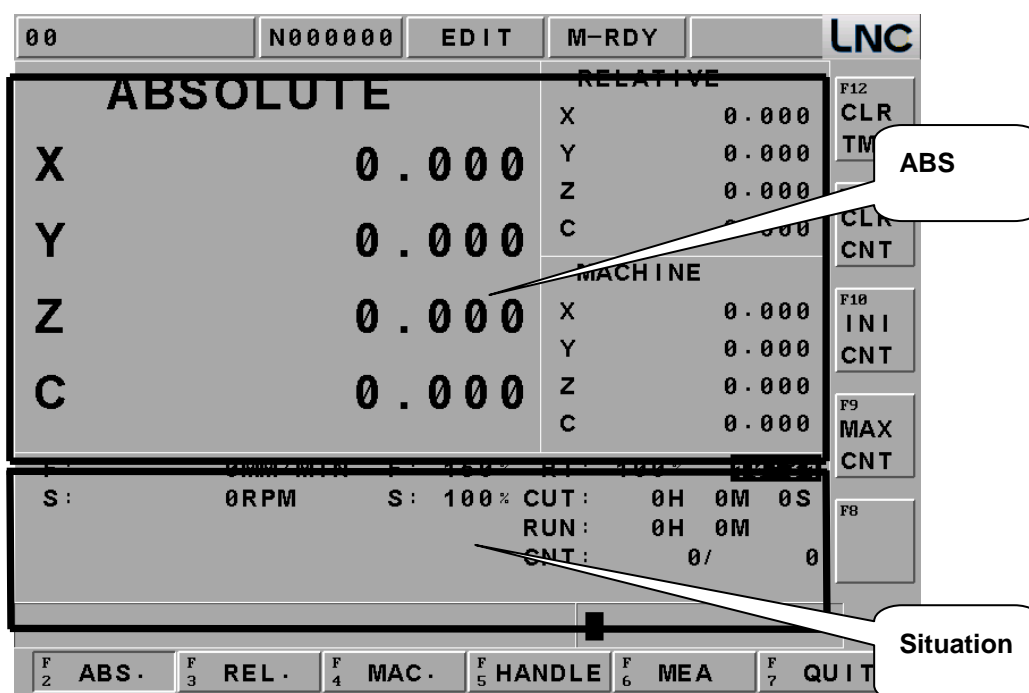
Note: This area is for main function buttons to use.

**10 : Sub-function button display**

Note: This area is for sub-function buttons to use.

### 1.3 POS Function (POS)

You will see POS page when you enter this system, or you can press <POS> key to enter into the coordinate display screen. Users will be able to choose 6 screen displays by pressing those function keys which are **[ABS.]**, **[REL.]**, **[MAC.]**, **[RST]** and **[QUIT]**. There are some command information that will be displayed in those screens, which will be described separately as following :



POS Function Display Screen

#### Actual Feed Rate

Feedrate(mm/min) : Feed rate of servo structure-is the composite speed for each servo axis. Unit is mm/min.

Feedrate percentage(%) : Feedrate percentage-with the feedrate dial to choose 15 selections from 0% to 150%.

Quick feed percentage (%) : Quick feed percentage-with the quick feed button to have 4 selections of <LOW> , <25%> , <50%> , <100%>.

Time : Current time.

F :	0MM/MIN	F :	150%	RT :	100%	10:31
S :	0RPM	S :	100%	CUT :	0H 0M 0S	
				RUN :	0H 0M	
				CNT :	0/ 0	

### Rotation situation

Rotational speed (rpm/min) : Current spindle actual rotation. Unit is rpm/min.

Rotational speed percentage(%) : Rotational speed percentage-with rotation button<UP>、<DOWN> to have 12 selections of 0%~120%.

F :	0MM/MIN	F :	150%	RT :	100%	10:31
S :	0RPM	S :	100%	CUT :	0H 0M 0S	
				RUN :	0H 0M	
				CNT :	0/ 0	

### Hint message display situation

Message : According to system situation, there are 7 messages as below :F0 (current federate is 0)、SBK (single block)、BDT (selective block jog)、DRN (dry run)、MLK (machine lock)、OPS (optional program pause)、ABS (absolute encoder reader)

F :	0MM/MIN	F :	0%	RT :	10%	11:50
S :	0RPM	S :	100%	CUT :	0H 0M 0S	
				RUN :	0H 0M	
				CNT :	0/ 0	
F0 SBK BDT DRN MLK OPS						

### Situation of CUT, RUN, CNT

CUT(H"M"S) : Every time when users press Cycle Start button, cutting time will reset(return zero) and start to time again until this working process finish.

RUN(H"M") : Running time is the total cutting time after every booting, until the system shuts down, the time will be reset to be zero. You can also use 【TIME CLEAR】 to make it to be zero.

CNT(current working pieces/max working pieces) : when CNC program reads M02, M30 or other M code at the end, system will sum up working pieces, this function need to use Pr.0089 and PLC to work.

F :	0MM/MIN	F :	150%	RT :	100%	10:31
S :	0RPM	S :	100%	CUT :	0H 0M 0S	
				RUN :	0H 0M	
				CNT :	0/ 0	

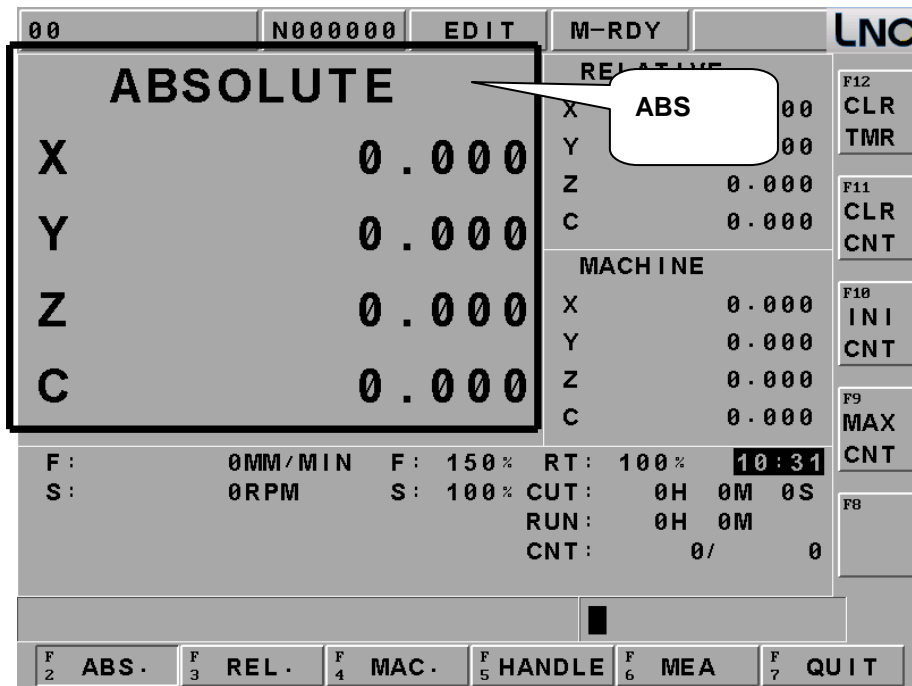
### Other

Time : Current time .

F :	0MM/MIN	F :	150%	RT :	100%	10:31
S :	0RPM	S :	100%	CUT :	0H 0M 0S	
				RUN :	0H 0M	
				CNT :	0/ 0	

### 1.3.1 ABS

Pressing **【ABS.】** to enter into absolute coordinate screen. Details will be shown in bigger words at the upper left side of the screen. Using the program home (means the workpiece coordinate) to show tool's current location, the absolute coordinate will be changed according to the movement of tool. This is called absolute coordinate.



Absolute coordinate is the program coordinate, which is [present value – tool offset = program value.]

- **【Time Clear】**

Press **【Time Clear】** to delete current total running time.

00	N000000	EDIT	M-RDY	LNC	
<b>ABSOLUTE</b>			<b>RELATIVE</b>		
X	0.000	X	0.000	F12	CLR TMR
Y		Y	0.000	F11	CLR CNT
Z		Z	0.000	F10	INI CNT
C			0.000	F9	MAX CNT
F: 0MM/MIN F: 150% RT: 100% 10:36			F8		
S: 0RPM S: 100% CUT: 0H 0M 0S					
RUN: 0H 0M					
CNT: 0/ 0					
<div style="border: 1px solid black; padding: 5px; margin: 10px auto; width: 80%;"> <p style="text-align: center;">Action Confirm</p> <p style="text-align: center;">CLEAR TIME?</p> </div>					
F2	OK	F3	CANCEL	F4	

- **【Workpiece Clear】**

Press **【Workpiece Clear】** to delete current workpiece-make it to be zero.

00	N000000	EDIT	M-RDY	LNC	
<b>ABSOLUTE</b>			<b>RELATIVE</b>		
X	0.000	X	0.000	F12	CLR TMR
Y		Y	0.000	F11	CLR CNT
Z		Z	0.000	F10	INI CNT
C			0.000	F9	MAX CNT
F: 0MM/MIN F: 150% RT: 100% 10:37			F8		
S: 0RPM S: 100% CUT: 0H 0M 0S					
RUN: 0H 0M					
CNT: 0/ 0					
<div style="border: 1px solid black; padding: 5px; margin: 10px auto; width: 80%;"> <p style="text-align: center;">Action Confirm</p> <p style="text-align: center;">CLREA COUNT?</p> </div>					
F2	OK	F3	CANCEL	F4	

- **【Initial Workpiece】**

Press **【Initial Workpiece】** to change current workpiece-make it to be current pieces.

00	N000000	EDIT	M-RDY	LNC	
<b>ABSOLUTE</b>			<b>RELATIVE</b>		F12 CLR TMR
X	0.000		X	0.000	
Y	0.000		Y	0.000	F11 CLR CNT
Z			Z	0.000	
C	0.000		C	0.000	F10 INI CNT
INPUT INI CNT 10			<b>MACHINE</b>		F9 MAX CNT
			Y	0.000	
			Z	0.000	F8
			C	0.000	
F: 0MM/MIN F: 150% RT: 100% 10:38 S: 0RPM S: 100% CUT: 0H 0M 0S RUN: 0H 0M CNT: 0/ 0					
F2 OK	F3 CANCEL	F4	F5	F6	F7

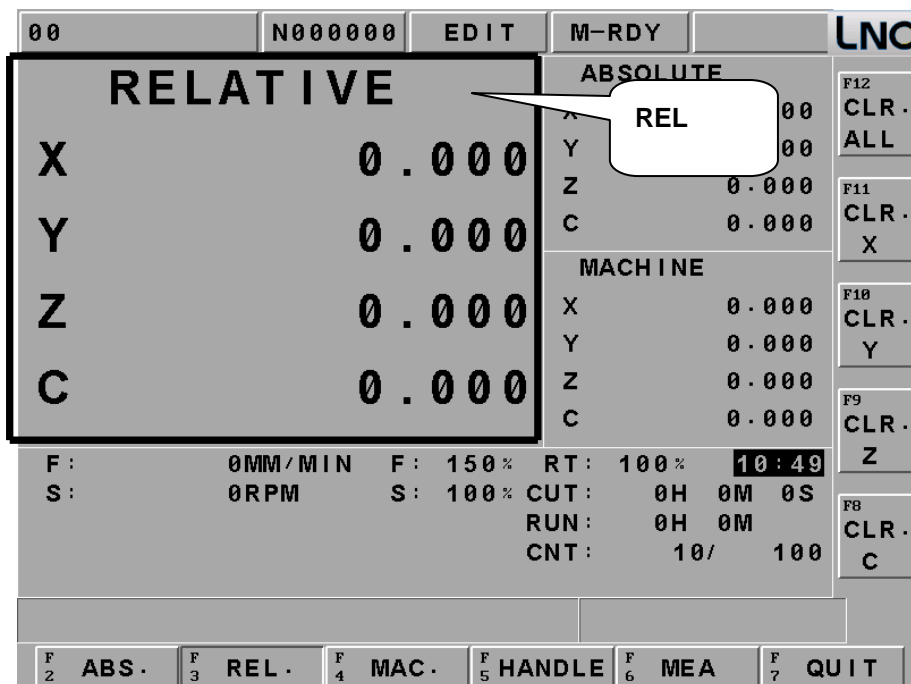
- **【Max Workpiece】**

Press **【Max Workpiece】** to change current max. workpiece-make it to be 100. When it comes to be 100 pieces, the controller will notify the workpieces have reached target.

00	N000000	EDIT	M-RDY	LNC	
<b>ABSOLUTE</b>			<b>RELATIVE</b>		F12 CLR TMR
X	0.000		X	0.000	
Y	0.000		Y	0.000	F11 CLR CNT
Z			Z	0.000	
C	0.000		C	0.000	F10 INI CNT
INPUT MAX CNT 100			<b>MACHINE</b>		F9 MAX CNT
			Y	0.000	
			Z	0.000	F8
			C	0.000	
F: 0MM/MIN F: 150% RT: 100% 10:39 S: 0RPM S: 100% CUT: 0H 0M 0S RUN: 0H 0M CNT: 10/ 0					
F2 OK	F3 CANCEL	F4	F5	F6	F7

## 1.3.2 REL

Pressing **【REL.】** to enter into the relative coordinate screen. Details will be shown in bigger words at the upper left side of the 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 at anytime, or enter the non-zero value directly.



The way for relative coordinate to return zero :

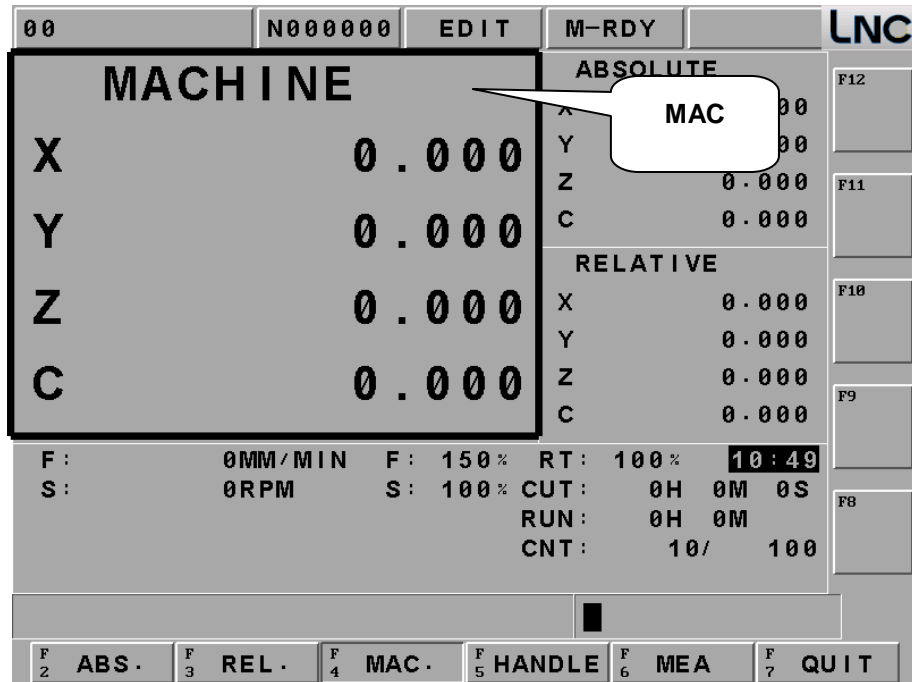
If want to set the 3 axes relative coordinate values to zero separately or at the same time, users only need to press the corresponding sub-function buttons **【CLR. X】**, **【CLR. Y】**, **【CLR. Z】**, **【CLR. ALL】** to execute the corresponding clean.

Setting way for relative coordinate :

If you 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.

## 1.3.3 MAC

Pressing 【MAC.】 to enter into the relative coordinate screen. Details will be shown in bigger words at the upper left side of the screen.



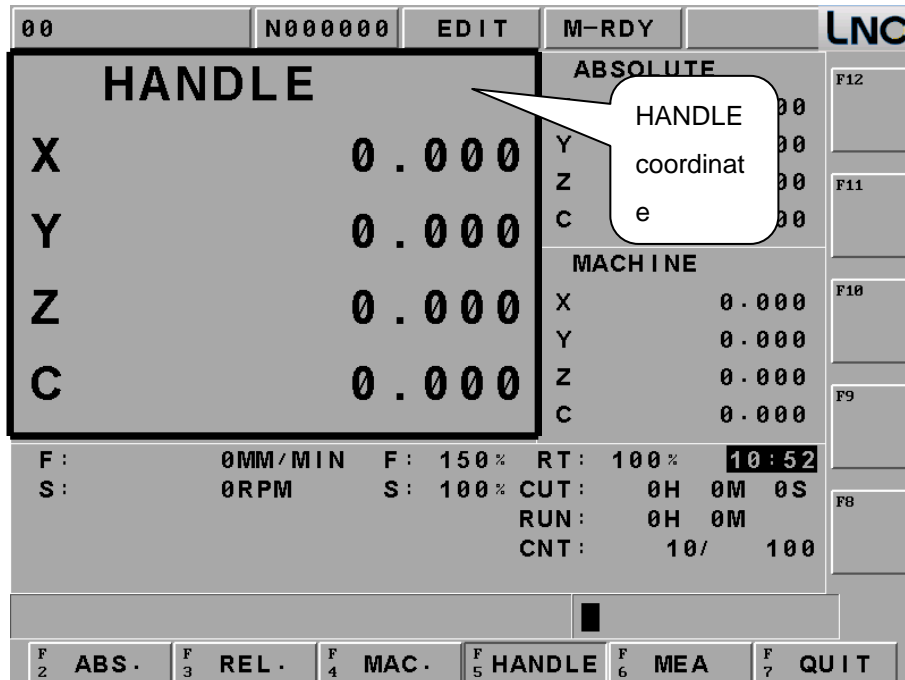
Machine coordinate is the distance of the present position corresponding to the reference point

**Note :** Each machine has its own reference point. For safety concern, please looking for the reference point whenever reboot machine before executing work.



### 1.3.4 HANDLE

At the auto mode to press【HANDLE】, users can use MPG interruption to increase or decrease the movement of tool to change the tool path.



**Note :** Because the absolute coordinate will not be changed by MPG interruption, therefore the tool path later will have certain offset with original program path. This offset need to be modified by manual return zero to clear. And about the offset value, you can refer to MPG coordinate.

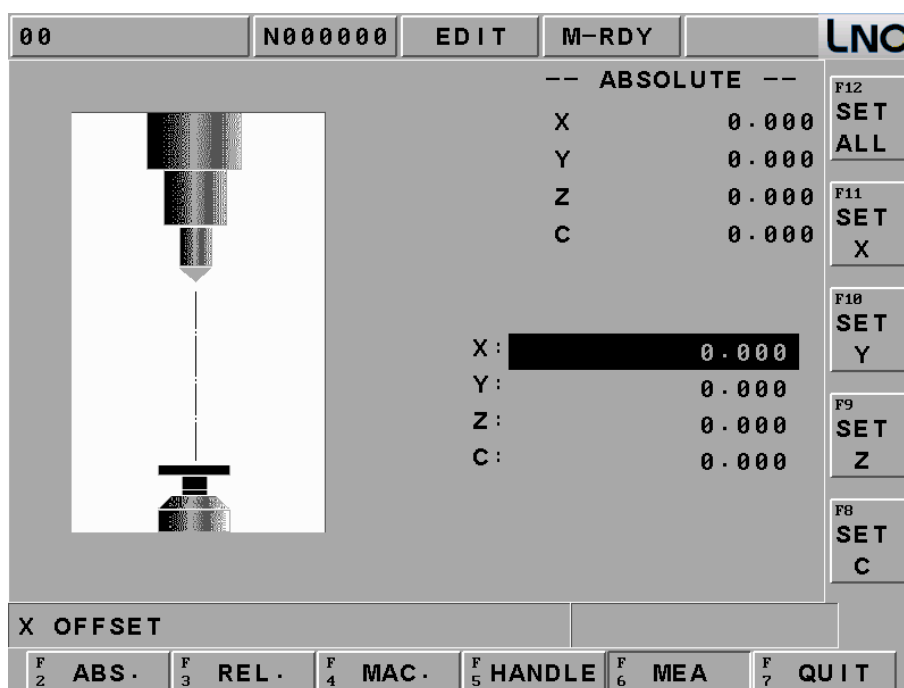
This function refer to software application-HANDLE axis selection signal(C BITS 066 ~ 071) and MPG ratio (REGISTER 014) .

### 1.3.5 MEA

Press **【MEA】** to enter main screen.

For program reference point setting, this controller provides setting methods for two types of program coordinates. One is to enter the position of the machine coordinate program definition at the [OFFSET]. The second method is using G92 to define.

This function page [MEA] is using the latter method, which is using the tool position to set the reference point of the new coordinate system. The original coordinate system will be the working coordinate system. Once set, the absolute value is referring to this coordinate system to calculate. This system provides the setting method like the following figure :



**【ALL SET】** Using G92 to redefine X, Y, and Z axes absolute reference point position.

**【SET X】** Using G92 to set X axis absolute coordinate value to 0.

**【SET Y】** Using G92 to set Y axis absolute coordinate value to 0.

**【SET Z】** Using G92 to set Z axis absolute coordinate value to 0.

**Example:**

1. Move Z Axis-100, make the tool to touch working part's surface, right now the absolute coordinate is -100, and the machine coordinate is Z Axis-100
2. Pressing **【SET Z】**, using G92 to set Z axis coordinate to 0. Re-define Z axis reference point of absolute coordinate.

**Note:**

1. If you want to cancel coordinate measure function, you can go to manual home to complete. But if you didn't go back to HOME, even if you change the tool, the measure will continue the compensation value, please be careful for this.
2. If you didn't use this function a lot, we suggest you can close this function with Parameter 0238.

## 1.3.6 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.

00	N000000	EDIT	NO-RDY	LNC	
<b>ABSOLUTE</b>			<b>RELATIVE</b>		F12
X	0.000		X	0.000	CLR
Y	0.000		Y	0.000	TMR
Z			Z	0.000	F11
C			C	0.000	CLR
			<b>MACHINE</b>		CNT
Z	INPUT PASSWORD ****		Y	0.000	F10
C	0.000		Z	0.000	INI
			C	0.000	CNT
F:	0MM/MIN	F: 150%	RT: 100%	10:54	F9
S:	0RPM	S: 100%	CUT: 0H 0M 0S		MAX
			RUN: 0H 0M		CNT
			CNT: 10/ 100		F8
F2	OK	F3	CANCEL	F4	
F5		F6		F7	

**Note :** System need be to under preparation unfinished situation.

## 1.4 PROG

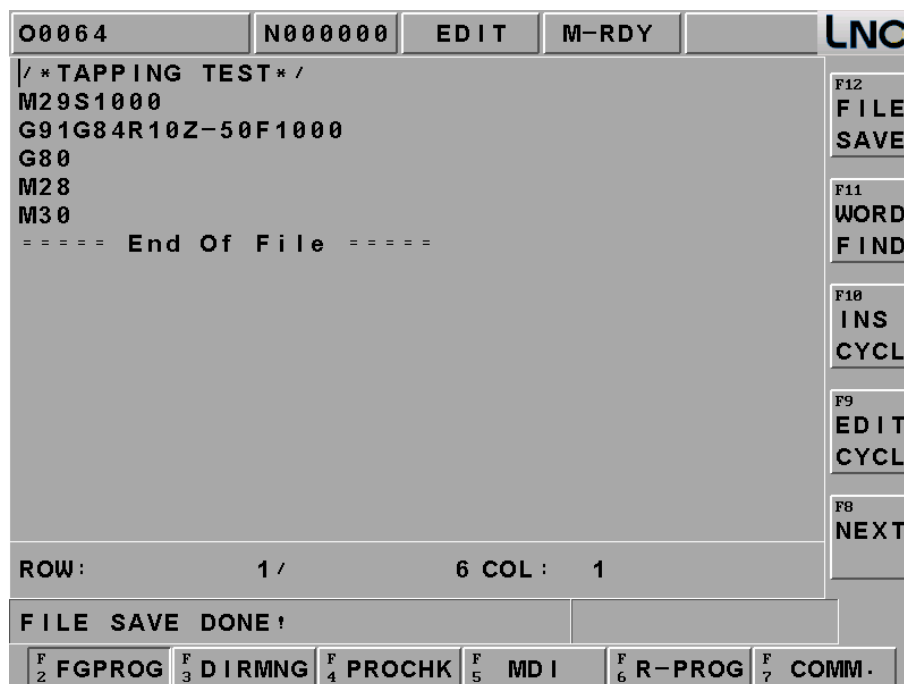
Pressing <PROG> to enter into program function group screen. This function group provides part program editing, file managing, checking, rebooting, RS232 transmitting and other related functions.

### 1.4.1 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. Press <confirm> to add space line, press <RESET> to make the cursor go back to head. Right side sub-function keys have below details :

- **【FILE SAVE】**

Press right side **【FILE SAVE】** function key to save current editing program or the file will save automatically when group page changing.



## ● 【WORD FIND】

You can input specific keywords to search, if there are two searching words in the program, you can search the second one again.

If the system can't find the word, it will show no words finding.



## ● 【INS CYCL】

This function is to provide each working method in order to produce program rapidly.

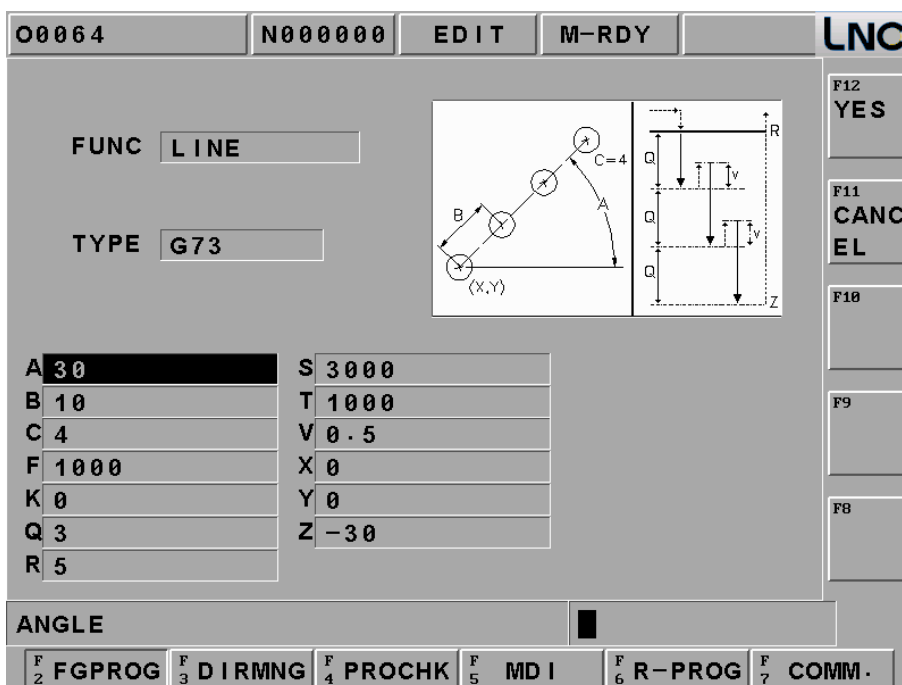
After pressing 【INS CYCL】 , the screen will be as below :



Pressing "YES" after selecting working method :



To press "YES" key to enter into the following Figure page. To move the highlighter to each parameter column and then press <YES> key :



Program code page is as following after pressing 【YES】 :

O0064	N000000	EDIT	M-RDY	LNC	
<pre>/*TAPPING TEST*/ M29S1000 G91G84R10Z-50F1000 G80 G101 A30. B10. C4 F1000. K0 M1 Q3. R5. S3000 M28 M30 ===== End Of File =====</pre>				F12 GOTO	
				F11 LDEL	
				F10 MARK	
				F9 UN MARK	
				F8 NEXT	
ROW: 5 / 7 COL: 1					
F2 FGPROG	F3 DIRMNG	F4 PROCHK	F5 MDI	F6 R-PROG	F7 COMM.

- **【EDIT CYCL】**

This function key is effective when the editing cursor is stop at **【INS CYCL】** command block or there is G Code in that **【INS CYCL】** block. This function key is for users to edit previous program. To press this key will enter into “Insert-3” page. If there is no assigned M Parameter (working style) value after G code, “Insert-2” page will occur. Operation and insert is totally the same. Each command’s detail description, please refer to **【CAM】** group introduction.

- **【GOTO】**

After pressing right side **【GOTO】** function key, you can input the line number directly at dialogue box input program and press enter to make the cursor to move to this location, as below.

O0064	N000000	EDIT	M-RDY	LNC
/ * TAPPING TEST * / M29S1000 G91G84R10Z-50F1000 G80 G101 A30. B10. C4 F1000. K0 M1 Q3. R5. S3000 M28 M30 = = = = End Of File = = = =				F12 GOTO
LN : 5				F11 LDEL
				F10 MARK
				F9 UN MARK
				F8 NEXT
ROW: 5 / 7 COL: 1				
F2 OK	F3 CANCEL	F4	F5	F6
F7				



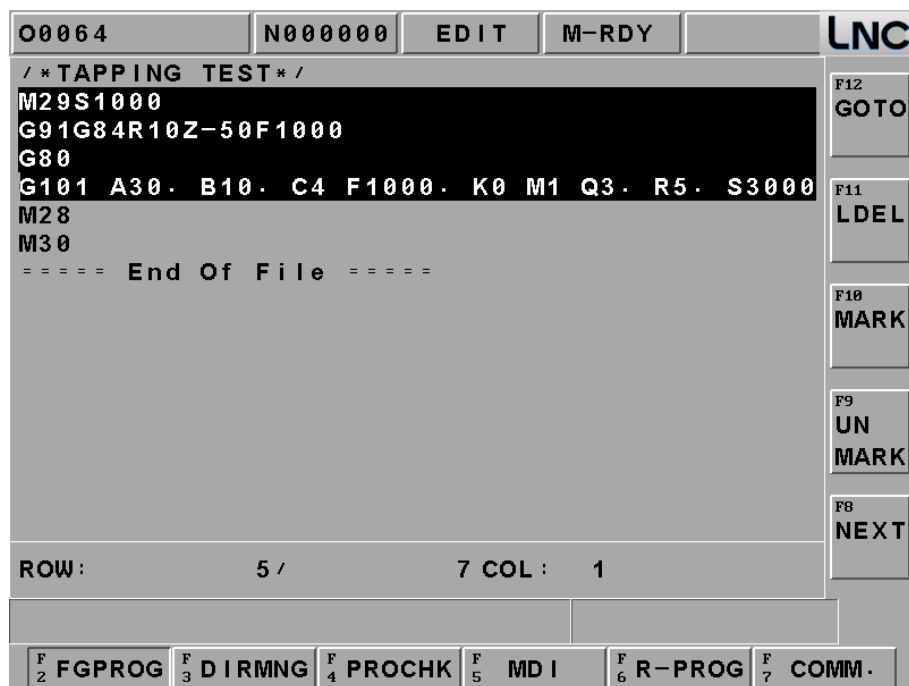
- **【LDEL】**

Able to delete a row of program codes at the cursor position. Note: It can not recover after deleting.

- **【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.

Note: After marking, you can use COPY and LDEL to do multi-line program editing.



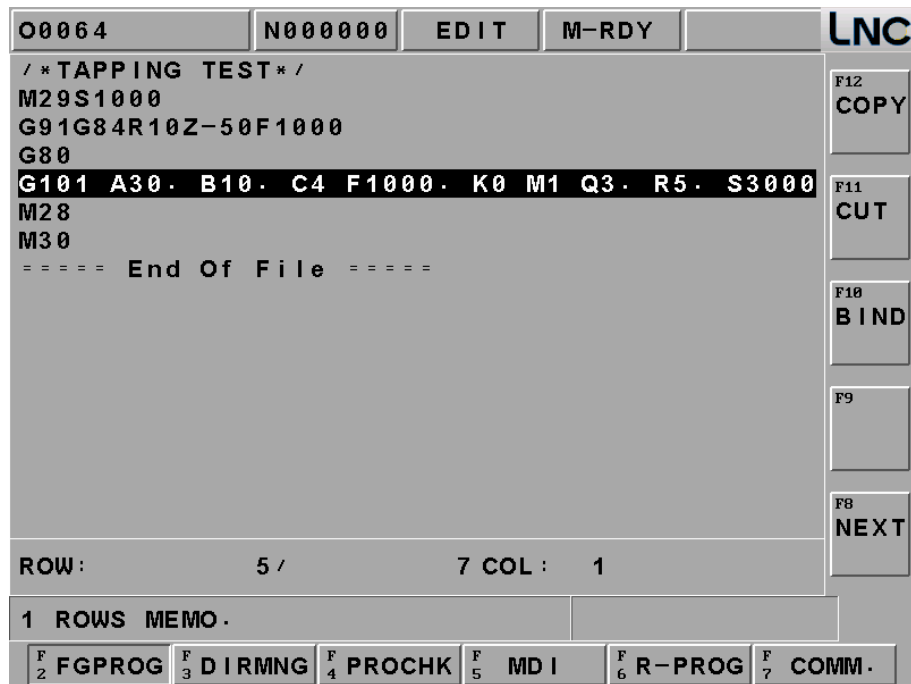
- **【UNMARK】**

- To cancel previous marking sign.

Note: If you will not use **【MARK】** anymore, please use **【UNMARK】** to prevent mistakes.

- **【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.

- **【Teach Mode】 / 【Delete Teach】**

Press this button to show ABS/REL/MAC at the down side for reference. At the right side of sub-function keys will show **【INSERT COORDINATE】** for users to insert current location's absolute coordinate quickly, detail is as below :

O0064		N000000		EDIT		M-RDY		LNC		
/ * TAPPING TEST * / M29S1000 G91G84R10Z-50F1000 G80 <b>G101 A30. B10. C4 F1000. X0 M1 Q3. R5. S3000</b> X0.000Y0.000Z0.000C0.000 M28 M30 = = = = End Of File = = = =										F12 THIN CANCEL
										F11 THIN INST
										F10
ABSOLUTE			RELATIVE			MACHINE				
X	0.000	X	0.000	X	0.000					
Y	0.000	Y	0.000	Y	0.000					
Z	0.000	Z	0.000	Z	0.000					
C	0.000	C	0.000	C	0.000					
ROW: 6 / 7 COL: 1										F9
										F8 HOME
F2 FGPROG		F3 DIRMNG		F4 PROCHK		F5 MDI		F6 R-PROG		F7 COMM.

## ● 【BGPROG】

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.

Note: Background file is 08999 permanently.

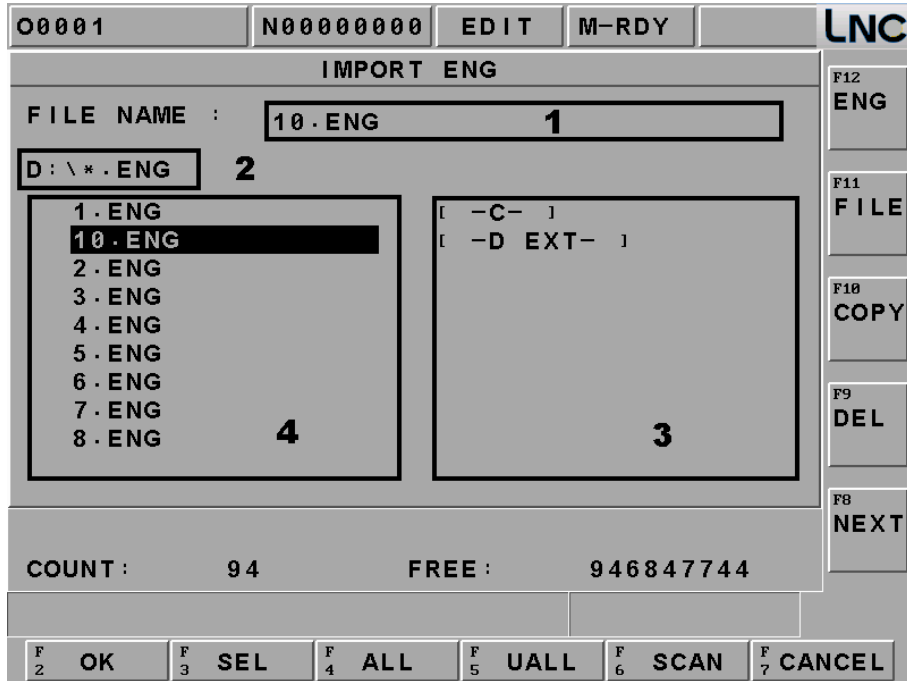


### 1.4.2 DIRMNG

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

- **【Import ENG】**

Press **【IMP ENG】** , you will see a window, you can choose or input import file path :



1 : show file name(you can input resource path and file name)

2 : import source file path.

3 : disk and directory

4 : file area

**【SEL】** : use line bar to select file.

**【YES TO ALL】** : select all ENG file.

**【NO TO ALL】** : cancel selection to all ENG.

**【CANCEL】** : cancel this function, return to DIRMNG.

O0001	N00000000	EDIT	M-RDY	LNC
IMPORT ENG				
FILE NAME :		3.ENG		
D:\*.ENG				
1.ENG v 10.ENG 2.ENG v 3.ENG 4.ENG 5.ENG 6.ENG 7.ENG 8.ENG		[ -C- ] [ -D EXT- ]		
COUNT : 94		FREE : 946847744		
F2 OK	F3 USEL	F4 ALL	F5 UALL	F6 SCAN
		F7 CANCEL		

Choose above file and press **【OK】** to see below.

O0001	N00000000	EDIT	M-RDY	LNC
<BATCH>		BATCH PROGRAM		
<RS232>		IMPORT ENG		
10. NC	SAVE Z	32	Y49.961	Z
3. NC	50	32	Y49.961	Z
ARITH	READY Z	ñσêçσèèμπσ		
G101-105	10			
G111-114	Z F	MENT * /		
G121-123	2000			
G131-133	CUT F			
KOR. NC	1000			
O0000	:M0:			
O0000000				
O0000001				
O0001				
O0011				
O0022				
G131-133	1128	04:28 AM 01/02/2002		
COUNT :	102	FREE : 946536448		
10.ENG: IMPORTING 0%				
F2	F3 CANCEL	F4	F5	F6
		F7		

Input value into above column and press **【OK】** to start transformation, you will see % at the hint line. After finishing, you will see 「ENG complete」, there will be a file in the DIRMNG.

O0001	N00000000	EDIT	M-RDY	LNC
<BATCH>	BATCH PROGRAM			F12 ENG
<RS232>	IMPORT ENG			F11 FILE
10.NC	SAVE Z	32 Y49.961 Z		F10 COPY
3.NC	50	32 Y49.961 Z		F9 DEL
ARITH	READY Z	Ñσêçσëèμπσ		F8 NEXT
G101-105	10			
G111-114	Z F			
G121-123	2000	MENT * /		
G131-133	CUT F			
KOR.NC	1000			
O0000	:M0:			
O0000000				
O0000001				
O0001				
O0011				
O0022				
G131-133	1128	04:28 AM	01/02/2002	
COUNT:	102	FREE:	946536448	
10.ENG: IMPORTING 0%				
F 2	F 3 CANCEL	F 4	F 5	F 6
				F 7

You can confirm this file at DIRMNG.

**ENG program :**

- Main program(xxx.nc), there are cutting and drilling according path attribute.

G65 P8991 T0 X-15.806 Y-28.855 Z22.500 A200 B1000 C50.000 D10.000;

...

G65 P8993 T0 X-15.806 Y-28.855 Z15.000 A200 B1000 C50.000 D10.000;

...

G65 P8992 T0 X12.217 Y-28.855 Z7.500 A200 B1000 C50.000 D10.000;

M30

- 3 sub-program for cutting, customer can modify this by themselves.

■ Head : O8991

;O8991: ENG CUT HEAD	Cutting path head sub program
;A(#1): Z FEEDRATE	Z axis feedrate
;B(#2): CUT FEEDRATE	Cutting speed
;C(#3): SAFE Z	Safe height Z
;D(#4): READY Z	Prepare height Z
;T(#20): TOOL NO	Tool number
;X(#24): FIRST X	1 X position
;Y(#25): FIRST Y	1 Y position
;Z(#26): FIRST Z	1 Z position
T#20	Change tool
G00 X#24 Y#25 Z#3	Move to safe height
Z(#26+#4)	Z move to prepare
G01 Z#4 F#1	Z move tool point
F#2	Appointed cutting speed
M99	

■ End : O8992

;O8992: ENG CUT TAIL	Cutting path head sub program
;A(#1): Z FEEDRATE	Z axis feedrate
;B(#2): CUT FEEDRATE	Cutting speed
;C(#3): SAFE Z	Safe height Z
;D(#4): READY Z	Prepare height Z
;T(#20): TOOL NO	Tool number
;X(#24): FIRST X	1 X position
;Y(#25): FIRST Y	1 Y position
;Z(#26): FIRST Z	1 Z position



G00 Z#3 M99	Move to safe height
----------------	---------------------

■ Change tool : O8993

;O8993: ENG CUT TOOL ;A(#1): Z FEEDRATE ;B(#2): CUT FEEDRATE ;C(#3): SAFE Z ;D(#4): READY Z ;T(#20): TOOL NO ;X(#24): FIRST X ;Y(#25): FIRST Y ;Z(#26): FIRST Z G00 Z#3 T#20 G00 X#24 Y#25 Z(#26+#4) G01 Z#4 F#1 F#2 M99	Cutting path head sub program Z axis feedrate Cutting speed Safe height Z Prepare height Z Tool number 1 X position 1 Y position 1 Z position Move to safe height Change tool Move to safe area Z move to prepare Z move tool point Appointed cutting speed
---	---

- Drill uses 3 sub program, customer can modify by themselves.

■ Head : O8981

;O8981: ENG DRILL HEAD ;A(#1): Z FEEDRATE ;B(#2): CUT FEEDRATE ;C(#3): SAFE Z ;D(#4): READY Z ;T(#20): TOOL NO ;X(#24): FIRST X ;Y(#25): FIRST Y ;Z(#26): FIRST Z T#20 G00 X#24 Y#25 Z#3 M99	Cutting path head sub program Z axis feedrate Cutting speed Safe height Z Prepare height Z Tool number 1 X position 1 Y position 1 Z position Change tool Move to safe area
---	---

■ End : O8982

;O8982: ENG DRILL TAIL ;A(#1): Z FEEDRATE ;B(#2): CUT FEEDRATE ;C(#3): SAFE Z ;D(#4): READY Z ;T(#20): TOOL NO ;X(#24): FIRST X ;Y(#25): FIRST Y ;Z(#26): FIRST Z G00 Z#3 M99	Cutting path head sub program Z axis feedrate Cutting speed Safe height Z Prepare height Z Tool number 1 X position 1 Y position 1 Z position Move to safe height
---	--

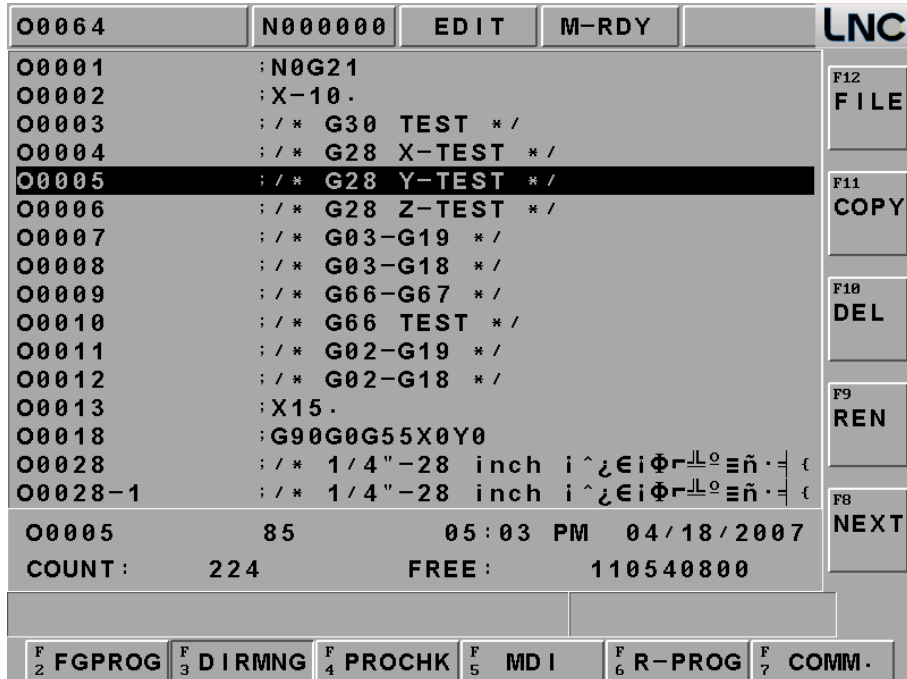
■ Change tool : O8983

;O8983: ENG DRILL TOOL ;A(#1): Z FEEDRATE ;B(#2): CUT FEEDRATE ;C(#3): SAFE Z ;D(#4): READY Z ;T(#20): TOOL NO ;X(#24): FIRST X ;Y(#25): FIRST Y ;Z(#26): FIRST Z G00 Z#3 T#20 G00 X#24 Y#25 M99	Cutting path head sub program Z axis feedrate Cutting speed Safe height Z Prepare height Z Tool number 1 X position 1 Y position 1 Z position Move to safe height Change tool Move to safe area
--	--

- **【FILE】**

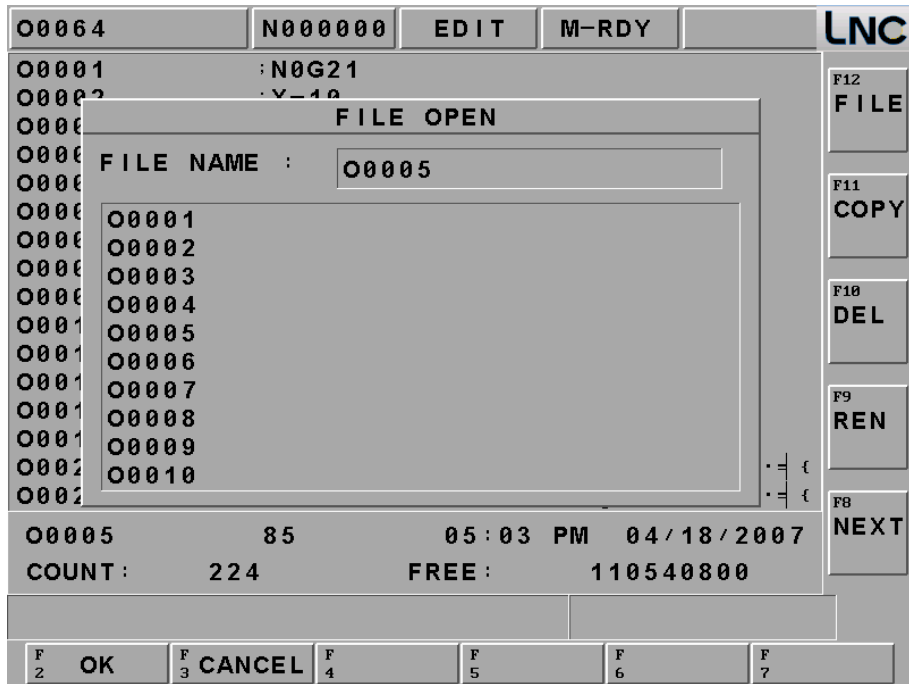
## Solution 1

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>

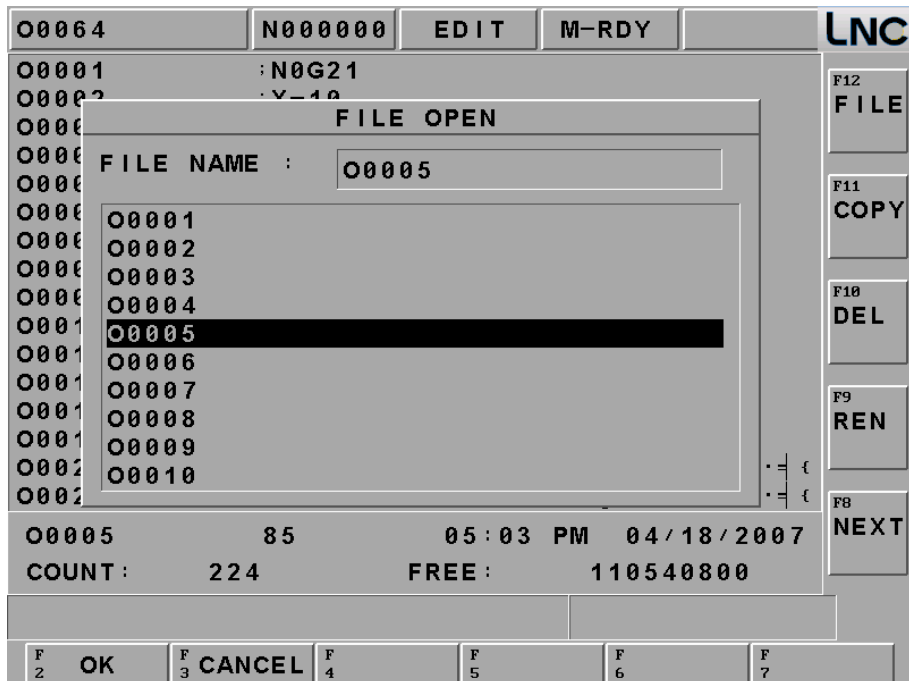


## Solution 2

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 <OK> to open the file. Or in the dialog box mode to press left or right button to switch line for cursor.



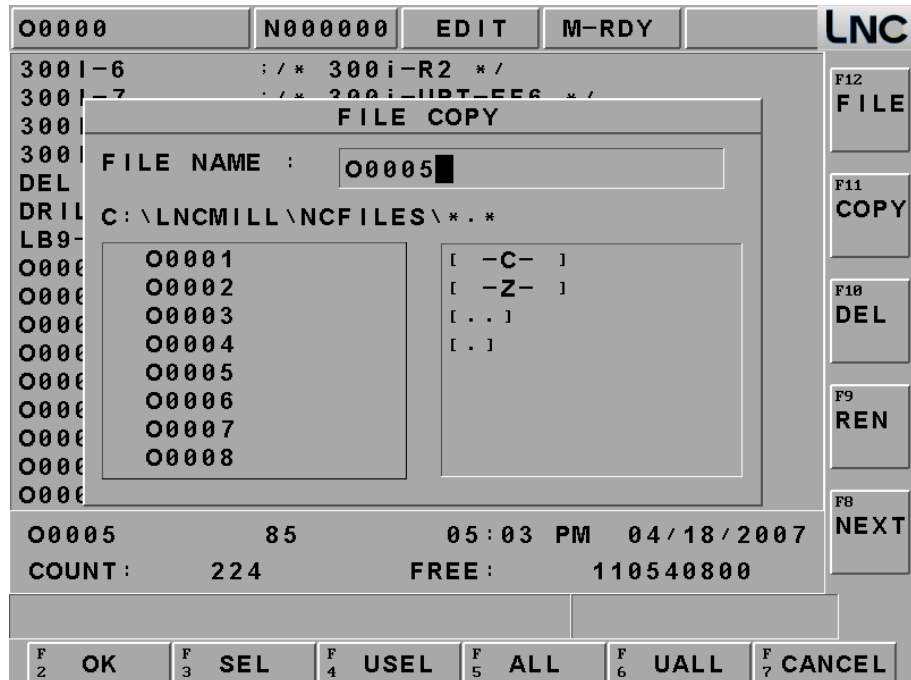
Or inside this window, move left/right to move line bar to open file.



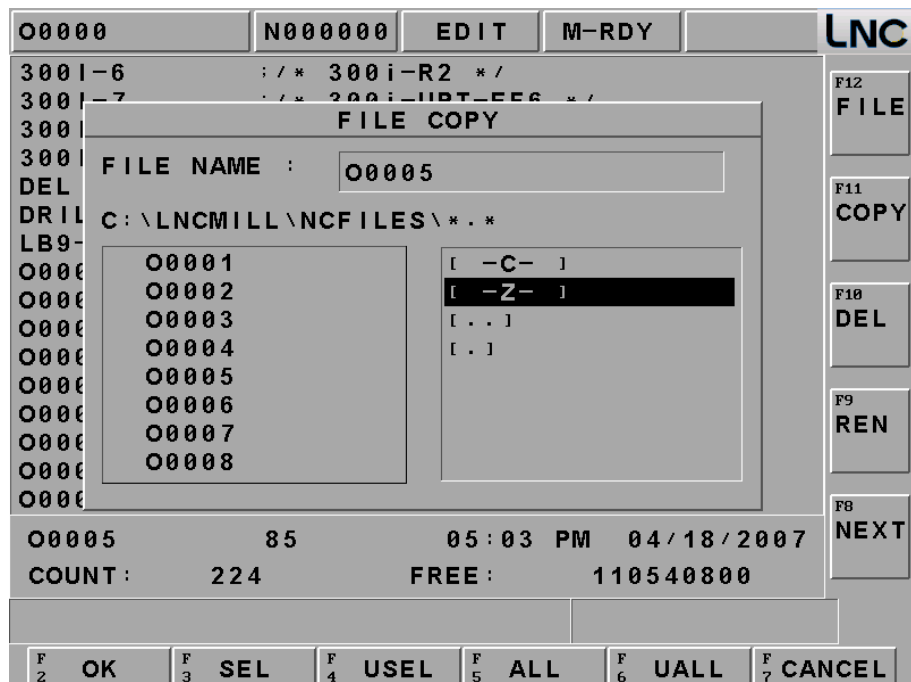
Note : Under **AUTO**, open file=background program; under **EDIT**, open file=current file. You can not open file at other mode.

## ● 【COPY】

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. Such as below :



Or in the dialog box mode to press left or right button to switch line for cursor to offer different path switch methods.



When input is complete, press confirm to enter target file's input page. At the target input page, you are 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. As below.

O0001	N00000000	EDIT	M-RDY	LNC	
<b>TARGET DIRECTORY</b>					
DIR NAME :		D:\NCFILES\			
[ -C- ] [ -D EXT- ] [ . . ] [ . ]					
COUNT : 102      FREE : 946421760					
F2 OK	F3 CANCEL	F4	F5	F6 SCAN	F7

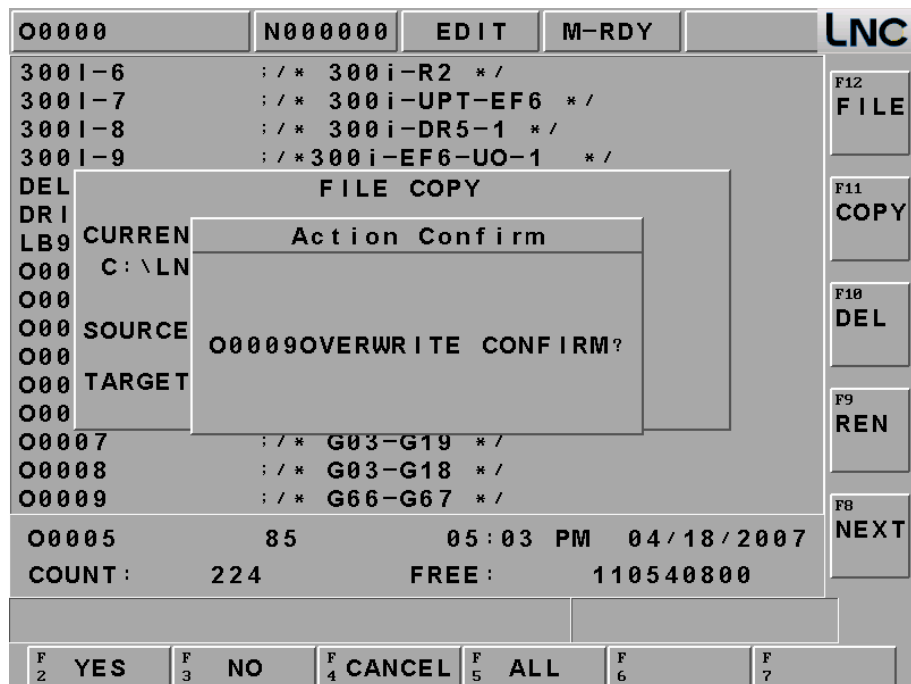
F12  
ENG  
  
 F11  
FILE  
  
 F10  
COPY  
  
 F9  
DEL  
  
 F8  
NEXT

Press **【OK】** to go to target file, you can input copy file name at target column :

O0000	N0000000	EDIT	M-RDY	LNC	
3001-6 : / * 300i-R2 * / 3001-7 : / * 300i-UPT-EF6 * / 3001-8 : / * 300i-DR5-1 * / 3001-9 : / * 300i-EF6-UO-1 * /					
<b>FILE COPY</b>					
DEL DRI LB9 CURRENT O00 C:\LNCMILL\NCFILES O00 O00 SOURCE O0005 O00 TARGET O0009 O00					
O0007 : / * G03-G19 * / O0008 : / * G03-G18 * / O0009 : / * G66-G67 * /					
O0005 85 05:03 PM 04/18/2007 COUNT : 224      FREE : 110540800					
F2 YES	F3 CALCEL	F4	F5	F6	F7

F12  
FILE  
  
 F11  
COPY  
  
 F10  
DEL  
  
 F9  
REN  
  
 F8  
NEXT

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

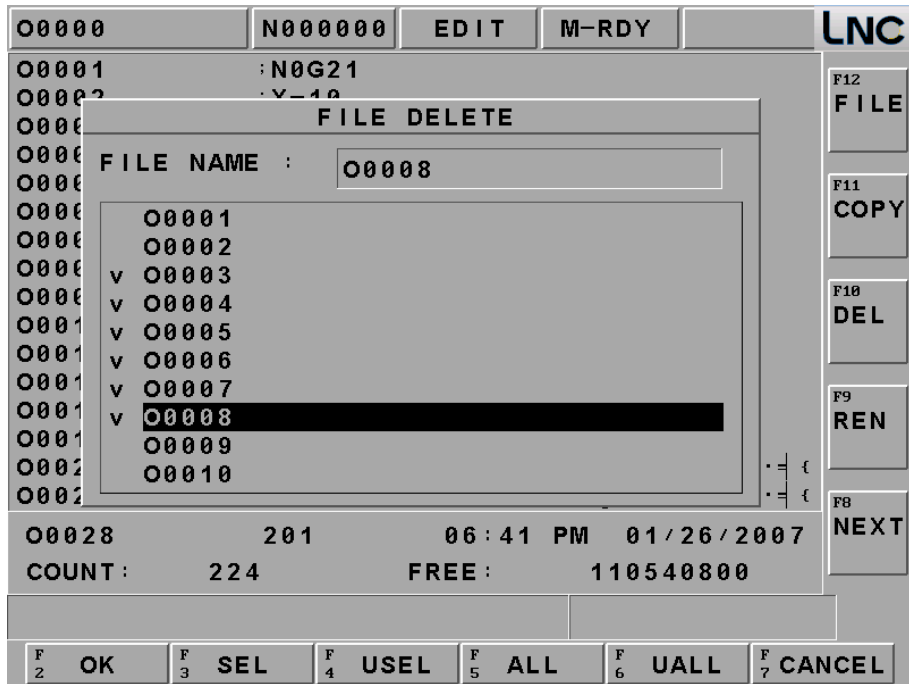


If the program is copying, the message-reminding dialog box will occur “Copying....”. When complete, the message-reminding dialog box will show “Copy Complete”.

### ● 【COPY】 MULTI-COPY

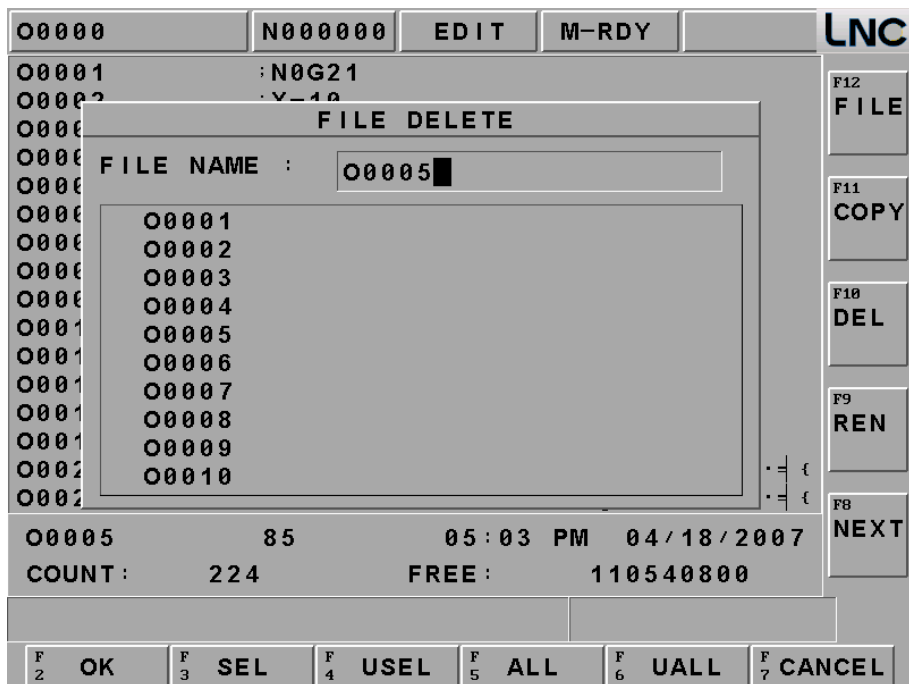
Except single file copy, our controller also offers multi-file copy function. In the dialogue box mode, you can press left or right button to move cursor to switch line to the left side. Press select to choose check box and press confirm later.

You can also use 【USEL】 【ALL】 【UALL】 to manage file.



- **【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. Press [OK], system will show O0005 DELETE FILE, after complete deleting, users can use PROCHK to confirm.

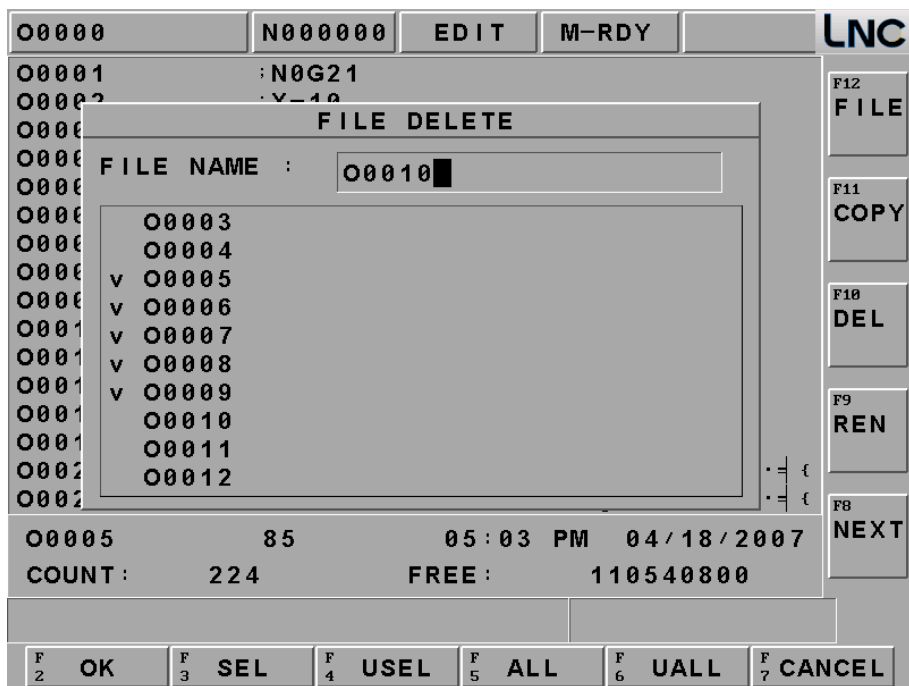




## ● 【DEL】 MULTI-COPY

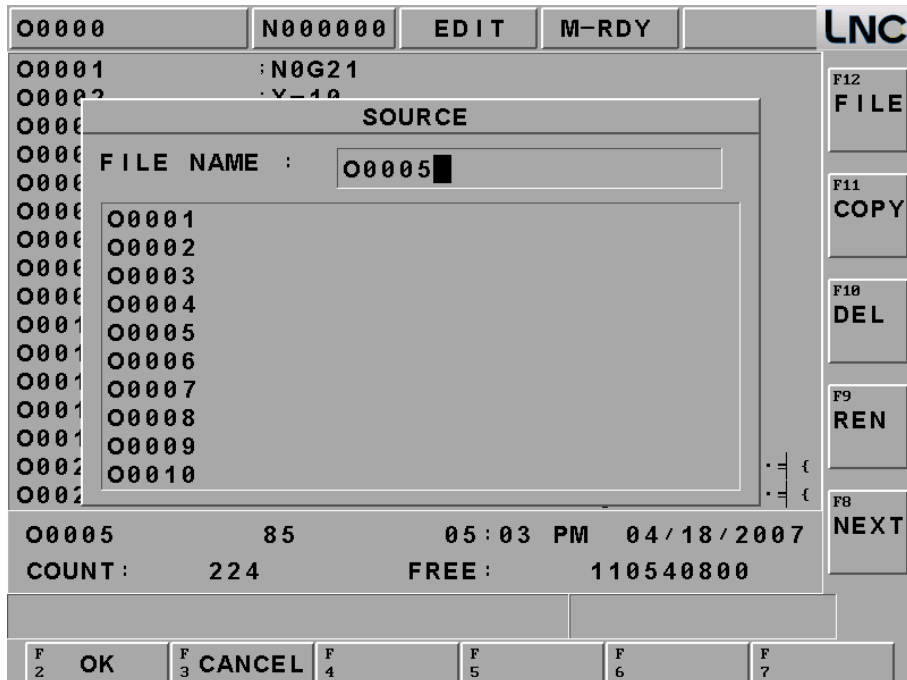
Except single file delete, our controller also offer multi-file delete function. In the dialogue box mode, you can press left or right button to move cursor to switch line to the left side. Press SEL to choose check box and press confirm to delete. After pressing confirm, the system will pop up hint message to double confirm if you really want to delete these files.

You can also use 【USEL】 【ALL】 【UALL】 to manage file.

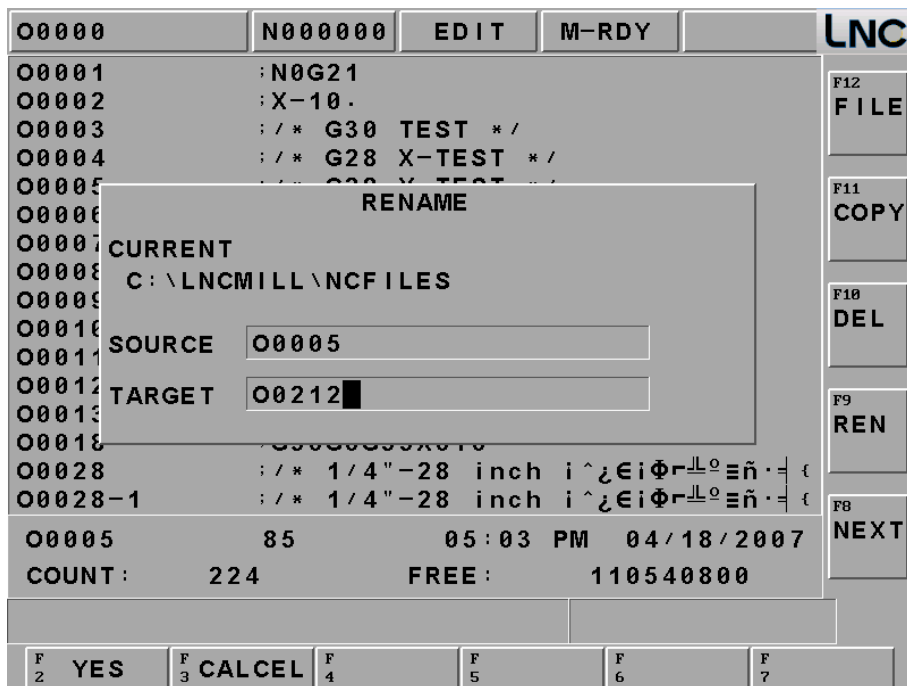


## ● 【REN】

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 OK button to enter into the file rename window.



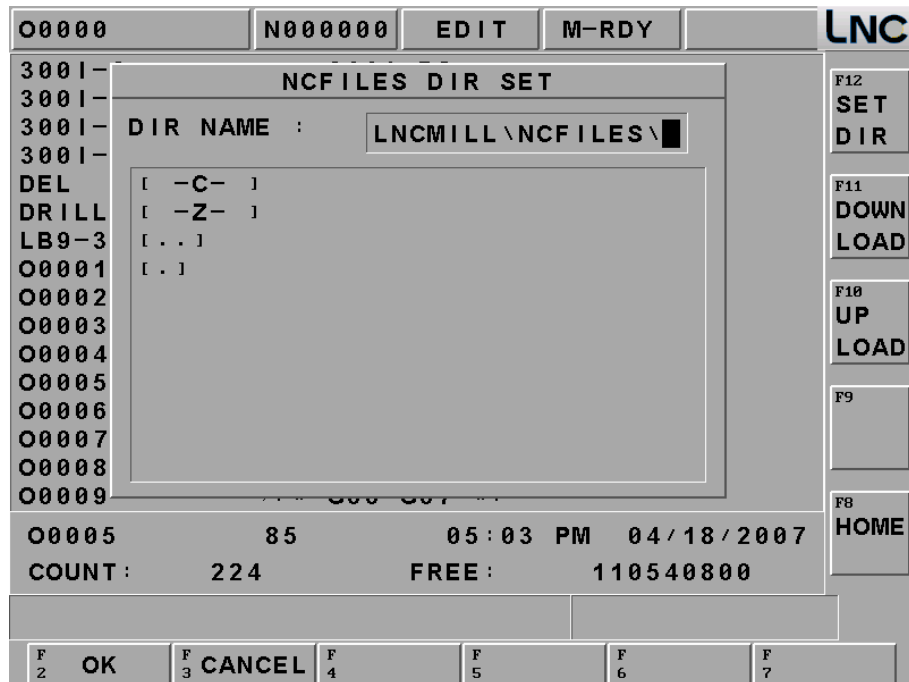
2. To enter the destination file name in the rename windows and then to press confirm button. After rename complete, users will be able to use program list to confirm.



## ● 【SET DIR】

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 NCFILES by using this dialog box or by entering work path directly.

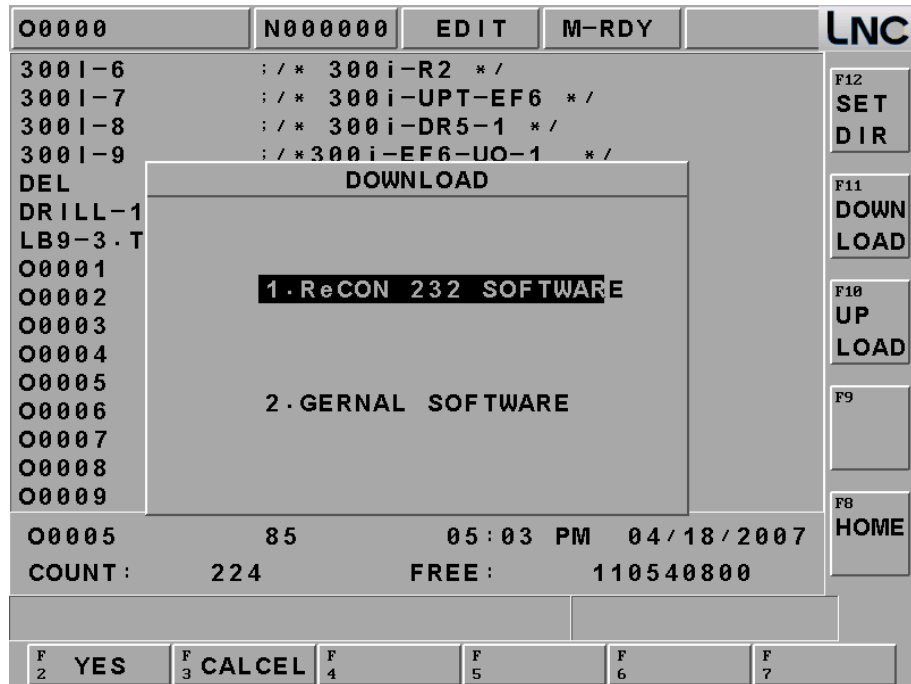
Note: The name of the directory can not be the same with our current controller's files.



## ● 【DOWNLOAD】

It means sending file from PC end to NC end, and also command NC end to save the file at NC end's external device.

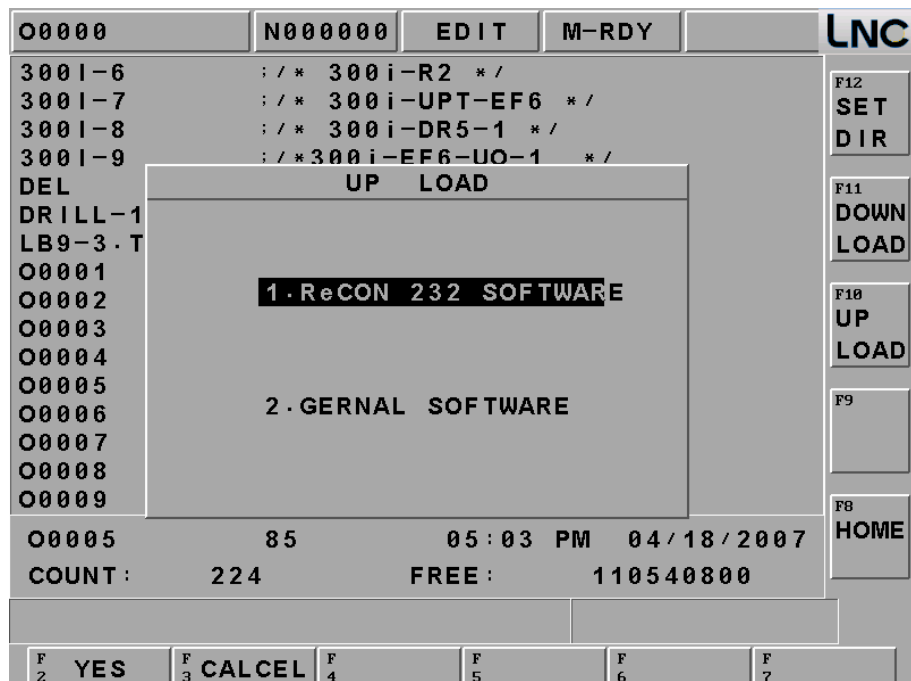
Note : As for detail information, please refer to RS232 communication software chapter.



#### ● 【UPLOAD】

It means sending file from NC end to PC end, and also command PC end to save the file at PC end's external device.

Note : As for detail information, please refer to RS232 communication software chapter.



### 1.4.3 PROCHK

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. :

- **【SINGLE BLOCK CHECK】**

You will see program content at the upper side of the monitor. The program that was executing right now will be highlighted. At the down side of the monitor is the absolute coordinate, dist to go , M/S/G/T CODE, exact speed and so on.

O0318		N000000		MEM	CSTART	LNC	
/ * BENZ * /							
G21							
G90							
M01							
T1							
ABSOLUTE		DIST TO GO		( G )			
X	0.000	X	0.000	G54	G01	G17	
Y	0.000	Y	0.000	G90	G23	G94	
Z	0.000	Z	0.000	G21	G40	G49	
C	0.000	C	0.000	G80	G98	G50	
				G67	G64	G69	
				G15	G50.1		
LN:	-1	FO:	150%	F	1000.000	H	0
AF:	0	RO:	100%	R		M	
AS:	0	SO:	100%	P		S	0
				Q		T	0
							HSP
F2	BGPROG	F3	DIRMNG	F4	PROCHK	F5	MDI
F6	R-PROG	F7	COMM.				

- **【AIC】**

This is function for digital controller model, system offers (Material 1~material 40) processing parameter memory function, all you need to do is to set up P.1932=1.

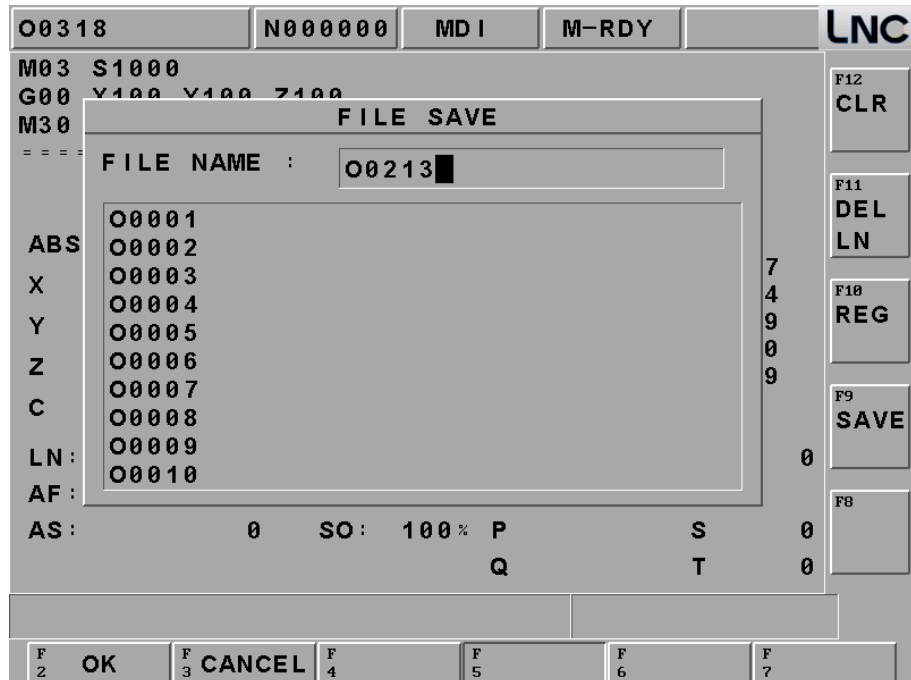
Press **【SEL AIC】** and move to the material you want.

O0001		N00000000		JOG	M-RDY	LNC	
FROM / 150 · 120 · 41						F12 AIC	
TOOL / MILL · 14 · 0 · 35							
color · 0 · 255 · 255							
STOCK / 120 · 100 · 40 · 0 · 0 · 0							
N1G1G90X150 · Y120 · Z0F30S500							
N2X110 · Y105 · F100							
ABSOLUTE		DIST TO GO		AIC		F11 CHK	
X	0.000	X	0.000	ITEM 1			
Y	0.000	Y	0.000	ITEM 2			
Z	0.000	Z	0.000	ITEM 3			
LN:	-1	FO:	100%	ITEM 4	G )		
AF:	0	RO:	10%	ITEM 5	01 G17		
AS:	0	SO:	100%	ITEM 6	23 G94	F10	
				ITEM 7	40 G49		
				ITEM 8	98 G50		
				ITEM 9	54 G69		
				ITEM 10	50.1	F9	
				ITEM 11			
				ITEM 12	0 M 0		
				ITEM 13	0	F8	
				ITEM 14	0 S 0		
				ITEM 15	0		
				ITEM 16	0 T 0		
						ITEM	
F2	OK	F3	CANCEL	F4		F6	



- **【SAVE】**

When you key-in multi-line command under MDI Mode, you can press **【SAVE】** and input file name you want to save at the dialogue box. Then the MDI file is complete.





### 1.4.5 PROGRAM RESTART

During program working, if the tool brakes, emergency stop happens, or other situations to make the working process stop, this function can search the interruption point (single block series or program code) to restart the processing. It can also be a quick exam function.

Note : The way to set R-PROG is to make Pr.650 to be 1 and use R-PROG 2 (call O9888)

R-PROG Point.

#### Program restart point :

- Allow input limit

When NC is at Auto>OK, you can input the position you want to search at the input line.

O5678	N000000	MEM	M-RDY	LNC
% N2 G21 N3 M01 N4 T08M06 N5 G90G00G54X0Y0  RES POINT            0  PROG LINE           -1            ( M )  RES POSITION    DIST TO GO X        0.000    X        0.000 Y        0.000    Y        0.000 Z        0.000    Z        0.000    ( S ) C        0.000    C        0.000    ( T )  F0				RES FIND  <input type="text"/>  <input type="text"/>  <input type="text"/>  <input type="text"/>
BGPROG	DIRMNG	PROCHK	MDI	R-PROG
				COMM.

## ● Program Line Search

Input R-PROG code (EX:400), press RESFIND.

O5678	N000000	MEM	M-RDY	LNC
N2 G21 N3 M01 N4 T08M06 N5 G90G00G54X0Y0  RES POINT 400 PROG LINE -1 (M) RES POSITION DIST TO GO X 0.000 X 0.000 Y 0.000 Y 0.000 Z 0.000 Z 0.000 (S) C 0.000 C 0.000 (T) F0				RES FIND  <input type="button" value=""/> <input type="button" value=""/> <input type="button" value=""/> <input type="button" value=""/>
BGPROG	DIRMNG	PROCHK	MDI	R-PROG
				COMM.

NC will start to search this processing file from the first line, when “400” window pops out at the screen, it means the search is completed.

O5678	N000398	MEM	B-STOP	LNC
N395 X23.195Z28.344 N396 X23.594Z28.169 N397 X24.096Z27.964 N398 X25.591Z27.271 N399 X26.666Z26.711  RES POINT PROG LINE RES POSITION X 25.591 Y 68.368 Y 68.368 Z 27.271 Z 30.508 (S) C 0.000 C 0.000 2300 (T) F0 8				<input type="button" value=""/> <input type="button" value=""/> <input type="button" value=""/> <input type="button" value=""/> <input type="button" value=""/>
HINT RES POINT FIND 7 3 6				
OK				

Note: Processing sub-program or MACRO is not in the searching field.

### ● Single Code Search

Input R-PROG single code (EX:N555), press RESFIND, NC will start to search this processing file from the first line, when program finds the same value (include processing sub-program and MACRO), it will show a pop up message window "Restart point has found". If this single code is the restart point, users can start the next step to process. If not, users can press NEXT to search a suitable section again.

O5678	N000555	MEM	B-STOP	LNC
N555 X14.988Z33.701 N556 X14.284Z33.725 N557 X-33.386Z33.732 N558 X-34.724Z33.682 N559 X-35.257Z33.644				
RES POINT N555				
PROG LINE 557 (M) 7 3 6				
RES POSITION DIST TO GO				
X	14.988	X	14.988	
Y	60.268	Y	60.268	
Z	33.701	Z	33.701	(S) 2300
C	0.000	C	0.000	(T) 8
F0				
BGPROG	DIRMNG	PROCHK	MDI	R-PROG COMM.

Note: Single code of canned cycle is not in the searching field.

If input format is not program or single code, it will show format error at NC end.

If you can not find the code or the single block that you input at the main program, NC will show "Can not find R-PROG point " message on window screen

1. **Program Code:** shows the section code of main program
2. **Restart point location:** press RESFIND to find restart point, it will show the absolute point location of program restart point.
3. **Dist to go:** press RESFIND to find restart point, it will show the distance for cutting tool to move from now to the restart point.

4. **(M)** : Press RESFIND to find restart point, the newest 14 M code will be arranged in order according to searching sequence.
  - During searching of R-PROG, if you look for M99(cutting of main loop program), NC program will stop and leave searching.
  - During searching of R-PROG, NC will show the newest 14 M code, but not include M code as below: M00(program stop), M01(select to stop) , M02(end program) , M30(end program & return head) , M98(call sub-program) , M99(return main program) , M codes(which can call macro programs)
5. **(S)** : Press RESFIND to find restart point, the newest S code will be arranged in order according to searching sequence.
6. **(T)** : Press RESFIND to find restart point, the newest 2 T code will be arranged in order according to searching sequence.

#### ● RESFIND

When users have input the restart point, press this key, NC will start to search and process restart. During searching, changing mode is not allowed. But if users change modes while searching, NC end will find the restart point firstly and change mode later.

#### ● Cycle Start

When you process Cycle Star in the process of RESFIND, you will need to confirm if every servo axis' actual position is on the restart point (no distance for every axis). If NC is on the restart point, NC will start process working at the restart point.

If the position is not on the restart point, when you press Cycle Start, NC will not have any response.

**EXIT R-PROG** : Below are the ways to exit R-PROG.

1. Find out restart point, when every axis turns back to restart point (no distance for every axis), press Cycle start to start processing, and complete R-PROG.
2. Press RESET to leave R-PROG, NC end has cleared searching data.
3. System switches to origin mode, turns back to the origin, and exit R-PROG.

#### 1.4.6 RS232

Clicking the **【COMM】** function button to receive and transmit programs between the controller and other PCs.  
Operation and description of sub-function buttons **【COMM】** and **【SETT】** are as following :

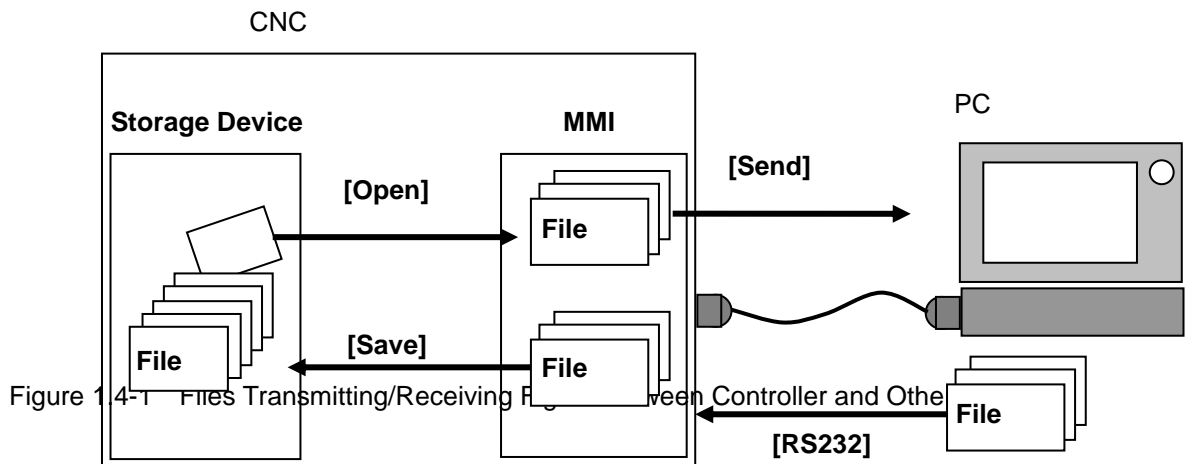
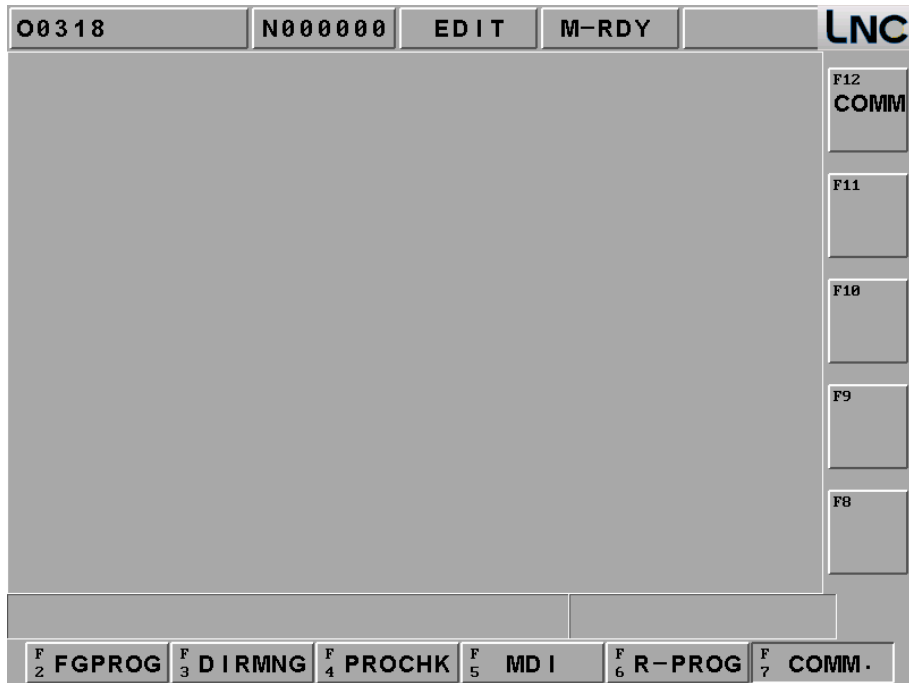


Figure 1-4-1 Files Transmitting/Receiving Between Controller and Other

- **【SEND】**

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.



Figure 1.4-2 send main screen

**【SEND】** Before pressing send button, the file select window will be pop up. After choosing the file you want to send, press OK to send by RS232.

**【READ】** Before PC starts transmitting, must pressing this function key in order to let the system to prepare to receive data.

**【SAVE】** When external device send a program file to controller's RS232 window, press this button to choose save directly with file path and name or save as other files.

**【CLEAR】** Clear all program files in the windows.

**【RESET】** Giving up file transmitting and resetting communication deal in order to build up connect with RS232.

Note : Detail information; please refer to RS232 communication software chapter.

## 1.5 OFFSET

Pressing <OFFSET> to enter into the offset function screens such as **【TL. OFF】**, **【TDC】**, **【MACRO】**, **【WORK】**, and **【TOOL】**. Users are able to modify these settings under MDI mode or machine ready mode.

### 1.5.1 TL. OFF

Pressing **【TL. OFF】** to enter into the below screen and to key-in data manually under MDI mode.

1. 99 sets of setting for Tool Offset. To switch the setting screen by using <PAGE↓> and <PAGE↑> key.
2. To move cursor on the wanted setting numbers. To key-in the setting value in the data enter rows and then to press <INPUT> key to write into the controller.
3. Absolute coordinate will change with tool offset every time the part program uses one tool offset number.

The value is

ABS = MAC–Outside Offset–G54~G59 Offset– (tool outlook compensation +tool wore out compensation)

O0318		N000000		EDIT		M-RDY		LNC		
NO.	RADIUS	ZLENGTH	XLENGTH	YLENGTH						
01	0.000	0.000	0.000	0.000					F12 ABS	
02	0.000	0.000	0.000	0.000						
03	0.000	0.000	0.000	0.000						
04	0.000	0.000	0.000	0.000	>	>			F11 INC	
05	0.000	0.000	0.000	0.000						
06	0.000	0.000	0.000	0.000				1 / 2		
07	0.000	0.000	0.000	0.000						
08	0.000	0.000	0.000	0.000	>	>			F10 NORU NT	
09	0.000	0.000	0.000	0.000						
10	0.000	0.000	0.000	0.000						
INPUT: ABS UNIT: MM					--PAGE: 1 / 10--					F9 MINU NT
RELATIVE					MACHINE					F8
X	0.000		X	0.000						
Y	0.000		Y	0.000						
Z	0.000		Z	0.000						
C	0.000		C	0.000						

F2 TL. OFF	F3 TDC	F4 MACRO	F5 WORK	F6	F7 TOOL
---------------	-----------	-------------	------------	----	------------

**【ABS】** : Input by absolute value, ex: value 0.500, input-0.1, and it will show -0.100.

**【INC】** : Input by increase value, ex: value 0.500, input-0.1, and it will show 0.400.

**【NORUNT】** : Input by normal unit, ex: value 0.500, input 1, and it will show 1.000.

**【MINUNT】** : Input by minimum unit, ex: value 0.500, input 1, and it will show 0.001.

### 1.5.2 TDC

When using different tool, you can use different TDC(TDC value and TDC time) to control.TDC will depend on users' experience to input setting. Compensation will base on processing time, the more processing time, the more compensation. It's a linear relationship, when it comes to the setting TDC, it will maintain the same. Meanwhile this function has no relation with current machine's position.

1. 99 sets of setting for TDC. To switch the setting screen by using <PAGE↓> and <PAGE↑> key.
2. To move cursor on the wanted setting numbers. To key-in the setting value in the data enter rows and then to press <INPUT> key to write into the controller.

O0318		N000000		EDIT	M-RDY	LNC	
NO.	TDC . VAL	TDC . TIME	NO.	TDC . VAL	TDC . TIME		
01	0	0.0	11	0	0.0	F12	
02	0	0.0	12	0	0.0		
03	0	0.0	13	0	0.0	F11	
04	0	0.0	14	0	0.0		
05	0	0.0	15	0	0.0	F10	
06	0	0.0	16	0	0.0		
07	0	0.0	17	0	0.0	F9	
08	0	0.0	18	0	0.0		
09	0	0.0	19	0	0.0	F8	
10	0	0.0	20	0	0.0		
--PAGE : 1 / 5--							
RELATIVE			MACHINE				
X	0.000		X	0.000			
Y	0.000		Y	0.000			
Z	0.000		Z	0.000			
C	0.000		C	0.000			
█							
F2	TL . OFF	F3	TDC	F4	MACRO	F5	WORK
F6		F7	TOOL				

#### NOTE :

1. TDC of each group can not be over Pr.0359.
2. TDC unit : um
3. TDC time unit : sec
4. With Pr.0358 : Tool length TDC function 0 ) ON 1 ) OFF
5. with Pr.0359 : Maximum allowable TDC input.
6. If pressing Reset, no matter changing the tool or not, please make TDC value to be zero.





00318		N000000		EDIT		M-RDY		LNC	
-------	--	---------	--	------	--	-------	--	-----	--

<b>NUM</b>	<b>VALUE</b>	<b>NUM</b>	<b>VALUE</b>
@001	0.000	@011	0.000
@002	0.000	@012	0.000
@003	0.000	@013	0.000
@004	0.000	@014	0.000
@005	0.000	@015	0.000
@006	0.000	@016	0.000
@007	0.000	@017	0.000
@008	0.000	@018	0.000
@009	0.000	@019	0.000
@010	0.000	@020	0.000

--PAGE: 4 / 53--

<b>RELATIVE</b>	<b>MACHINE</b>
X	X
Y	Y
Z	Z
C	C

F2 TL . OFF	F3 TDC	F4 MACRO	F5 WORK	F6	F7 TOOL
-------------	--------	----------	---------	----	---------

## 1.5.4 WORK

Pressing **【WORK】** button to enter into the working coordinate setting screen.

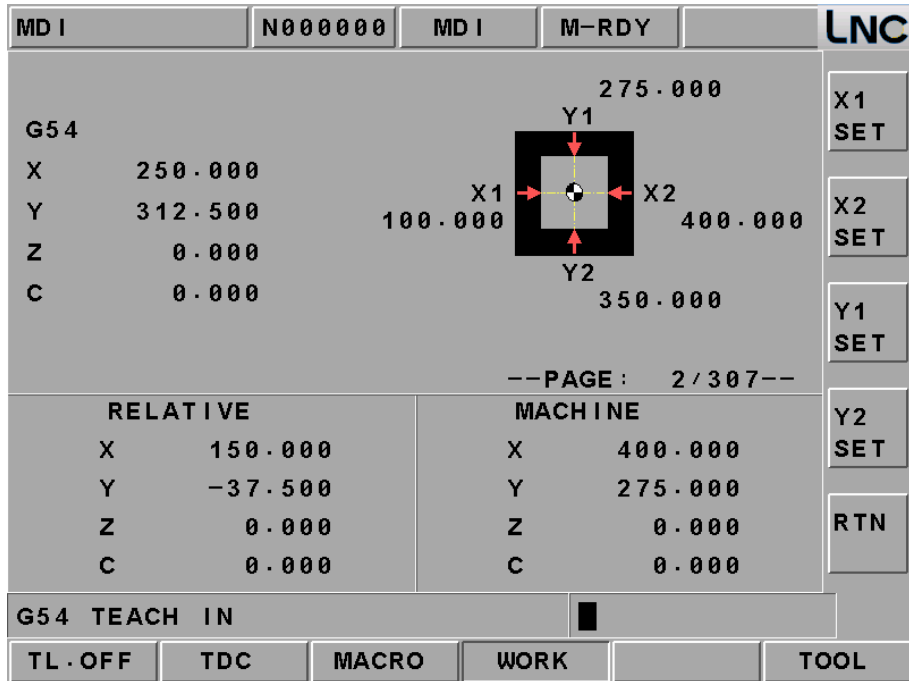
1. The controller provides working coordinate system for users. Users can execute the following settings such as G0、G54、G55、G56、G57、G58、G59、G54P1~P300. Also, using <PAGE↓> and <PAGE↑> key to switch screen.
2. Using direction keys to move cursor to the wanted change column and to key-in the setting value Xxxx (i.e. : X100 or Z200 ) in the enter rows. Then, the selected coordinate value will be updated immediately by pressing <INPUT> key.
3. 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.)
4. Set in the auto coordinate-gathering when the cursor is at G54~G59. So able to set the present machine coordinates automatically in the part coordinate setting page.

O0318		N000000		EDIT		M-RDY		LNC	
0		G54		G55				F12 SQU RE	
X	0.000	X	0.000	X	0.000			F11 CIRC LE	
Y	0.000	Y	0.000	Y	0.000			F10 TEAC H IN	
Z	0.000	Z	0.000	Z	0.000			F9 GOTO	
C	0.000	C	0.000	C	0.000			F8	
						--PAGE: 1 / 103--			
RELATIVE			MACHINE						
X	0.000	X	0.000						
Y	0.000	Y	0.000						
Z	0.000	Z	0.000						
C	0.000	C	0.000						
F2 TL·OFF		F3 TDC		F4 MACRO		F5 WORK		F6	
								F7 TOOL	

Controller provides 3 sets of working coordinate system for users to set easily :

**【SQUARE】** : Go to square work setting page, follow **【SQUARE】** figure X1,X2,Y1,Y2 step to use measuring instruments to touch workpiece and then press X1set,X2 set,Y1 set,Y2 set. Controller will automatically find the center of workpiece.

Press **【return】** to go back to WORK main page.



X1 SET : Set square 1 point X1 Axis's machine coordinates value into X1 working coordinates.

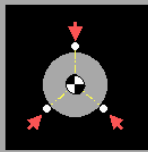
X2 SET : Set square 2 point X2 Axis's machine coordinates value into X2 working coordinates.

Y1 SET : Set square 3 point Y1 Axis's machine coordinates value into Y1 working coordinates.

Y2 SET : Set square 4 point Y2 Axis's machine coordinates value into Y2 working coordinates.

**【CIRCLE】** : Go to circle work setting page, follow **【CIRCLE】** figure P1,P2,P3 step to use measuring instruments to touch workpiece and then press P1set,P2 set,P3. Controller will automatically find the center of workpiece.

Press **【return】** to go back to WORK main page.

O5678		N000000		JOG	M-RDY	LNC	
G54				P1 X1 400.000 Y1 275.000		P1 SET	
X	167.654			P2 X2 501.876 Y2 452.976		P2 SET	
Y	526.143			P3 X3 193.129 Y3 867.330		P3 SET	
Z	0.000						
C	0.000						
--PAGE: 2/307--							
RELATIVE		MACHINE					
X	25.475	X	193.129				
Y	341.187	Y	867.330				
Z	1.000	Z	1.000			RTN	
C	0.000	C	0.000				
G54 TEACH IN							
TL OFF		TDC		MACRO		WORK	
						TOOL	

**P1 SET** : Set circle point 1 X, Y Axis's machine coordinates value into P1 X1,Y1 working coordinates.

**P2 SET** : Set circle point 2 X, Y Axis's machine coordinates value into P2 X2,Y2 working coordinates.

**P3 SET** : Set circle point 3 X, Y Axis's machine coordinates value into P3 X3,Y3 working coordinates.

**【TEACH IN】** : Go to **【TEACH IN】** work setting page, use measuring instruments to touch workpiece and then press X set, Y set, Z set, C set. Controller will automatically set each axis.  
Press **【return】** to go back to WORK main page.

MD I		N000000		MD I		M-RDY		LNC																																									
<table> <tr> <td colspan="2">0</td> <td colspan="2">G54</td> <td colspan="2">G55</td> <td colspan="2"></td> </tr> <tr> <td>X</td> <td>0.000</td> <td>X</td> <td>267.654</td> <td>X</td> <td>0.000</td> <td colspan="2"></td> </tr> <tr> <td>Y</td> <td>0.000</td> <td>Y</td> <td>626.143</td> <td>Y</td> <td>0.000</td> <td colspan="2"></td> </tr> <tr> <td>Z</td> <td>0.000</td> <td>Z</td> <td>-50.000</td> <td>Z</td> <td>0.000</td> <td colspan="2"></td> </tr> <tr> <td>C</td> <td>0.000</td> <td>C</td> <td>0.000</td> <td>C</td> <td>0.000</td> <td colspan="2"></td> </tr> </table>								0		G54		G55				X	0.000	X	267.654	X	0.000			Y	0.000	Y	626.143	Y	0.000			Z	0.000	Z	-50.000	Z	0.000			C	0.000	C	0.000	C	0.000			SET ALL	
0		G54		G55																																													
X	0.000	X	267.654	X	0.000																																												
Y	0.000	Y	626.143	Y	0.000																																												
Z	0.000	Z	-50.000	Z	0.000																																												
C	0.000	C	0.000	C	0.000																																												
								SET X																																									
								SET Y																																									
--PAGE: 1/103--								DN																																									
RELATIVE				MACHINE				RTN																																									
X	0.000	X	267.654																																														
Y	0.000	Y	626.143																																														
Z	0.000	Z	-50.000																																														
C	0.000	C	0.000																																														
G54 TEACH IN																																																	
TL-OFF		TDC		MACRO		WORK		TOOL																																									

**SET ALL** : Set current machine's each axis coordinate into **【TEACH IN】** s working coordinates.

**X SET** : Set X Axis' machine coordinates value into working coordinates which the cursor stops.

**Y SET** : Set Y Axis' machine coordinates value into working coordinates which the cursor stop.

**Z SET** : Set Z Axis' machine coordinates value into working coordinates which the cursor stop.

**C SET** : Set C Axis' machine coordinates value into working coordinates which the cursor stop.

Press **【GO TO】** to enter work page for users to search quickly, sub-coordinate only G54 can be used.

O0318		N000000		EDIT		M-RDY		LNC	
G54P198		G54P199		G54P200				F12 SQUA RE	
X	0.000					0.000			F11 CIRC LE
Y	0.000					0.000			F10 TEAC H IN
Z	0.000					0.000			F9 GOTO
C	0.000					0.000			F8
						E: 69 / 103--			
RELATIVE				MACHINE					
X	0.000	X	0.000						
Y	0.000	Y	0.000						
Z	0.000	Z	0.000						
C	0.000	C	0.000						
F2	OK	F3	CANCEL	F4		F5		F6	

### 1.5.5 TOOL

Pressing **【TOOL】** button to enter into the tool setting screen.

Offer users the tool numbers' management. When you move the cursor to the tool cap number, set the number you want to change and press conform, the tool number will be change right away.

O0318		N000000		EDIT	M-RDY	LNC
SP-NO	T-R	PRE-T	T-RP	PRE-TP	F12 CHG	
0	0	1	0	1		
POT D	POT D	POT D	F11			
001 1	011 11	021 21				
002 2	012 12	022 22				
003 3	013 13	023 23	F10			
004 4	014 14	024 24				
005 5	015 15		F9			
006 6	016 16					
007 7	017 17		F8			
008 8	018 18					
009 9	019 19					
010 10	020 20					
F2 TL-OFF		F3 TDC	F4 MACRO	F5 WORK	F6	F7 TOOL

**Note :** The number limit of tool can be changed by Pr.5, also depend on the machine's tools.

#### 【TOOL CHANGE】

Pressing **【TOOL CHANGE】**, controller will change spindle tool number with standby tool number. You will need to use cycle start button to change them.



## 1.6 EASY-CAM

(Note : M515i 、M516i 、M516id did not support this function.)

### 1.6.1 Preface

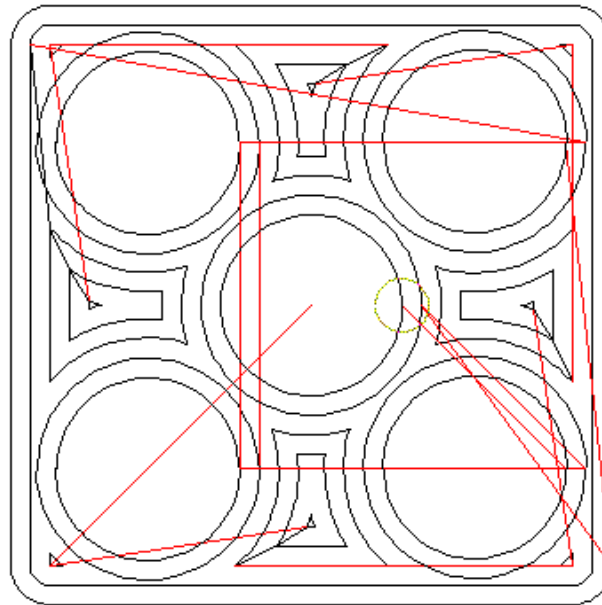
This function provides 2D pocketing work of one allowable island. The function area has 「Rough-Cutting」 and 「Fine-Cutting」 working types. The CAM function in this controller allows users to produce auto NC path for its pocketing of any Profile shape. It means that it can define working piece's geometry shape direction on the controller. Also, it will produce the corresponding post NC code after setting the related working parameters.

#### EASY CAM Function Specification :

- File input will be 2 types, controller editing one and AutoCAD DXF file.
- Regardless of the file source, one project can only contain 10 graphs.
- Regardless of the file source, lines and curves which composite to a graph can no be over 100 parts.
- Only allow defining one closed Profile and it is limited inside the closed Profile. Only can define maximum 6 islands.
- Elements that are combined by Profile and Island must be closed.
- Island must be surrounding by Profile. So, the island must be defined with the Profile.
- Cannot have any intersect or self-intersect between islands. Also, cannot have any point or line connecting/cutting between islands.
- For profile and island, neither can have intersection nor have connection/cutting between them.
- At DXF FILE IN, close profile, island, random milling path didn't have any limitation for graph and line, because it can be deleted later.
- The DXF FILE will differentiate profile, island and path with different colors. In the color plate of AutoCAD, details will be as below: ADxfOutloopColor=5 (Bleu, Profile) , DxfIslandColor=3 (Green, Island) , random milling path=6 (purple), other color's line will be considered as random path.
- The current profile or island's mode can't be mixed with random milling path, therefore we have below restrictions :
  - If DXF IN have random line or unassigned color ==>
  - a 、Those lines include close profile or island will turn to be random lines.
  - b 、You can only add random line at controller.
  - c 、You can only save as a milling path processing file.
- If the existing files in the project are close profile or island, then you can only add close profile and island.
- If the existing files in the project are random lines, then you can only add random lines.
- If this is a new project that didn't add any file, you can add all types of files that has mentioned at former paragraph.
- Because AUTO\_CAD support all kind of software to produce different graphs, in order to have clear analysis of graph, below are some rules :

- a 、 Figure shape entity must be LWPOLYLINE, even using plane command to draw Figure.
- b 、 Need to use color to indicate those files that you are going to file in.
- c 、 The sequence of NC CODE post will depend on the sequence of making graph.

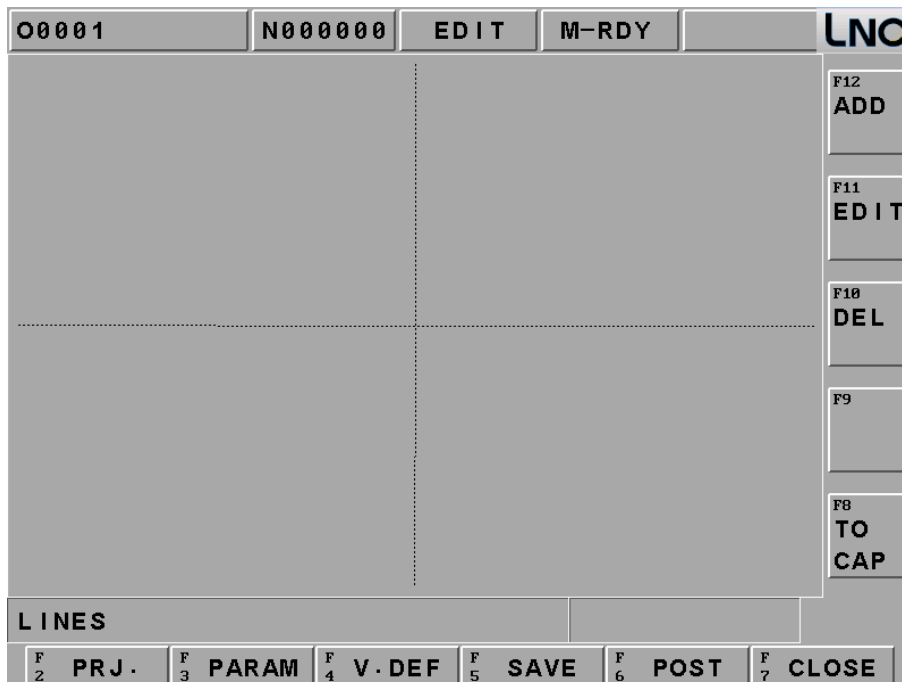
- The using pocking calculation method—Horizontal Surrounding ( Arachnoid. As the below figure )



### 1.6.2 Screen Operation Description

**Main** : Press <CAM> to go to EASY CAM.

**Note** : Need to add new file or open old file, otherwise, can not edit here.



**PRJMNG** : Press PRJMNG will switch to this page.

O0001		N000000		EDIT		M-RDY		LNC	
FILE MNG								FILES 9	
DEMOA DEMOB DEMOC DEMOD DEMOE DEMOF DEMOG DEMOH DEMOI									
OUT	1	ISLAND	1	OTHER	0				
DATE	11 / 16 / 01	SIZE	838						
COMM									
F2	OPEN	F3	NEW	F4		F5	DELETE	F6	DXF IN
						F7	RETURN		

#### Buttons :

- 【OPEN】 : open an old project.
- 【ADD】 : add a new project.
- 【DEL】 : delete an old project.
- 【DXF IN】 : execute AutoCAD DXF IN function.
- 【RETURN】 : to return Easy-Cam screen.

#### Columns :

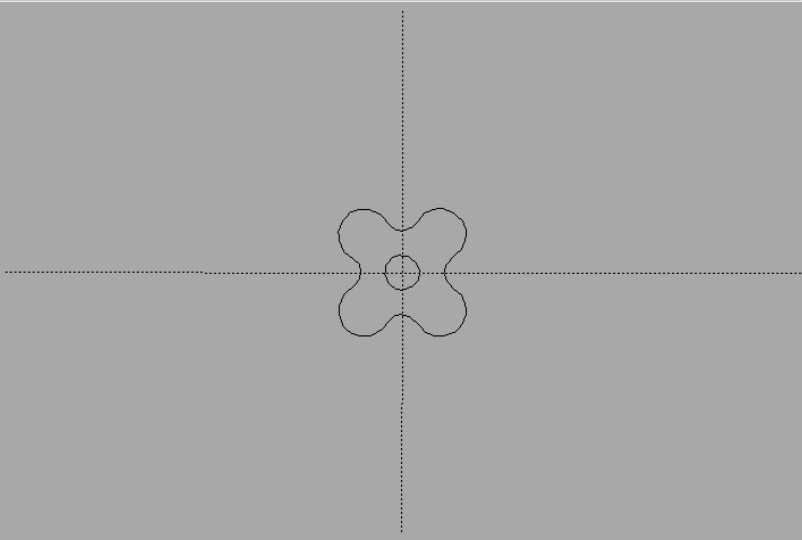
- FILES : show current project numbers.
- DATE : show last saved date.
- SIZE : show the file size that yellow highlight stay.
- COMM : show file's note.

- OPEN

Move to DEMOA, press 【OPEN】

O0001		N000000		EDIT		M-RDY		LNC	
FILE MNG								FILES 9	
DEMOA DEMOB DEMOC DEMOD DEMOE DEMOF DEMOG DEMOH DEMOI								F12  F11  F10  F9  F8	
OUT		1		ISLAND		1		OTHER 0	
DATE		11/16/01		SIZE		838			
COMM									
F2		OPEN		F3		NEW		F4	
				F5		DELETE		F6 DXF IN	
								F7 RETURN	

Open as below.

O0001		N000000		EDIT		M-RDY		LNC	
								F12 ADD	
								F11 EDIT	
								F10 DEL	
								F9	
								F8 TO CAP	
F2		PRJ.		F3		PARAM		F4	
				F5		SAVE		F6 POST	
								F7 CLOSE	

- **NEW**

Press **【NEW】** , input new file name, as CAI, OK.

00001		N000000		EDIT	M-RDY	LNC	
FILE MNG						FILES	9
DEMOA DEMOB DEMOC DEMOD DEMOE DEMOF DEMOG DEMOH DEMOI						F12 OK	
						F11 CAN CEL	
						F10	
						F9	
						F8	
OUT	1	ISLAND	1	OTHER	0		
DATE	11/16/01	SIZE	838				
COMM							
INPUT FILE NAME					CAI		
F2 OPEN	F3 NEW	F4	F5 DELETE	F6 DXF IN	F7 RETURN		

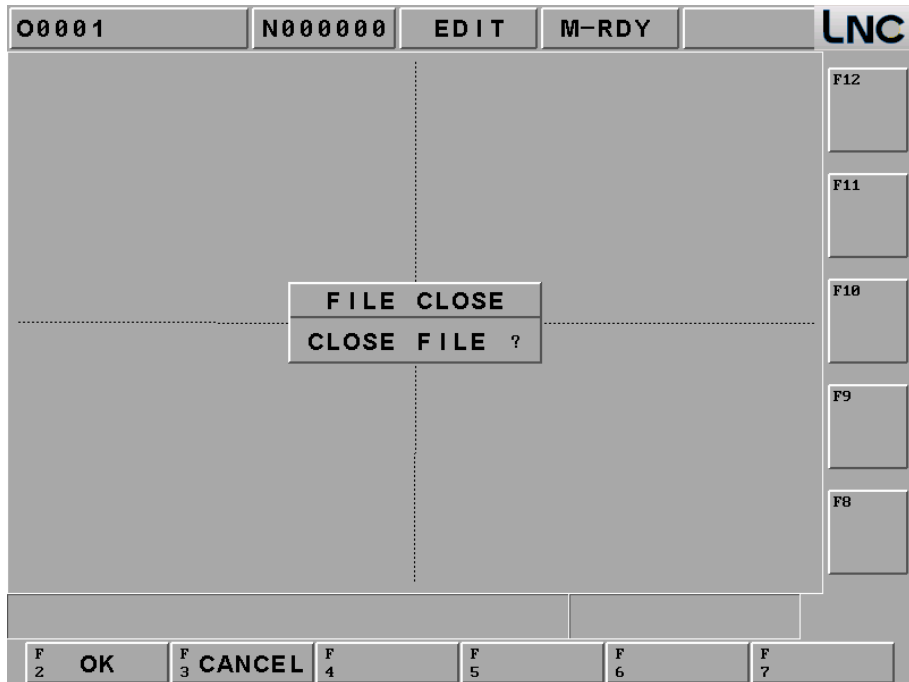
Then screen will return main page to wait for drawing.

00001		N000000		EDIT	M-RDY	LNC	
						F12 ADD	
						F11 EDIT	
						F10 DEL	
						F9	
						F8 TO CAP	
F2 PRJ.	F3 PARAM	F4 V-DEF	F5 SAVE	F6 POST	F7 CLOSE		

## ● CLOSE

Press **【CLOSE】** to confirm.

**Note** : If one project did not close, can not process next step.



## ● DELETE

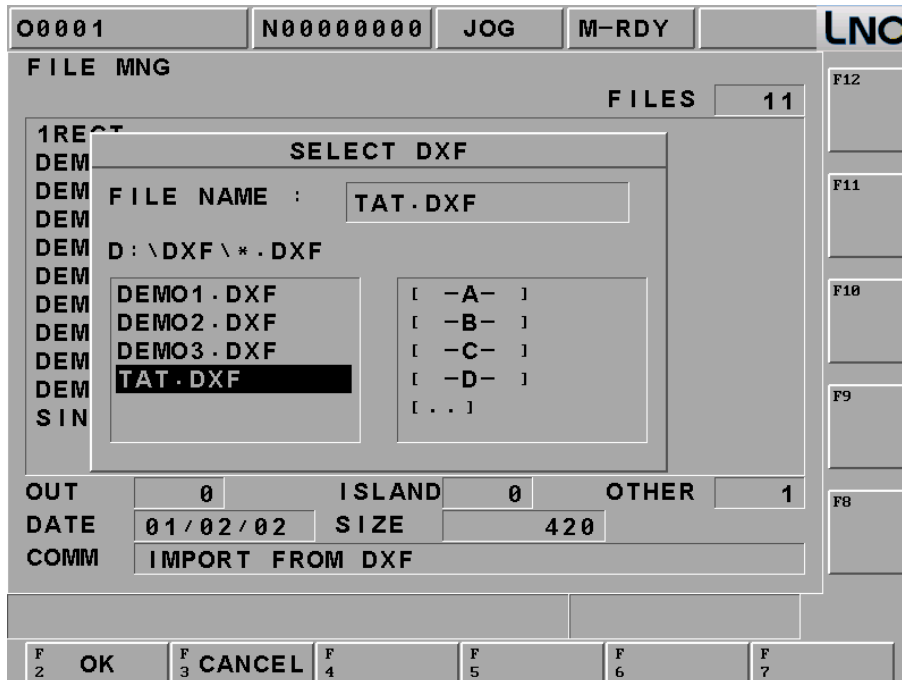
Move to old file and press delete, for example, delete CAI.

Press **Y** to delete.

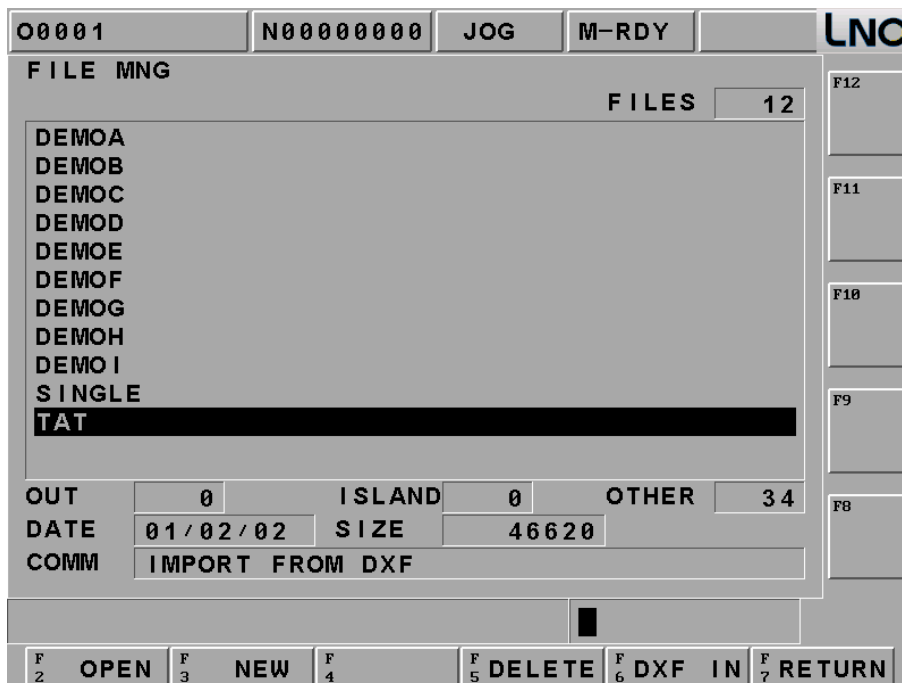


### ● Import AutoCAD DXF file

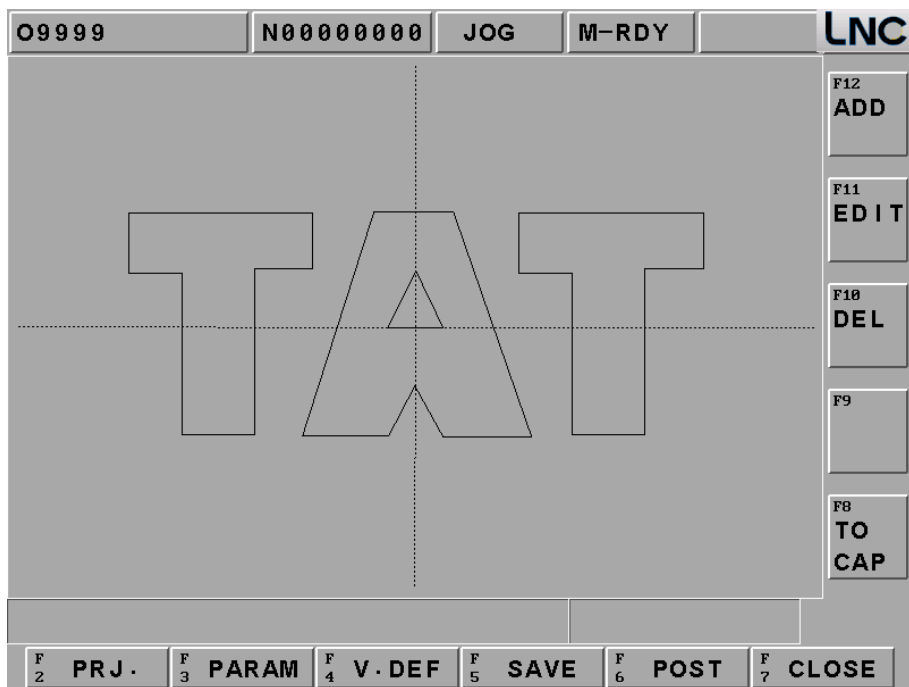
At Main screen, press **【DXF IN】**, pop up DXF dialogue : Hold cross handle to move yellow cursor to appoint DXF source EX: Ethernet connection is at M:\EASYCAM\TAT\TAT.DXF. press **【CONFIRM】** to input.



At FILE MNG page's hint line show : DXF IN success, hold cross handle to move yellow cursor to choose TAT file, press **【OPEN】** to run the DXF file which was input already.



Open this DXF file to check if the transmission is correct.



Parameter Specification : This page is mainly to define the related CAM calculation and the output parameters.

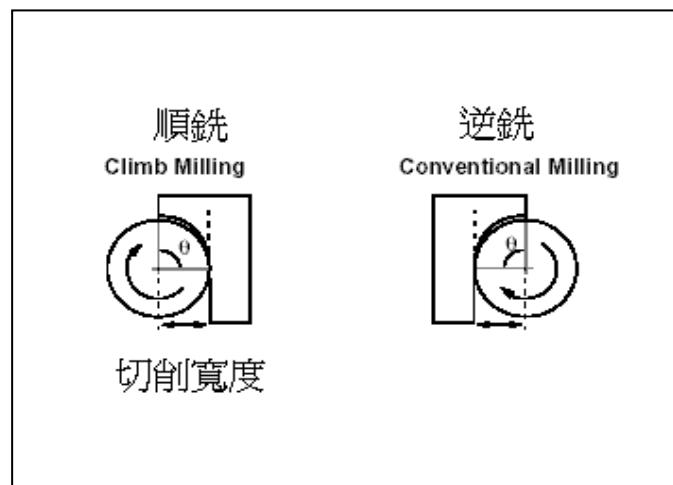
O0001		N000000		EDIT		M-RDY		LNC	
STR · Z	0.000	OFF · L	1						
END · Z	-2.000	H CODE	1						
DEP · H	2.000	RPM · S	1500						
CUT · W	5.000	XY · F	1800						
FIN · W	0.000	Z · F	1200						
ENTRY	2.000	TYPE	0						
TRAVER	4.000	M08	0						
DIA · T	8.000	APPRO	0						
NO · T	1	APP · L	0.000						
START PLANE · Z									
F2	F3	F4	F5	F6	F7	RETURN			

- Starting Surface : This is an absolute Z coordinate program, which is used to assign the starting cutting level.
- Ending Surface : This is an absolute Z coordinate program, which is used to assign the ending cutting level.
- Depth Height : Tooling depth of Z coordinate, the depth setting will not more then the depth of end plane.
- Cutting Width : Every time's feed amount on XY pane for the assigned tool. This value should not be larger



than the tool diameter.

- Fine-Cutting Preservation Amount : The preservation amount for already-formed size (for fine mill-cutting).
- Feed Height : The absolute Z coordinate program, which is use to set the starting point of mill-cutting. Tool will cut working part in G01 speed after arriving to the point in G00 speed. So, the setting value of that point must be greater than [Starting Surface].
- Moving Height : The absolute Z coordinate program, which is to set the ascent height when tool working is switching position during mill-cutting procedure. Tool will cut into the working part in G01 speed after arriving to the point in G00 speed. So the setting value of this point must be greater than [Starting Surface].
- Tool Diameter : This value must be greater than zero.
- Tool Number : The timing codes (M06 T) when the designated tool is doing tool-exchanging job, i.e., if this value is 1, M06 T01 will be produced
- Tool Length Offset : To choose tool length offset, setting 0 means to use the +ive direction offset G43; setting 1 means to use -ive direction offset G44.
- Spindle Rotational Speed : S machine structure means spindle rotational speed
- Flat Feed : F function, define tool's cutting feed rate is F when flat cutting.
- Z-Axis Feed : F function, is to define the cutting feedrate is F when cutting toward Z-direction.
- Climb/Conventional Cutting : To decide the cutting type of the tool path program. Two cutting types are [Climb] and [Conventional]. [Climb] is used in rough-cutting and [Conventional] is used in smooth-cutting.



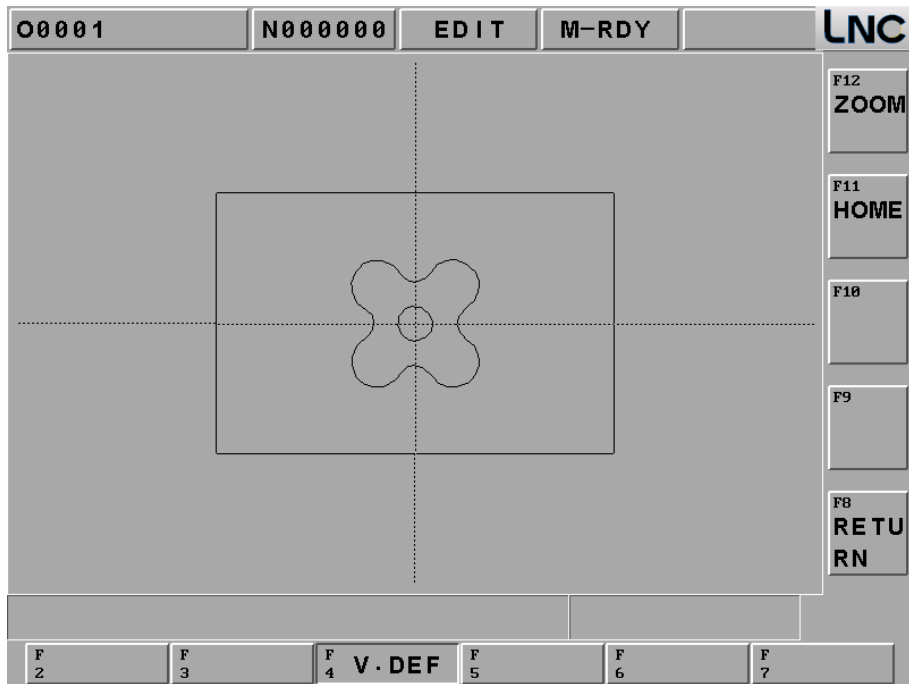
- Soluble oil : To decide whether or not to use the cutting function (M8).
- Fine-Cutting Feed : To decide the Tool Approach type when doing [Fine-Cutting]. Two types of fine-cutting feed which are vertical feed and arch feed. The preserve length before vertical feed is the radius of the tool. The arch radius is the radius for the tool feed.
- Feed distance : the percentage of cutting tool's diameter.

**Window Enlarge/Reduce** : This page is to define relating CAM window enlarge/reduce specification.

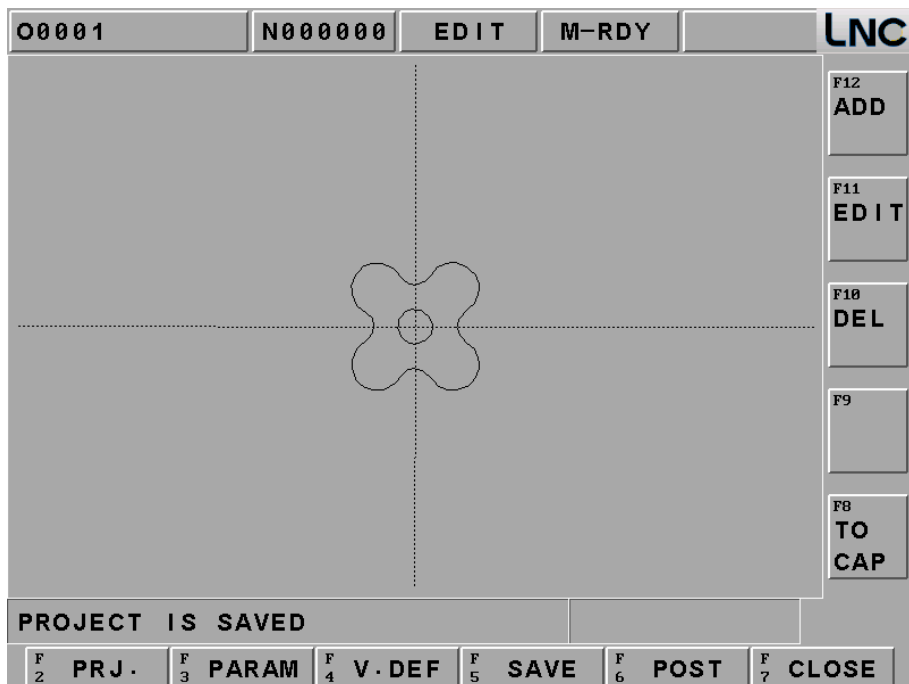
Function keys are as below:

Press ENLARGE to see larger yellow window, press CONFIRM to enlarge graph.

Press PROPER to show proper size.

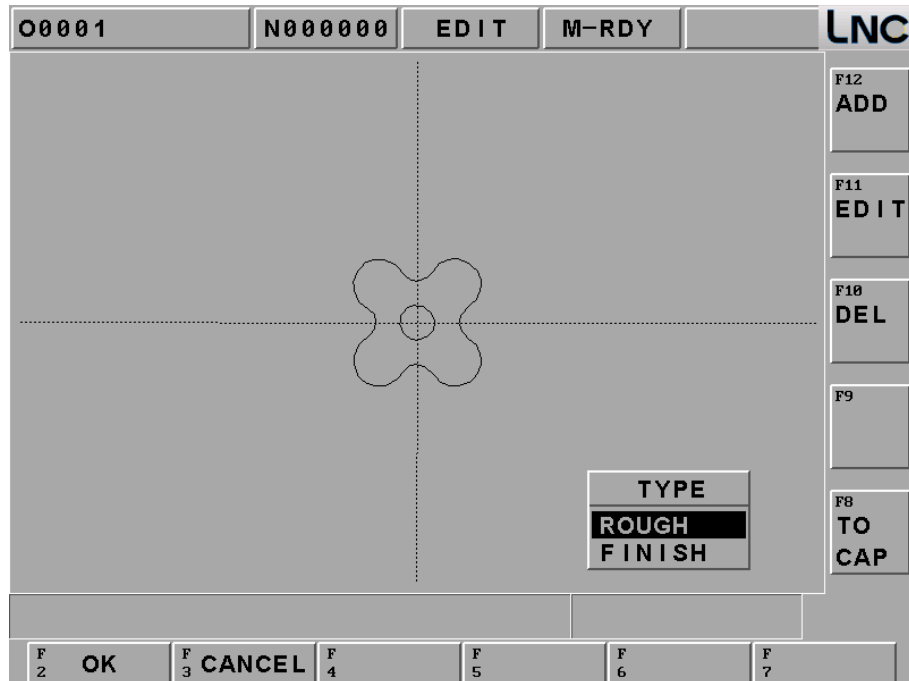


**SAVE** : Press SAVE, message hint row shows save complete, this project was saved.

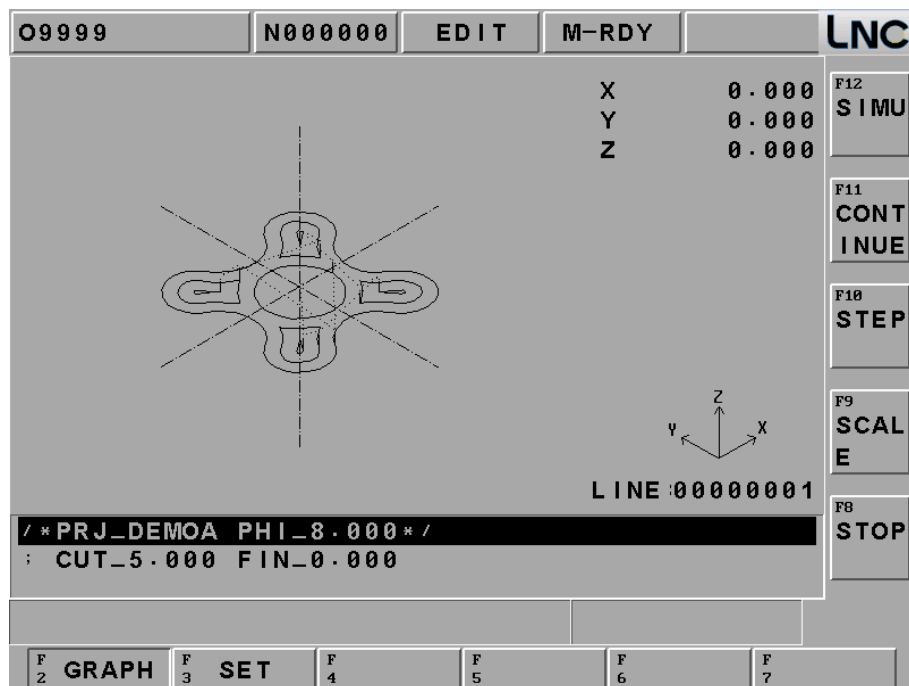


**POST** : Press Post, it will show different NC file content according to line mode, close profile or island of the project.

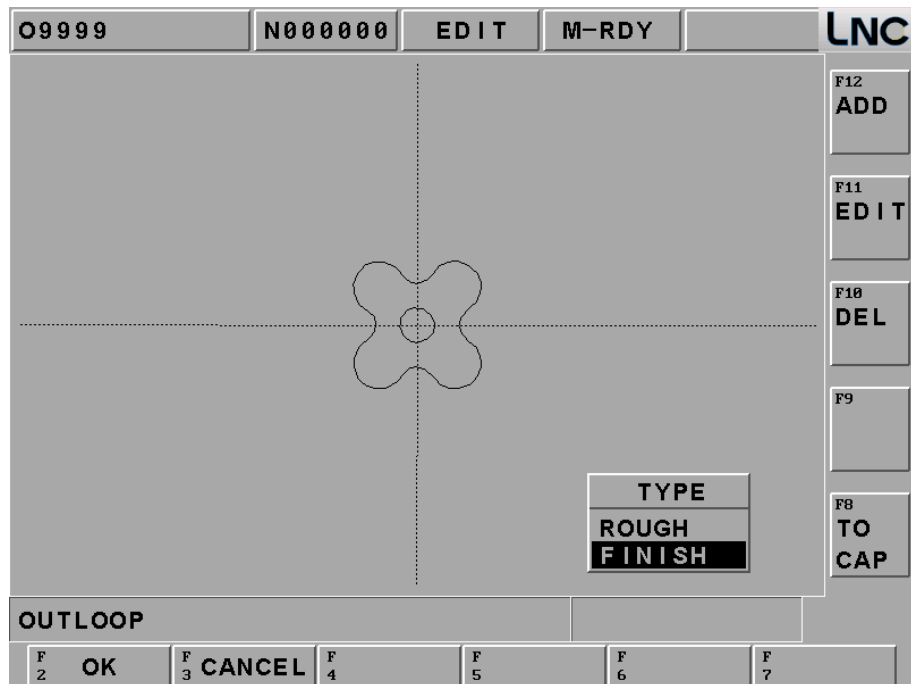
- Shows ROUGH mode of close profile and island.



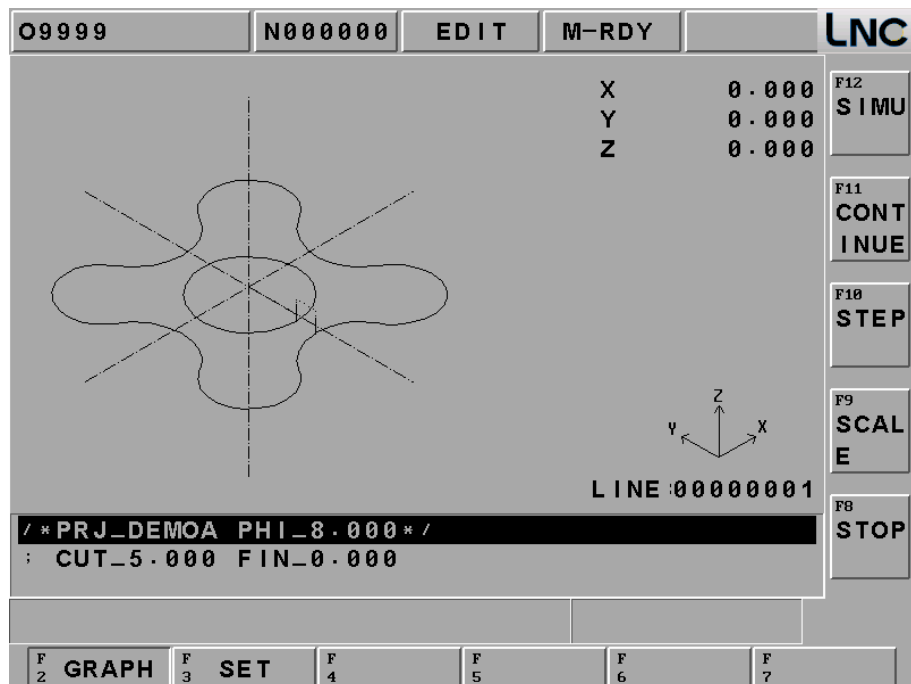
Shows ROUGH mode of close profile and island.



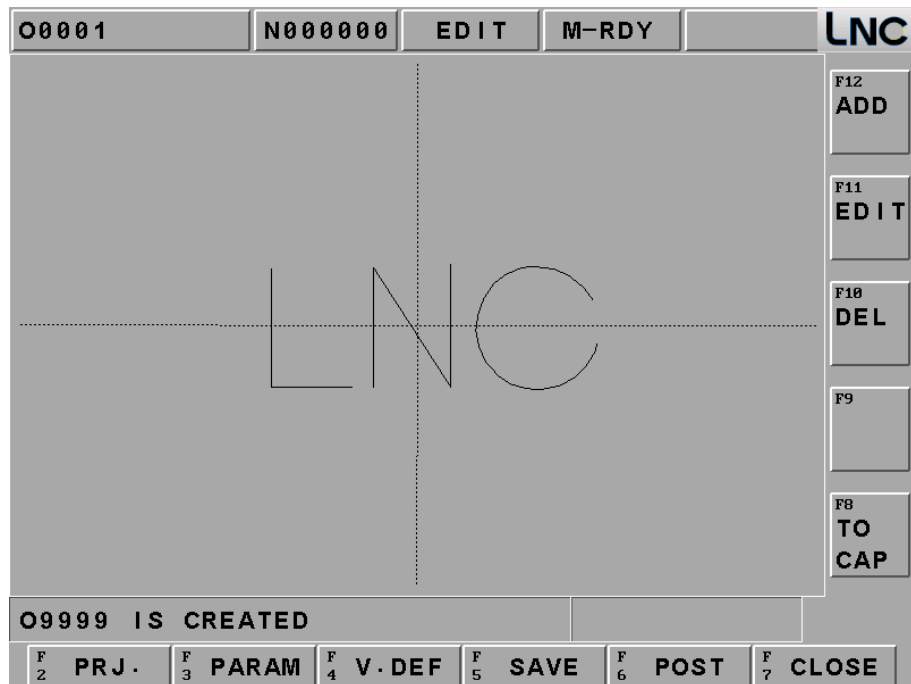
- Shows FINISH mode of close profile and island.



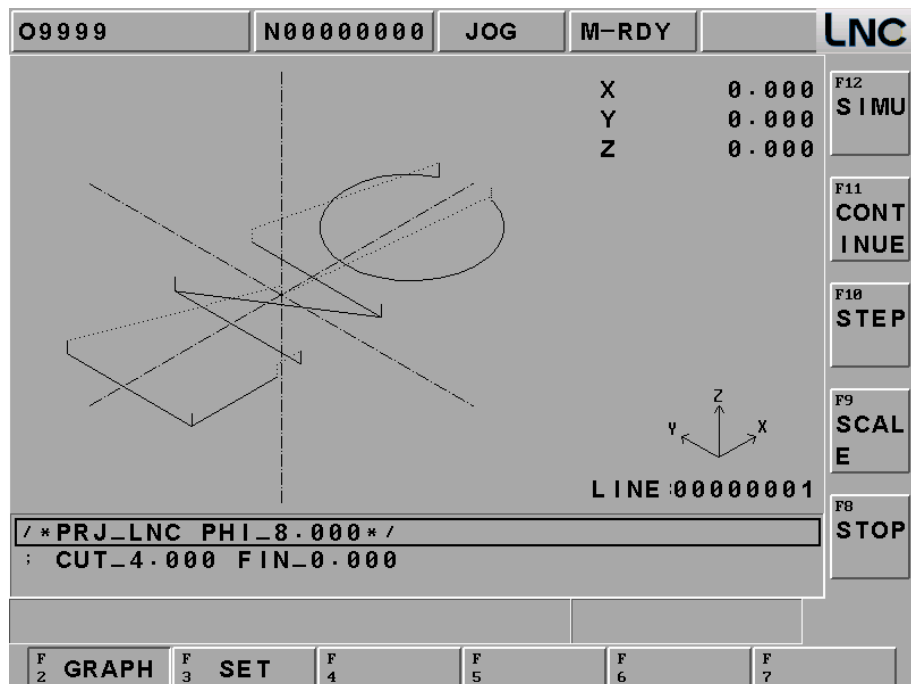
Preview of tool path's graph under FINISH mode.



3. If this is only a line, it will only shows path.

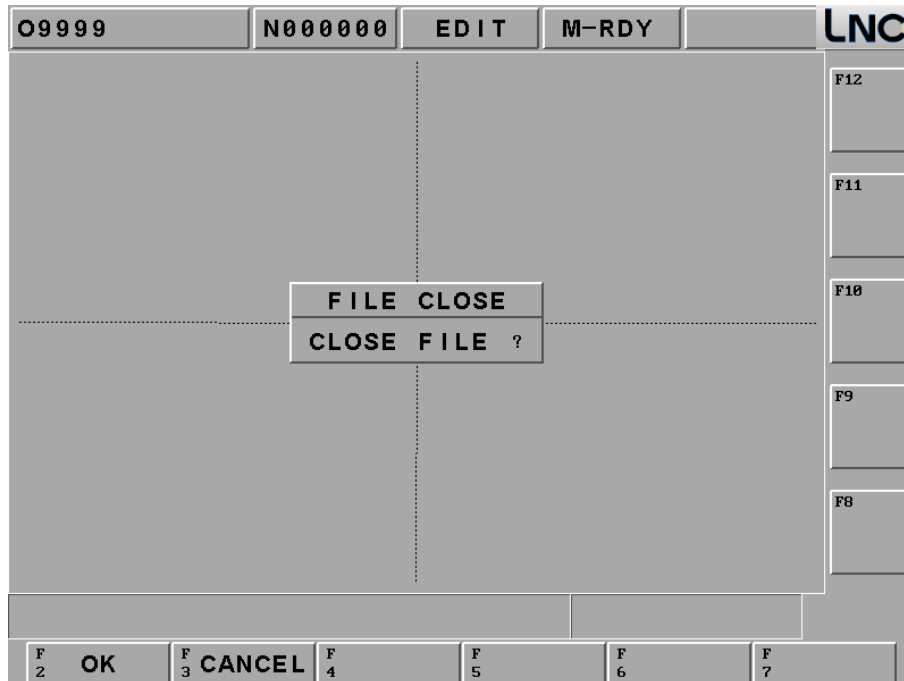


Preview of tool path's graph under LINE mode.

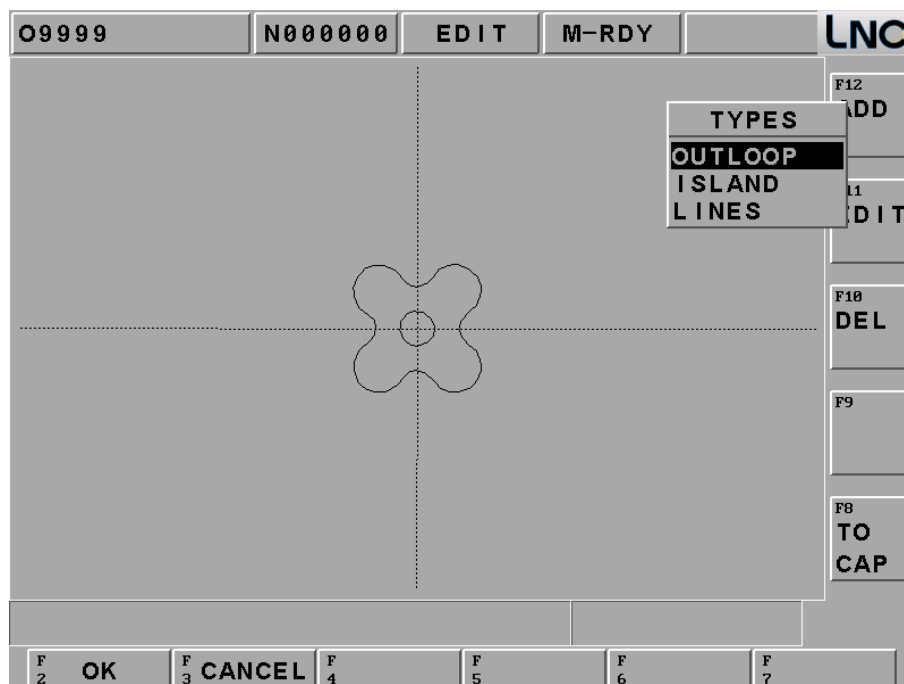


**CLOSE** : Close current project.

Press CLOSE to close this project, pop out window shows confirm close, then press CONFIRM, return to main page. If you didn't close this window, you will not be able to return main page.



**ADD** : The EASY CAM file input way is not just the DXF IN, but also has another solution by pressing ADD at NC end to choose outloop, island, line.



ADD : Press ADD to select close profile

Line: Sets starting and terminal points, draws line.

ARC CW: Sets starting and terminal points, draws a ARC CW.

ARC CCW: Sets starting and terminal points, draws a ARC CCW

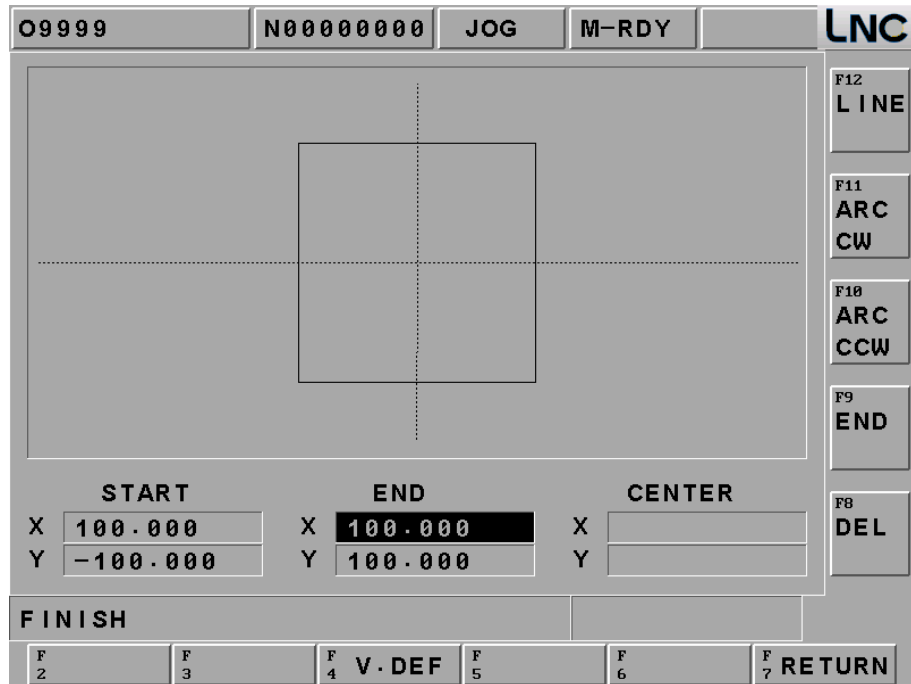
END: Use direct line to close unsealed line.

DEL: Delete the final line.

The screenshot shows the LNC Milling Large Screen Operation Manual interface. At the top, there are status bars displaying 'O9999', 'N000000', 'EDIT', and 'M-RDY'. The main display area is a coordinate grid with a dashed line representing the 'LINE' function. Below the grid, there are input fields for 'START', 'END', and 'CENTER' points. The 'START' point is set to X100Y100, the 'END' point is set to X-100Y100, and the 'CENTER' point is set to X100Y100. Below these fields, there is a 'START POINT X' field and a 'V-DEF' field. The bottom of the screen has a row of function keys: F2, F3, F4 V-DEF, F5, F6, and F7 RETURN. On the right side of the screen, there are four function keys: F12 LINE, F11 ARC CW, F10 ARC CCW, and F9 END. Below these is a F8 DEL key.

Press ADD to choose OUTLOOP and make a square :

1. Press LINE, Sets starting point as X100Y100, terminal point as X-100Y100, press OK.
2. Press LINE, Sets terminal point as X-00Y-100, press OK.
3. Press LINE, Sets terminal point as X100Y-100, press OK.
4. Press LINE, Sets terminal point as X100Y100, press OK.
5. Press Return to go back main page



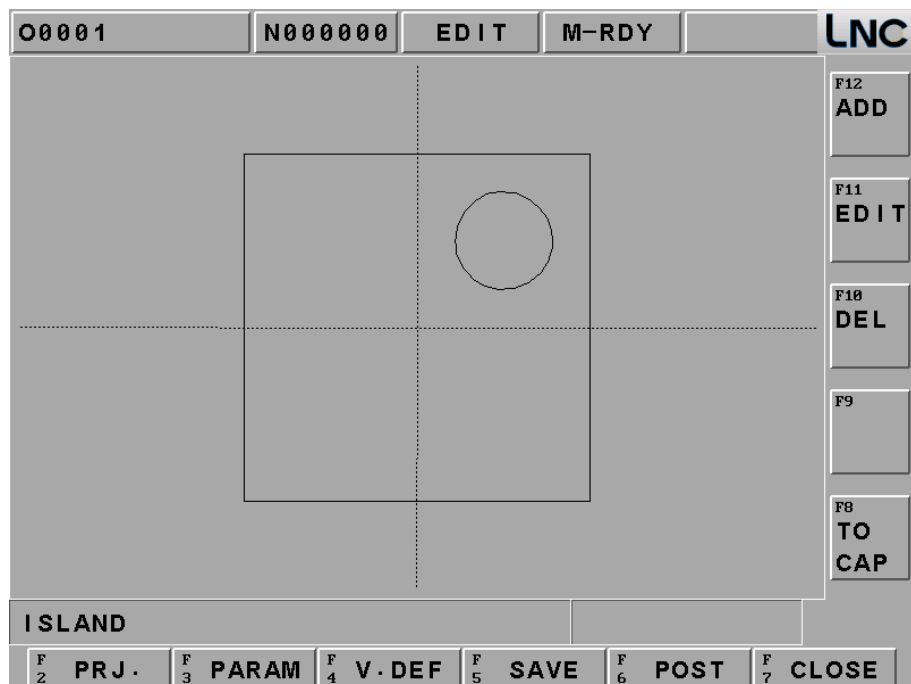
Press ADD, choose island to make 2 round islands.

Create islands in the square that we've just created.

Draw the first island.

1. Press ARC CW, sets starting point as X70Y50, terminal points as X30Y50, center of circle as X50Y50, press OK.
2. Press ARC CW, sets terminal points as X70Y50, center of circle as X50Y50, press OK.
3. Press return to go back to main page.

Then you can create the first island.

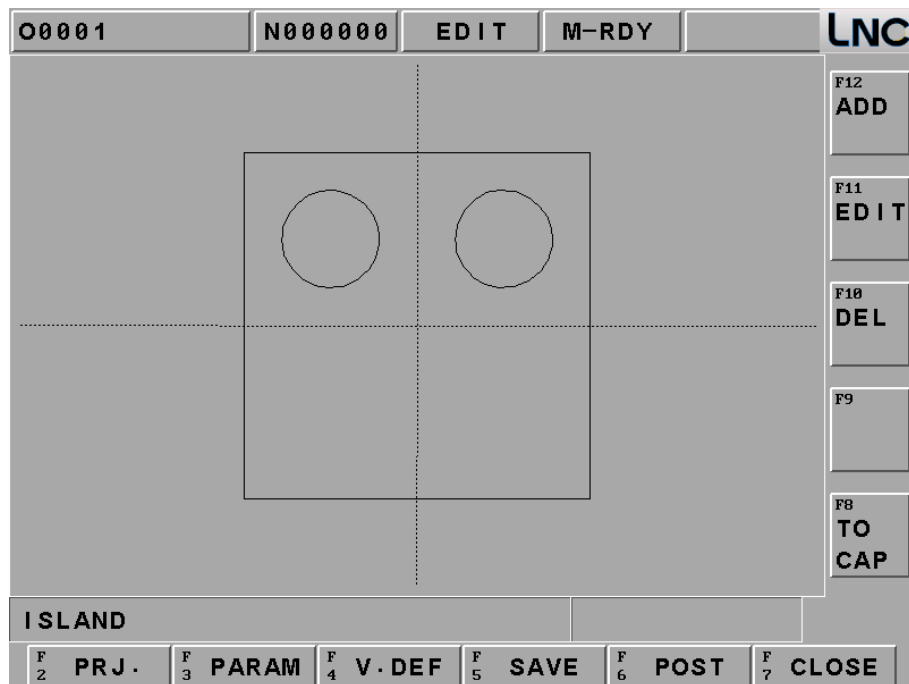




Create the second island.

1. Press ARC CCW, sets starting point as X-30Y50, terminal points as X-70Y50, center of circle as X-50Y50, press OK.
2. Press ARC CCW, sets terminal points as -30Y50, center of circle as X-50Y50, press OK.
3. Press return to go back to main page.

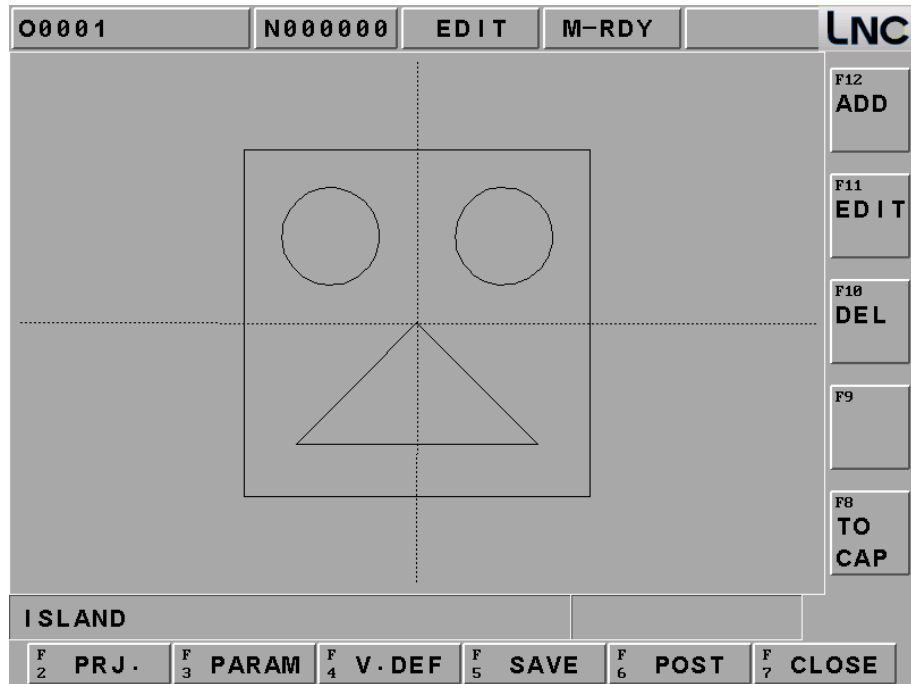
Then you can create the second island.



Create a triangle islands :

1. Press LINE, Sets starting point X70Y-70, terminal point as X-70Y-70, press OK.
2. Press LINE, Sets terminal point as X0Y0, press OK.
3. Press CLOSE, sets terminal point as X70Y-70, system will close the line automatically
4. Press return to go back to main page.

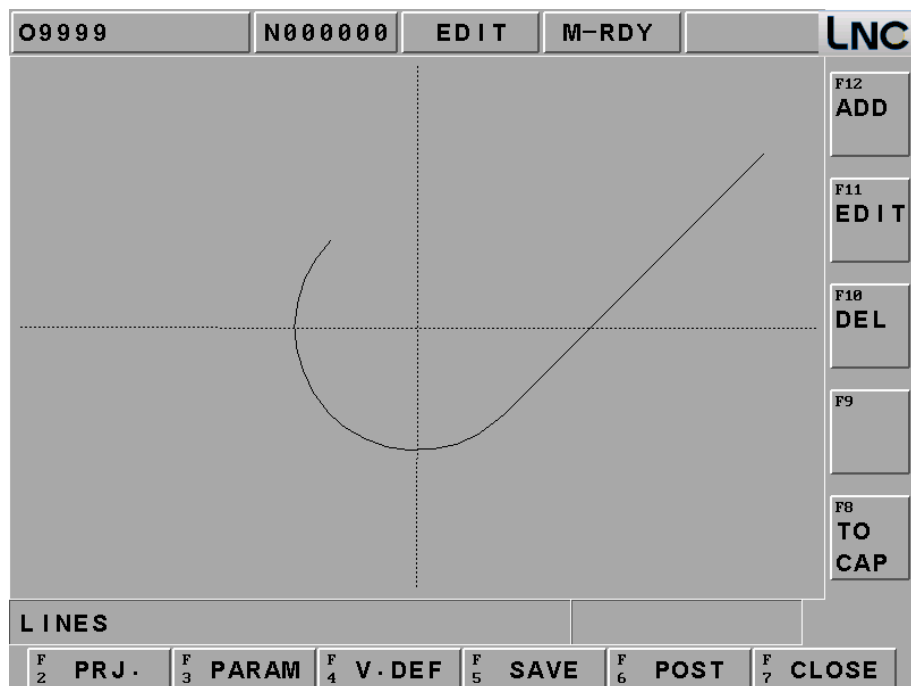
Then you can create a triangle island inside the square.



Make two random lines.

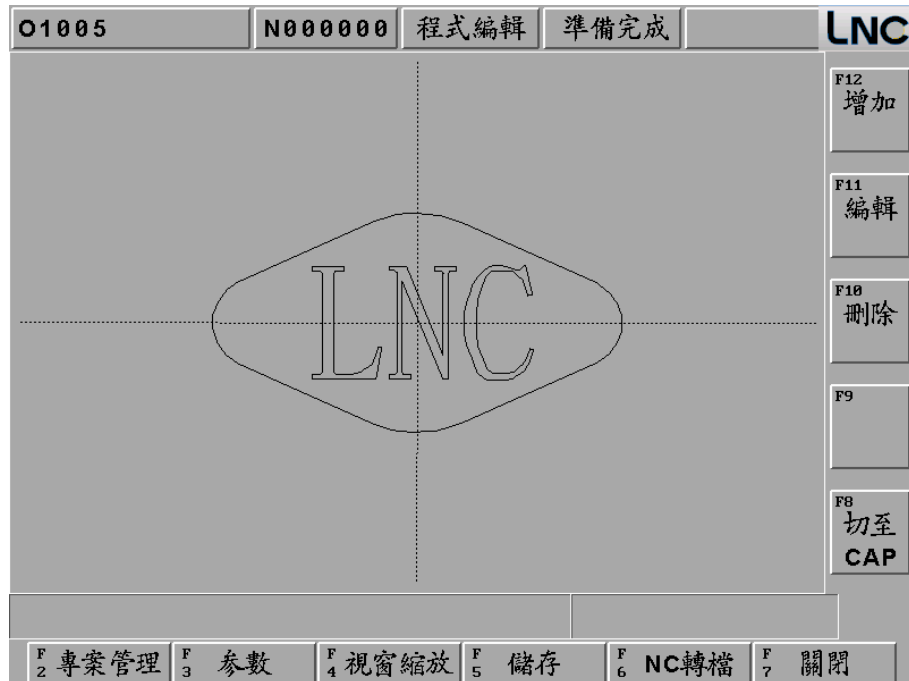
1. Press ARC CCW, sets starting point as X-44.650Y50.120, terminal point as X44.650Y-50.120, center of circle as X0Y0.
2. Press OK to make a curve.
3. Press LINE, sets terminal points as X230.180Y115.160, press OK.
4. Press return to go back to main page

Then you can create 2 random lines.



**EDIT :**

Use ↑↓ to change to island or outloop, Yellow line in the monitor will become your modification target. Press edit to modify.

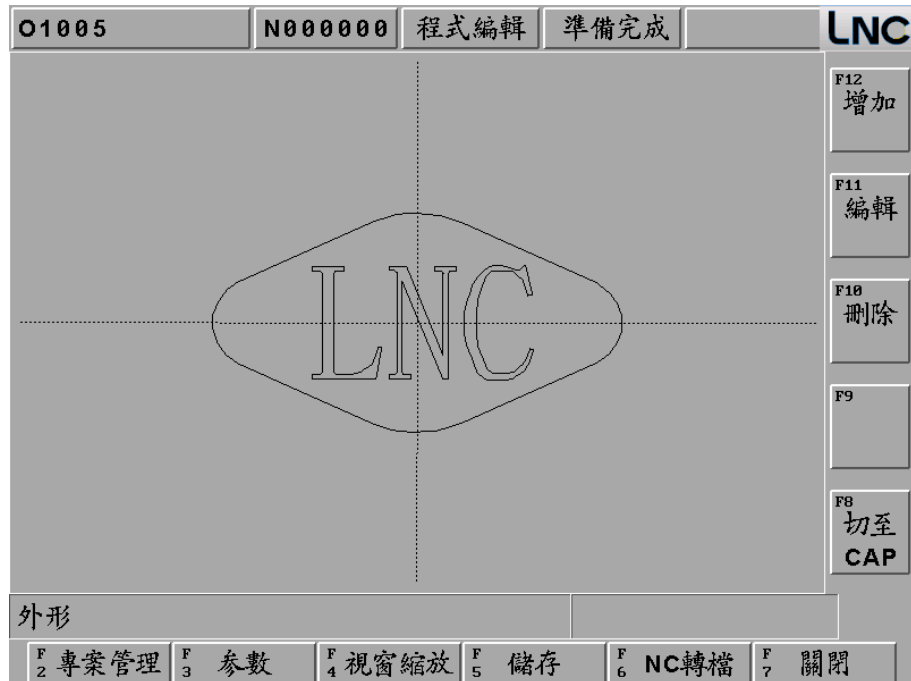


Enter into Edit screen to add line or delete.

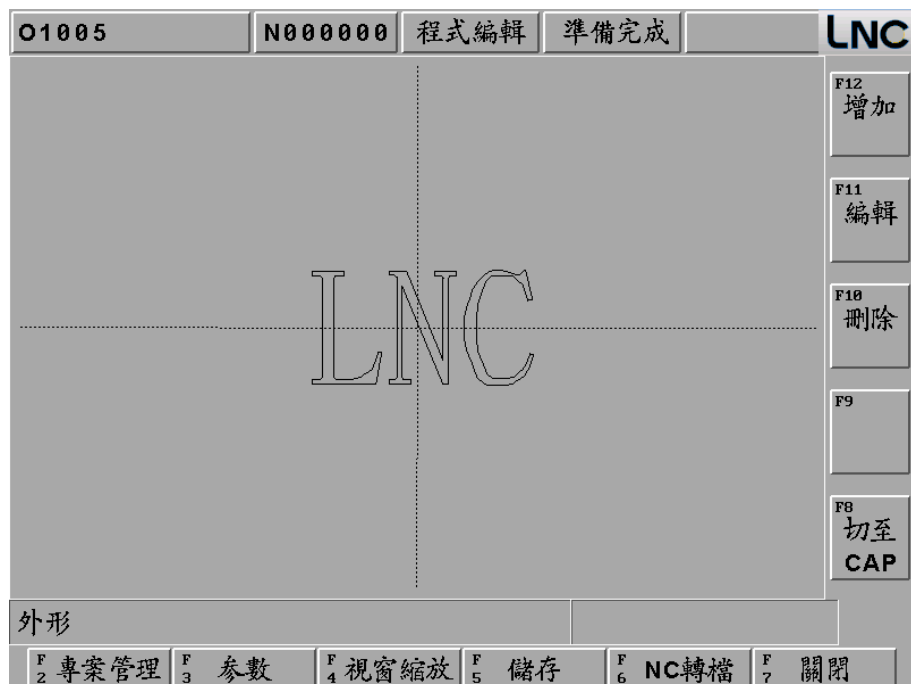
O1005	N000000	程式編輯	準備完成	LNC	
				<div style="margin-bottom: 10px;">F12 線</div> <div style="margin-bottom: 10px;">F11 順圓</div> <div style="margin-bottom: 10px;">F10 逆圓</div> <div style="margin-bottom: 10px;">F9 閉合</div> <div>F8 刪除</div>	
起點		終點		圓心	
X	15.124	X	15.124	X	
Y	26.384	Y	28.668	Y	
起點 X					
F 2	F 3	F 4 視窗縮放	F 5	F 6	F 7 返回

**DELETE :**

Use ↑↓ to change to island or outloop, yellow line in the monitor will become your modification target. Press delete to cancel the yellow line right now.



After canceling the outline, there are still three islands.



**Note :** After Press POST, the solution to O9999 file can not be modified after post :

1. Under EDIT mode, hint line shows the file is under protection, modification is not allowed. Please make the parameter 94 to be 1 and press RESET to make the parameter valid, and then you can modify programs after O9000.

2. Under EDIT mode, go to PMNG, choose copy file.

If you Copy file of O9999 to new file of O0408, then open O0408 to change

**Note :** If your file is too big to be modified on controller, please modify it at PC firstly, and then send back to controller later.

## 1.7 MILLING FIGURE CONVERSATION

### 1.7.1 REFACE

This function provides a G code structure that is easier to use. There are four working methods: drill-machining, facing, side-cutting and pocketing. So users do not need to remember complicate G code types and parameter definitions. Users only need to follow the instruction of Figure Panel to key-in the related information. And then pressing gear shifting button in order to produce the needed part program.

### 1.7.2 Function Specification

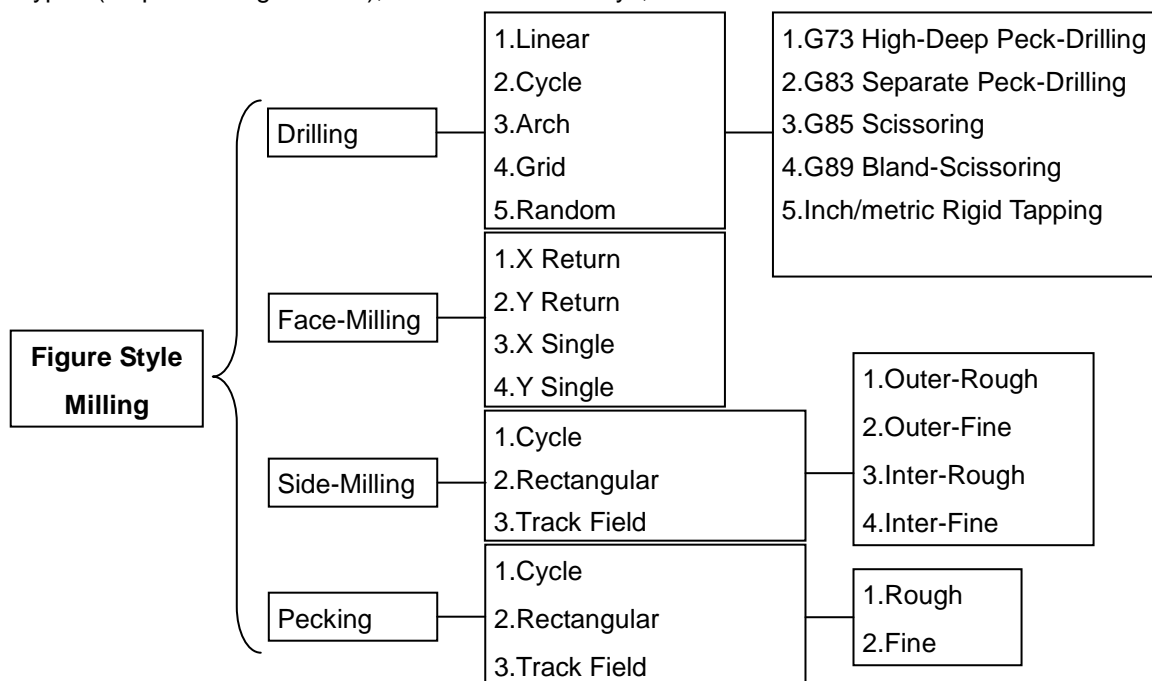
In order to have more elasticity for this function, so LNC Technology Co., Ltd. has designed projects and production management structure. Also, the followings are some restrictions due to the concern of the actual use :

- Project name is D0000~D9999, totally 10000 units.
- Every project allows having more than one production procedure. 99 production procedures are the maximum.

Every project has one set of common parameters. Every production procedure has one set of cutting parameter.

### 1.7.3 Process type

This function offers 4 types (15 processing method), total 47 ones of ways, as below :



**1<sup>st</sup> Type:** Drill-Machining, There are 5 working styles, which are G73, G83, G85, G89 and TAP. 5 working methods are as following:

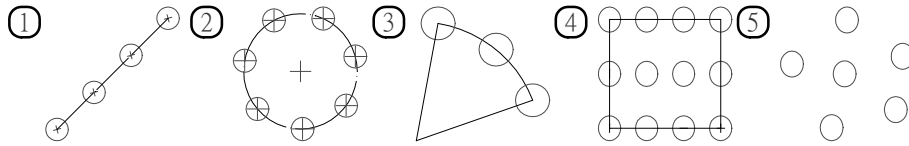


Figure 1.7-1 Drill-Machining

1. Linear type orientation ( G101 )
2. Circle type orientation ( G102 )
3. Arch type orientation ( G103 )
4. Square type orientation ( G104 )
5. Random type orientation ( G105 )

**2<sup>nd</sup> Type:** Facing-milling, 4 working methods are as following:

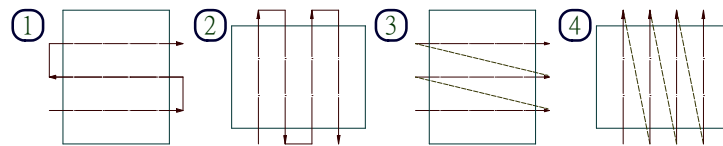


Figure 1.7-2 Face-Milling

1. X-axis two directions surface ( G111 )
2. Y-axis two directions surface ( G112 )
3. X-axis single direction surface ( G113 )
4. Y-axis single direction surface ( G114 )

**3<sup>rd</sup> Type:** Side-Milling, there are 4 types of this working methods which are external rough-cutting, external fine-cutting, internal rough-cutting, and internal fine-cutting. 3 working methods are as below:

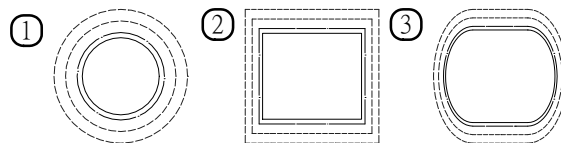


Figure 1.7-3 Side-Cutting

1. Circular side-cutting ( G121 )
2. Square shape side-cutting ( G122 )
3. Tracking field side cutting ( G123 )



**4<sup>th</sup> Type** : Pocketing, there are two working styles of this working methods which are rough-cutting and fine-cutting. 3 working methods are as below:

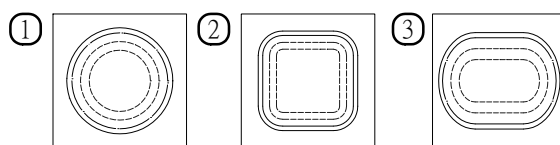


Figure 1.7-4 Pocketing

1. Circular pecking ( G131 )
2. Square pecking ( G132 )
3. Run-way pecking ( G133 )

#### 1.7.4 Cutting Parameter Setting

The needed setting parameters and definitions are different due to different working methods. Please refer to the description of arguments of the corresponding G code in the program manual.

#### 1.7.5 Operation Screen Description

This system's operation is divided into 4 types and they are described as below :

- **PMNG**

This function page mainly provides add/delete/modify/check for production and procedure and save/gear-shifting and exit for projects. Every production procedure must edit 4 items. Tool number and compensation can be inputted directly. G code and cutting parameter can continue editing after clicking the input key. Since CAD/CAM and figure transmitting are in the same function group, so screen is able to switch to CAD/CAM editing page by pressing CAM button. Please refer below as the related screen.

Users can use the direction key up/down to move the yellow cursor and also to decide the wanted editing production procedures or items. The delete key (del) can delete the production procedure at the row where the yellow cursor occurs.

O9999	N0000000	EDIT	M-RDY
<b>LNC</b>			
NAME <input type="text"/>			
NO G CODE T H CUT P	NO G CODE T H CUT P		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	<input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		

F12 ADD  
F11 COPY  
F10 MOVE DOWN  
F9 MOVE UP  
F8 TO CAM

---

F2 PMNG	F3 COMP	F4	F5 SAVE	F6 POST	F7 CLOSE
---------	---------	----	---------	---------	----------

- PMNG

This function page mainly provides open, copy, rename, delete and note for projects. All operating methods are to key-in data before clicking on function keys expect opening files. If key-in a new file name, then a new file will be opened. Otherwise, the system will open the file that is in the file list where the yellow cursor occurs. Return button is to switch back to the figure transmitting main function page. Note: please open the file at that page before start editing at the main function page. Please refer below for the related screen.

Users not only can use the up/down key to move the yellow cursor but also use the pgup/pgdn key to do extensively moving. The notice display of the file is within the bracket of the file list.

O9999		N000000		EDIT		M-RDY		LNC	
PROJEC		1							
D0000		(							
DATE		01/08/2002		SIZE		258			
F2		F3		F4		F5		F6	
F7		F8		F9		F10		F11	
F12		F13		F14		F15		F16	
F17		F18		F19		F20		F21	
F22		F23		F24		F25		F26	
F27		F28		F29		F30		F31	
F32		F33		F34		F35		F36	
F37		F38		F39		F40		F41	
F42		F43		F44		F45		F46	
F47		F48		F49		F50		F51	
F52		F53		F54		F55		F56	
F57		F58		F59		F60		F61	
F62		F63		F64		F65		F66	
F67		F68		F69		F70		F71	
F72		F73		F74		F75		F76	
F77		F78		F79		F80		F81	
F82		F83		F84		F85		F86	
F87		F88		F89		F90		F91	
F92		F93		F94		F95		F96	
F97		F98		F99		F100		F101	
F102		F103		F104		F105		F106	
F107		F108		F109		F110		F111	
F112		F113		F114		F115		F116	
F117		F118		F119		F120		F121	
F122		F123		F124		F125		F126	
F127		F128		F129		F130		F131	
F132		F133		F134		F135		F136	
F137		F138		F139		F140		F141	
F142		F143		F144		F145		F146	
F147		F148		F149		F150		F151	
F152		F153		F154		F155		F156	
F157		F158		F159		F160		F161	
F162		F163		F164		F165		F166	
F167		F168		F169		F170		F171	
F172		F173		F174		F175		F176	
F177		F178		F179		F180		F181	
F182		F183		F184		F185		F186	
F187		F188		F189		F190		F191	
F192		F193		F194		F195		F196	
F197		F198		F199		F200		F201	
F202		F203		F204		F205		F206	
F207		F208		F209		F210		F211	
F212		F213		F214		F215		F216	
F217		F218		F219		F220		F221	
F222		F223		F224		F225		F226	
F227		F228		F229		F230		F231	
F232		F233		F234		F235		F236	
F237		F238		F239		F240		F241	
F242		F243		F244		F245		F246	
F247		F248		F249		F250		F251	
F252		F253		F254		F255		F256	
F257		F258		F259		F260		F261	
F262		F263		F264		F265		F266	
F267		F268		F269		F270		F271	
F272		F273		F274		F275		F276	
F277		F278		F279		F280		F281	
F282		F283		F284		F285		F286	
F287		F288		F289		F290		F291	
F292		F293		F294		F295		F296	
F297		F298		F299		F300		F301	
F302		F303		F304		F305		F306	
F307		F308		F309		F310		F311	
F312		F313		F314		F315		F316	
F317		F318		F319		F320		F321	
F322		F323		F324		F325		F326	
F327		F328		F329		F330		F331	
F332		F333		F334		F335		F336	
F337		F338		F339		F340		F341	
F342		F343		F344		F345		F346	
F347		F348		F349		F350		F351	
F352		F353		F354		F355		F356	
F357		F358		F359		F360		F361	
F362		F363		F364		F365		F366	
F367		F368		F369		F370		F371	
F372		F373		F374		F375		F376	
F377		F378		F379		F380		F381	
F382		F383		F384		F385		F386	
F387		F388		F389		F390		F391	
F392		F393		F394		F395		F396	
F397		F398		F399		F400		F401	
F402		F403		F404		F405		F406	
F407		F408		F409		F410		F411	
F412		F413		F414		F415		F416	
F417		F418		F419		F420		F421	
F422		F423		F424		F425		F426	
F427		F428		F429		F430		F431	
F432		F433		F434		F435		F436	
F437		F438		F439		F440		F441	
F442		F443		F444		F445		F446	
F447		F448		F449		F450		F451	
F452		F453		F454		F455		F456	
F457		F458		F459		F460		F461	
F462		F463		F464		F465		F466	
F467		F468		F469		F470		F471	
F472		F473		F474		F475		F476	
F477		F478		F479		F480		F481	
F482		F483		F484		F485		F486	
F487		F488		F489		F490		F491	
F492		F493		F494		F495		F496	
F497		F498		F499		F500		F501	
F502		F503		F504		F505		F506	
F507		F508		F509		F510		F511	
F512		F513		F514		F515		F516	
F517		F518		F519		F520		F521	
F522		F523		F524		F525		F526	
F527		F528		F529		F530		F531	
F532		F533		F534		F535		F536	
F537		F538		F539		F540		F541	
F542		F543		F544		F545		F546	
F547		F548		F549		F550		F551	
F552		F553		F554		F555		F556	
F557		F558		F559		F560		F561	
F562		F563		F564		F565		F566	
F567		F568		F569		F570		F571	
F572		F573		F574		F575		F576	
F577		F578		F579		F580		F581	
F582		F583		F584		F585		F586	
F587		F588		F589		F590		F591	
F592		F593		F594		F595		F596	
F597		F598		F599		F600		F601	
F602		F603		F604		F605		F606	
F607		F608		F609		F610		F611	
F612		F613		F614		F615		F616	
F617		F618		F619		F620		F621	
F622		F623		F624		F625		F626	
F627		F628		F629		F630		F631	
F632		F633		F634		F635		F636	
F637		F638		F639		F640		F641	
F642		F643		F644		F645		F646	
F647		F648		F649		F650		F651	
F652		F653		F654		F655		F656	
F657		F658		F659		F660		F661	
F662		F663		F664		F665		F666	
F667		F668		F669		F670		F671	
F672		F673		F674		F675		F676	
F677		F678		F679		F680		F681	
F682		F683		F684		F685		F686	
F687		F688		F689		F690		F691	
F692		F693		F694		F695		F696	
F697		F698		F699		F700		F701	
F702		F703		F704		F705		F706	
F707		F708		F709		F710		F711	
F712		F713		F714		F715		F716	
F717		F718		F719		F720		F721	
F722		F723		F724		F725		F726	
F727		F728		F729		F730		F731	
F732		F733		F734		F735		F736	
F737		F738		F739		F740		F741	
F742		F743		F744		F745		F746	
F747		F748		F749		F750		F751	
F752		F753		F754		F755		F756	
F757		F758		F759		F760		F761	
F762		F763		F764		F765		F766	
F767		F768		F769		F770		F771	
F772		F773		F774		F775		F776	
F777		F778		F779		F780		F781	
F782		F783		F784		F785		F786	
F787		F788		F789		F790		F791	
F792		F793		F794		F795		F796	
F797		F798		F799		F800		F801	
F802		F803		F804		F805		F806	
F807		F808		F809		F810		F811	
F812		F813		F814		F815		F816	
F817		F818		F819		F820		F821	
F822		F823		F824		F825		F826	
F827		F828		F829		F830		F831	

- COMM

Common parameter is G100's parameter, it will be used in all production process for the same projects. This page's main function is to set the local parameters of all project production procedures. Pressing return button will be able to switch back to the main function page of the figure transmitting after complete setting. Please refer below as the related screen :

**FIX** : If the tool's cutting width is fixed or not. 1 : YES , 0 : NO.

**DIAM** : Tool diameter ( mm, this value is default, if users didn't set federate for cutting parameter, controller will use default to process.

**ESC** : Z Axis escape height ( mm ) means the cutting safe height(first cutting point)

**FEED** : Feedrate ( mm/min ), this value is default, if users didn't set federate for cutting parameter, controller will use default to process.

**CUT** : Set the percentage(%) for tool in to tool diameter

**SPIN** : Spindle rotation ( rpm ) , this value is default, if users didn't set federate for cutting parameter, controller will use default to process.

**CUTP** : 1. First cutting point: common parameter escape amount.

2. Last cutting point: cutting parameter (R) point.

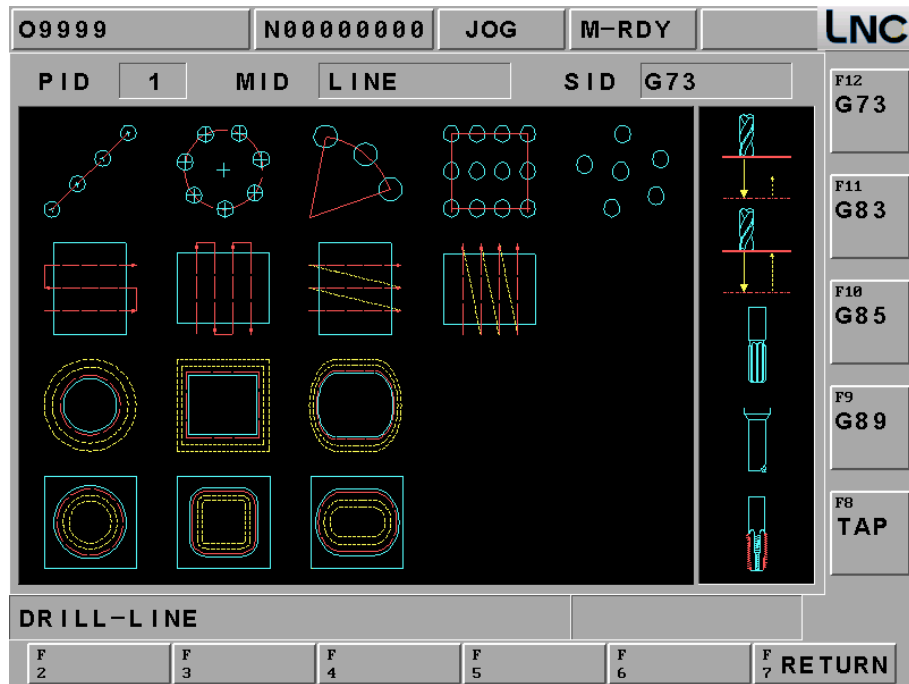
O9999	N000000	EDIT	M-RDY	LNC	
COMM					F12
FIX	0				F11
DIAM	8.000				F10
ESC	20.000				F9
FEED	120				F8
CUT	20				
SPIN	2500				
UP	0				
1 : ON , 0 : OFF					
F2	F3	F4	F5	F6	F7 RETURN

- G Code

This page's main function is to select working method. Pressing the return button to switch to the figure transmitting main function page after selecting.

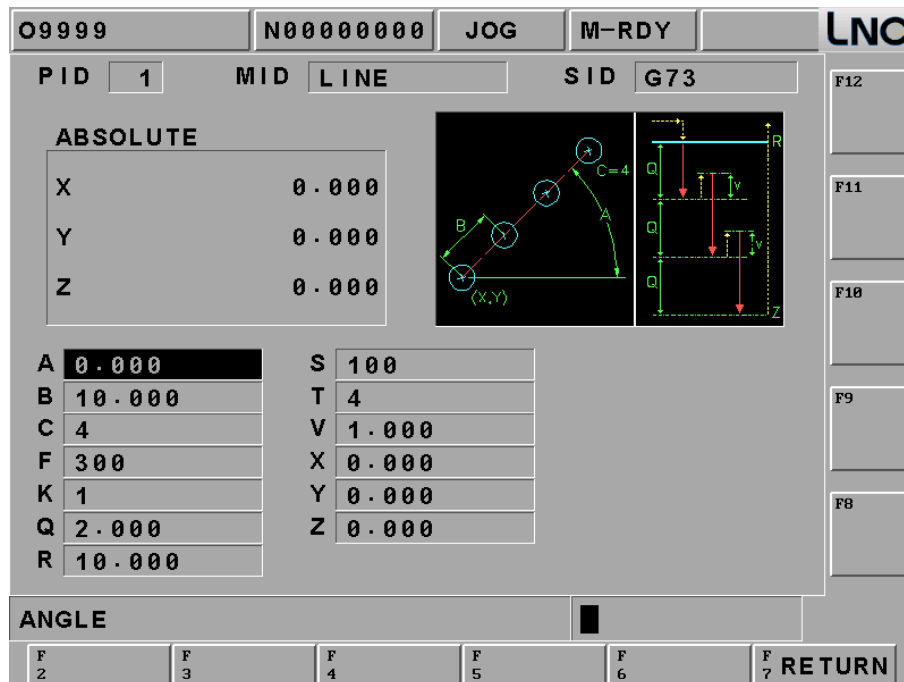
Please refer below as the related screen.

Using the direction keys up/down/left/right to move the yellow square before pressing the function button at the right side in order to complete the selecting motion. The figure at the right side and the function buttons will be different according to the item in the yellow square at the left side.



# ● CUT PARAM

This page is mainly to set the cutting parameter of a single production procedure in a project. Pressing the return button to switch back to the figure transmitting main function page after complete setting. Please refer below as the related screen :



The screenshot shows the LNC Milling Large Screen Operation Manual interface. At the top, there are status bars for 'O9999', 'N000000000', 'JOG', 'M-RDY', and the 'LNC' logo. Below these are fields for 'PID 1', 'MID LINE', and 'SID G73'. The main area is divided into several sections:

- ABSOLUTE**: A table with columns for X, Y, and Z coordinates, all set to 0.000.
- ANGLE**: A table with columns for A, B, and C angles, all set to 0.000.
- PARAMETERS**: A table with columns for S, T, V, X, Y, and Z, all set to 100, 4, 1.000, 0.000, 0.000, and 0.000 respectively.
- Diagram**: A diagram showing a cutting path with points A, B, and C, and a coordinate system (X, Y, Z).
- Buttons**: A row of buttons labeled F2, F3, F4, F5, F6, and F7, with F7 labeled 'RETURN'.

Users only need to use the figure shape reminding at the upper right side and then to key-in the related information in order complete setting.

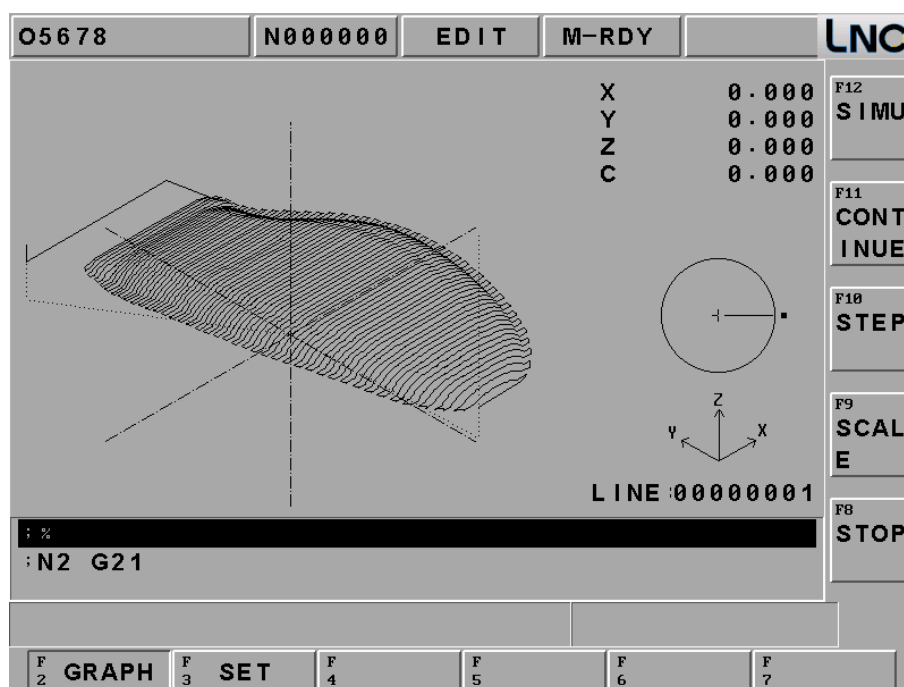
## 1.8 GRAPH

### 1.8.1 Function

Pressing **【GRAPH】** to enter into this function group. The present working path and that of the preview programs will occur in **【GRAPH】** screen. Users are able to set the view angle and the display range of the path display in **【SET】** screen.

### 1.8.2 GRAPH

The path display screen is as the below Figure. Coordinate value that is displayed at the upper-right side is the absolute coordinate of the present tool position. Lower-right side displays the coordinate view angle.



**【PRGSIM】**: when program is ok, press this button to draw program's working path.

**【CONSIM】**: press this button to re-draw the working program.

**【STPSIM】**: Press this button to show step by step of working program.

**【SCALE】**: Press this button to show 1 square, and under this square this graph will be enlarged. Users can use <PAGE↑><PAGE↓> to adjust the range. And use<↑↓> to move this square. When you move to place you want to see, press <↵> or **【SCALE】** to update the screen to be the square.

**【STOP】**: when processing **【SIMU】** 或 **【CONTINUE】**, press this button to stop processing.

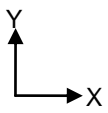
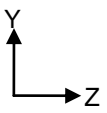
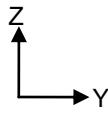
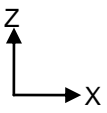
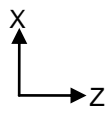
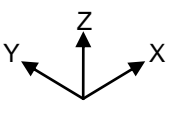
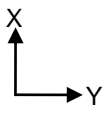
Note 1: Cursor is green color, G00 is red color and G01, G02 and G03 are yellow color.

Note 2: Cursor is green color, G00 is blue color, and G01, G02, and G03 are pink color. Center line is bright-green color.

### 1.8.3 SET

O5678	N000000	EDIT	M-RDY	LNC
<b>AXES</b> 0 ( 1=XY, 2=YZ, 3=ZX, 4=YX, 5=ZY, 6=XZ, 0=XYZ )				F12
<b>METHOD</b> 2 ( 0=MANUAL, 1=SIMU-ALL, 2=SIMU-CUT )				F11
<b>RAGE</b> ( MAX. ) X=      136504      Y=      110968      Z=      46106				F10
<b>RAGE</b> ( MIN. ) X=      -138156      Y=      -124332      Z=      17147				F9
<b>MARGIN</b> 0				F8
<b>AUTO ER</b> 1      ( 0=NO, 1=YES )				
<b>KEEP SI</b> 1      ( 0:NO 1:YES )				
<b>AUTO SI</b> 0      ( 0:AUTO 1:MANUAL )				
F2 GRAPH	F3 SET	F4	F5	F6
				F7

- **Drawing Surface** : the using coordinate view angles are ( 1=XY, 2=YZ, 3=ZX, 4=YX, 5=ZY, 6=XZ, 0=XYZ ) when key-in into the path display screen.

SET	GRAPH	SET	GRAPH
1=XY		5=ZY	
2=YZ		6=XZ	
3=ZX		0=XYZ	
4=YX			



- **Setting Method** : the drawing range of the entering **【SET】** screen (0=manual, 1= preview result-full travel, 2= preview result-cutting travel).
  - 0) **MANUAL** : Preview the drawing range as the reading manual setting's max/min values.
  - 1) **SIMU-ALL → full traveling ( including moving path )** : Preview the drawing range as the reading part program path's max/min values, the smallest value.
  - 2) **SIMU-CUT →cutting traveling ( only including cutting path )** : Preview the drawing range as the max and/or min values of the reading cutting path, the smallest value.
- **Drawing Range (max)** : Setting to use the manual method to draw the largest value of X, Y, and Z axes range.
- **Drawing Range (min)** : Setting to use the manual method to draw the smallest value of X, Y, and Z axes range.
- **AUTO ER** : Assumed under the condition of not executing program preview. Users are able to choose whether or not to delete the previous path display screen when enable cutting function while the machine is doing the path display ( 0= Not Delete , 1=Auto Delete ) .
- **KEEP SI** : At auto mode, you can set if you want to use previewed graph or to preserve preview.
  - 0) NO : show exact working path.
  - 1) YES : show current preview path.
- **AUTO SI** : Switch to **【GRAPH】** to see if the auto preview is processing.
  - 0 : Start auto preview function. When users change to path page, system will start to do preview automatically and draw program path.
  - 1 : Close auto preview function . When users change to path page, system will not start to do preview automatically and will not draw program path.

## 1.9 DGNOS

Pressing <DGNOS> and 6 main function screens occur, which are 【ALARM】, 【IOCSA】, 【MLC2】,【SYSTEM】, 【COMM】,【SYSU】 and 【CIRCUL】. Users are able to know the condition of HMI signal and machine condition from the DGNOS function screen in order to do the maintenance and the system testing,

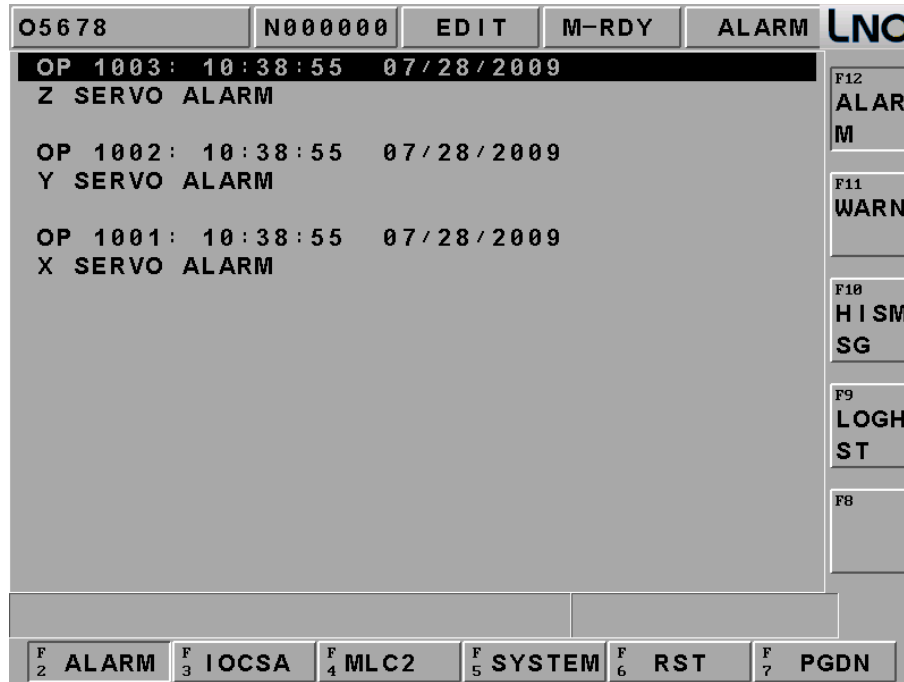
### 1.9.1 ALARM

Clicking 【ALARM】 to enter into the 【ALARM】, 【WARN】, 【HISMSG】 and 【LOGHST】. When there is any alarm message or warning message from MLC, the alarm message will occur on the screen. Users can use this screen to remove any irregular condition of the controller.



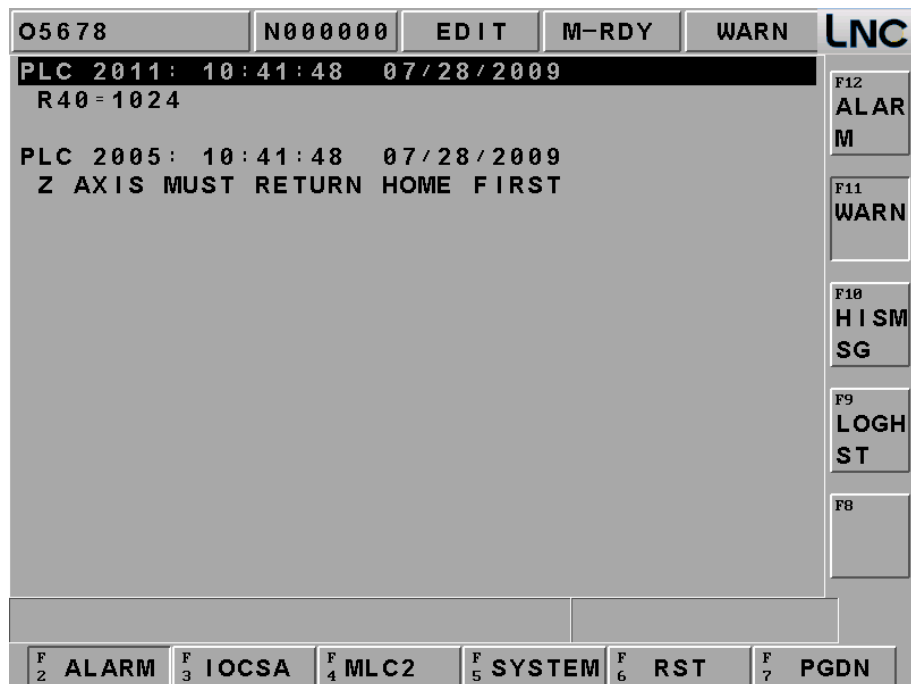
- **【Alarm】**

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.



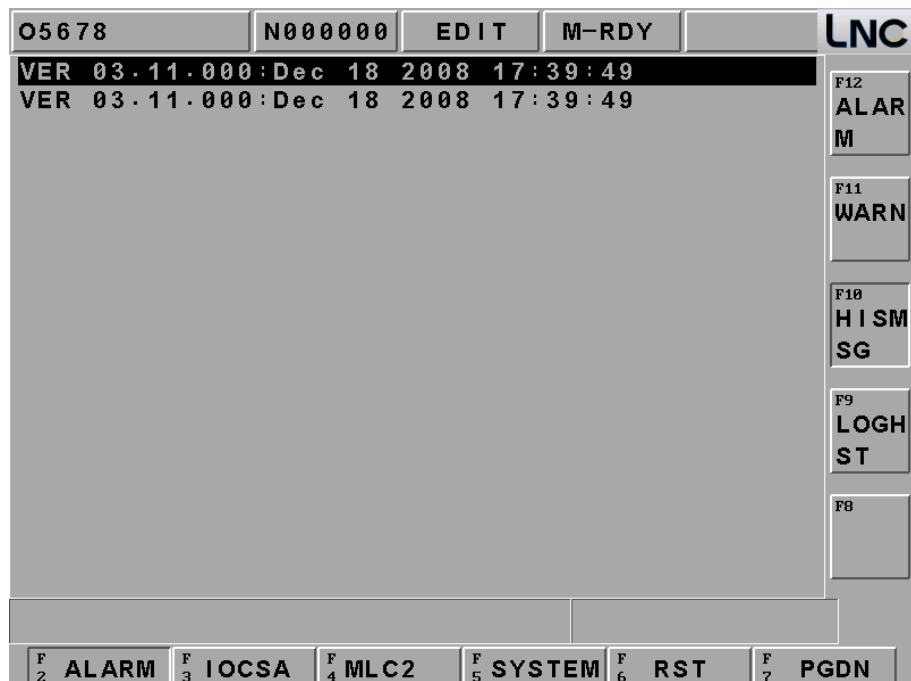
- **【Warn】**

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 LOW" 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】**

System version and program debugging message will occur. This page displays the system current running condition.



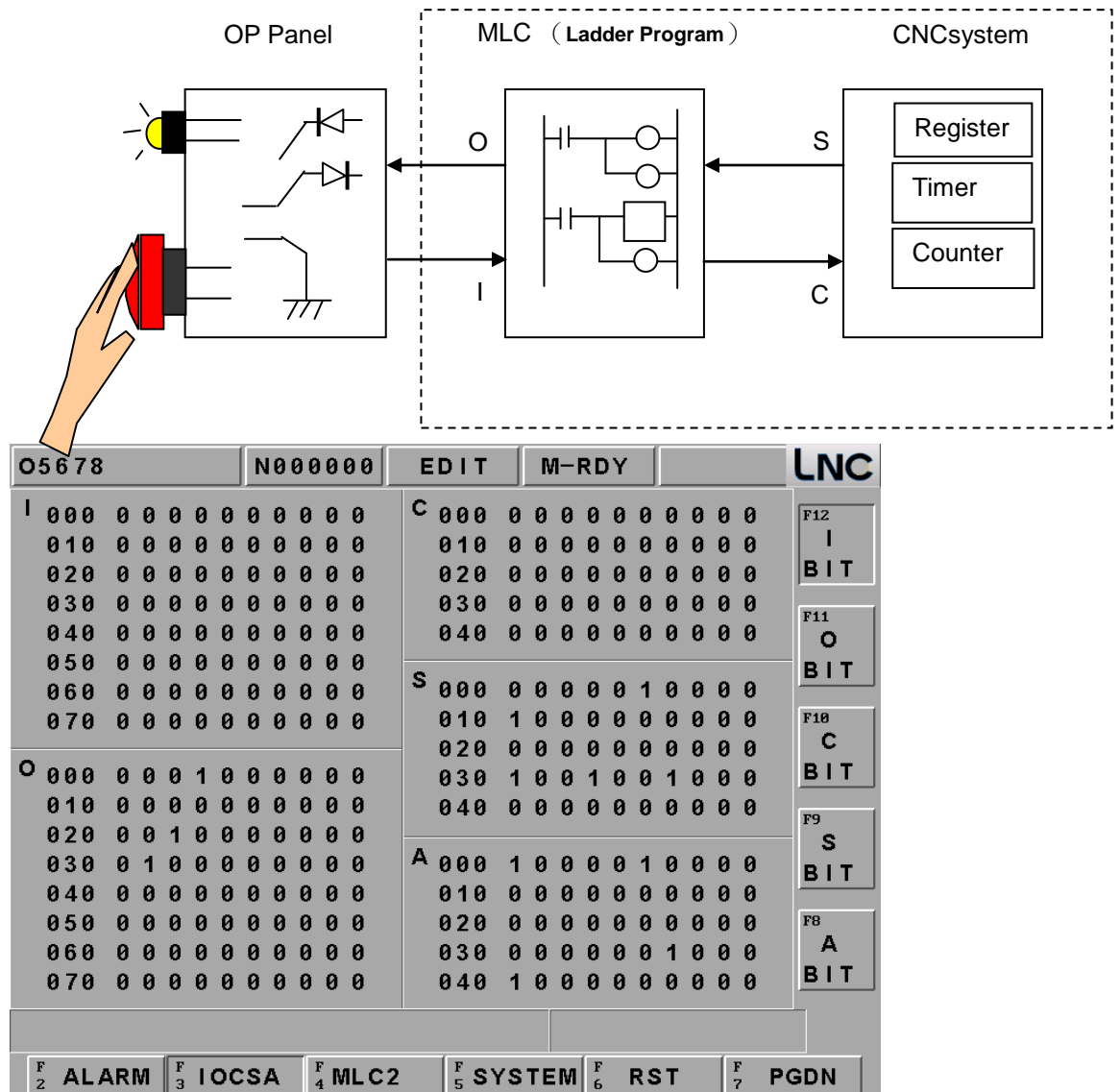
- **【LOGHST】**

Able to display system's all alarms/alarm records, even delete the alarm, the records will not disappear.



## 1.9.2 IOCSA

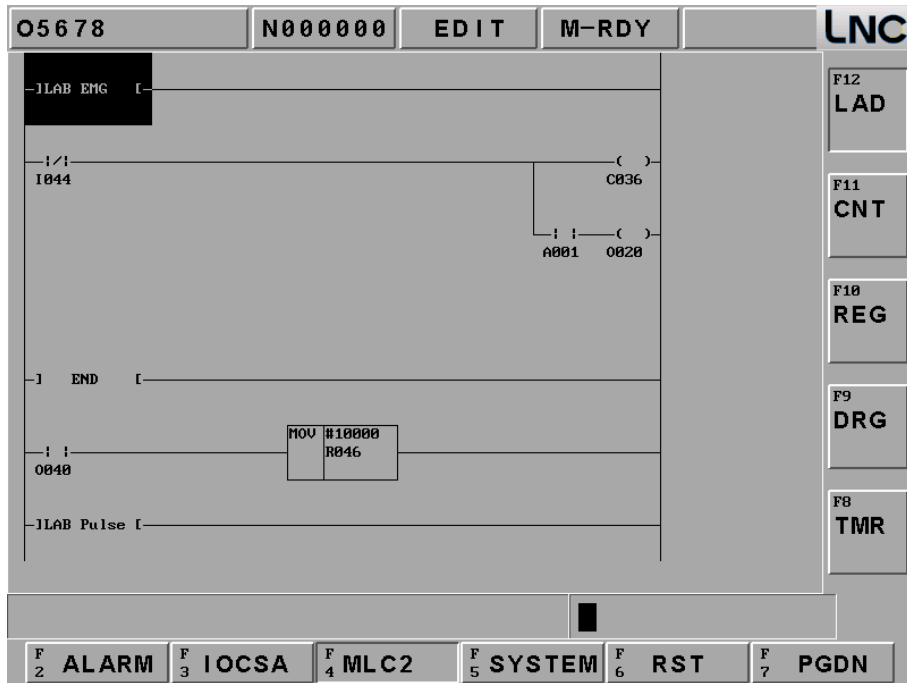
【IOCSA】 is to check the screen of the I/O or system internal condition which including I, O, C, S, A.



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,

## 1.9.3 MLC2

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



Users are able to add number on the wanted searching English 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.

Press **【CNT】** to check data as below.

O5678		N000000		EDIT		M-RDY		LNC	
NUM	CUR . VASET . VAL	NUM	CUR . VASET . VAL						
000	0 0	016	0 0						
001	0 0	017	0 0						
002	0 0	018	0 0						
003	0 0	019	0 0						
004	0 0	020	0 0						
005	0 0	021	0 0						
006	0 0	022	0 0						
007	0 0	023	0 0						
008	0 0	024	0 0						
009	0 0	025	0 0						
010	0 0	026	0 0						
011	0 0	027	0 0						
012	0 0	028	0 0						
013	0 0	029	0 0						
014	0 0	030	0 0						
015	0 0	031	0 0						

F2 ALARM
F3 IOCSA
F4 MLC2
F5 SYSTEM
F6 RST
F7 PGDN

F12 LAD  
 F11 **CNT**  
 F10 REG  
 F9 DRG  
 F8 TMR

Press **【REG】** to check data as below.

O5678		N000000		EDIT		M-RDY		LNC	
NUM	VALUE	NUM	VALUE	NUM	VALUE				
000	0	016	15	032	0				
001	0	017	15	033	0				
002	0	018	4	034	0				
003	0	019	0	035	0				
004	0	020	0	036	0				
005	0	021	0	037	0				
006	0	022	0	038	0				
007	0	023	0	039	0				
008	0	024	0	040	0				
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	1	030	0	046	0				
015	10	031	0	047	0				

F2 ALARM
F3 IOCSA
F4 MLC2
F5 SYSTEM
F6 RST
F7 PGDN

F12 LAD  
 F11 CNT  
 F10 **REG**  
 F9 DRG  
 F8 TMR



Press 【TMR】、【REG】、【DRG】、【TMR】 to see data as below.

O5678

N000000

EDIT

M-RDY

LNC

NUM	VALUE	NUM	VALUE	NUM	VALUE
000	0	016	16	032	0
001	1	017	17	033	0
002	2	018	18	034	0
003	3	019	19	035	0
004	4	020	20	036	0
005	5	021	21	037	0
006	6	022	22	038	0
007	7	023	23	039	0
008	8	024	24	040	0
009	9	025	0	041	1000
010	10	026	0	042	1000
011	11	027	0	043	1000
012	12	028	0	044	1000
013	13	029	0	045	1000
014	14	030	0	046	1000
015	15	031	0	047	1000

F12  
LAD

F11  
CNT

F10  
REG

F9  
DRG

F8  
TMR

F2 ALARM

F3 IOCSA

F4 MLC2

F5 SYSTEM

F6 RST

F7 PGDN

Press 【TMR】 to check data as below.

O5678

N000000

EDIT

M-RDY

LNC

NUM	CUR	VASET	VAL	NUM	CUR	VASET	VAL
000	0	0		016	0	0	
001	0	0		017	0	0	
002	0	0		018	0	0	
003	0	5		019	0	5	
004	0	5		020	0	0	
005	0	5		021	0	0	
006	0	2		022	0	0	
007	0	2		023	0	0	
008	0	60		024	0	0	
009	0	1		025	0	0	
010	0	30		026	0	0	
011	0	5		027	0	0	
012	0	2		028	0	0	
013	0	0		029	0	0	
014	0	10		030	0	0	
015	0	0		031	0	0	

F12 LAD

F11 CNT

F10 REG

F9 DRG

F8 TMR

F2 ALARM

F3 IOCSA

F4 MLC2

F5 SYSTEM

F6 RST

F7 PGDN

#### 1.9.4 SYSTEM

Pressing **【SYSTEM】** and the screen is as below. (System data is to display screen of the system maintenance variable. This function is for designers and technical people to use.)

- **【SYSTEM】**

Input numbers and press <Enter>, you can find it.

O5678		N000000		EDIT		M-RDY		LNC	
NUM	VALUE	NUM	VALUE	NUM	VALUE	NUM	VALUE		
000	0	020	0	040	0			F12	GBL
001	0	021	0	041	0				
002	0	022	0	042	0			F11	H·D
003	0	023	0	043	0				
004	0	024	0	044	0				
005	0	025	0	045	0			F10	
006	0	026	0	046	0				
007	0	027	0	047	0				
008	0	028	0	048	0				
009	0	029	0	049	0				
010	0	030	0	050	0			F9	
011	0	031	0	051	0				
012	0	032	0	052	0				
013	0	033	0	053	0				
014	0	034	0	054	0				
015	0	035	0	055	0				
016	0	036	0	056	0				
017	0	037	0	057	0				
018	0	038	0	058	0				
019	0	039	0	059	0				

-- PAGE : 1 / 3 --

F2	ALARM	F3	IOCSA	F4	MLC2	F5	SYSTEM	F6	RST	F7	PGDN
----	-------	----	-------	----	------	----	--------	----	-----	----	------

- **【H.D.】**

There are different items in the diagnosis function item. 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 is correct. If the question mark (?) changes to across (X) after diagnosis on the right windows side, it indicates this item has error. Users are able to get the possible error message from the solving method in order to check and to solve the problem.

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.

O5678	N000000	EDIT	M-RDY	LNC
EPCIO: PCC3850-2				GBL
DOG: X=0 Y=0 Z=0 C=0				H·D
INDEX: X=1 Y=1 Z=1 C=1				
ITEMS: 1) SET1 REMOTE I/O MASTER O				
2) SET1 REMOTE I/O SLAVE0 O				
3) SET1 REMOTE I/O SLAVE1 O				
4) SET2 REMOTE I/O MASTER O				
5) SET2 REMOTE I/O SLAVE0 O				
6) SET2 REMOTE I/O SLAVE1 O				
7) AXIS1 ENCODE DETECT O				
8) AXIS2 ENCODE DETECT O				
9) AXIS3 ENCODE DETECT O				
10) AXIS4 ENCODE DETECT O				
ANSWER: CHECK I/O SET1 CONNECTOR				
-- PAGE: 1 / 2 --				
<div>ALARM</div> <div>IOCSA</div> <div>MLC2</div> <div>SYSTEM</div> <div>RST</div> <div>PGDN</div>				

- **【Version】**

Press **【Version】** to see version number, press Ok to close this message.

O9999	N000000	EDIT	M-RDY	LNC		
NUM	VALUE	NUM	VALUE	NUM	VALUE	F12
000	0	020	0	040	0	GBL
001	0	021	0	041	0	F11
002	0	022	0	042	0	
003	0	023	0	043	0	H·D
004	0	024	0	044	0	
005	0	025	0	045	0	F10
006	0	026	0	046	0	
007	0	027	0	047	0	F9
008	0	028	0	048	0	
009	0	029	0	049	0	F8
010	0	030	0	050	0	
011	0	031	0	051	0	VER
012	0	032	0	052	0	
013	0	033	0	053	0	
014	0	034	0	054	0	
015	0	035	0	055	0	
016	0	036	0	056	0	
017	0	037	0	057	0	
018	0	038	0	058	0	
019	0	039	0	059	0	
-- PAGE: 1 / 3 --						
<div>F2 OK</div> <div>F3</div> <div>F4</div> <div>F5</div> <div>F6</div> <div>F7</div>						

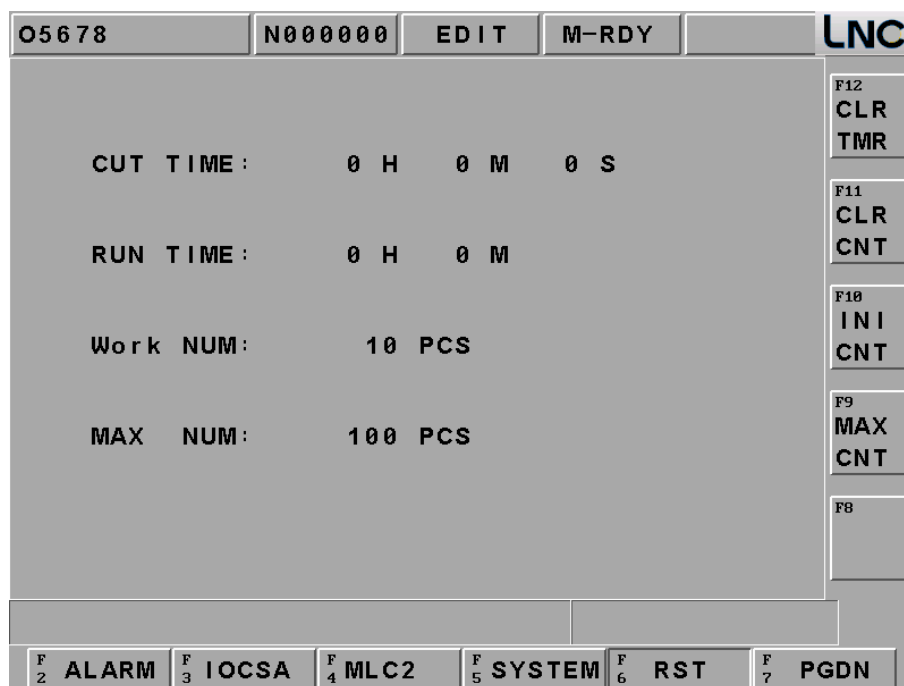
### 1.9.5 Parameter

When pressing **【RST】**, monitor's left side will show buttons as below, **【CLRTMR】**、**【CLRCNT】**、**【INICNT】**、**【MAXCNT】**, you can use this buttons to clear operation time and parts.

**Cutting Time** : When users press "Cycle Start" button on OP, cutting time will reset to zero and then start timing until the part program finishes.

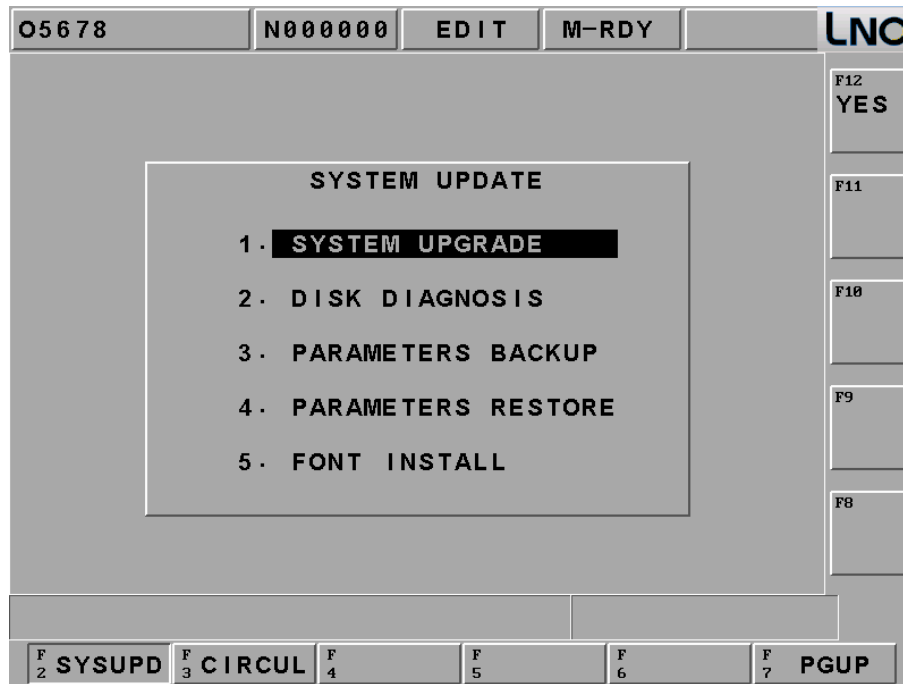
**Working Time** : Working time is the sum of cutting time every time rebooting. It will be set to zero until exit (returns to zero). Using **【RST】** + **【RUNTIME】** will return the working time back to zero immediately.

**Working Piece** : When CNC has read M02, M03 or other M codes, the system will adding up the working piece automatically. Using **【RST】** + **【PARTCONT】** will return the working piece number back to zero.



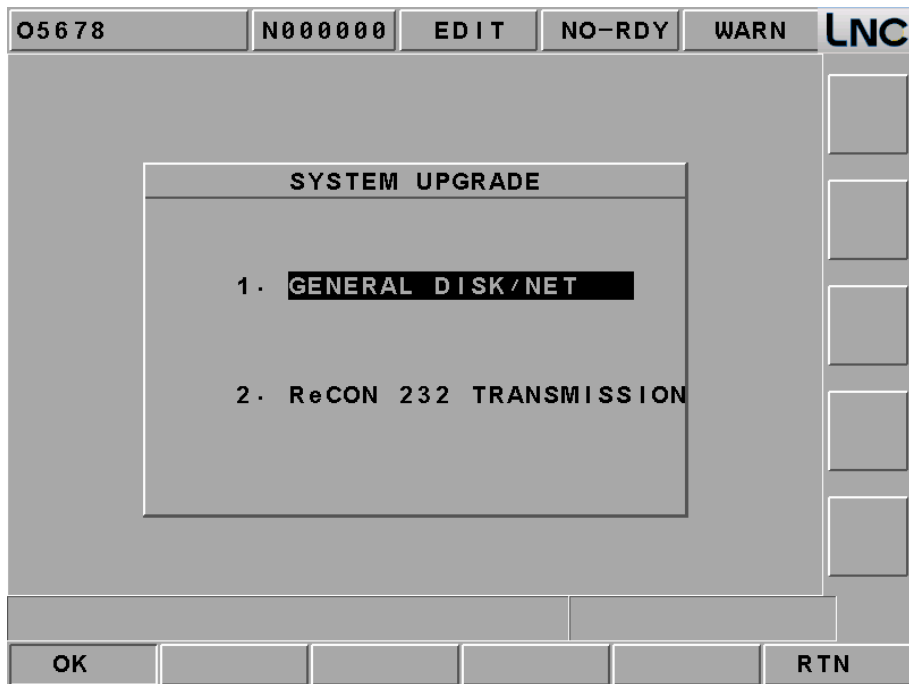
### 1.9.6 SYSUPD

This function can be executed only under that condition that CNC is not ready. Please pressing【SYSUPD】button after pressing the EMG-STOP key. At this time, a screen which function is selected by the cursor (like Figure 2.10-3) will occur. This screen allows users to select the wanted working item. Each function is listed as below :

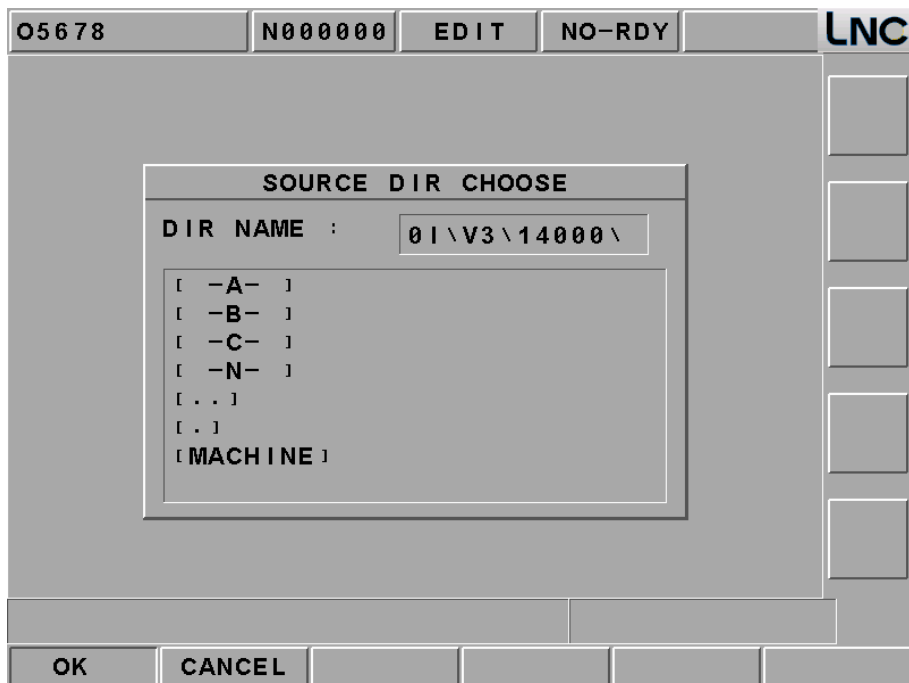


## ● 【UPGRADE】

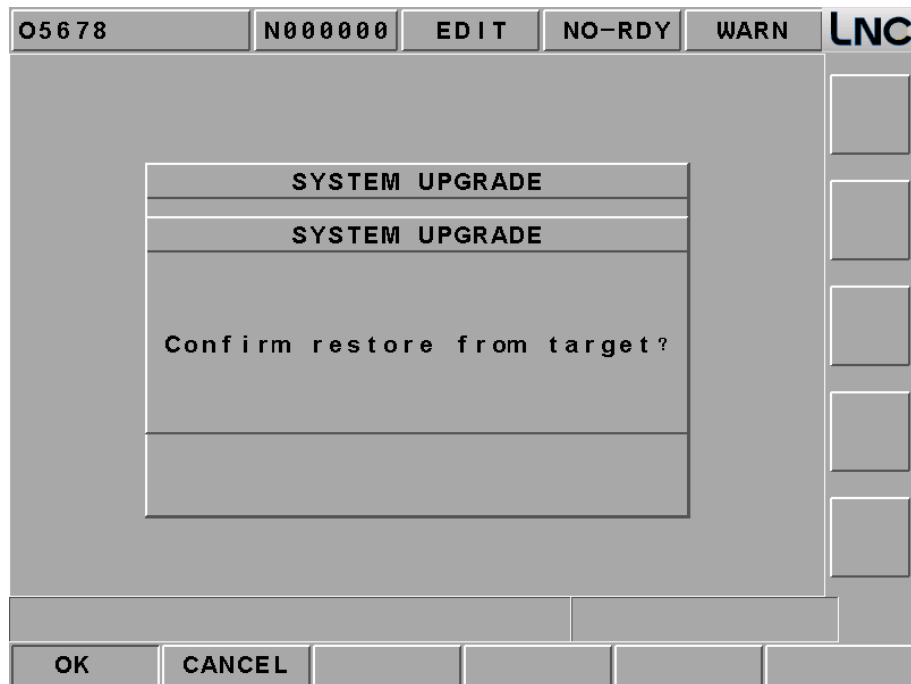
This function offers two ways for update: general disk/RS232 transmission. Please install LNC Technology Co., Ltd. latest version software if choosing this function. Pressing 【YES】 , the installation screen will occur. There is installation program instruction during installation so you only need to follow the instruction to upgrade the system.



At the system upgrade file , move cursor to the line and press 【OK】



System will double confirm if you want to do upgrade, make sure if you really want to do upgrade and press **【OK】**



Then system will turn to DOS page as below to show current version and update version, press **【Y】** and system will process upgrade automatically.

```
=====
WELCOME TO INSTALL LNC-M515i SERIES
=====
Current Version:
    Not install

Installing Version:
    M515i_VER_03.20.000

Are you sure to install?[Y,N]?
```

Please wait, file preparing.

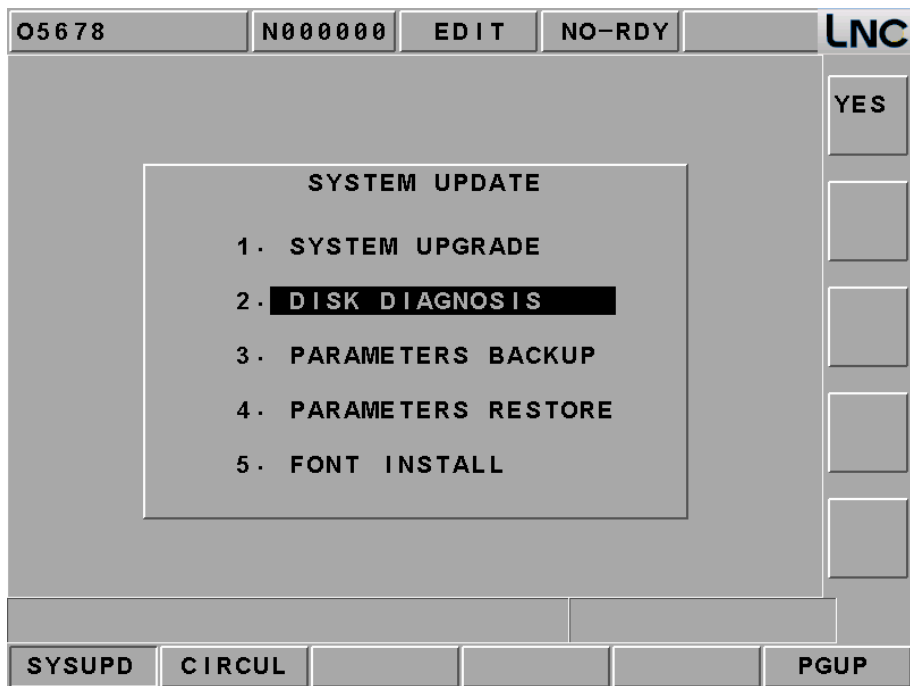
```
Installing...
Please wait,file preparing ...
```

System upgrade is completed, please press any key to reboot and you can use new version software later.

```
Wonderful ! Fully install finish!!

Press any key to continue . . .
```

- **【H.D CHECK】**



Selecting this function, a confirm dialog box will occur :





Pressing the confirm button to return back to DOS mode. 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 :

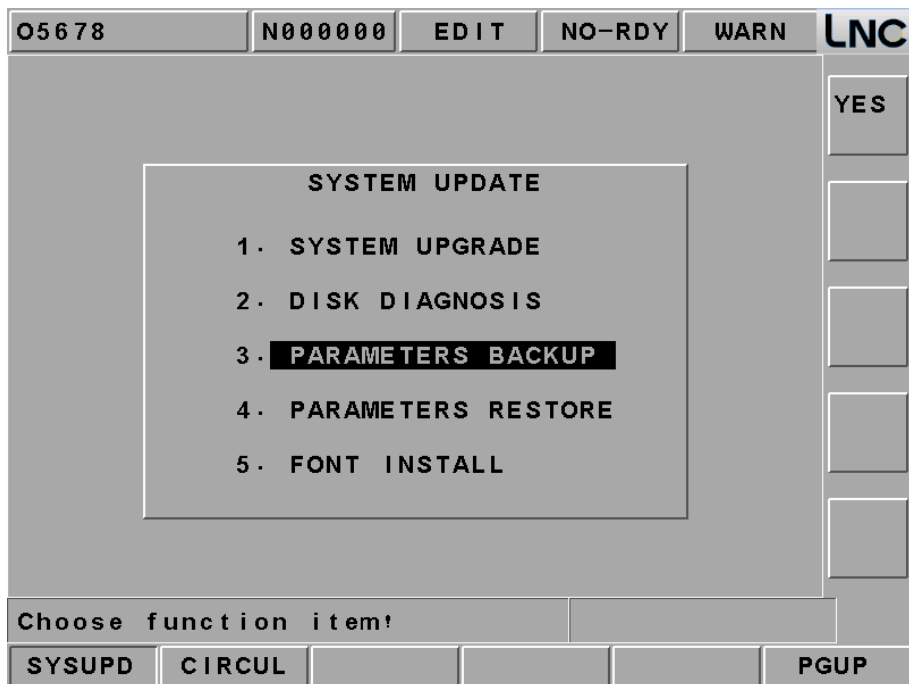
As below :

```
1.PCscan
  Virus checking
2.DISK doctor
  disk diagnostic and errer-fix
3.DEFRAG
  Disk access performance enhancement

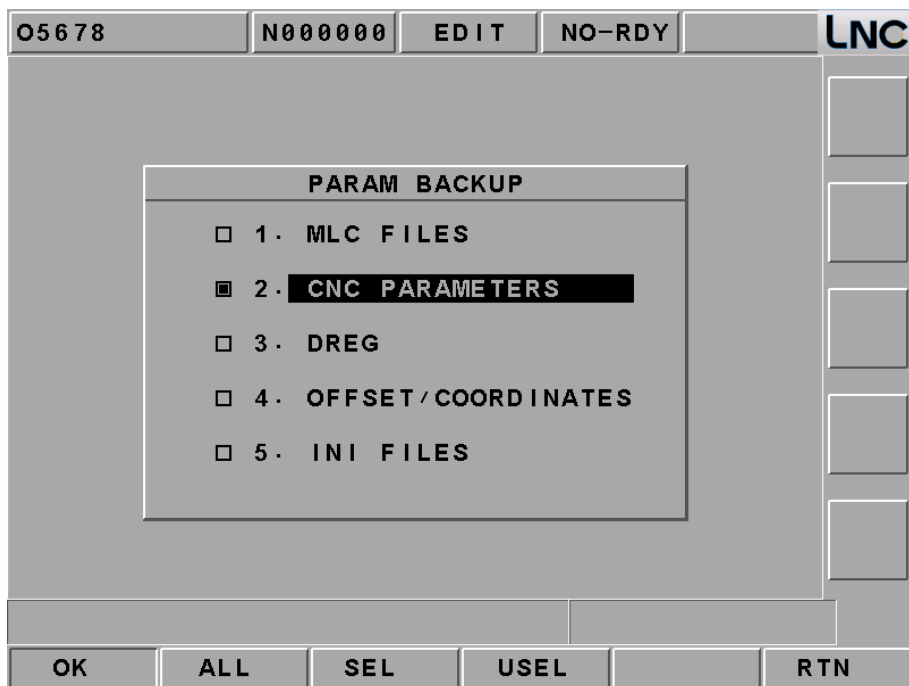
0.Quit

Choise an Option[0,1,2,3]?
```

● **【Backup】**



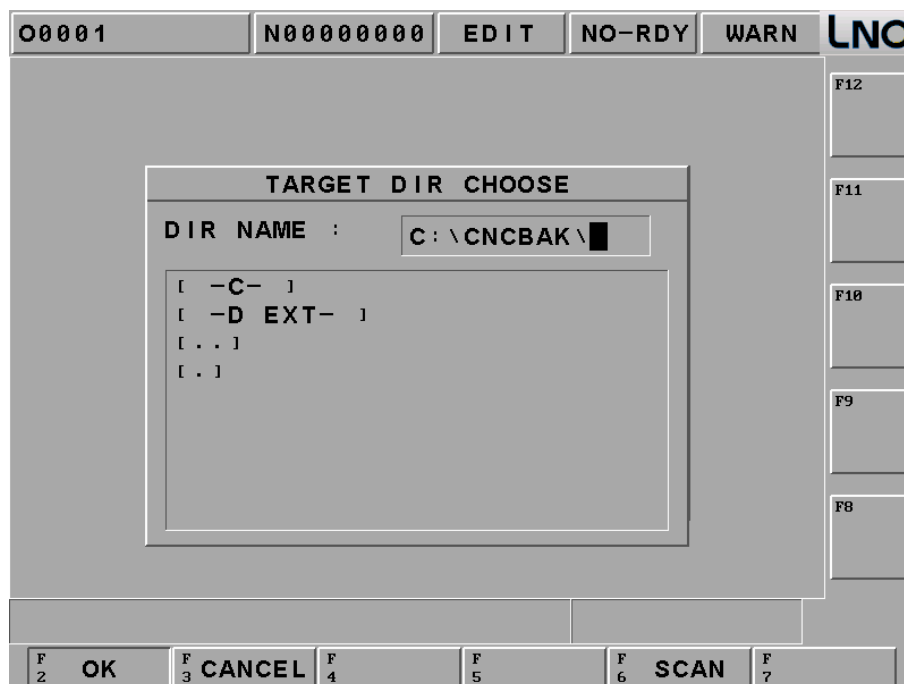
You will see below later, choose the one you want.



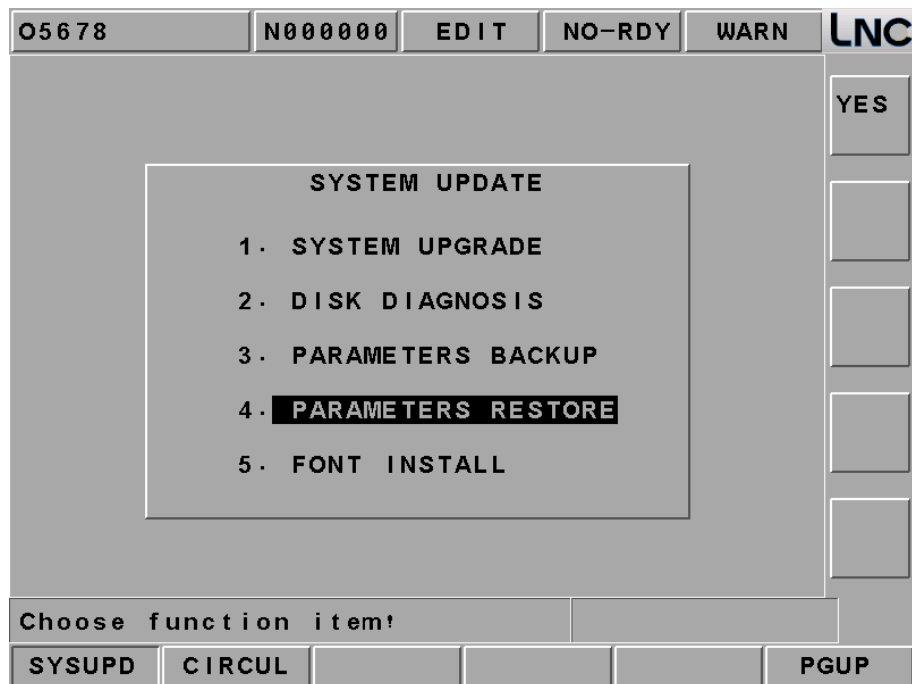
Select backup method as below.



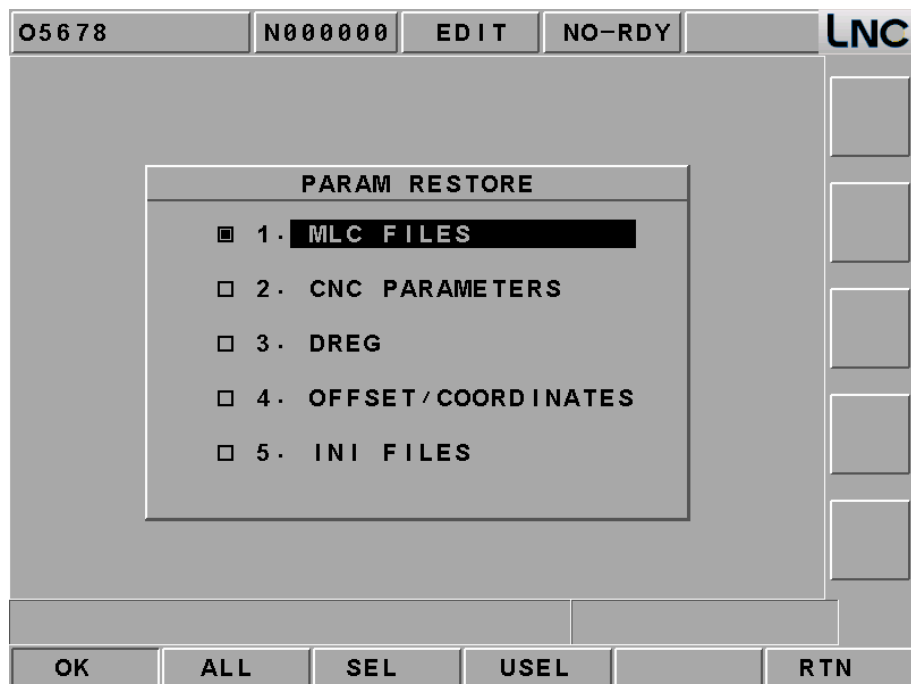
Press **【OK】** to see path, select the path you want t and complete backup.



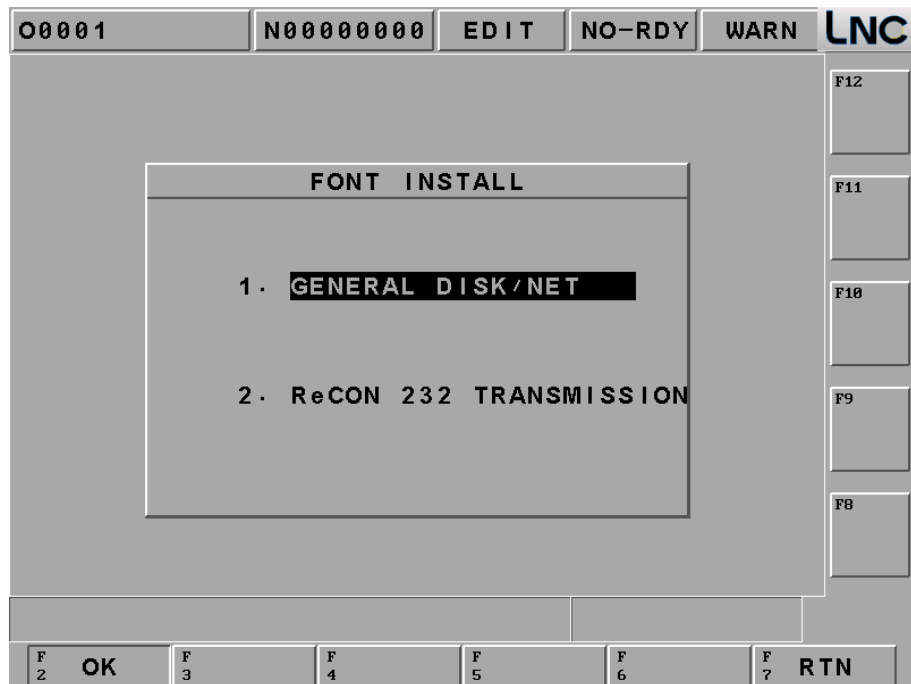
● **【Restore】**



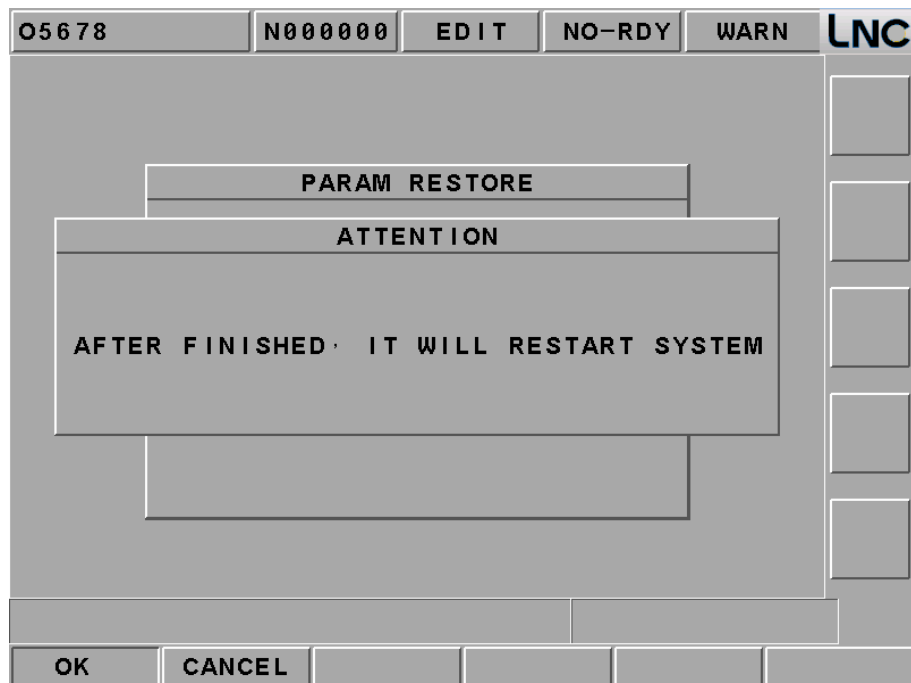
You will see below later, choose the one you want.



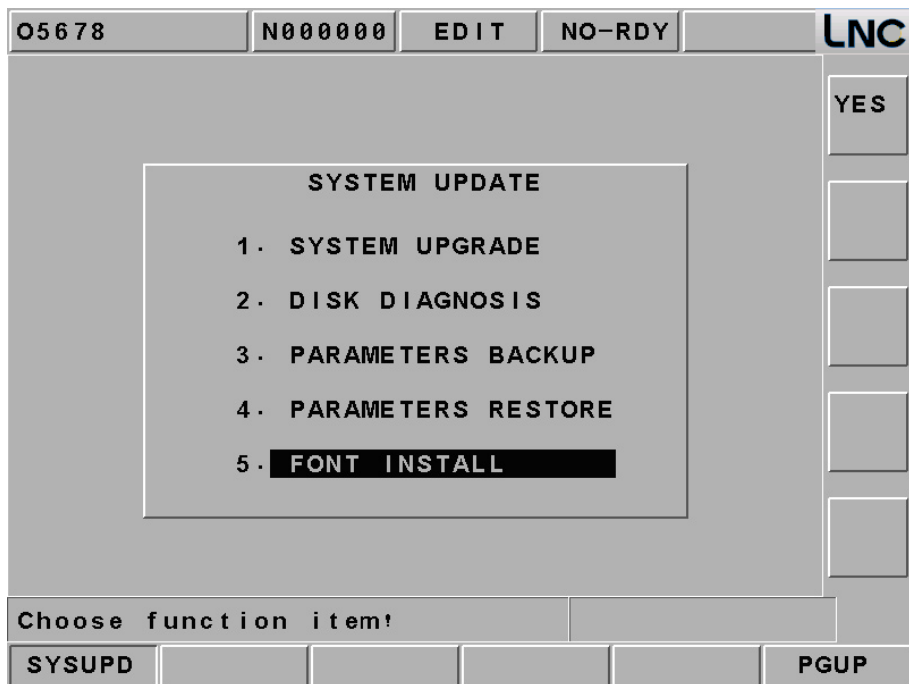
Select method as below.



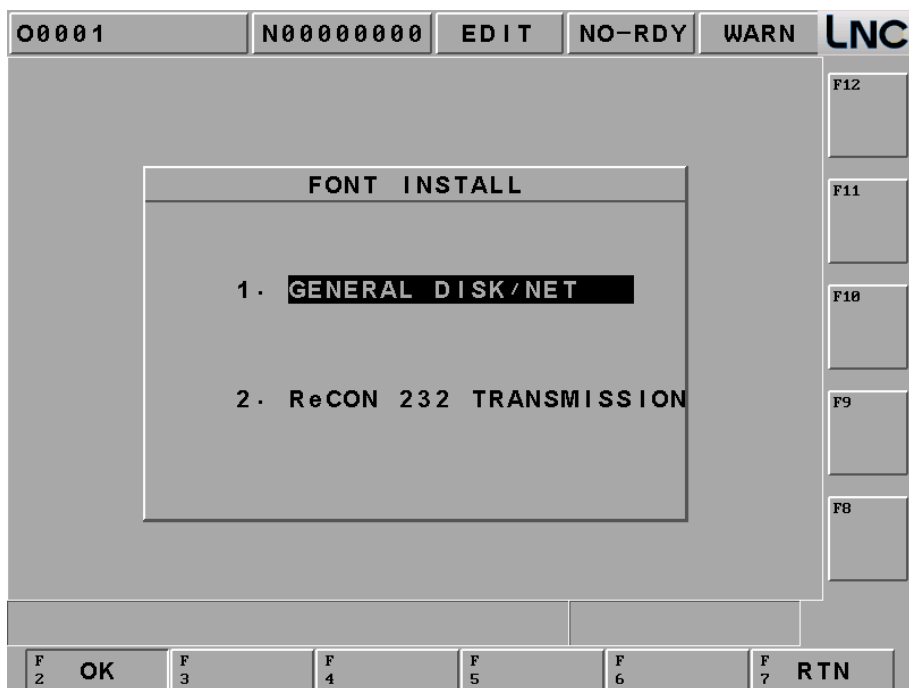
Select source path and press **【OK】** to restore.



● 【Font】



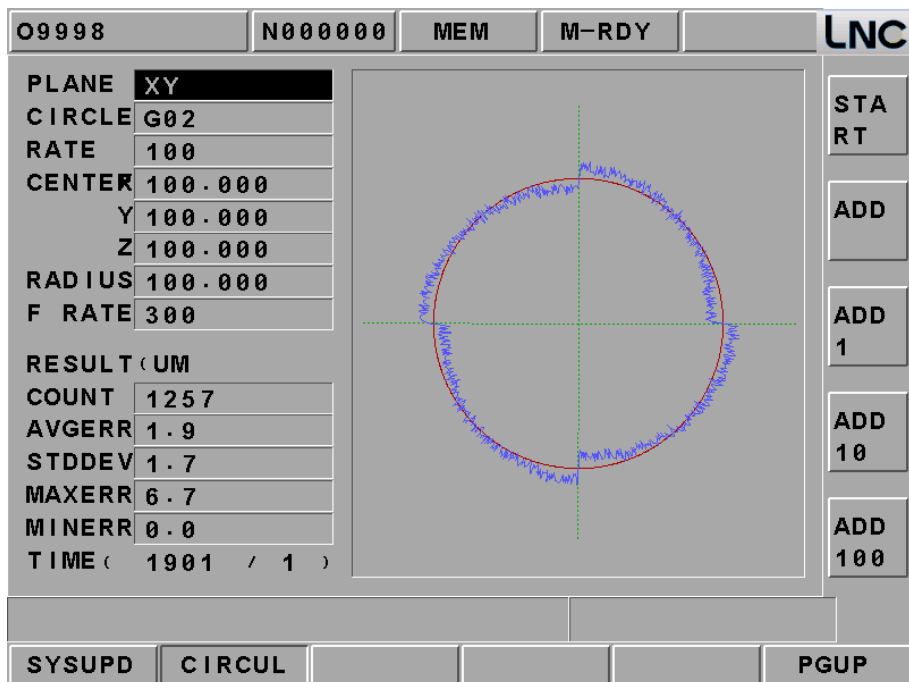
Select method as below.



Select source path and press 【OK】 to continue.



### 1.9.7 CIRCUL



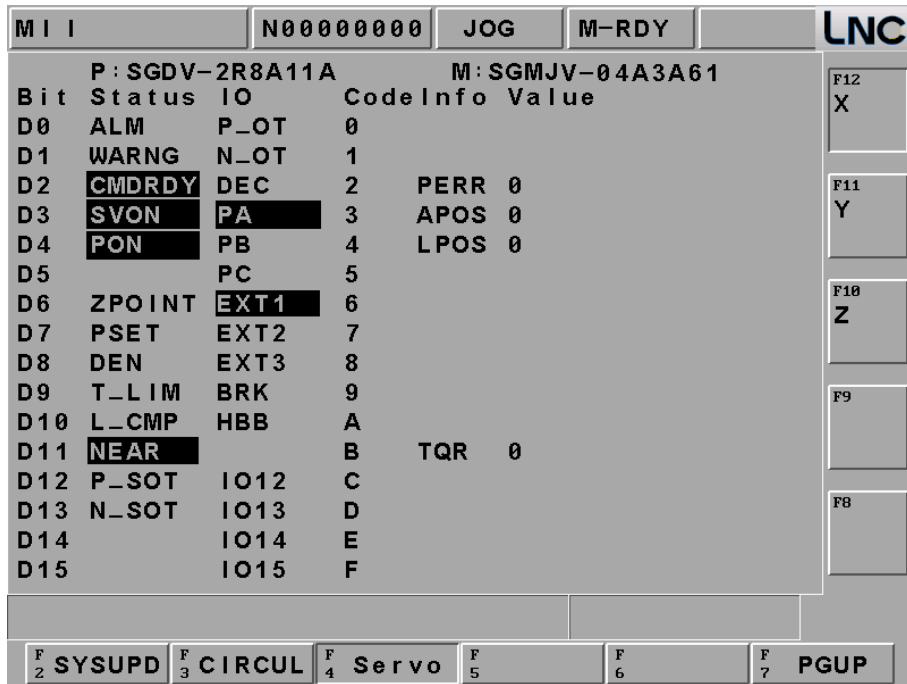
- 【Panel】** Able to choose any one of XY, YZ, ZX panes as the circular pane from the popup menu after pressing <input>.
- 【CW/CCW】** Able to choose CW (G02) or CCW (G03) as the cycling motion from the popup menu after pressing <input>.
- 【Sample Period】** Setting the sample interval time (ms) for estimation error.

<b>【Center Point XYZ】</b>	Absolute coordinates ( X 、 Y 、 Z ) of the entered center point.
<b>【Radius】</b>	Setting radius of circular ( mm ) .
<b>【Feedrate】</b>	Setting feedrate of circular ( mm/min ) .
<b>【Sample Point #s】</b>	Total numbers of the sample estimation: Sample point numbers= ( 2*PI*circular radius*60*1000 ) ／ ( feedrate * sample interval ) .  <b>Note</b> : PI= 3.1415926
<b>【Average Error】</b>	Display average error value.  Average error =total/sample point numbers
<b>【Standard Error】</b>	The difference between the data information and the balance number.
<b>【Max Error】</b>	The largest circular error value.
<b>【Min Error】</b>	The smallest circular error value.
<b>【Ratio】</b>	Display the circular error's zoom-in ratio ( N / 1 ) , N is the ratio.  <b>Note:</b> the canned cycle error is the distance between each simple point and the center point and also the different value between each simple point and cycle radius.
<b>【Ratio +/-】</b>	To set the error zoom-in ratio of the sub-function operation button. Pressing this key, this button will be sagged and <b>【Ratio -】</b> will occur. Pressing the key again, this button will be raising and <b>【Ratio +】</b> will occur.
<b>【1】</b>	When the upper key displays <b>【Ratio +】</b> , the ratio will increase by 1 when pressing this button. When the upper button display <b>【Ratio -】</b> , the ratio will decrease by 1 when pressing this button.
<b>【10】</b>	when the upper button displays <b>【Ratio +】</b> , the ratio will increase by 10 when pressing this button. When the upper button display <b>【Ratio -】</b> , the ratio will decrease by 10 when pressing this button.
<b>【100】</b>	when the upper button displays <b>【Ratio +】</b> , the ratio will increase by 100 when pressing this button. When the upper button display <b>【Ratio -】</b> , the ratio will decrease by 100 when pressing this button.
<b>【Starting】</b>	After canned cycle parameter setting is completed, pressing this button in order to indicate the setting is completed. Wait until the users press CYCLE START button in order to start the canned cycle motion. Pressing this button again, the key will raise, which means the canned cycle motion is canceled.



## 1.9.8 Servo Info

Digital model can use servo info page to understand current servo drive and motor situation. :



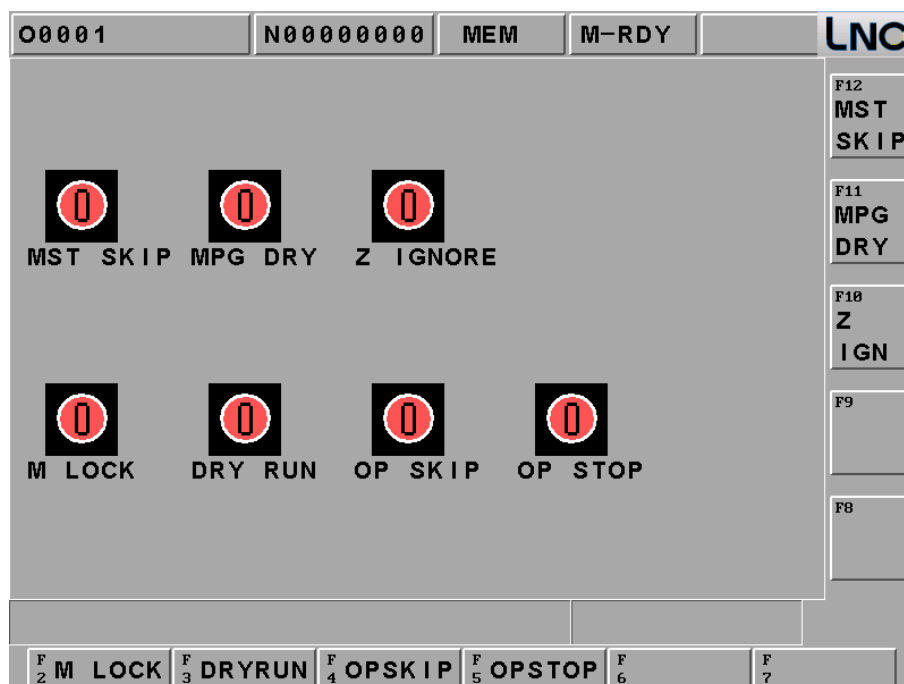
Servo Status(1)		
D0	ALM	Alarm
D1	WARNG	warning
D2	CMDRDY	Received command
D3	SVON	Servo ON
D4	PON	Main power ON
D5		
D6	ZPOINT	Z point
D7	PSET	Positioning OK
D8	DEN	Output OK
D9	T_LIM	Torque limit arrive
D10	L_CMP	Lock complete
D11	NEAR	Near positioning range
D12	P_SOT	(+)OT signal
D13	N_SOT	(-)OT signal
D14		
D15		

Servo Status(2)		
<b>0</b>	<b>P_OT</b>	CW protection input
<b>1</b>	<b>N_OT</b>	CCW protection input
<b>2</b>	<b>DEC</b>	Home DEC <b>LS</b> input
<b>3</b>	<b>PA</b>	Encoder <b>A</b> input
<b>4</b>	<b>PB</b>	Encoder <b>B</b> input
<b>5</b>	<b>PC</b>	Encoder <b>C</b> input
<b>6</b>	<b>EXT1</b>	1 external lock signal input
<b>7</b>	<b>EXT2</b>	2 external lock signal input
<b>8</b>	<b>EXT3</b>	3 external lock signal input
<b>9</b>	<b>BRK</b>	Brake output
<b>A</b>	<b>HBB</b>	Stop singla input or <b>HWBB1/HWBB2</b>
<b>B</b>		
<b>C</b>	<b>IO12</b>	At <b>Pn81E.0 CN1</b> input signal
<b>D</b>	<b>IO13</b>	At <b>Pn81E.1 CN1</b> input signal
<b>E</b>	<b>IO14</b>	At <b>Pn81E.2 CN1</b> input signal
<b>F</b>	<b>IO15</b>	At <b>Pn81E.3 CN1</b> input signal
<b>P</b>	Drive model number	
<b>M</b>	Motor model number	
<b>PERR</b>	Servo lag	
<b>APOS</b>	Current <b>Z</b> signal encoder value	
<b>LPOS</b>	<b>Z</b> signal lock encoder value	
<b>TQR</b>	CW/CCW torque %, over <b>120%</b>	

## 1.10 SOFTPL

(Note : M515i 、M516i 、M516id did not support this function)

Press <SOFTPL> to enter into the diagrams display screen. There are 7 function selection switches which are corresponding to 7 function buttons on the screen and they are **【M LOCK】** , **【DRY RUN】** , **【OP STOP】** , **【MST SKIP】** , **【MPG DRY】** , **【Z IGNORE】**



Press once=ON, press again to cancel this function.

Details for soft panel :

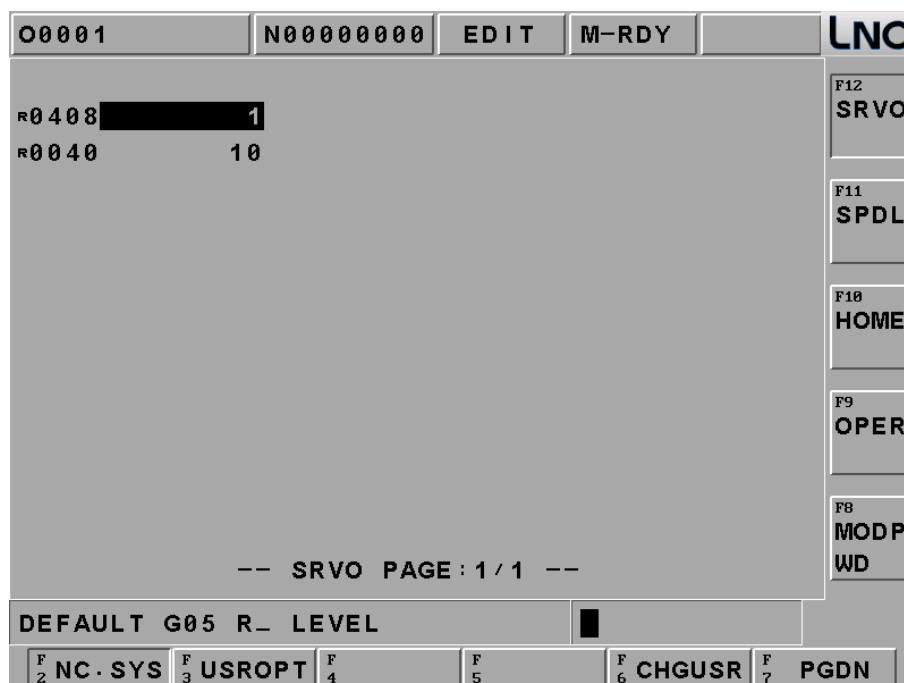
Function Switches	ON	OFF
<b>M LOCK</b>	The controller will continue executing program while the program is executing. But, the moving command of the servo axis will NOT be outputted. So actually, the servo axis is completed stop (no movement).	The moving command of the servo axis will be outputted with the program while the program is executing.
<b>DRY RUN</b>	It will follow the parameter's G01 velocity, not the program's velocity(the velocity of Pr,1004 setting)	The designated speed of the servo feed program.

Function Switches	ON	OFF
<b>OP SKIP</b>	Program begins with "/" symbol will be skipped and not be executed.	Single block that begins with "/" symbol will be executed normally.
<b>OP STOP</b>	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.
<b>MST Ignore</b>	Ignore M, S, T codes commanding.	Consider M, S, T codes commanding.
<b>MPG DRY RUN</b>	System starts to process, hand wheel can control the program processing, and the program coordinate and servo axis to change together. The faster MPG runs, the faster program runs. But MPG will not faster then program feetrage. When MPG stops, the program will also stop. MPG CCW will not control program.	Hand wheel can not modify the program coordinate and let the servo axis moves according to it.
<b>Z Ignore</b>	Ignore any command from Z Axis	Process any command from Z Axis.

## 1.11 PARAM

Pressing <PARAM> and the parameter setting screen occurs. There are several main function menus such as 【NC. SYS】 , 【USROPT】 , 【TLIM】 , 【NET SET】 , 【MODPWD】 and 【CHGUSR】 .

### 1.11.1 System



Pressing 【NC. SYS】, it will show the page on the screen, you can use 【CHGUSR】button to change parameter level. Moving the highlighter to the wanted modify parameter. The message hint section will have parameter description. Key-in the wanted modify parameter value into the column and then press <INPUT>, it will show a password dialogue on the screen, and then press default password <0000>, and you can change the parameter. You only need to input password one time and then you don't need to input password anymore.

According to its effective time, there are 4 types of system parameter. It will be marked in front of the parameter. :

1. ⚡ (Effective after re-power)
2. R (Effective after RESET)
3. ⌚ (Effective after rebooting)
4. Empty (Effective immediately)

User must fully understand each parameter's definition before modifying any parameter. Two types of user status: end-user and machine maker. Different user status will have different sub-function keys.

**As 「machine maker」**

Press 【NC. SYS】 and go to parameter setting page, there are several sub-function keys, including 【SRVO】、【MAC】、【SPDL】、【MPG】、【COMP】、【ZRN】、【OP】 and 【CHGUSR】

As 「end-user」

Press 【NC. SYS】 and go to parameter setting page, there are 4 sub-functions files, including 【servo】、【SPDL】、【ZRN】、【OP】、【CHGUSR】, End-Users has relatively few functions.

### ● 【Change password】

This function for you to change parameter functions, press 【CHGUSR】, follow the step to change password.

**Note** : Exit system password is not here.

「Machine maker」

If you press 【CHGUSR】 under Machine maker's status, the password you changed will be under Machine Maker. :

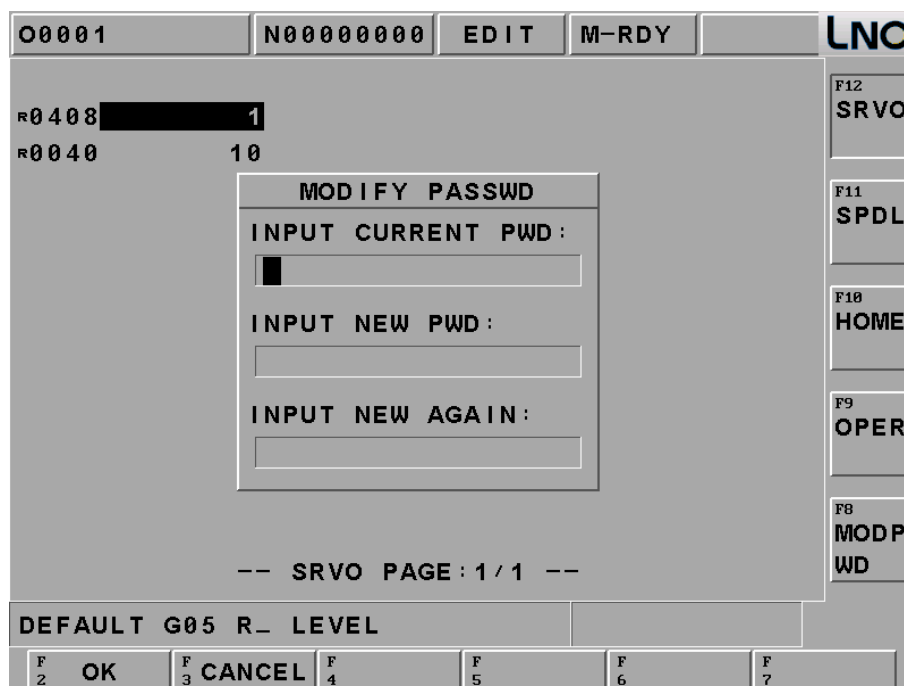
- 1.From End-User to enter in Machine Maker's password.
- 2.Under Machine Maker Status to change parameter's password. :

「End-user」

If you press 【CHGUSR】 under End-User's status, the password you changed will be under End-User.

- 1.Input current End-User's password.
- 2.Input new End-User's password
- 3.Input again the new password

Press 【OK】 to finish.



### 1.11.2 USROPT

Press **【USROPT】** and enter to **【PARAM 1】** page, there are 40 sets of parameters. The way to change the parameter is the same to system parameter, please refer to last chapter.

O9998	N000000	EDIT	M-RDY	LNC
001	0	PROGRAM PROTECT		F12 PARAM 1
002	0	ZRN RQUIRE		F11 PARAM 2
003	2	HOME 1ST		F10
004	0	RPD 50%		F9
005	24	TURRENT TOOL NO		F8
006	0	DR INTRLCK		
007	0	LUB. MTHD		
008	5	LUB. ON TIMER		
009	30	LUB. OFF TIMER		
010	0	POWER OFF		
-- PAGE : 1 / 4 --				
D051 = 0 : EDITABLE 1 : PROTECT				
F2 NC·SYS	F3 USROPT	F4 TLIM	F5 NET SET	F6 F7

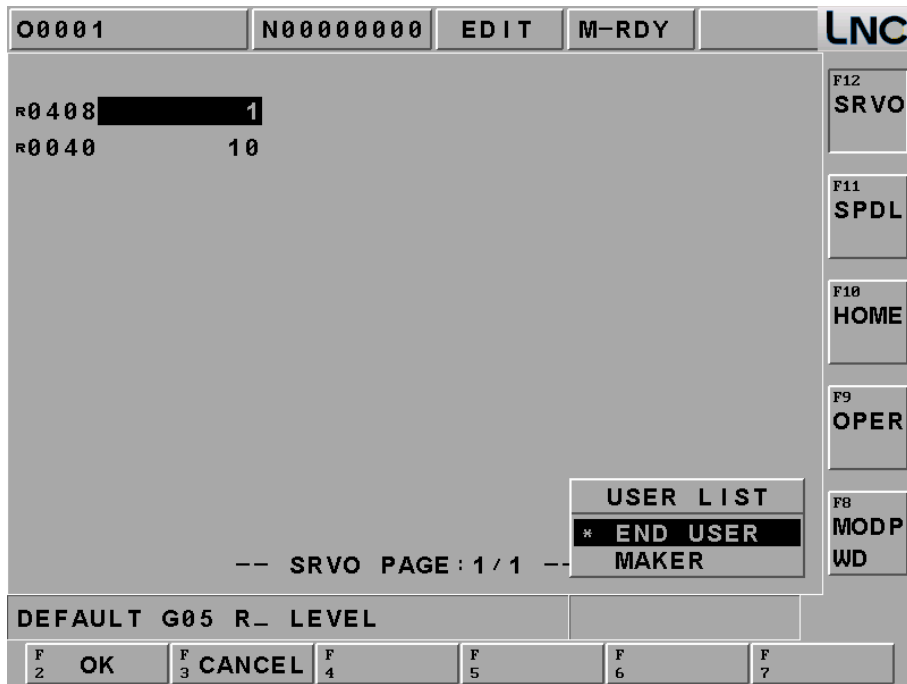
Press **【PARAM 2】** page, there are 40 sets of parameters. The way to change the parameter is the same to system parameter, please refer to last chapter.

O9998	N000000	EDIT	M-RDY	LNC
001	0	N/A		F12 PARAM 1
002	0	N/A		F11 PARAM 2
003	0	N/A		F10
004	0	N/A		F9
005	0	N/A		F8
006	0	N/A		
007	0	N/A		
008	0	N/A		
009	0	N/A		
010	0	N/A		
-- PAGE : 1 / 4 --				
R000 =				
F2 NC·SYS	F3 USROPT	F4 TLIM	F5 NET SE	F6 F7

### 1.11.3 Switch user

Press **【Switch user】**, the one with 「\*」 is your current identity.

This function is for you to switch identity. From end-user to machine maker, you will need to input password.



### 1.11.4 TLIM

Press **【TLIM】** and enter to this page.

This function can let machine makers or distributor to trace or give limit time to customers which have installments. Machine makers can lock or unlock this area to control.



O9998	N000000	EDIT	M-RDY	LNC
DATE : 2009/07/28  TIME : 10:56:39  DEADLINE 2009/07/28  REG VER:FF				F12 ACT  F11 PWD.  F10  F9  F8
EXECUTE REGISTER				
F2 NC·SYS	F3 USROPT	F4 TLIM	F5 NET SET	F6 F7

■ How the system judge the due day and the condition when the system due

System will check the due date when the system power on, the checking way is as below :

1. If system' date is more then the due date.
2. Shipping date from factory + users' cumulative date ( note ) > the due date.

( Note ) Because this system allows users to change the setting of date, therefore cumulative time is needed, the system will add hours and minutes to become dates.

Any of these two conditions above is effective, system will judge it as due, at this time, ( CYCLE START ) function will not be effective. If you press ( CYCLE START ) after due date, the system will show ALARM message as below :

【OP 1017 system has expired, please contact your machines makers.】

## 1.11.5 Net

Press **【NET SET】** to enter this page, the page will show two ways for you, such as **【NET】** and **【ReCON】**。

The operation method is as below :

Change to <PARAM>, press **【CHGUSR】** and change to machine maker level and input P651 and press INPUT, system will search P651 and set this parameter to be as below( 0 means NET method ; 1 means ReCON method ), after setting, please reboot the system.

I. The connection of **【NET】** and the PC setting :

**【SAVE】** : Follow below line to complete and press save to save setting.

**【RECONNECT】** : After **【SAVE】** , press EMG( emergency stop button) and pres **【ReCONNECT】** , the system will reconnect again.

O5678	N00000000	JOG	M-RDY	LNC																																												
NC NAME : LNCM515 USERNAME : LNCDOS GROUP : WORKGROUP				F12 SAVE																																												
<table border="1"> <thead> <tr> <th>STATUS</th> <th>PC NAME</th> <th>SHARE DIR</th> <th>PWD .</th> </tr> </thead> <tbody> <tr><td>G ) X</td><td></td><td></td><td></td></tr> <tr><td>H ) X</td><td></td><td></td><td></td></tr> <tr><td>I ) X</td><td></td><td></td><td></td></tr> <tr><td>J ) X</td><td></td><td></td><td></td></tr> <tr><td>K ) X</td><td></td><td></td><td></td></tr> <tr><td>L ) X</td><td></td><td></td><td></td></tr> <tr><td>M ) X</td><td></td><td></td><td></td></tr> <tr><td>N ) X</td><td>DEREK</td><td>CNC</td><td>*****</td></tr> <tr><td>O ) X</td><td></td><td></td><td></td></tr> <tr><td>P ) X</td><td></td><td></td><td></td></tr> </tbody> </table>				STATUS	PC NAME	SHARE DIR	PWD .	G ) X				H ) X				I ) X				J ) X				K ) X				L ) X				M ) X				N ) X	DEREK	CNC	*****	O ) X				P ) X				F11 RECON F10 F9 F8
STATUS	PC NAME	SHARE DIR	PWD .																																													
G ) X																																																
H ) X																																																
I ) X																																																
J ) X																																																
K ) X																																																
L ) X																																																
M ) X																																																
N ) X	DEREK	CNC	*****																																													
O ) X																																																
P ) X																																																
F2 NC·SYS	F3 USROPT	F4 TLIM	F5 NET SET	F6																																												

Definition as below :

**Controller Name** : This is the controller's name in the net, different controllers will have different names, and the name can not be the same with the PC's. Its default name will be LNCDOS, therefore if there are many controllers in the same Net area, you will need to modify this section.

**User Name** : This is the name for controller to log in PC, except the name of Window98, at the PC side, you need to add the same user name. If you use Guest to log in, usually the default name will be LNCDOS. Since one PC can let one user to log in several times, even if you have a lot of controllers in the same NET, you still don't have to change this column here.

**Group Name** : Controller can choose a group name to join in. This name will depend on the NET where user's controller is.

**Disk Code** : After connecting, the disk's code, if connection success, it will show ☉, if fails it will show X.

**PC Name** : The PC name that controller is going to connect with.

**Share File** : The share files on the PC

**Password** : This is the password for controller to log into PC. Also means PC end LNCDOS user's password.

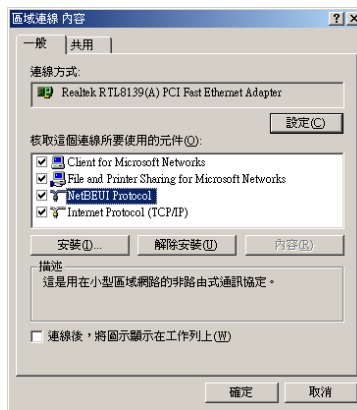
### PC end NET Connection Setting and Steps.

A. Confirm the network card, cables and software are ready to use ; Make sure if Network setting has [NetBEUI Protocol] and [File and printer sharing for Microsoft Networks] service, if not, please install this protocol firstly.

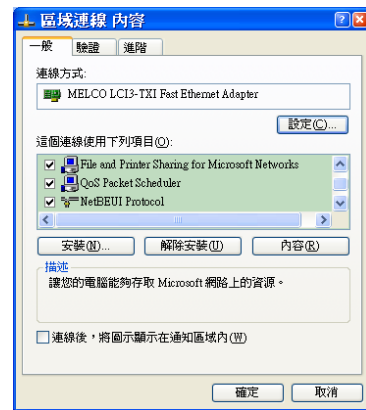
**Note** : Nwlink NetBIOS, this protocol can't be drove, otherwise the connection will fail.



WINDOWS-98

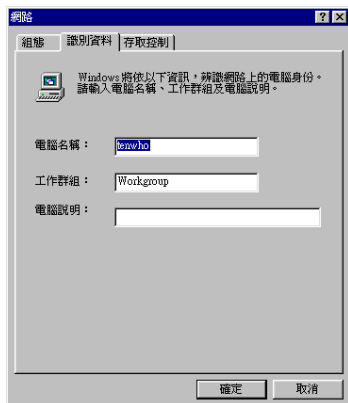


WINDOWS-2000



WINDOWS-XP

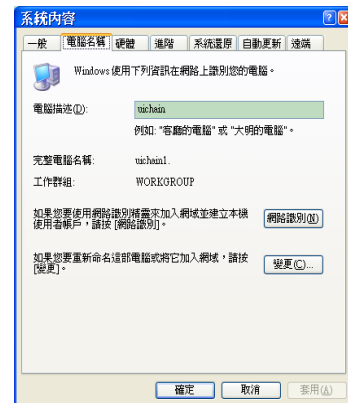
B. Confirm [PC name] and [Work Group] is correct setting. ( Controller side's PC name and work group name should be the same, and vice versa )



WINDOWS-98



WINDOWS-2000



WINDOWS-XP

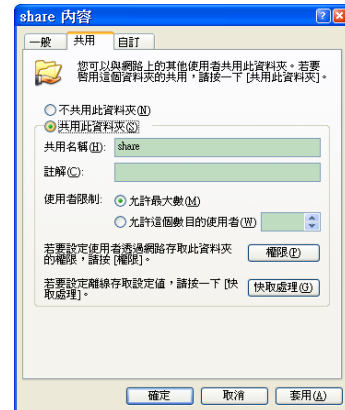
C.To share PC end's file, and make sure the share name is the same with controller side.



WINDOWS-98



WINDOWS-2000



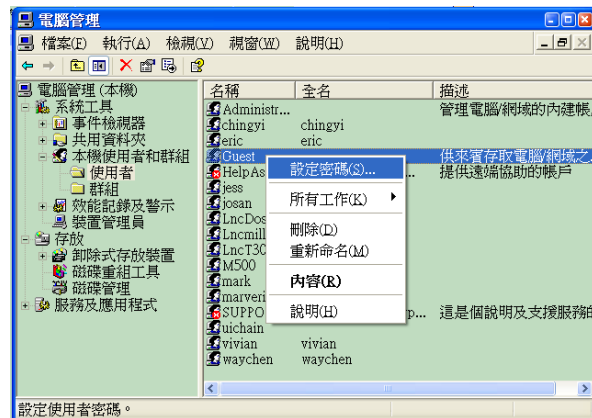
WINDOWS-XP

D. Add new PC end users, make sure if users are the same with controller side.

(Window98 : If you have files to share, no need to add new users. )



WINDOWS-2000



WINDOWS-XP

## II. Use 【ReCON】 to connect and the setting of PC end :

O9998	N000000	HOME	M-RDY	LNC																		
IP : <b>172</b> . 23 . 165 . 204 NETMASK : 255 . 255 . 255 . 0 GATEWAY : 172 . 23 . 165 . 254																						
<table border="1"> <thead> <tr> <th>NUM</th> <th>LEV</th> <th>IP</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>R/W</td> <td>192 . 168 . 1 . 100</td> </tr> <tr> <td>2</td> <td>R/W</td> <td>172 . 23 . 165 . 205</td> </tr> <tr> <td>3</td> <td>R/W</td> <td>172 . 23 . 165 . 46</td> </tr> <tr> <td>4</td> <td>R/W</td> <td>0 . 0 . 0 . 0</td> </tr> <tr> <td>5</td> <td>R/W</td> <td>0 . 0 . 0 . 0</td> </tr> </tbody> </table>					NUM	LEV	IP	1	R/W	192 . 168 . 1 . 100	2	R/W	172 . 23 . 165 . 205	3	R/W	172 . 23 . 165 . 46	4	R/W	0 . 0 . 0 . 0	5	R/W	0 . 0 . 0 . 0
NUM	LEV	IP																				
1	R/W	192 . 168 . 1 . 100																				
2	R/W	172 . 23 . 165 . 205																				
3	R/W	172 . 23 . 165 . 46																				
4	R/W	0 . 0 . 0 . 0																				
5	R/W	0 . 0 . 0 . 0																				
CONNECT 172 . 23 . 165 . 46 (SF) 172 . 23 . 165 . 46 (MT) REQUEST																						
1 : R 2 : R/W 3 : EXCLUDE																						
NC . SYS	USROPT	TLIM	NET SET																			

Use ↑↓←→ to move direction buttons to move to wanted line and input data into, press INPUT and you will see data was shown on the screen.

Details are as below :

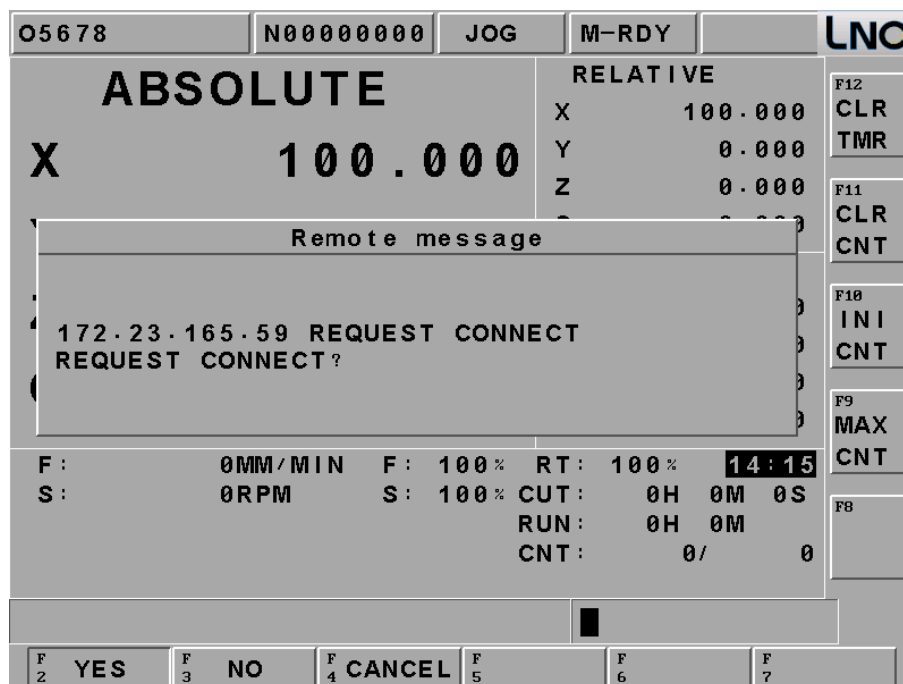
- 1. IP** : This address can not be repeated, otherwise the net will not be connected.
- 2. NET MASK** : Please set this gnet mask the same with PC end.
- 3. GATEWAY** : Please set this gateway the same with PC end.

After setting, shut down and reboot system, the NET setting is complete. You can wait for PC end to process **ReCON** software to connect.

## Controller and ReCON Connection

### a. Controller and ReCON software first connection

When PC end's ReCON software to connect with controller for the first time, controller will pop out below dialogue, it will show IP address and ask if you want to connect to controller.



Press **【Y】**: Means you allow this IP's PC to connect with controller from now on. At this time, connecting PC's IP will show on controller.

Press **【N】**: Means you are not allowed this IP's PC to connect with controller, next time controller will not accept any request from this IP.

Press **【CANCEL】**: Means you cancel this time's connection from PC end. Next time, when PC end wants to connect, it will show the same dialogue to ask your permission.

### b. Change connecting PC's permissions.

This is the function for controller to define the permission of PC end. Move the cursor to this area and set value from 1~3 and press INPUT, then you can change the permission. Setting value detail as below :

- 1 : Set to be 1-only reading, connecting PC can only read, but can not write.
- 2 : Set to be 2-reading and writing are all ok, connecting PC can read and write.
- 3 : Set to be 3-means rejection. This IP's PC is not able to connect with controller.

### c. Cancel connecting PC's IP address

If you want to cancel the Ip address, you can move cursor to this are and press INPUT to make the data to be 0. Next time when you connect to PC, it will show a dialogue again to request you fill up the IP address.

#### d. Check current connecting controller

you can check at 『current connection』 area to see if there is any PC connected to controller right now, if there is IP address on it, it means right now has connecting PC. But if not, it means there is no connection. As for **ReCON** connection detail, please refer to RECON relating setting chapters.

#### 1.11.6 Servo parameter

**Note :** Identity need to be 「Machine maker」

Digital controller is with the high speed communication servo system, you can use 【servo parameter】 to check and adjust all servo parameters on controller :

NO	X	Y	Z	1 / 3
+000	0	0	0	0
+001	0	0	0	0
+002	100	100	100	100
+006	2	2	2	2
+007	0	0	0	0
+008	4000	4000	4000	4000
+009	10	10	10	10
+00B	100	100	0	0
+00C	0	0	0	0
100	400	400	400	400
101	2000	2000	2000	2000
102	400	400	400	400
103	100	100	100	100
104	400	400	400	400
105	2000	2000	2000	2000
106	400	400	400	400

1 / 131

ITEM

F2 NC-SYS F3 USROPT F4 SERVO F5 AIC-II F6 CHGUSR F7 PGDN

According to valid time, there are 2 types of parameter :

↳ (valid after reboot machine power) , we use red mark to put before parameter numbers.

Blank (valid immediately)

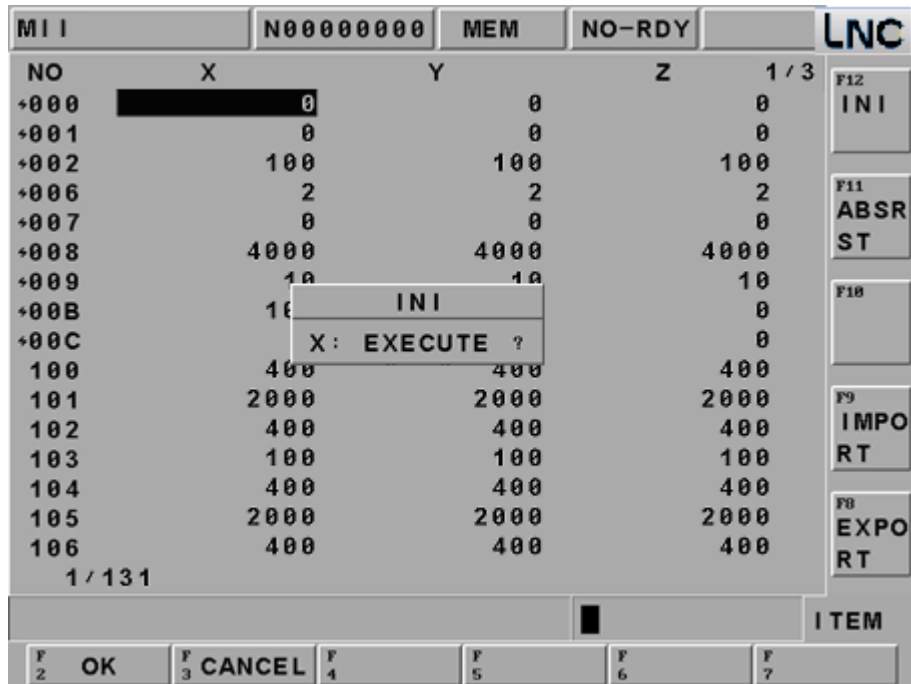
Because the servo parameter is the base of servo drive, you will need to be much more careful on parameter adjusting and changing. You will need to read the servo manual first and then change this parameter.

Servo parameter input method :

Move line bar to the parameter that you want to change, input value directly and ok, but if there is a H number at the value, you will need to input value with H, otherwise this modification will fail.

- 【Initial parameter】

Under Not ready, move line bar to X axis, press initial parameter, it will return to default value. This setting method is the same to other parameter. After initiating, you will need to reboot controller to see this new value.



### ● 【Reset ABS】

When servo axis is ABS, move line bar to X parameter and press reset ABS. After pressing ok, you can reset X axis.

Note : detail setting refer to M-II digital communication technical document.





## ● 【Import】

Under Not-ready, you can import the servo parameter file into controller. Import file must be controller's **.TXT** file.



## ● 【Export】

Under Not-ready, you can export the servo parameter file out of controller, you don't have to backup servo parameters in a very complicated way.



## 1.11.7 AIC-II

**Note** : Level must be 「machine maker」 and P.1932=1, you can see this page.

With digital communication configuration, controller can offer 40 sets of smart cutting function. (Material 1~Material 40). Via 【AIC-II】 this page, you can set up many different sets of combination to get the high speed/high precision functions for different material, settings as below :

O0001		N000000000		NO-RDY		LNC	
NO	ITEM 1	ITEM 2	1 / 40				
001	60	60			F12		
002	20	20					
003	0	0			F11		
004	60	60					
005	2000000	2000000			F10		
006	500000	500000					
007	800000	800000			F9		
008	400	400					
009	2000	2000			F8		
010	400	400					
011	0	0					

1 / 11

LINEAR ACC/DEC TIME				ITEM	
F2 NC·SYS	F3 USROPT	F4 SERVO	F5 AIC-II	F6 CHGUSR	F7 PGDN

Material definition :

- 001 : Cutting linear ACC/DEC time ms
- 002 : Cutting bell shape ACC/DEC time ms
- 003 : Cutting smooth ACC/DEC time ms
- 004 : Cutting ACC/DEC time ms
- 005 : Circular clamp speed um/min
- 006 : Circular clamp min. speed um/min
- 007 : Corner reference speed um/min
- 008 : Speed loop gain (0.1Hz)
- 009 : Speed loop integral time constant (0.01m)
- 010 : Position loop gain (0.1/s)
- 011 : Feedforward(%)

**Note** : If you change the material column value, you will need to go back to ( PROG ) > program check > 【select AIC】 to resign again, this parameter will be valid.

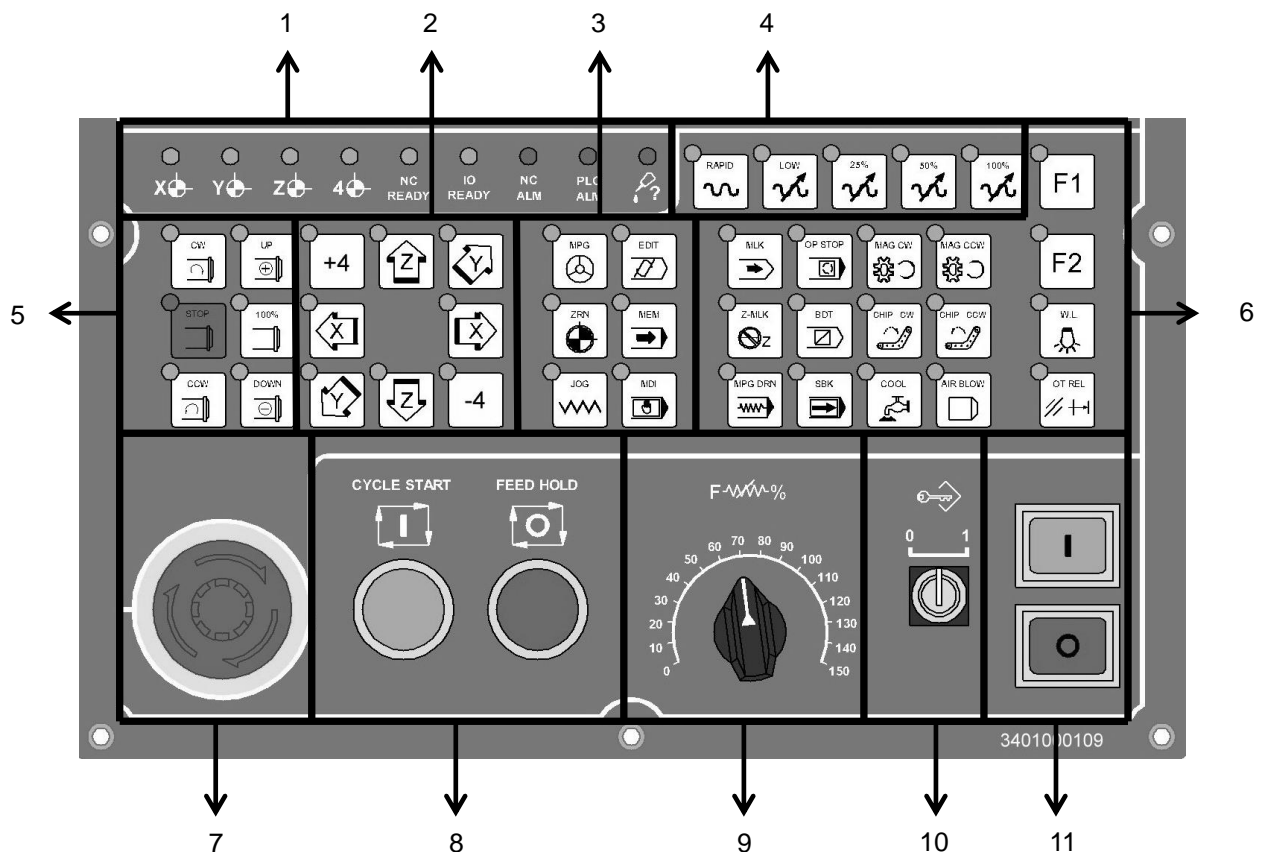
## 2 OP

Operation panel will base on machine makers' different demands to do various designs. Here only describing operation methods and functions for normal using buttons.

### 2.1 OP

There are 11 sections according to different functions :

- (1) LED SIGNAL
- (2) AXIS SELECTION
- (3) MODE SELECTION
- (4) RAPID MODE
- (5) SPINDLE&RATIO
- (6) AUXILIARY
- (7) EMERGENCY STOP
- (8) CYCLESTART & FEEDHOLD
- (9) FEEDRATE
- (10) PROGRAM LOCK
- (11) POWER ON/OFF

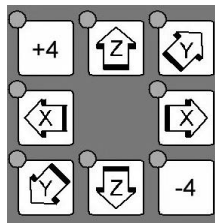


## 2.2 LED SIGNAL



- 1 : X HOME
- 2 : Y HOME
- 3 : Z HOME
- 4 : 4<sup>th</sup> HOME
- 5 : POWER ON, LED ON
- 6 : I/O MOTION CARD SELF HARDWARE TESTING, CONFIRM OK, LED ON
- 7 : ALARM/WARNING ON, LED ON
- 8 : PLC OT, LED ON
- 9 : LUB LOW, LED ON

## 2.3 AXIS SELECTION



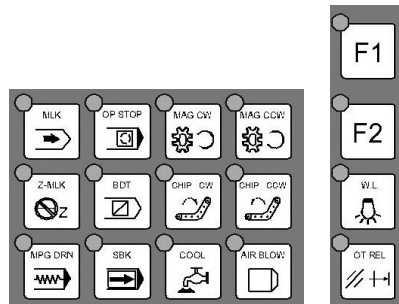
These buttons are used to command moving axis direction under JOG and ZRN modes.

For example, pressing +X key under JOG mode, it will move to +X direction, and release this key, it will stop moving. Other axis has the same functions.

Pressing +X under ZRN mode, X axis will go plus direction to return home. Other axis has the same functions.

## 2.4 Additional Function Selection

We offer several usual additional function buttons to help your operation much easier. Like OT release, or use PLC to define buttons for yourself, details are as below :



### 1. MLK

Control machine lock button, press this button to turn on, and press again to turn off. When the program is processing, controller will continue to execute program, but servo axis' moving command will not output, therefore servo axis will stay stop.

### 2. Z-MLK

Control Z Axis ignore button, press this button to turn on, and press again to turn off.

### 3. MPG DRN

Control MPG dry run button, press this button to turn on, and press again to turn off.

When system cycle start, MPG can control the operation and change the program coordinate to move with servo axis. When MPG runs faster, the program will be executed faster, but it will be faster than the program federate command. When MPG stops, the program will stop.

### 4. OP STOP

Control optional stop button, press this button to turn on, and press again to turn off.

When program was processed to M01, it will stop, but if users want to continue, users need to press CYCLE START to start again.

### 5. BDT

Control optional skip button, press this button to turn on, and press again to turn off. Those program's head with this mark "/", this line's program will be skip.

## **6. SBK**

Control single block button, press this button to turn on, and press again to turn off. When single block button is ON, program will be processed by single block, and will not have continuous actions, every single blocks need you to press CYCLE START again and again to operate.

## **7. MAG CW**

This is manual magazine CW button, under manual mode, ( here means JOG · RAPID · MPG ) , press this button ( CW LED ON ) , magazine will rotate CW, until you release this button, magazine will stay at next position. This button's situation will not maintain, in the other words, when you release this button, the function will stop. (Lights will be OFF)

## **8. MAG CCW**

This is manual magazine CCW button, movement is the same with MAG CW.

## **9. CHIP CW**

Control chip conveyor CCW button, press this button to turn on, until you release this button, the function will stop.

## **10. CHIP CCW**

Control chip conveyor CCW button, press this button to make chip conveyor to rotate CCW, until you release this button, the function will stop. This button's situation will not maintain, in the other words, when you release this button, the function will stop. (Lights will be OFF)

## **11. COOL**

Control coolant button, press this button to turn on, and press again to turn off.

## **12. AIR BLOW**

Control air blow button, press this button to turn on, and press again to turn off.

## **13. F1 、 F2**

F1 : This buttons are for machine makers to define.

F2 : This buttons are for machine makers to define.

## **14. W.L**

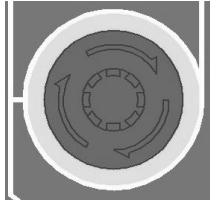
Control working lamp switch, press this button to turn on, and press again to turn off.

## 15. OT REL

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, please 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 ignore this over traveling emergency situation temporarily. This implies that users can use hand wheel or axis-direction key to move servo axis back to the travel range. Releasing (OT REL) button at this time, in order to let the system to continue travel checking. If everything is working normally, which means “CNC Ready” will replace “CNC Not Ready”, and then continue operating. If other warning messages occur, please press [RESET] button before returning back to normal. Please be careful about moving direction, and moving speed while moving the servo structure in order to prevent collusion.

**Note:** It is possible that over traveling situation occurs when “NOT Ready” condition occurs without any prediction. Please put the over traveling condition into the checking item when searching for reasons.

## 2.5 EMG-STOP

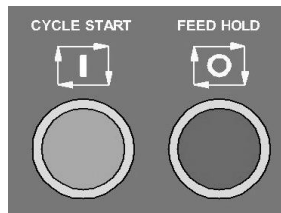


Using this button under the condition of danger or emergency and all motions will stop. To cancel is to turn the button by following the arrow direction. When the button is jump up automatically, the emergency stop is released. When the button is pressed, the system is in “Not Ready” condition (condition 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.

Note : When you press EMG-STOP, you will need to do return ZRN every time while you release it. But absolute encoder is not included in.



## 2.6 CYCLE START&FEED HOLD



### CYCLE START

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

#### a. Auto executing in MEM mode

When a program is selected in MEM mode, pressing “CYCLE START” button to execute the program. The light will remain ON during program executing time until the program execution is finished. Before executing program, the three axes must return home. If not returning back to home, 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 home.

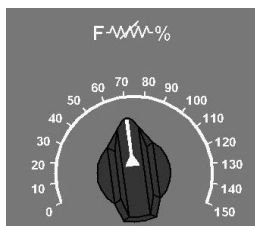
#### b. Manual executing in MDI mode

Users can key-in a block of program commanding such as G91 G01 X100. Z100 in MDI mode, then press “CYCLE START” button 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

### FEED HOLD

Press this button to stop program executing temperately. During the pause time, “FEED HOLD” light will be ON. M, S and T will remain in the current condition. Please press the “CYCLE START” button again in order to continue executing the unfinished program

## 2.7 Feedrate/Rapid Rate Adjusting Button



This is the feedrate/rapid rate adjusting button

FEEDRATE :

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

## 2.8 PROGRAM PROTECTION LOCK

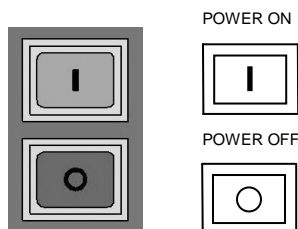


Users can use this program protection lock to lock program edit mode, after locking users can only start program processing via file management, but users can not modify this program.

0 : PROGRAM PROTECTION OFF

1 : PROGRAM PROTECTION ON

## 2.9 POWER ON/OFF



When users press power ON, it will turn on the CNC controller's power, on the contrary, when users press power OFF. System will wait for a while to release servo's electricity and shut down controller's power.

### 3 PC Keyboard

Operation panel will base on machine makers' different demands to do various designs. Here only describing operation methods and functions for normal using buttons. Buttons with green words need to use with ALT, Buttons with green words need to use with CTRL.

#### 3.1 OP

There are 9 sections according to different functions :

- (1) AXIS SELECTION
- (2) MODE SELECTION
- (3) RAPID MODE
- (4) SPINDLE&RATIO
- (5) PROGRAM AUXILIARY
- (6) AUXILIARY
- (7) CYCLESTART
- (8) FEEDHOLD
- (9) FEEDRATE



### 3.2 AXIS SELECTION

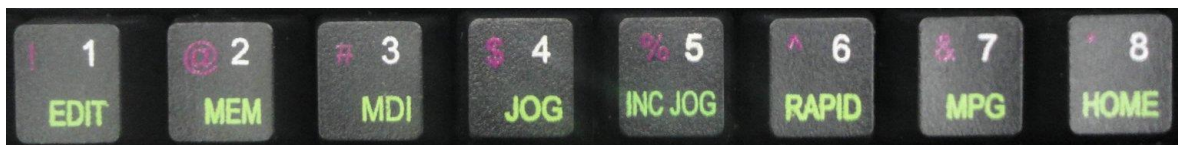


These buttons are used to command moving axis direction under JOG and ZRN modes.

For example, pressing +X key under JOG mode, it will move to +X direction, and release this key, it will stop moving. Other axis has the same functions.

Pressing +X under ZRN mode, X axis will go plus direction to return home. Other axis has the same functions.

### 3.3 MODE SELECT



There are 7 modes on this operation panel, which are: (EDIT) (MEM) (MDI) (MPG) (ZRN) (JOG) (RAPID)

Note : RAPID will be introduce at next chapter.

- (1). (EDIT)  
Users can edit new program or modify old one.
- (2). (MEM)  
Users can do auto program executing.
- (3). (MDI)  
Users can execute single block program, modify parameters and setting data.
- (4). (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 ratios, 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.



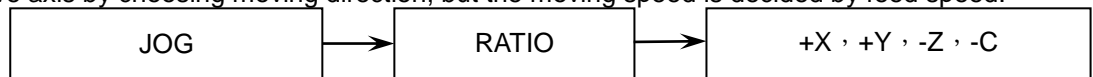
**(5). ZRN**

Operating home return of each axis. When changing to this mode, pressing home return direction button of each axis (the same button 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 point. 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 users to define machine to complete HOME RETURN motion. Please do the Home Return procedure before doing other part program every time reboot the machine in order to ensure each coordinate's accuracy.

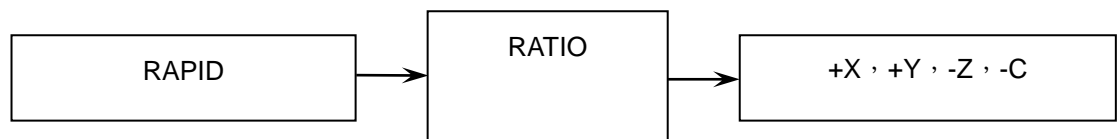
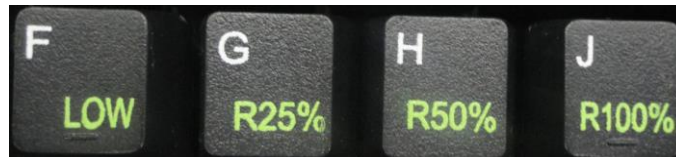
Meanwhile if customer use servo system with absolute encoder, when you return home you don't have to reach DOG, you can directly go back to home and complete this procedure.

**(6). JOG**

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

**(7). RAPID**

Under this mode, users can choose axis direction to move axis, moving speed will depend on Rapid % buttons. Under Rapid mode, there are 3 selections for you to choose, G00 when executing working program, RAPID mode and return ZRN's front section rate. Rapid % buttons has 4 type to select LOW 、25% 、50% 、100%. This speed depends on Parameter 40 to control.



### 3.4 SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH



#### Spindle Operation key

Under manual mode ( here is JOG 、 RAPID 、 MPG ) , spindle rotation can be controlled by these 3 keys.

CW : Rotate in CW direction.

STOP : Stop rotating

CCW : Rotate in CCW direction.

Under Manual mode, the rotation speed command is 0% ~ 120% at CW or CCW. If users want to switch CW and CCW, they need to do STOP first.

#### Spindle speed turn key

When spindle is under MEM or manual mode, user can use M3( or M4 ) Sxx.....to turn on spindle. Rotation speed of spindle can be tuned by UP + 、 DOWN — and 0% ~ 120%.

For example, give command of M3 S1000, turn key is at 120%, exact rotation speed is 1200PRM.

If at 10% 100PRM, press 100% key to return back to 1000PRM.

### 3.5 PROGRAM AUXILIARY

We offer several usual additional function buttons to help your operation much easier. Like OT release, or use PLC to define buttons for yourself, details are as below :



#### 1. SBK

Control single block button, press this button to turn on, and press again to turn off. When single block button is ON, program will be processed by single block, and will not have continuous actions, every single blocks need you to press CYCLE START again and again to operate.

#### 2. MST

MST, press this button to let MST ignore function ON, press again to release this function. Part program with " M " " S " " T " code will be ignored and will not be executed.

#### 3. Z-MLK

Control Z Axis ignore button, press this button to turn on, and press again to turn off.

#### 4. MPG DRN

Control MPG dry run button, press this button to turn on, and press again to turn off.

When system cycle start, MPG can control the operation and change the program coordinate to move with servo axis. When MPG runs faster, the program will be executed faster, but it will be faster than the program federate command. When MPG stops, the program will stop.

#### 5. OP STOP

6. Control optional stop button, press this button to turn on, and press again to turn off.

7. When program was processed to M01, it will stop, but if users want to continue, users need to press CYCLE START to start again.

#### 8. BDT

Control optional skip button, press this button to turn on, and press again to turn off. Those program's head with this mark "/", this line's program will be skip

#### 9. MLK

Control machine lock button, press this button to turn on, and press again to turn off. When the program is processing, controller will continue to execute program, but servo axis' moving command will not output, therefore servo axis will stay stop.

### 3.6 AUXILIARY



#### 1. MAG CW

This is manual magazine CW button, under manual mode, ( here means JOG · RAPID · MPG ) , press this button ( CW LED ON ) , magazine will rotate CW, until you release this button, magazine will stay at next position. This button's situation will not maintain, in the other words, when you release this button, the function will stop. (Lights will be OFF).

#### 2. MAG CCW

This is manual magazine CCW button, movement is the same with MAG CW.

#### 3. CHIP CW

Control chip conveyor CCW button, press this button to turn on, until you release this button, the function will stop.

#### 4. CHIP CCW

Control chip conveyor CCW button, press this button to make chip conveyor to rotate CCW, until you release this button, the function will stop. This button's situation will not maintain, in the other words, when you release this button, the function will stop. (Lights will be OFF).

#### 5. COOL

Control coolant button, press this button to turn on, and press again to turn off.

#### 6. AIR BLOW

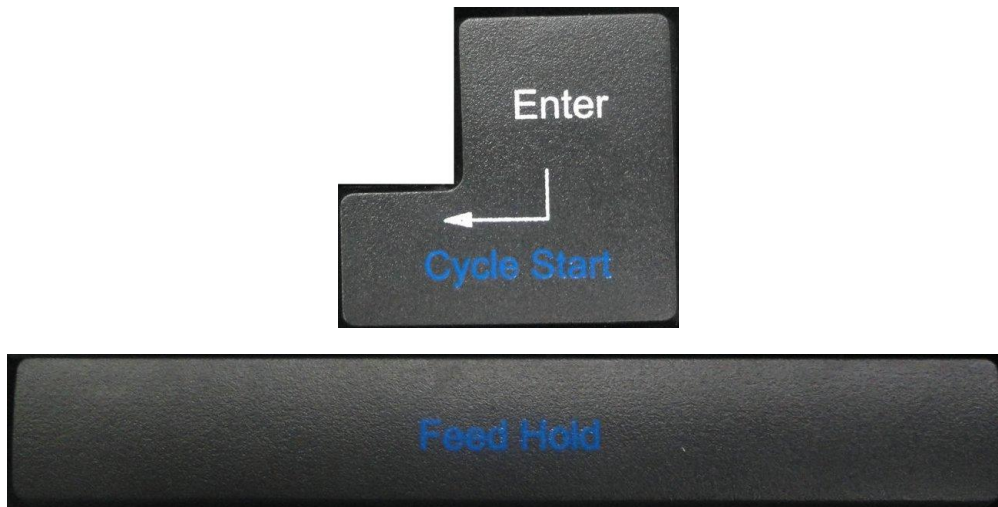
Control air blow button, press this button to turn on, and press again to turn off.

#### 7. W.L

Control working lamp switch, press this button to turn on, and press again to turn off.



### 3.7 CYCLE START & FEED HOLD



#### CYCLE START

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

##### a. Auto executing in MEM mode

When a program is selected in MEM mode, pressing “CYCLE START” button to execute the program. The light will remain ON during program executing time until the program execution is finished. Before executing program, the three axes must return home. If not returning back to home, 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 home.

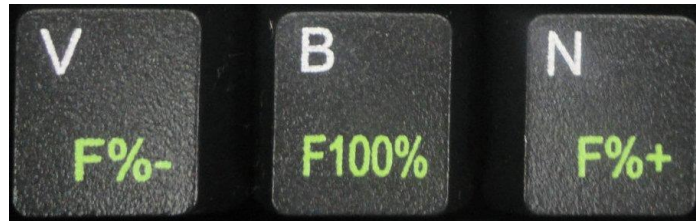
##### b. Manual executing in MDI mode

Users can key-in a block of program commanding such as G91 G01 X100. Z100 in MDI mode, then press “CYCLE START” button 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

#### FEED HOLD

Press this button to stop program executing temperately. During the pause time, “FEED HOLD” light will be ON. M, S and T will remain in the current condition. Please press the “CYCLE START” button again in order to continue executing the unfinished program

### 3.8 Feedrate



This is the feedrate/rapid rate adjusting button

FEEDRATE :

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

## 4 RS232 COMMUNICATION SOFTWARE

### 4.1 Transmission line preparation

Any type of communication will only be connected successfully when hardware and software of both controller and user's PC are under good condition. Hence, hardware function checking must be done before transmission.

#### Hardware

Confirm COM PORT : Confirm if BIOS is ON and can operate COM port (ENABLE COM PORT)

Confirm specification of Transmission line :

Usually PC side has two kinds of com ports-9 pin and 25 pin, LNC controller was fixed to use 9 pin types, so you will have two ways to follow.

1

NC(DB9PIN Jack)	PC end(DB9PIN Jack)
pin2(RD) -----	pin3(TD)
pin3(TD) -----	pin2(RD)
pin5(SG) -----	pin5(SG)

2

NC(DB9PIN Jack)	PC end(DB25PIN Jack)
pin2(RD) -----	pin3(TD)
pin3(TD) -----	pin2(RD)
pin5(SG) -----	pin7(SG)

## 4.2 PC end installation (use LNC ReCON 232 software)

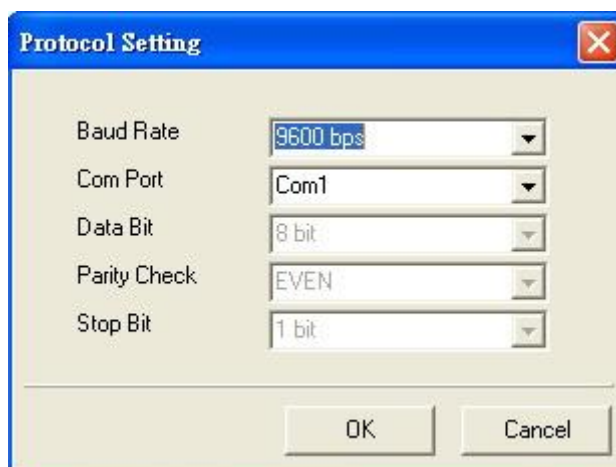
5. Clicks CD> Software\ ReCON-232 V1.25.0010, or double click setup.exe icon, the driver will start to install automatically.
6. According to Wizard's direction to follow next step or click the items to continue.
7. After installation, the desktop will have an icon of ReCON – 232, also at the start> program file.

### PC end

8. After installation, click ReCON – 232 Icon at the desktop, or press the button at start/ program file.
9. Press Start, below window will show on the screen.



10. Press Setup, below window will show on the screen.



11. Users can move the mouse cursor to the right side's table list to select transmission mode. The setting here needs to be the same with PARAM setting at the machine, otherwise, the transmission may fail.  
After communication setting, press OK, at this time, Start button will pop out. Press Start to complete PC end's preparation.



12. If you want to re-set system, you can press stop to end up transmission and press start again.
13. Press EXIT to leave software.

### 4.3 NC end

#### 【Setting】

Please edit the setting of RS232 to be 1812~1817, the setting of RS232 between two sides need to be the same and correct, otherwise transmission or DNC may fail.

No.	Name	Suggest settings (use COM1)	Remark
P1806	COM1 address(0:Disable 744~1016)	1016	
P1807	COM2 address(0:Disable 744~1016)	760	
P1808	COM1 stop code(3~7)	4	
P1809	COM2 stop code(3~7)	3	
P1810	COM1 functions (0 none 1file 2 absolute 3 axis)	1	Use COM1
P1811	COM2 functions (0 none 1file 2 absolute 3 axis)	0	Can't be the same with P1810
P1812	Transmission speed (bps)(0~4)	2	Set RS-232 transmission speed (bps) 0:2400 1:4800 2:9600 3:19200 4:38400
P1813	Transmission file bit(0:7 1:8 bit)	1	
P1814	Transmission stop bit (0:1 1:2)	0	
P1815	Transmission parity bit (0:N 1:E 2:O)	1	
P1816	Transmission mode (0: terminating machine 1:host)	0	
P1817	Transmission code (0ASCII 1EIA 2ISO)	0	

## Communication connection (RS232) - 【 DNC direct transmission functions 】

DNC function is a very useful tool for CAD/CAM system users, especially when the controller is professional one. (Not PC BASED one) Because their memories are usually very small, CAD/CAM program may not be able to upload at one time. Thus, processing while sending function plays a very important role. Also, because this special reason, it can't be compiled by controller called by program or jumped directions.

In our whole series of controllers, we equip with high amount of storage devices. We can transfer files into controller firstly and continue to processing later. (This is called inner DNC functions) It can show better performance.

However, DNC's old processing while sending function was still remained (this is called external DNC functions); it is also a very useful tool.

Steps are as below:

1. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
2. Use PROG group, change to EDIT mode.
3. Go to 【DIRMNG】 , select 《RS232》 DNC PROGRAM and press OK.



4. Select 1. LNC ReCON 232 software to upload.

O0313	N000000	EDIT	M-RDY	LNC	
<BATCH>	BATCH PROGRAM			F12 FILE	
<RS232>	DNC PROGRAM			F11 COPY	
O0000	: / * LAYSER MEA. MOVEMENT * /			F10 DEL	
O0001	: : FROM / 150 · 120 · 41			F9 REN	
O0003	DNC			F8 NEXT	
O0004	<div>1 · ReCON 232 SOFTWARE</div> <div>2 · GERNAL SOFTWARE</div>				
O0005					
O0006					
O0007					
O0008					
O0009					
O0010					
O0011					
O0012					
O0115					
O0121					
COUNT :		25	FREE :	493797376	
F2 YES	F3 CALCEL	F4	F5	F6	F7



5. Go to PC end disk and select DNC file, press **【YES】**

O0313		N000000		EDIT	M-RDY	LNC	
DNC							
[ -C- ]		O0000				F12 FILE	
[ -D- ]		O0001				F11 COPY	
[ -E- ]		O0115				F10 DEL	
[ . . ]		O0121				F9 REN	
[ . ]		O5678				F8 NEXT	
SOURCE		O5678					
CURRENT		E:\CNC\NCFILES\CAT\					
F2 YES		F3 INTO		F4 REFH		F5 CALCEL	
F6		F7					

6. Now the file will turn to be RS232 and change mode to MEM.

RS232		N000000		EDIT	M-RDY	LNC	
=== File too big, Load partial ===							
F12 FILE SAVE		F11 WORD FIND		F10 INS CYCL		F9 EDIT CYCL	
F8 NEXT							
ROW:		1 /		0 COL:		1 PROC	
PROGRAM PROTECTED, EDIT INHIBI							
F2 FGPROG		F3 DIRMNG		F4 PROCHK		F5 MDI	
F6 R-PROG		F7 COMM.					

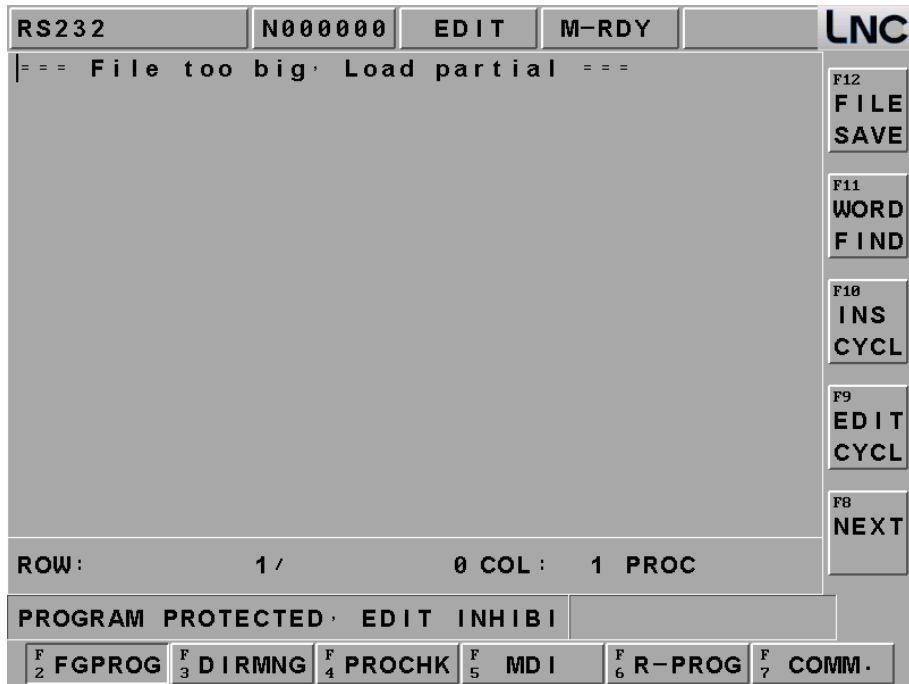
7. Press 【CYCLE START】, start to do DNC.

RS232		N000000		MEM		M-RDY		LNC	
; %								F12 CHK	
; N2 G21									
; N3 M01									
; N4 T08M06									
N5 G90G00G54X0Y0								F11 CUR	
ABSOLUTE				DIST TO GO				(G)	
X	0.000	X	0.000	G54	G01	G17			
Y	0.000	Y	0.000	G90	G23	G94			F10 NEXT
Z	0.000	Z	0.000	G21	G40	G49			
C	0.000	C	0.000	G80	G98	G50			
				G67	G64	G69			F9
				G15	G50.1				
LN:	1	FO:	0%	F	1000.000	H	0		
AF:	0	RO:	0%	R		M			F8
AS:	0	SO:	100%	P		S	0		
F0				Q		T	0		
<div style="display: flex; justify-content: space-between;"> <span>F2 BGPROG</span> <span>F3 DIRMNG</span> <span>F4 PROCHK</span> <span>F5 MDI</span> <span>F6 R-PROG</span> <span>F7 COMM.</span> </div>									

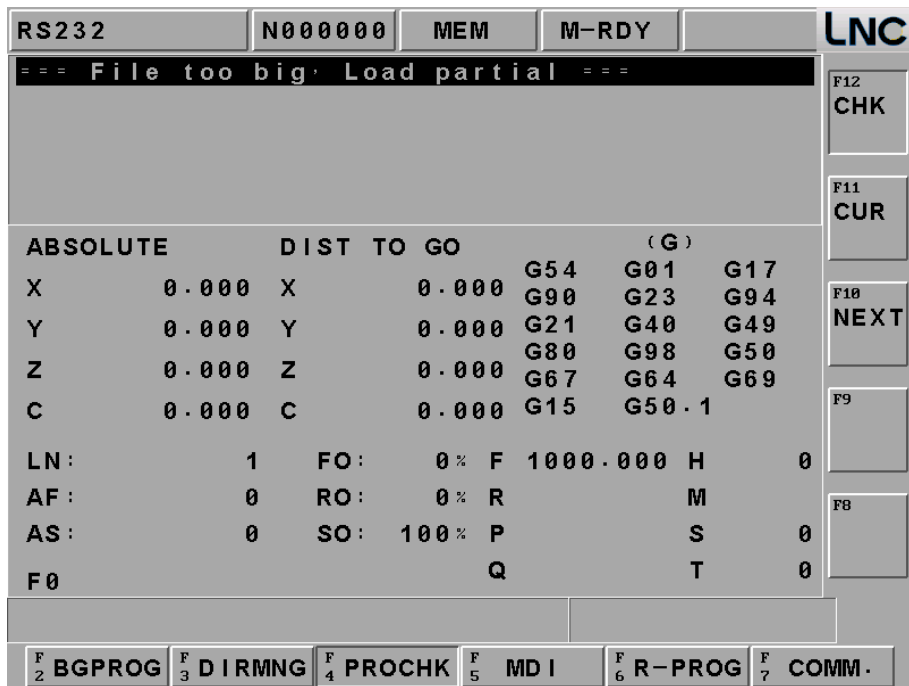
8. Select 2. General software to upload. (Use SM8 sample)

RS232		N000000		EDIT		M-RDY		LNC	
< BATCH >		BATCH PROGRAM							
< RS232 >		DNC PROGRAM							
O0000		; / * LAYSER MEA. MOVEMENT * /							
O0001		; : FROM / 150.120.41							
O0003		DNC							
O0004									
O0005									
O0006									
O0007		1. ReCON 232 SOFTWARE							
O0008									
O0009									
O0010									
O0011		2. GERNAL SOFTWARE							
O0012									
O0115									
O0121									
COUNT :		25		FREE :		494411776			
<div style="display: flex; justify-content: space-between;"> <span>F2 YES</span> <span>F3 CALCEL</span> <span>F4</span> <span>F5</span> <span>F6</span> <span>F7</span> </div>									

9. Now the file will turn to be RS232 and change mode to MEM.



10. Screen change to [PROCHK] and NC end is at stand by mode, wait for PC end to transfer files.



11. Before press Send button at PC end, please make sure NC end is at stand by>Cycle-Start mode. In other words, receiver must be prepared, so the sender can send out files, or after sending, press Cycle-Start to do DNC.

RS232		N000578		MEM		CSTART		LNC																																																									
N551 X17.752Z33.356																																																																	
N552 X17.086Z33.454																																																																	
N553 X16.009Z33.574																																																																	
N554 X15.X-33.386Z33.732																																																																	
N558 X-34.724Z33.682																																																																	
<table border="0"> <tr> <td colspan="2">ABSOLUTE</td> <td colspan="2">DIST TO GO</td> <td colspan="4">(G)</td> </tr> <tr> <td>X</td> <td>-29.047</td> <td>X</td> <td>-5.158</td> <td>G54</td> <td>G01</td> <td>G17</td> <td></td> </tr> <tr> <td>Y</td> <td>60.268</td> <td>Y</td> <td>0.000</td> <td>G90</td> <td>G23</td> <td>G94</td> <td></td> </tr> <tr> <td>Z</td> <td>33.718</td> <td>Z</td> <td>0.016</td> <td>G21</td> <td>G40</td> <td>G49</td> <td></td> </tr> <tr> <td>C</td> <td>0.000</td> <td>C</td> <td>0.000</td> <td>G80</td> <td>G98</td> <td>G50</td> <td></td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>G67</td> <td>G64</td> <td>G69</td> <td></td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>G15</td> <td>G50.1</td> <td></td> <td></td> </tr> </table>										ABSOLUTE		DIST TO GO		(G)				X	-29.047	X	-5.158	G54	G01	G17		Y	60.268	Y	0.000	G90	G23	G94		Z	33.718	Z	0.016	G21	G40	G49		C	0.000	C	0.000	G80	G98	G50						G67	G64	G69						G15	G50.1		
ABSOLUTE		DIST TO GO		(G)																																																													
X	-29.047	X	-5.158	G54	G01	G17																																																											
Y	60.268	Y	0.000	G90	G23	G94																																																											
Z	33.718	Z	0.016	G21	G40	G49																																																											
C	0.000	C	0.000	G80	G98	G50																																																											
				G67	G64	G69																																																											
				G15	G50.1																																																												
LN:	571	FO:	100%	F12	000.000	H	0																																																										
AF:	11653	RO:	100%	R		M																																																											
AS:	0	SO:	100%	P		S	0																																																										
				Q		T	0																																																										
								HSP																																																									
F2 BGPROG		F3 DIRMNG		F4 PROCHK		F5 MDI		F6 R-PROG																																																									
								F7 COMM.																																																									

### Communication connection (RS232) - 【 general software 】

At below content, about the software of PC end, we will take SMART CAM's Communication V8.02 as an example.

Note :

1. Please make sure communication setting should be the same with NC end. (Like above picture) The port setting will depend on exact connection.
2. Communication software on PC end could be "share ware— comEDITcnc" which was provided by SoftServo Software or others, as long as this software could provide RS232 functions.

PC end operation steps :

Make sure PC end COM1 is free and communication setting is the same with NC end.

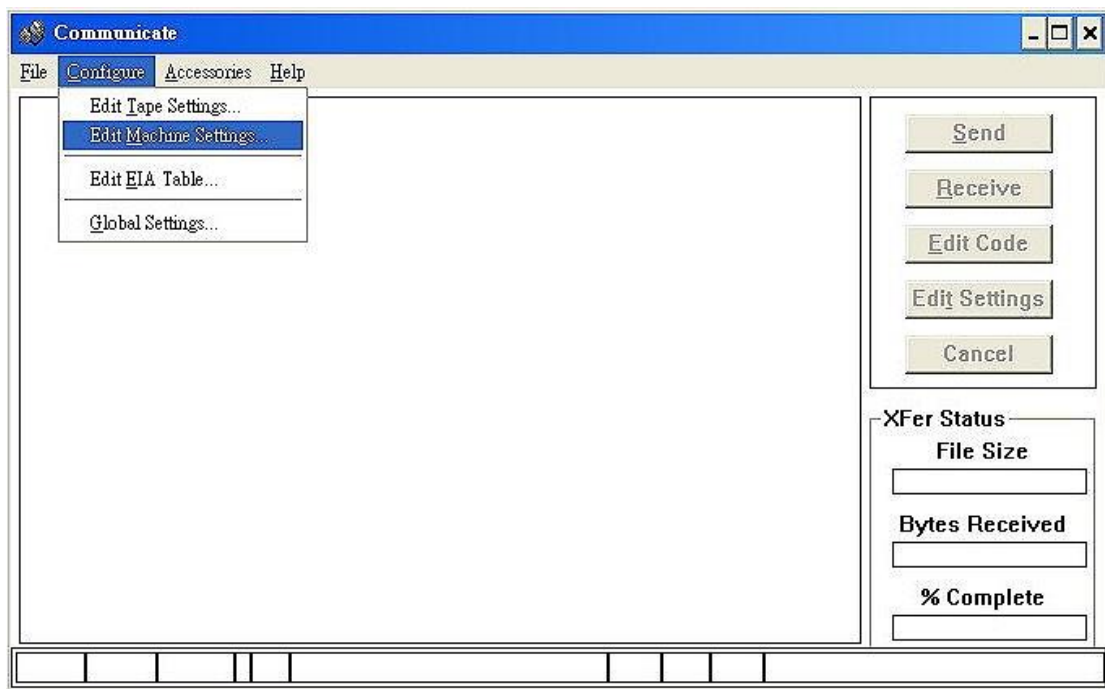
Select the file you want to transfer.

Press Send button, NC end will start to receive.

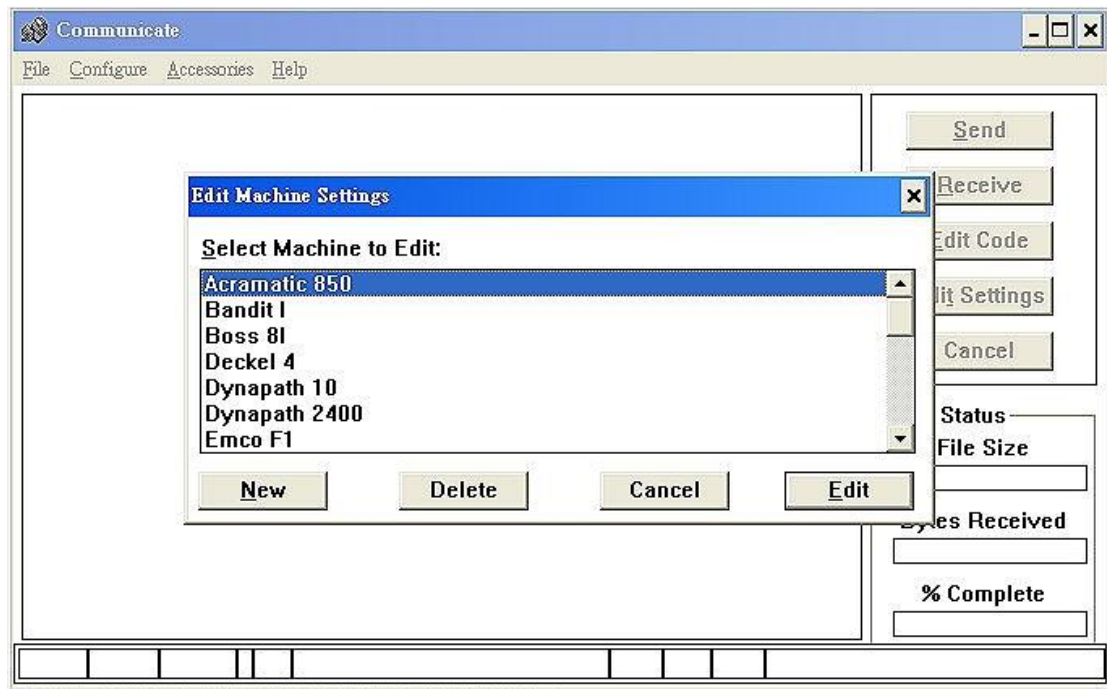
(For safety, we recommend that you make single block enable and minimize override at NC end)

When "Transfer is complete" shows on PC end, it means the transmission is complete.

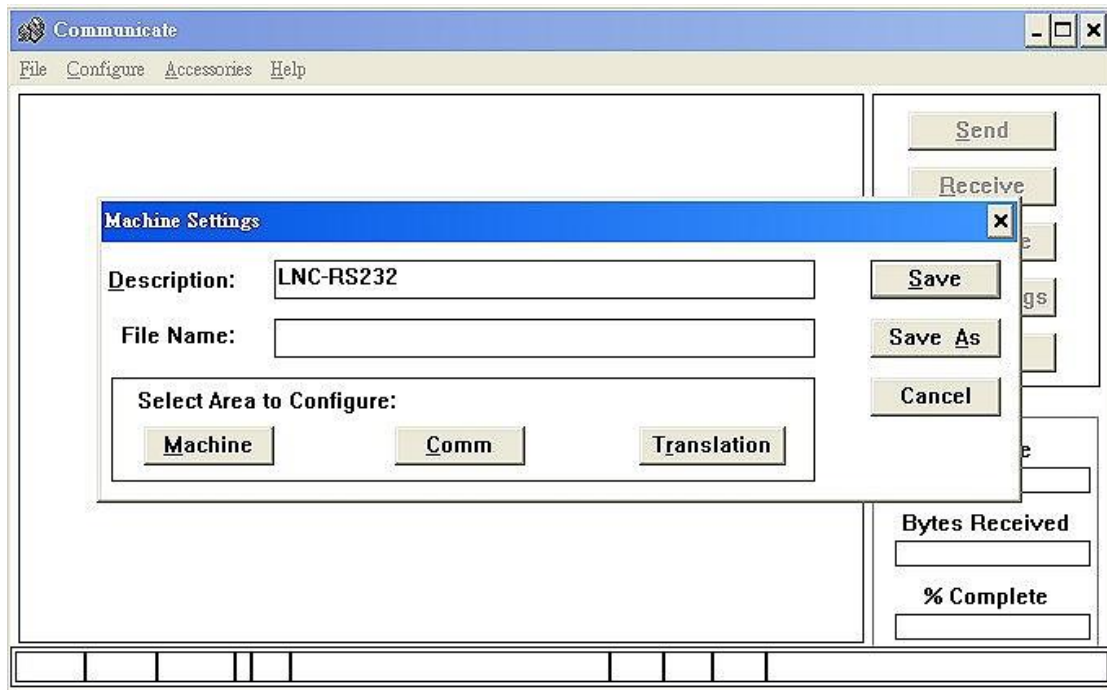
Below is the window of PC end's communication setting page. Go to [Configure]> [Edit Machine Settings] choose a correct setting to connect with LNC.



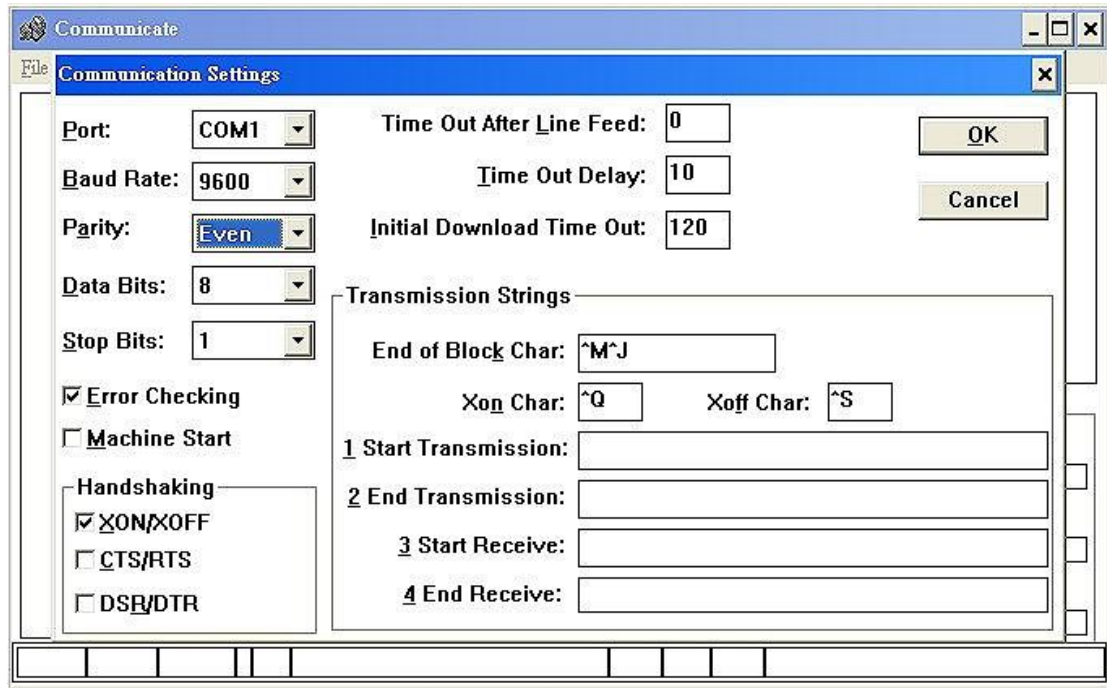
Press [NEW]



Description could be LNC-RS232 (or defined by user)



Go to [Comm]>Communication setting, other tables can be the same.



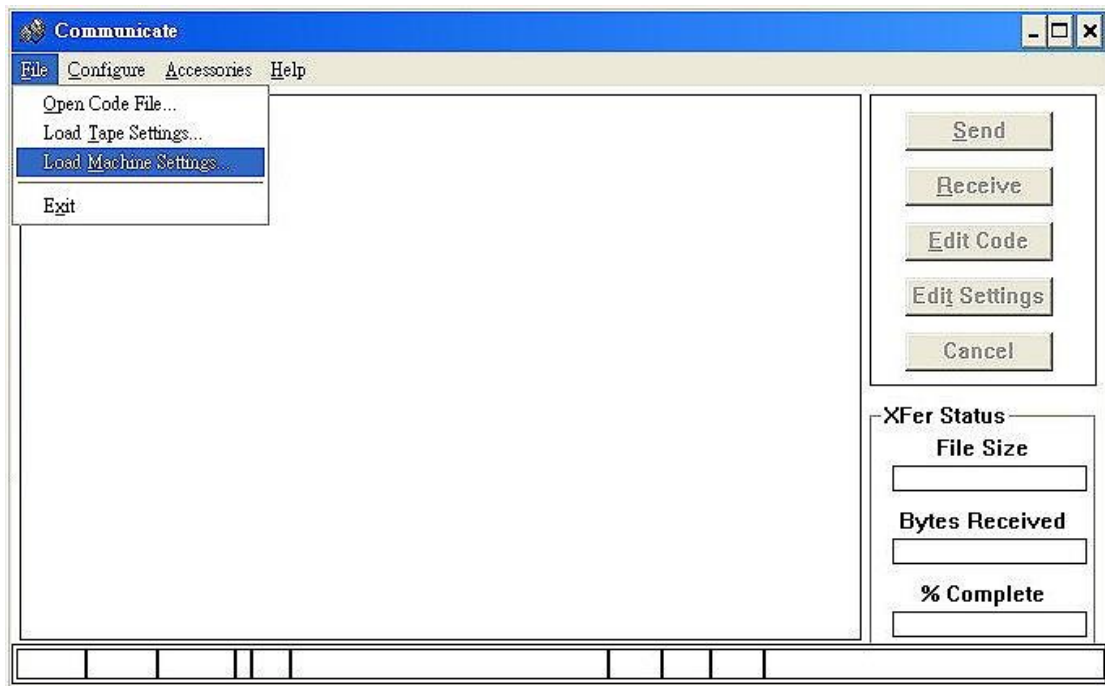
Please click XON/XOFF at HandShaking line; this is a transmission control method which conducts bits by software. If receiver wants to stop transmission from sender, receiver will send a Xon Char (ASCII No.19 bit : ^Q) to sender and stop. But if receiver wants to recover transmission, he will send a Xoff Char(ASCII No. 17 bit : ^S) to sender. This is the way to control sender.

Press [Save], save the file at SM8>C:\SM8 and name it as LNCRS232.

You only need to edit machine setting at the first time, after that all you need to do is to select the same setting.

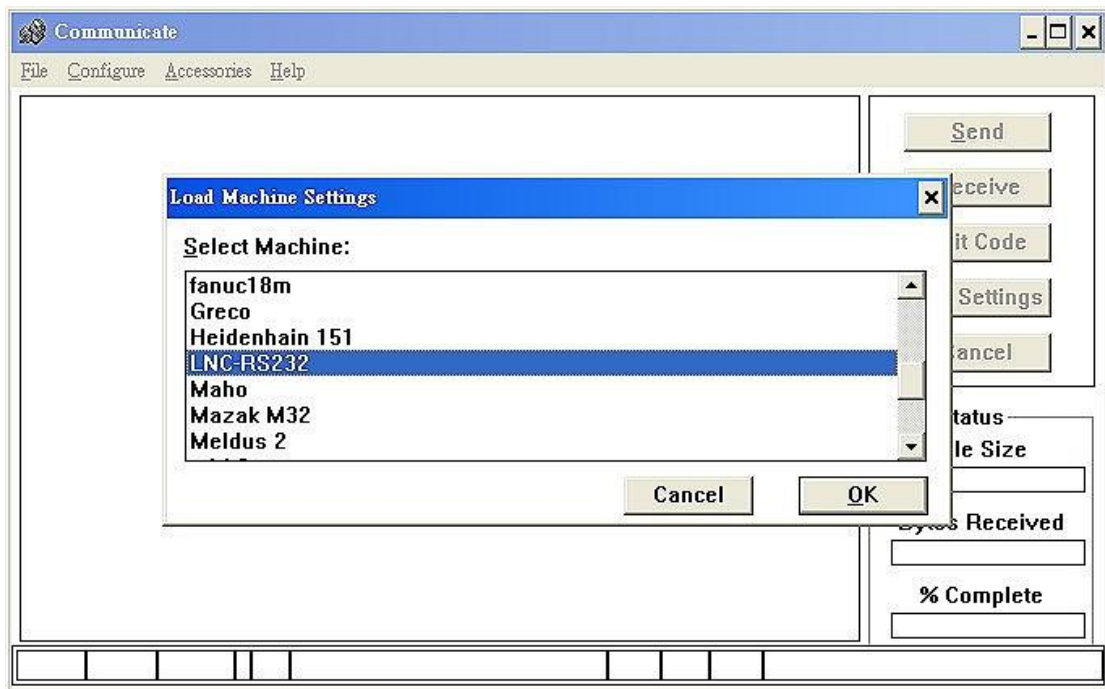


Go to [File]> [Edit Machine Settings]

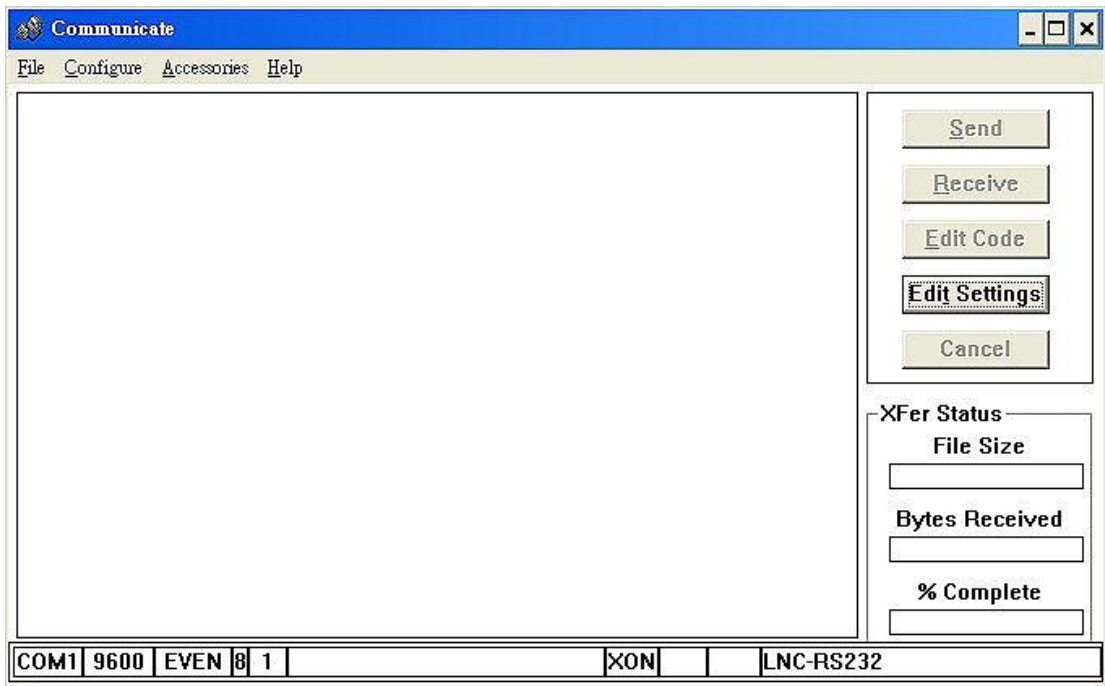




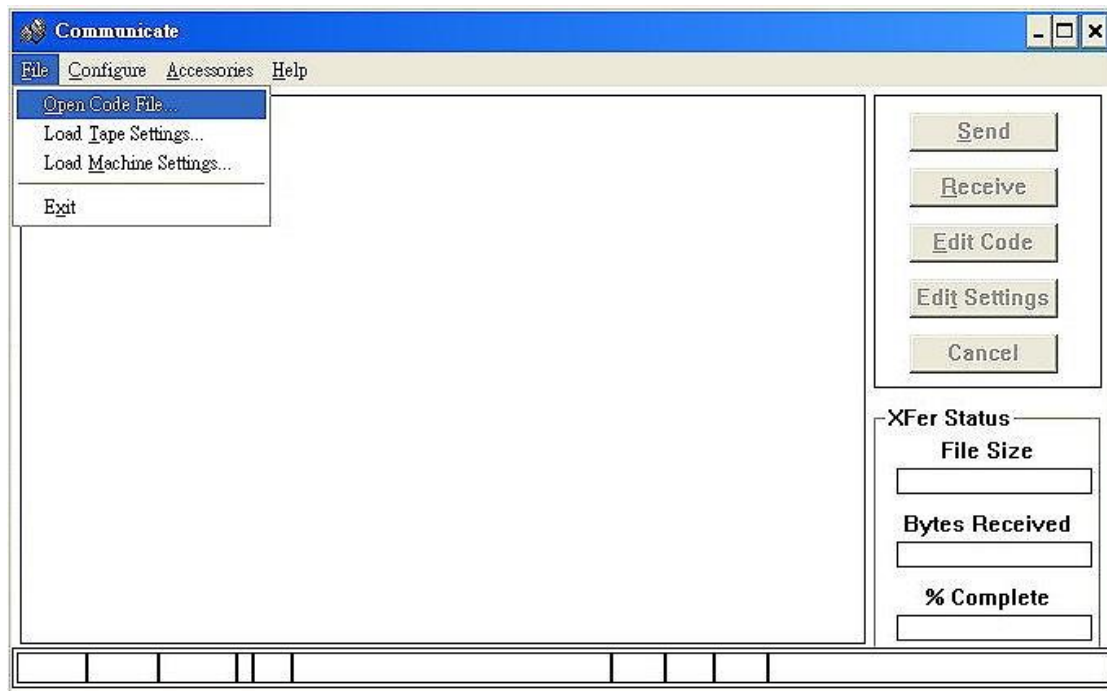
Select the setting of [LNC-RS232] that you just made.



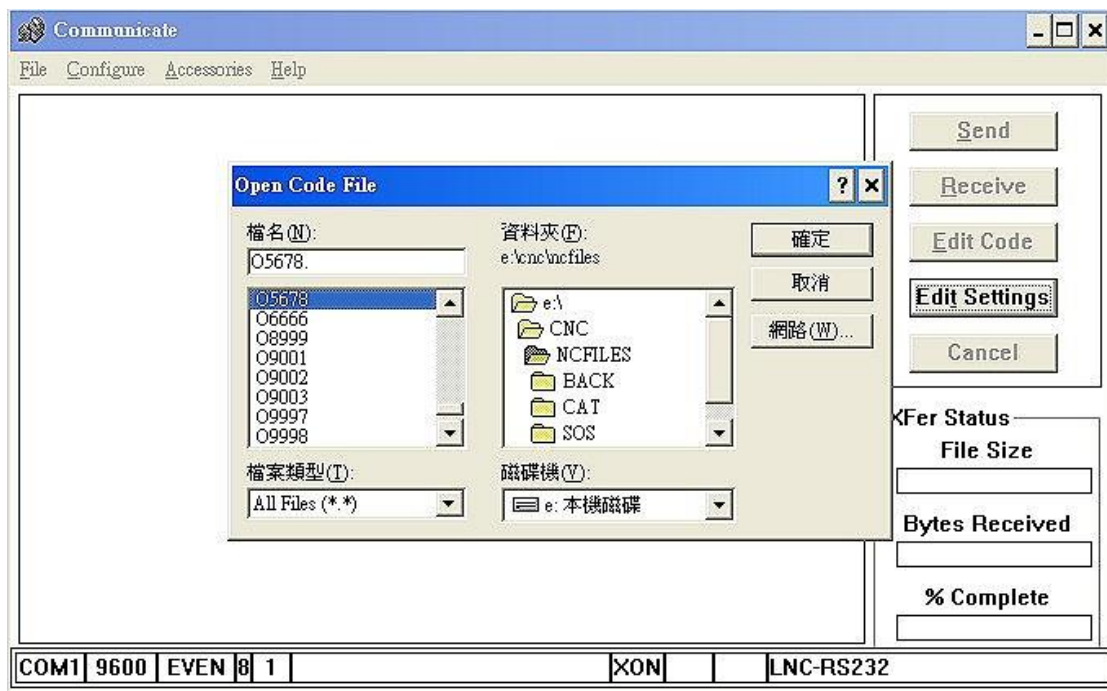
At this time, [Edit Settings] button will pop out, you will see COM1 , 9600 , EVEN , 8 , 1 at down side.



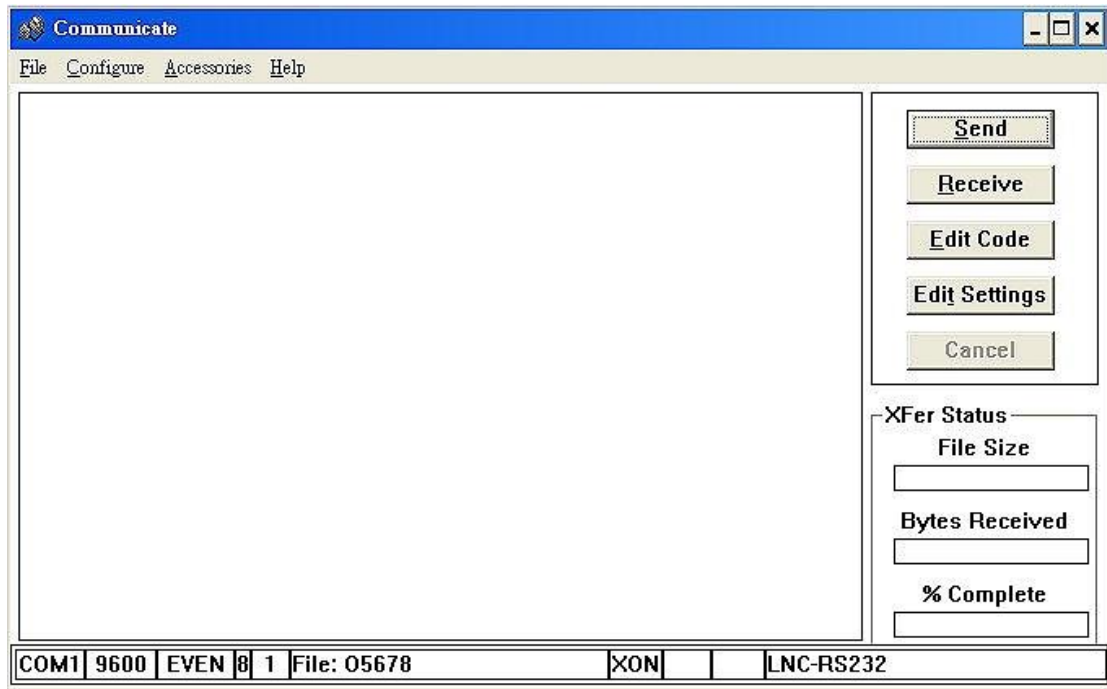
Press file and go to [Edit]> [Open Code File]



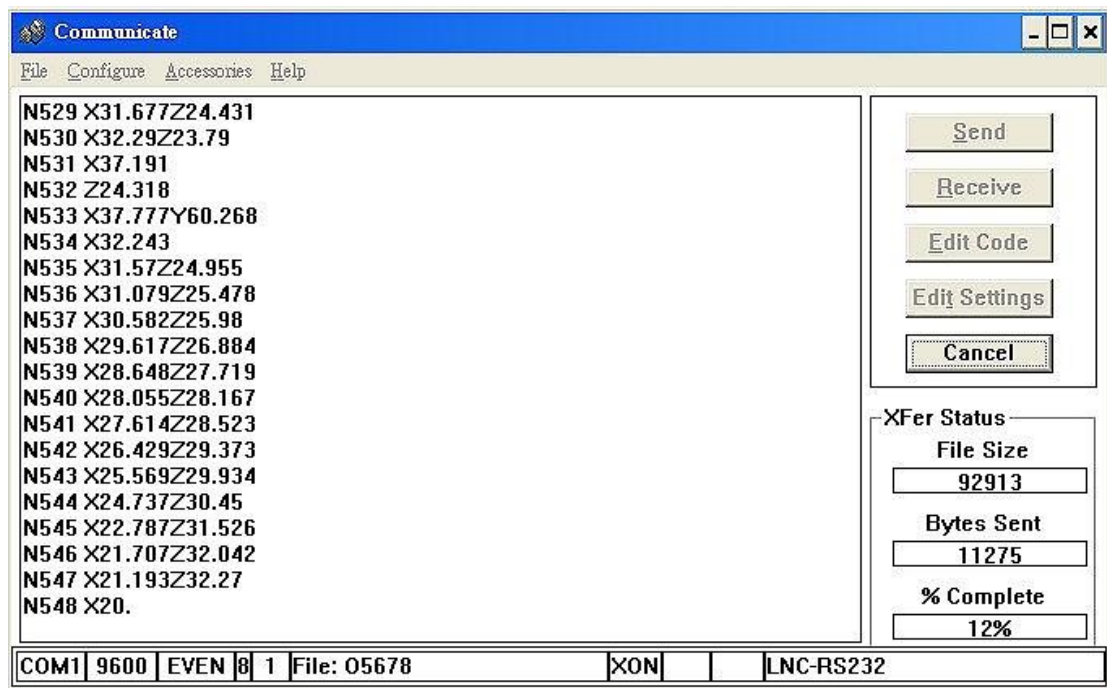
Select the file that you want to do DNC transfer and press OK.



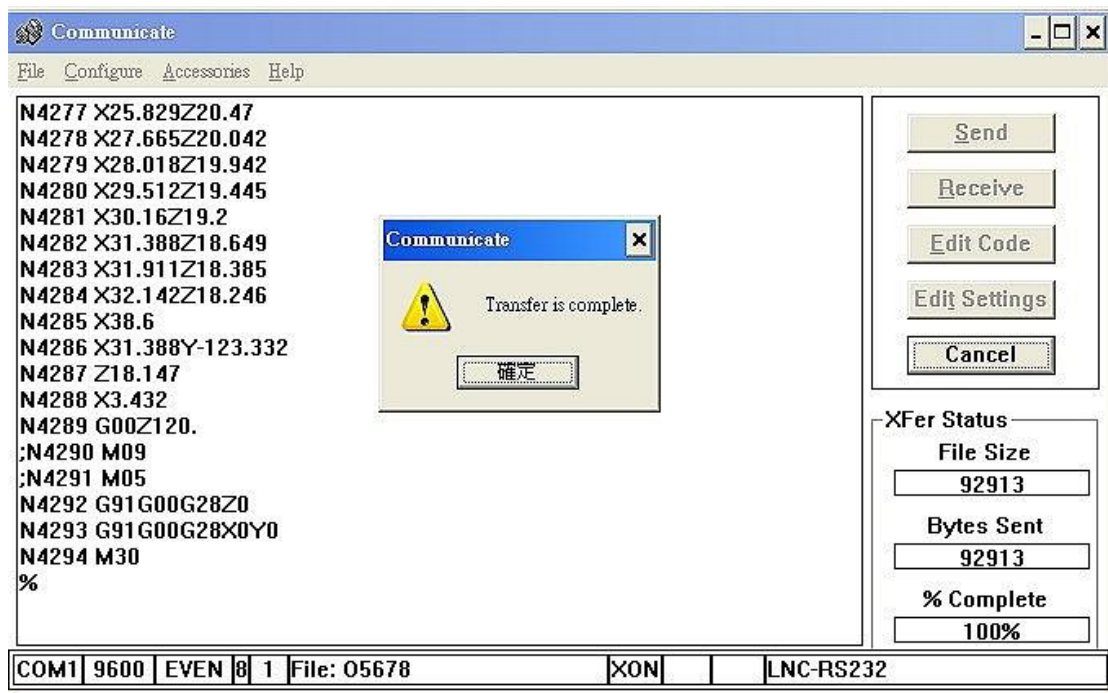
Press [Send]



Start to upload, press 【CYCLE START】 and start to do DNC.



When you see "Transfer is complete" at PC end, it means the transferring is ok.



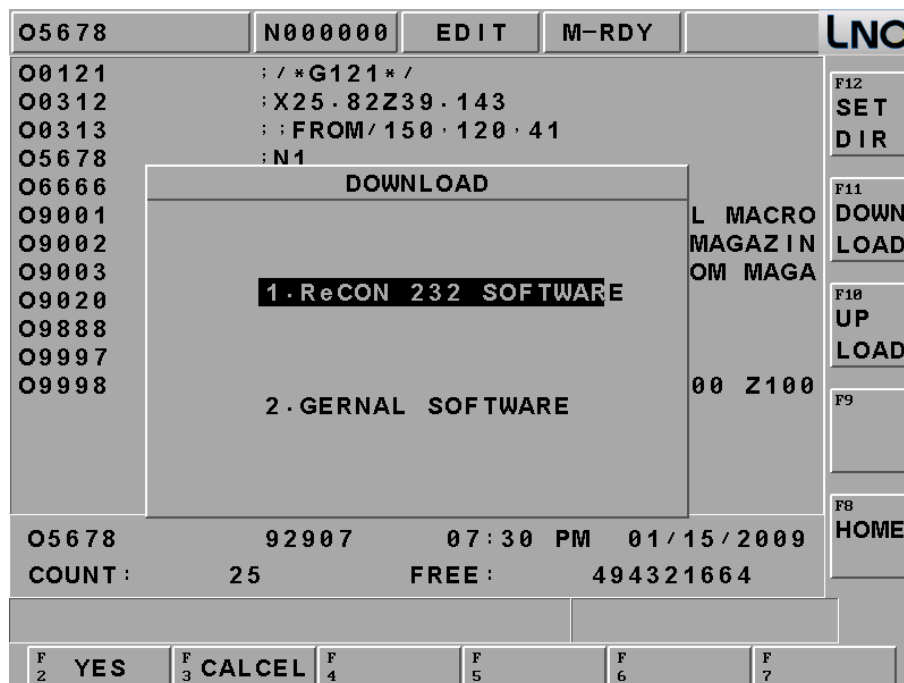
#### 4.4 Communication (RS232) - 【 DNC file upload and download functions 】

##### 4.4.1 Path Program Upload & save (using ReCON software)

This is a device that could make you transfer the file from PC end to NC end and also save the file at NC end(H.DD/Disk ON Chip/Disk ON Modular), Details as below :

Operation for NC&PC end

14. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
  15. Use PROG group, change to EDIT mode.
  16. Go to 【DIRMNG】 and choose the file that you want to transfer to PC end and press [DOWN LOAD]
- Select 1. LNC ReCON 232 software to upload.



17. Choose the file O5678 you want to download from controller and press OK

O5678	N000000	EDIT	M-RDY	LNC							
O0121	; /*G121*/			CRO ZIN AGA  100	F12 SET DIR						
O0312	; Y25 82720 442				F11 DOWN LOAD						
O0312	DOWNLOAD ( NC )				F10 UP LOAD						
O5678	FILE NAME : O5678				F9						
O6666					F8 HOME						
O9009	O0009										
O9010	O0010										
O9011	O0011										
O9012	O0012										
O9888	O0115										
O9999	O0121										
O9999	O0312										
	O0313										
	O5678										
	O6666										
O5678 92907 07:30 PM 01/15/2009											
COUNT : 25 FREE : 494321664											
<table border="1"> <tr> <td>F2 OK</td> <td>F3 SEL</td> <td>F4 USEL</td> <td>F5 ALL</td> <td>F6 UALL</td> <td>F7 CANCEL</td> </tr> </table>						F2 OK	F3 SEL	F4 USEL	F5 ALL	F6 UALL	F7 CANCEL
F2 OK	F3 SEL	F4 USEL	F5 ALL	F6 UALL	F7 CANCEL						

18. Set PC end's file path and upload file name. (You could choose new name or save at old file.

O5678	N000000	EDIT	M-RDY	LNC							
DOWNLOAD ( PC )				CRO ZIN AGA  100	F12 SET DIR						
[ -C- ]	O0000		F11 DOWN LOAD								
[ -D- ]	O0001										
[ -E- ]	O0115										
[ . . ]	O0121										
[ . ]	O5678										
TARGET		O0115			F10 UP LOAD						
CURRENT		E:\CNC\NCFILES\CAT\									
<table border="1"> <tr> <td>F2 YES</td> <td>F3 INTO</td> <td>F4 REFH</td> <td>F5 CALCEL</td> <td>F6</td> <td>F7</td> </tr> </table>						F2 YES	F3 INTO	F4 REFH	F5 CALCEL	F6	F7
F2 YES	F3 INTO	F4 REFH	F5 CALCEL	F6	F7						

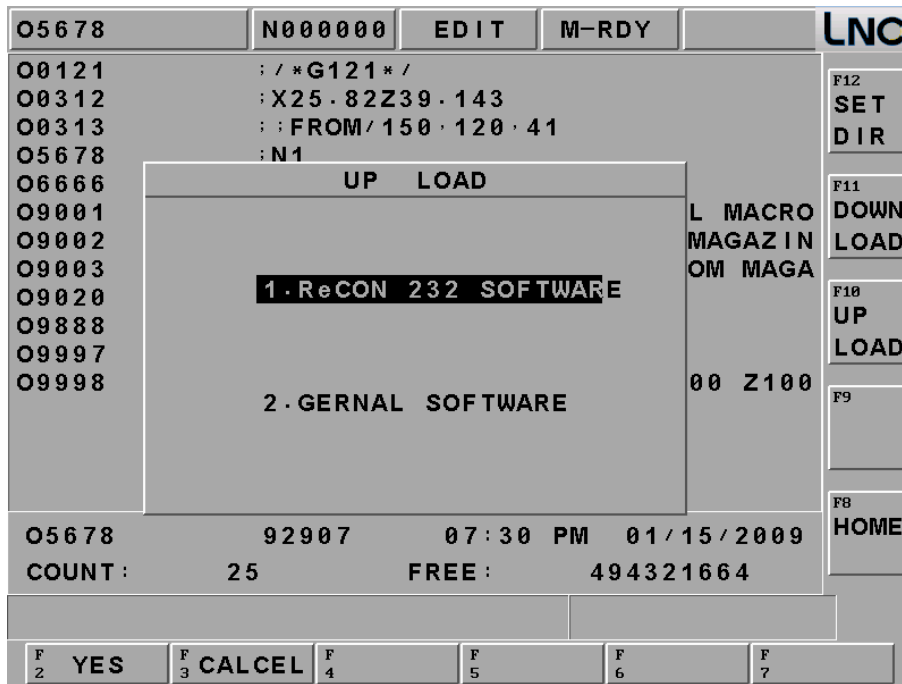
19. Check if file name is ok at NC and PC end, press OK to upload.

O5678	N000000	EDIT	M-RDY	LNC	
O0121	; / *G121* /			F12 SET DIR	
O0312	: X25.82Z39.143				
DOWNLOAD					F11 DOWN LOAD
NC WORK PATH					
SOURCE	C: \LNCMILL \NCFILES \O5678				F10 UP LOAD
PC WORK PATH					
TARGET	E: \CNC \NCFILES \CAT \O0115				F9
					F8 HOME
O5678	92907	07:30 PM	01/15/2009		
COUNT:	25	FREE:	494321664		
F2 OK	F3 CANCEL	F4	F5	F6	F7

#### 4.4.2 Path Program Upload & save (using ReCON software)

Operation Steps for NC&PC end

20. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
21. Use PROG group, change to EDIT mode.
22. Go to [DIRMNG] and choose the file that you want to transfer to PC end and press [UP LOAD].  
Select 1. LNC ReCON 232 software to upload.





23. Select save path and upload name at PC end.

O5678	N000000	EDIT	M-RDY	LNC	
UP LOAD ( PC )					
[ -C- ]	O0000	<div style="float: right;"> F12 SET DIR  F11 DOWN LOAD  F10 UP LOAD  F9  F8 HOME </div>			
[ -D- ]	O0001				
[ -E- ]	O0115				
[ . . ]	O0121				
[ . ]	O5678				
SOURCE		O0001			
CURRENT		E : \CNC\NCFILES\CAT\			
<div style="display: flex; justify-content: space-between;"> <span>F2 YES</span> <span>F3 INTO</span> <span>F4 SEL</span> <span>F5 USEL</span> <span>F6 PGDN</span> <span>F7 CALCEL</span> </div>					

24. At NC end, you could input new name or save at the same file.

O5678	N000000	EDIT	M-RDY	LNC	
O0121	; / *G121* /				
O0312	; Y25 82720 142				
O031	UP LOAD ( NC )				
O567	FILE NAME :	O0012	<div style="float: right;"> F12 SET DIR  F11 DOWN LOAD  F10 UP LOAD  F9  F8 HOME </div>		
O666	O0009				
O900	O0010				
O900	O0011				
O900	O0012				
O988	O0115				
O999	O0121				
O999	O0312				
	O0313				
	O5678				
	O6666				
O5678		92907	07:30 PM	01/15/2009	
COUNT :		25	FREE :	494321664	
<div style="display: flex; justify-content: space-between;"> <span>F2 OK</span> <span>F3 CANCEL</span> <span>F4</span> <span>F5</span> <span>F6</span> <span>F7</span> </div>					

25. Confirm and press [OK]

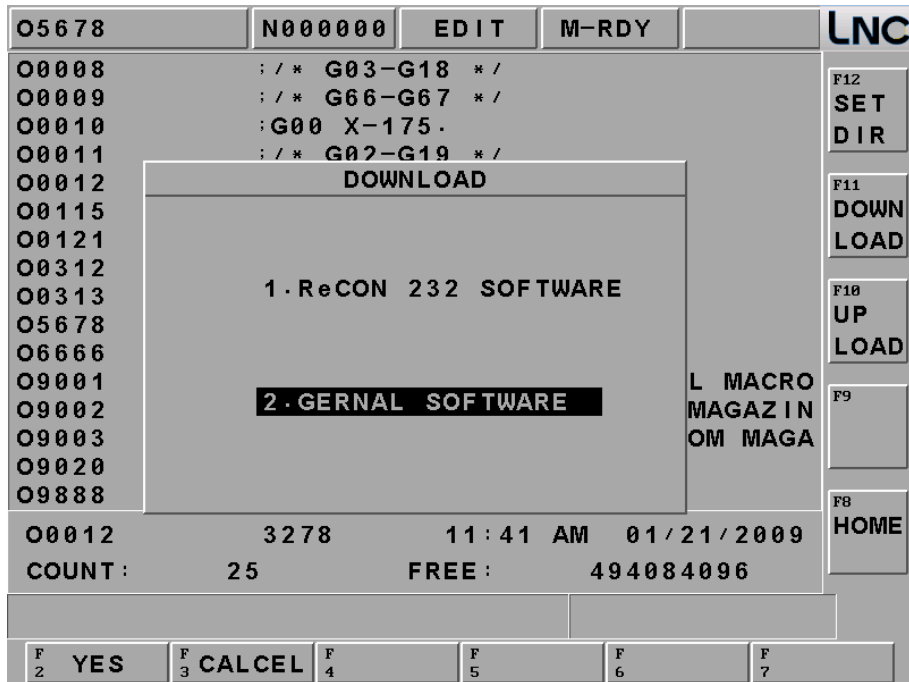
O5678	N000000	EDIT	M-RDY	LNC
O0121	: / * G121 * /			F12 SET DIR
O0312	: X25.82Z39.143			
UP LOAD				
PC WORK PATH				F11 DOWN LOAD
SOURCE	E:\CNC\NCFILES\CAT\O0001			
NC WORK PATH				F10 UP LOAD
TARGET	C:\LNCMILL\NCFILES\O0012			
				F9
				F8 HOME
O5678	92907	07:30 PM	01/15/2009	
COUNT :	25	FREE :	494321664	
F2 OK	F3 CANCEL	F4	F5	F6
F7				

26. Upload file successful.

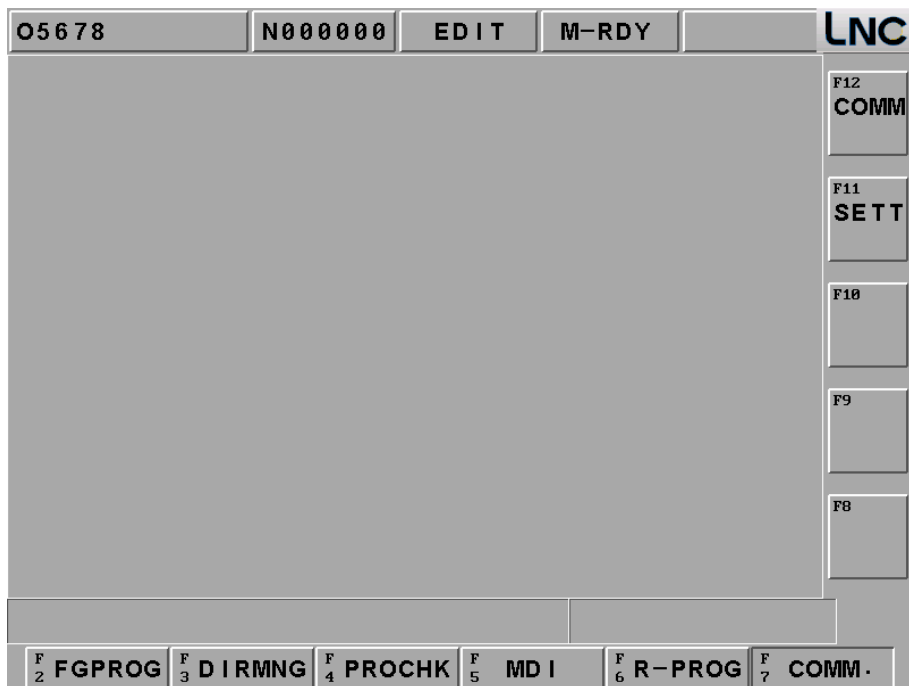
O5678	N000000	EDIT	M-RDY	LNC
O0008	: / * G03-G18 * /			F12 SET DIR
O0009	: / * G66-G67 * /			
O0010	: G00 X-175.			
O0011	: / * G02-G19 * /			F11 DOWN LOAD
O0012	: FROM/150.120.41			
O0115	: G21			F10 UP LOAD
O0121	: / * G121 * /			
O0312	: X25.82Z39.143			
O0313	: FROM/150.120.41			F9
O5678	: N1			F8 HOME
O6666	: N1			
O9001	: / * O9001 IS THE M06 CALL MACRO			
O9002	: / * M91 PUT THE TOOL TO MAGAZIN			
O9003	: / * M90 TAKE THE TOOL FROM MAGA			
O9020	: M99			
O9888	: O9888			
O0012	3278	11:41 AM	01/21/2009	
COUNT :	25	FREE :	494084096	
F2 FGPROG	F3 DIRMNG	F4 PROCHK	F5 MDI	F6 R-PROG
F7 COMM.				

#### 4.4.3 Path Program Upload & save (using general software)

27. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
28. Use PROG group, change to EDIT mode.
29. Go to **【DIRMNG】** and choose the file that you want to transfer to PC end and press [DOWN LOAD].  
Select 1. General software to upload.



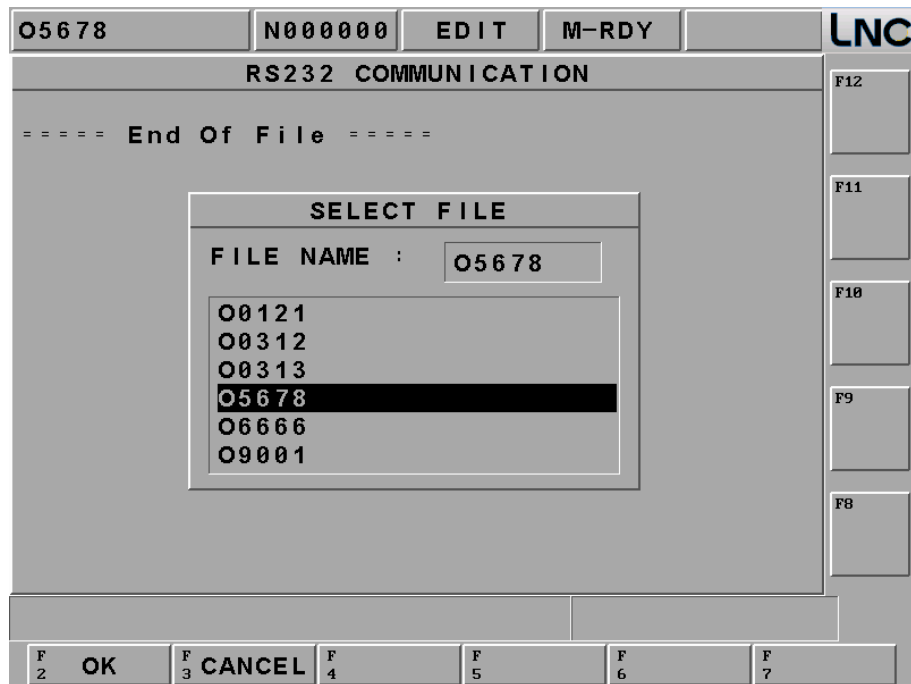
30. Go to Communication page and press [COMM] at the down right side.



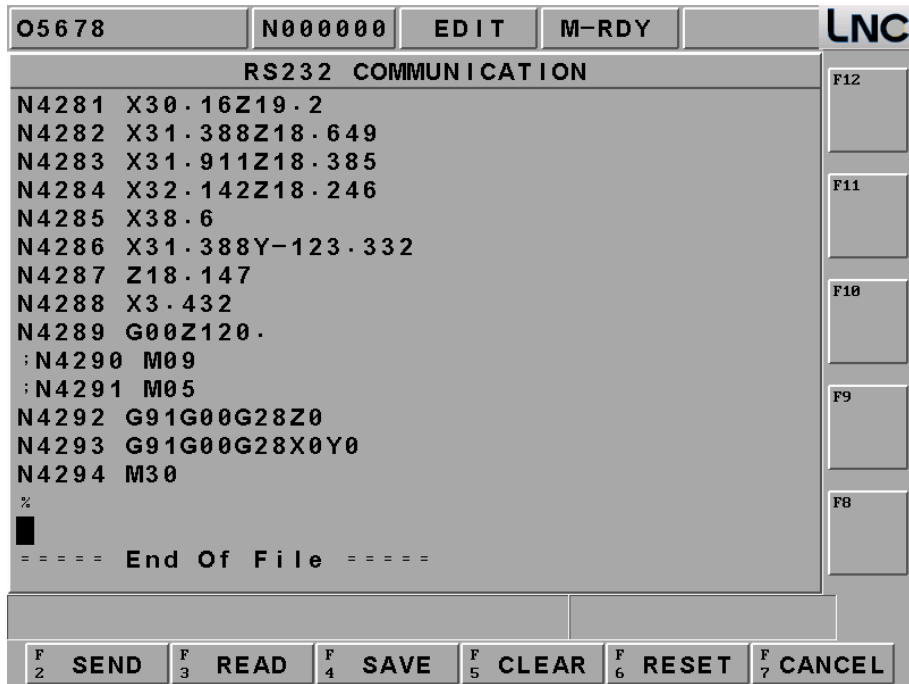
31. Go to RS232 COMMUNICATION file download page.



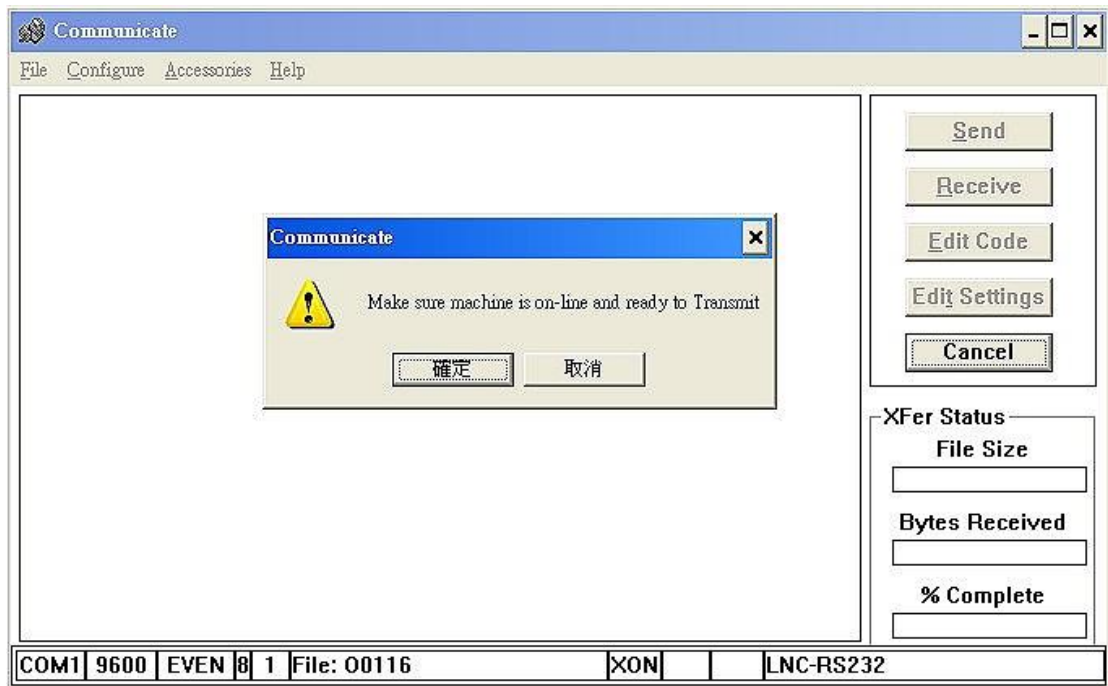
32. Press [SEND], start to transfer files from NC end to PC end, select name and press [OK]



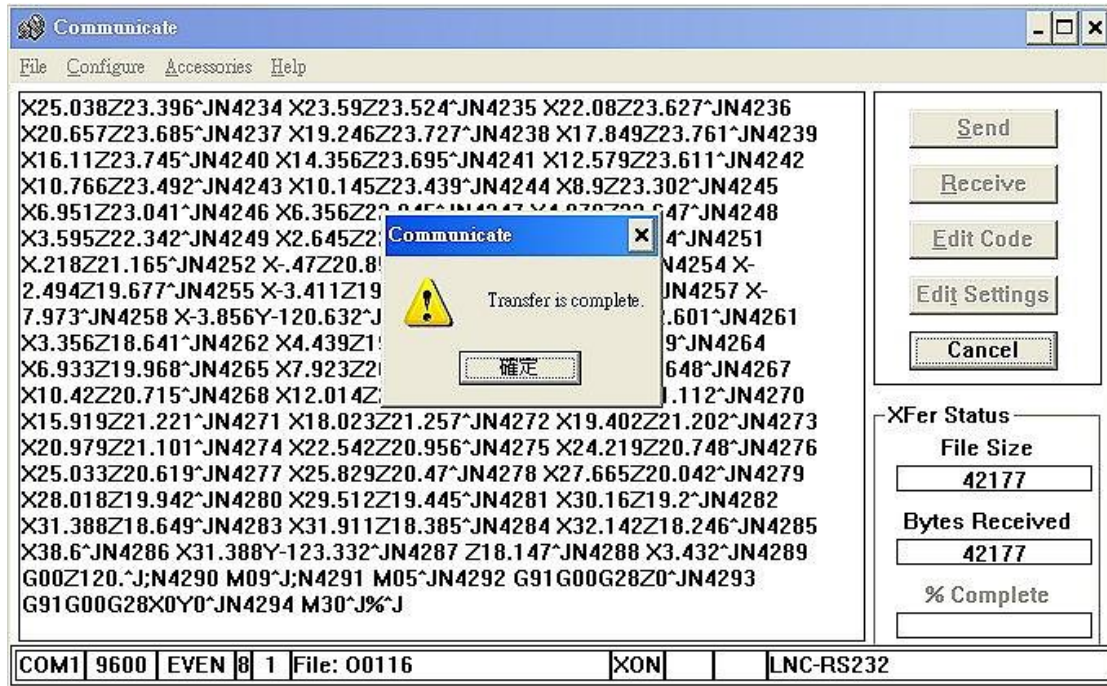
33. NC end's file will start to transfer to PC end.



34. After communicate setting at PC end, open a blank file for transferring, press [Receive], confirm to deliver file, press [Confirm].

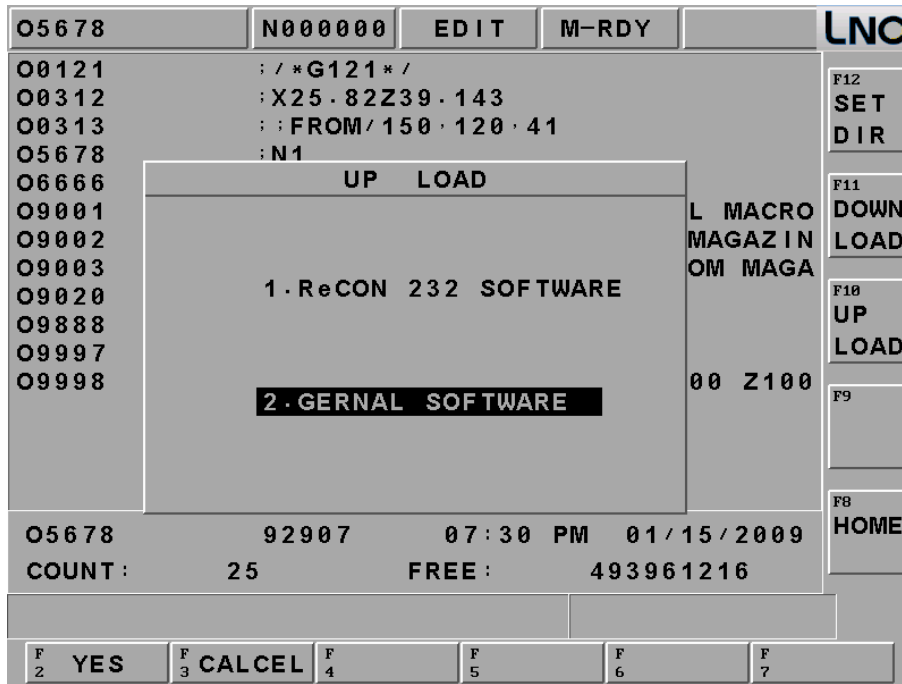


35. After transfer is completed, press [OK]

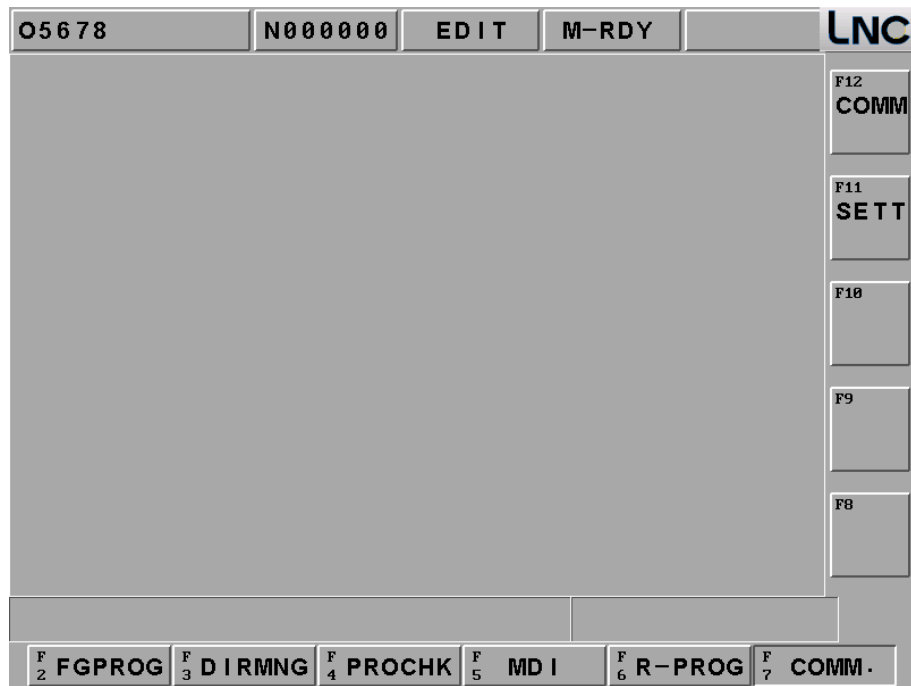


#### 4.4.4 Path Program Upload & save (using general software)

36. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
37. Use PROG group, change to EDIT mode.
38. Go to **【DIRMNG】** and choose the file that you want to transfer to PC end and press [UP LOAD].  
Select 1. General software to upload.



39. Go to communication page and press [COMM] at the down right side.

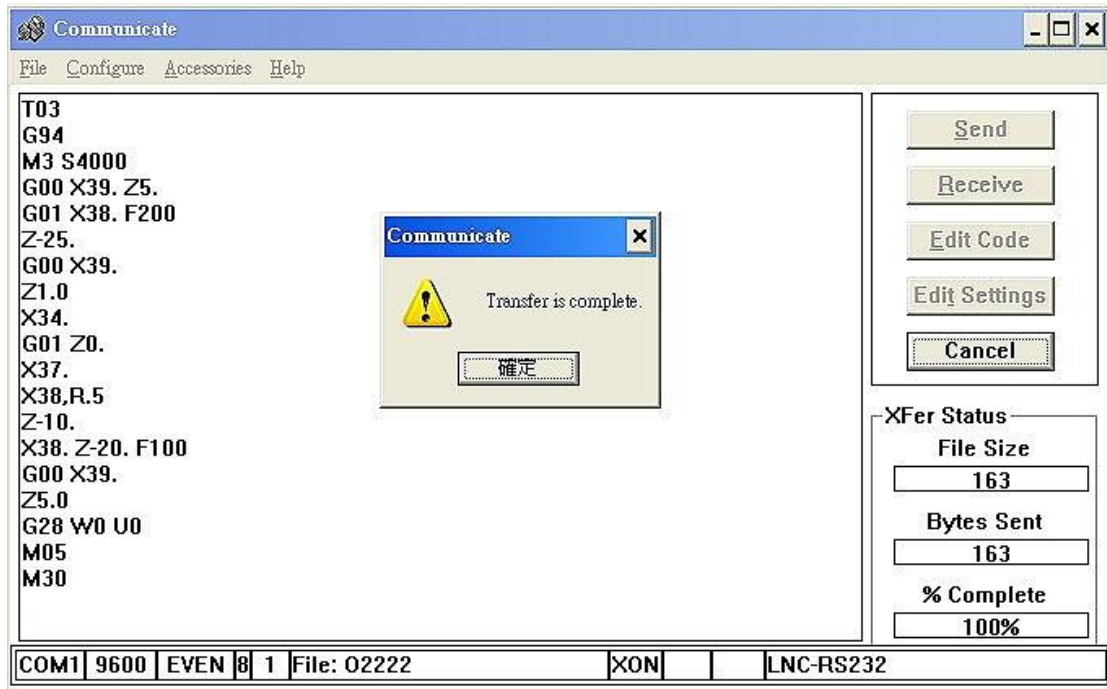


40. Go to RS232 communication.

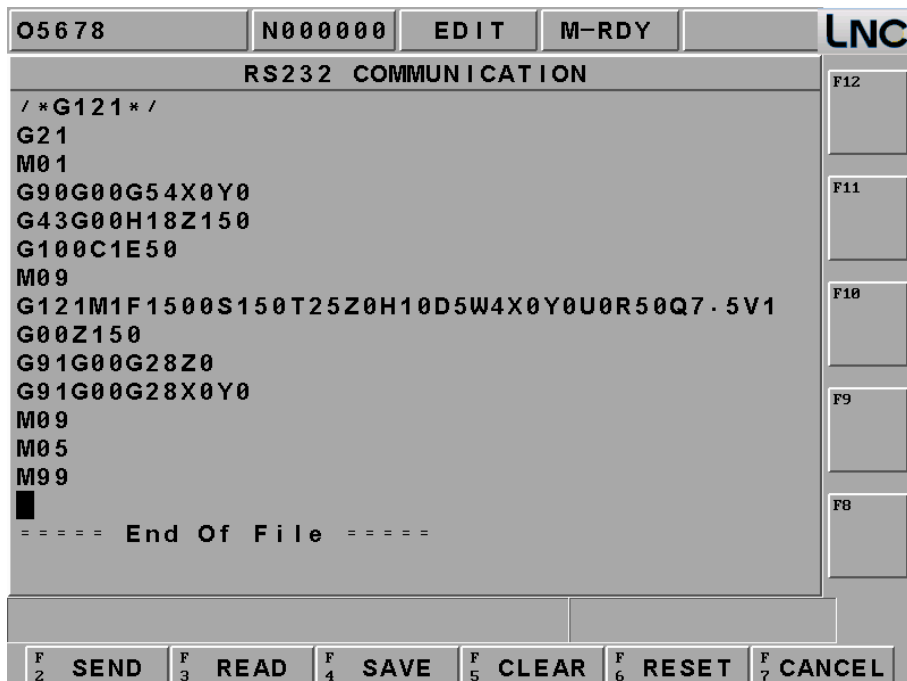




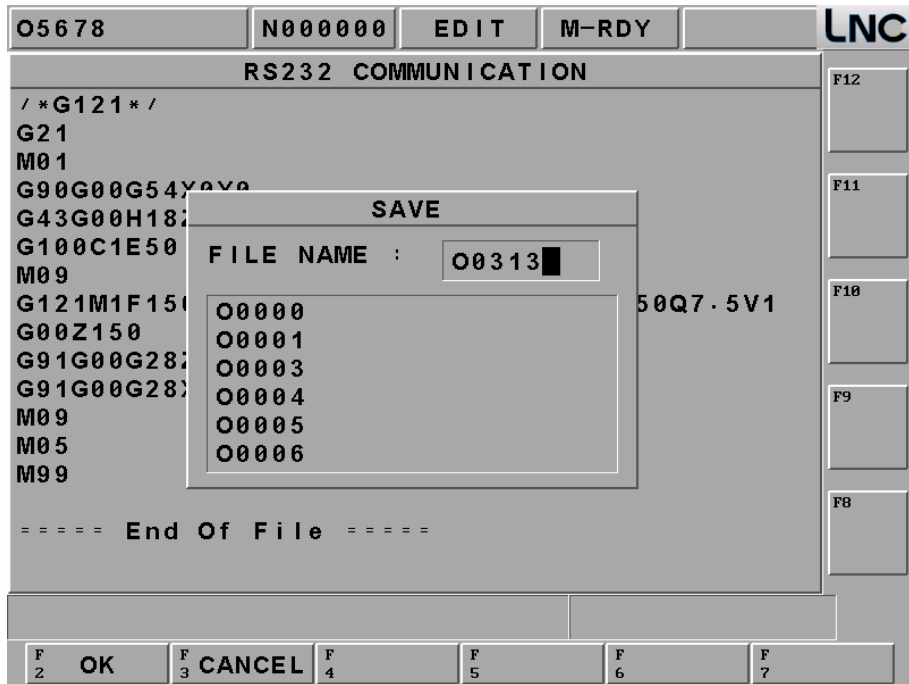
41. After communicate setting at PC end, open an old file for transferring, press [Send], confirm to upload file, press [OK], upload complete .



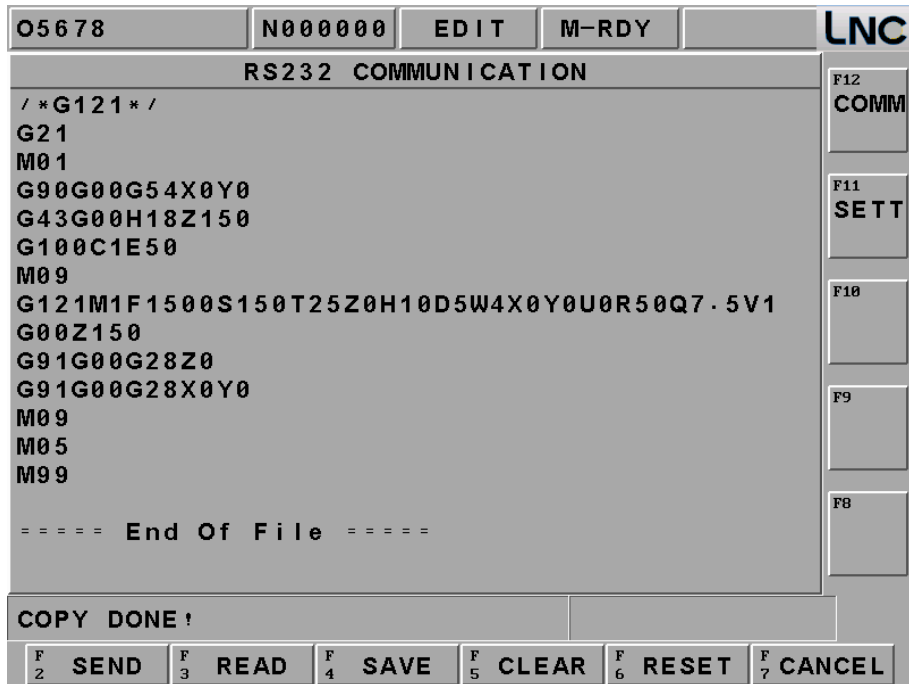
42. PC end will start to receive the files from NC end automatically.



43. After receiving, press SAVE with a new name or SAVE at the same file.



44. Press OK, Copy complete.



45. Go to [DIRMNG], you could see O0312 has saved at [DIRMNG]

O5678	N000000	EDIT	M-RDY	LNC
O0011	: / * G02-G19 * /			F12 SET DIR
O0012	: : FROM / 150 · 120 · 41			
O0115	: G21			F11 DOWN LOAD
O0121	: / * G121 * /			
O0312	: X25 · 82Z39 · 143			F10 UP LOAD
<b>O0313</b>	<b>: / * G121 * /</b>			F9
O5678	: N1			F8 HOME
O6666	: N1			
O9001	: / * O9001 IS THE M06 CALL MACRO			
O9002	: / * M91 PUT THE TOOL TO MAGAZIN			
O9003	: / * M90 TAKE THE TOOL FROM MAGA			
O9020	: M99			
O9888	: O9888			
O9997	: / * TOOL CHANGE * /			
O9998	: G90 G00 X200 · 000 Y100 · 000 Z100			
O0313 162 11:55 AM 01/21/2009				
COUNT : 25 FREE : 493838336				
<div> <div>F2 FGPROG</div> <div>F3 DIRMNG</div> <div>F4 PROCHK</div> <div>F5 MDI</div> <div>F6 R-PROG</div> <div>F7 COMM ·</div> </div>				

46. At last, check file amount and graphic preview; see if there is any mistake.

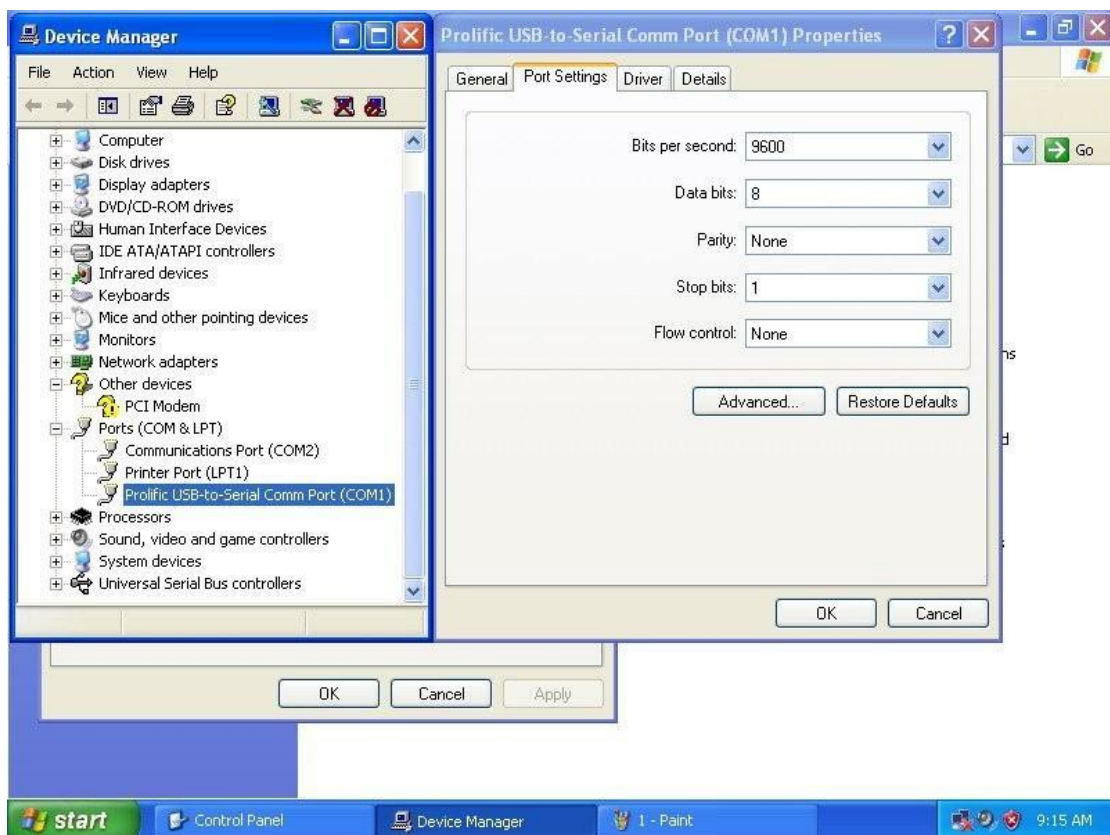
O0313	N000000	EDIT	M-RDY	LNC
/ * G121 * /				F12 GOTO
G21				
M01				F11 LDEL
G90G00G54X0Y0				
G43G00H18Z150				F10 MARK
G100C1E50				
M09				F9 UN MARK
G121M1F1500S150T25Z0H10D5W4X0Y0U0R50Q7 · 5V1				F8 NEXT
G00Z150				
G91G00G28Z0				
G91G00G28X0Y0				
M09				
M05				
M99				
= = = = End Of File = = = =				
ROW: 1 / 14 COL: 1				
<div> <div>F2 FGPROG</div> <div>F3 DIRMNG</div> <div>F4 PROCHK</div> <div>F5 MDI</div> <div>F6 R-PROG</div> <div>F7 COMM ·</div> </div>				

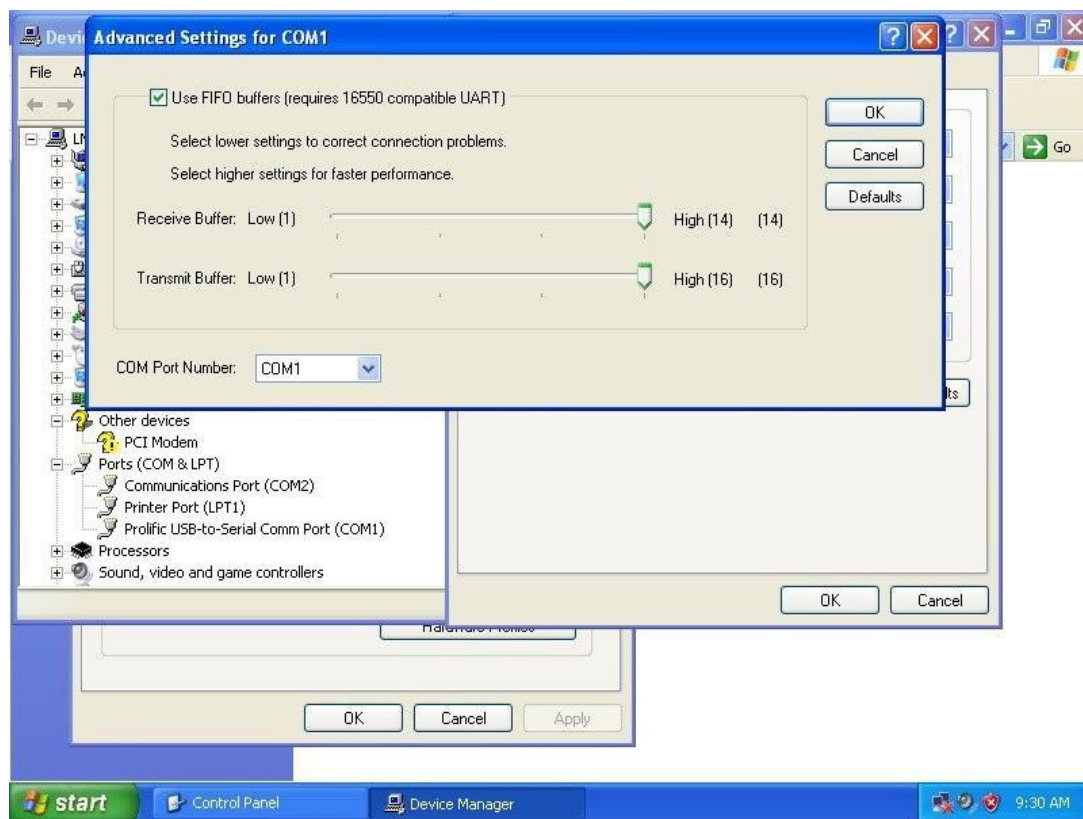
#### 4.5 Setting & Attention for USB to RS232 port

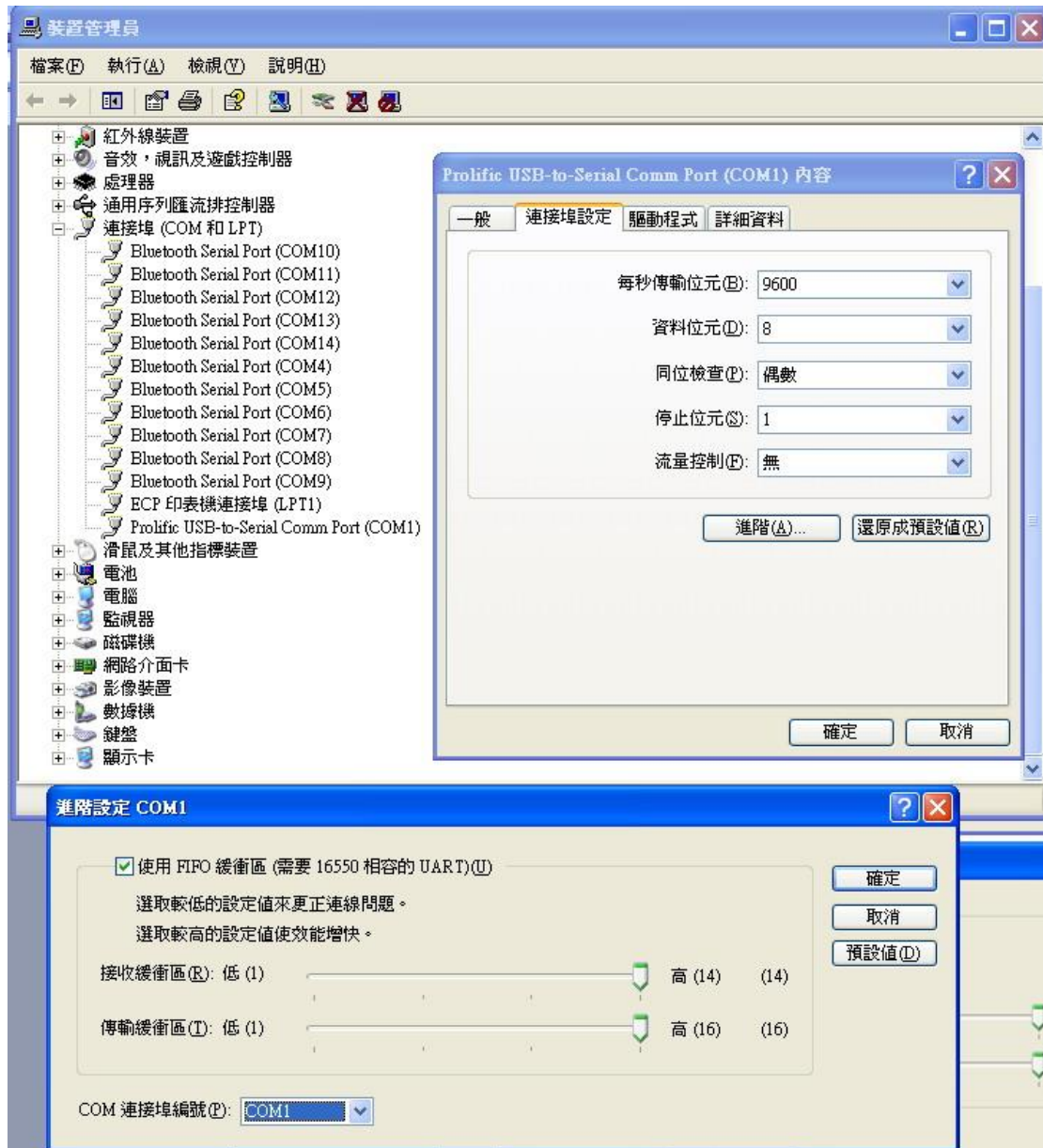
If you are NOTE BOOK user, you may not find COM1 pin all the time, at this time, you could use USB to transfer to RS232. Besides installing drivers of USB to RS232, you may still need to pay attention to below details :

If this is not COM1 with using [USB to RS232], please open device manager> further setting and change the setting to be COM1 and make sure other setting is the same with NC end.

[Device Manager] > [Start] > [Control Panel] > [System] > [Hardware] > [Device Manager] > [Ports] or [COM&LPT], you can find detail as below.







## 5 Alarm & Warning

When the system alarm message (ALARM) occurs, operation will stop. User must check the whole machine according to the alarm message. If able to solve problems, then only need to click RESET again to clear the condition. (However, if need to change parameter, then must exit the system software and then re-enter for the system to work.)

Three types of alarm message which are MOT (MOTION) ALARM, OP (OPERATION) ALARM or INT (INTERPRETATION) ALARM. Definitions of the above alarm message are as following :

### 5.1 OP OPERATION ALARM

#### OP 1001 : X SERVO ALARM

#### OP 1002 : Y SERVO ALARM

#### OP 1003 : Z SERVO ALARM

#### OP 1004 : 4<sup>TH</sup> SERVO ALARM

- (1) Alarm message from SERVO driver.
- (2) Please check ERROR message from the SERVO driver to know the cause.
- (3) Please re-booting.

#### OP 1013 : SWITCH TO CONTROL/MONITOR WHILE X MOVING

- (1) Trigger C bit 130 while X moving.
- (2) Press RESET to clear move command and C 130

#### OP 1014 : SWITCH TO CONTROL/MONITOR WHILE Y MOVING

- (1) Trigger C bit 131 while Y moving.
- (2) Press RESET to clear move command and C 131

#### OP 1015 : SWITCH TO CONTROL/MONITOR WHILE Z MOVING

- (1) Trigger C bit 132 while Z moving
- (2) Press RESET to clear move command and C 132

#### OP 1016 : SWITCH TO CONTROL/MONITOR WHILE 4 MOVING

- (1) Trigger C bit 133 while 4 moving
- (2) Press RESET to clear move command and C 133

#### OP 1017 : LNC SYSTEM EXPIRED

- (1) Please contact with distributor or machine makers.

#### OP 1018 : DESIRED MACRO VARIABLES NOT EXIST.

- (1) Check Macro variables.

#### **OP 1019 : DESIRED MACRO VARIABLES OVER RANGE**

- (1) Check Macro variable value.

#### **OP 1020 : OVER MLC TRAVEL LIMIT**

- (1) Check each axis hardware travel limit.

#### **OP 1021 : GEAR SIGNAL ERROR**

- (1) Check if gear is correct.

#### **OP 1024 : SWITCH TO CONTROL/MONITOR WHILE 5 MOVING**

- (1) Trigger C bit 168 while 5 moving
- (2) Press RESET to clear move command and C 168

#### **OP 1025 : SWITCH TO CONTROL/MONITOR WHILE 6 MOVING**

- (1) Trigger C bit 169 while 6 moving
- (2) Press RESET to clear move command and C 169

#### **OP 1027 : HSP LONG, NEED TO CONTACT WITH SYSTEM PROVIDER.**

Please contact with distributor or machine makers

#### **OP 1029 : X AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1030 : X AXIS OVER MLC TRAVEL LIMIT (-)**

#### **OP 1031 : Y AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1032 : Y AXIS OVER MLC TRAVEL LIMIT (-)**

#### **OP 1033 : Z AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1034 : Z AXIS OVER MLC TRAVEL LIMIT (-)**

#### **OP 1035 : 4TH AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1036 : 4TH AXIS OVER MLC TRAVEL LIMIT (-)**

#### **OP 1037 : 5TH AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1038 : 5TH AXIS OVER MLC TRAVEL LIMIT (-)**

#### **OP 1039 : 6TH AXIS OVER MLC TRAVEL LIMIT (+)**

#### **OP 1040 : 6TH AXIS OVER MLC TRAVEL LIMIT (-)**

OP1029~OP1040:

- (1) Press rest to reboot system
- (2) Move to the opposite side out of limit manually

#### **OP 1100 : 5 SERVO AXIS ERROR**

- (1) 5 servo drive send out ALARM
- (2) Check servo drive ERROR message to know the reason.
- (3) Reboot.



**OP 1101 : 6 SERVO AXIS ERROR**

- (1) 6 servo drive send out ALARM
- (2) Check servo drive ERROR message to know the reason
- (3) Reboot

**5.2 OP**

**OP 6001 : X AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6002 : X AXIS OVER MLC TRAVEL LIMIT (-)**

**OP 6003 : Y AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6004 : Y AXIS OVER MLC TRAVEL LIMIT (-)**

**OP 6005 : Z AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6006 : Z AXIS OVER MLC TRAVEL LIMIT (-)**

**OP 6007 : 4TH AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6008 : 4TH AXIS OVER MLC TRAVEL LIMIT (-)**

**OP 6009 : 5TH AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6010 : 5TH AXIS OVER MLC TRAVEL LIMIT (-)**

**OP 6011 : 6TH AXIS OVER MLC TRAVEL LIMIT (+)**

**OP 6012 : 6TH AXIS OVER MLC TRAVEL LIMIT (-)**

OP6001~OP6012:Check each axis hardware travel limit.

## **5.3 INT**

### **INT 3001 : NO SUCH TOKEN**

- (1) Part program enter data has invalid symbols or characters.
- (2) Modify program error.
- (3) Press RESET to clear the wrong warning message.

### **INT 3002 : GRAMMAR ERROR**

- (1) Part program enter data has invalid symbols or characters.
- (2) Modify program error.
- (3) Press RESET to clear the wrong warning message.

### **INT 3003 : OUT OF NODE MEMORY**

- (1) MACRO grammar has too complicate showing program such as too many brackets.
- (2) Simplify complexity degree or make it to be 2 blocks.
- (3) Press RESET to clear the wrong warning message.

### **INT 3004 : EXECUTE NODE ERROR**

- (1) System executes mathematics calculation that is not allow to execute
- (2) Press RESET to clear the wrong warning message.

### **INT 3005 : FUNCTION ERROR**

- (1) System executes invalid function that is not allow to execute. (Won't happen under normal system condition.)
- (2) System error, please contact the supplier.

### **INT 3006 : DIVIDED BY 0**

- (1) MACRO is divided by 0.
- (2) Modify numerator of the division. Must NOT be 0.
- (3) Press RESET to clear the wrong warning message.

### **INT 3007 : VARIABLE OVER RANGE**

- (1) One/some of local variables, common variables and global variables are out of range.
- (2) Modify variable numbers that are out of their number range.
- (3) Press RESET to clear the wrong warning message.

**INT 3008 : MACRO DOMAIN ERROR**

- (1) MACRO function domain error. If square (SQRT) argument is negative or ATAN arguments are two zeros.
- (2) Modify domain.
- (3) Press RESET to clear the wrong warning message.

**INT 3010 : NOT ALLOWABLE DECIMAL POINT**

- (1) NC address has not allowable decimal point.
- (2) Modify decimal point in NC address.

**INT 3011 : WORD DATA OVER RANGE**

- (1) NC address word data is out of range.
- (2) Modify word data in NC address.
- (3) Press RESET to clear wrong warning message.

**INT 3012 : MACRO ILLEGAL MACRO PARAMATER INPUT ( G 、 L 、 N 、 O 、 P )**

- (1) Illegal arguments ( G 、 L 、 N 、 O 、 P ) in MACRO program.
- (2) Correcting these illegal arguments.
- (3) Press RESET to clear wrong warning message.

**INT 3013 : PART PROGRAM HAS GRAMMER ERROR.**

- (1) Input data has grammar error.
- (2) Part program error.
- (3) Press RESET to clear wrong warning message.

**INT 3050 : TOOL DIAMETER IS 0**

- (1) Modify tool diameter setting.

**INT 3051 : ILLEGAL RPM GIVEN**

- (1) Part program command has rotation speed command over maximum setting.
- (2) Modify error in part program.

**INT 3052 : ILLEGAL FEEDRATE GIVEN**

- (1) Part program command has feed rate command over maximum setting.
- (2) Modify error in part program.

**INT 3053 : ( D ) EACH CUT DOWN DEPTH IS 0**

- (1) When using G101~G133, check if setting cutting depth, if setting is equal to 0 or smaller than 0, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program to prevent cutting depth is equal to 0 or smaller than 0.

**INT 3054 : ( H ) TOTAL DEPTH IS 0**

- (1) When using G101~G133, check if setting total cutting depth, if setting is equal to 0 or smaller than 0, Alarm
- (2) Check program, if has this situation .
- (3) Revise program to prevent total cutting depth is equal to 0 or smaller than 0.

**INT 3055 : ESCAPE LOWER START POINT.Z**

- (1) G100 can set up return amount, when using G101~G133, check if the setting of cycle return R point is higher then setting, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program to make cycle return R lower than return amount.

**INT 3056 : ( W ) EACH CUT WIDTH IS 0**

- (1) When using G101~G133, check if setting cutting width, if setting is equal to 0 or smaller than 0, Alarm
- (2) Check program, if has this situation.
- (3) Revise program to prevent cutting width is equal to 0 or smaller than 0.

**INT 3060 : HOLES TOO DENSITY**

- (1) When using G101~G133, check if hole distance too small, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program to prevent too small hole.

**INT 3061 : HOLES COUNT MUST >=2**

- (1) When using G101~G133, check if hole numbers are small or equal to 1, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program to prevent hole numbers are small or equal to 1.

**INT 3062 : R MUST > Z**

- (1) When using G101~G133, check if this Z axis hold bottom coordinate setting is higher than cycle return R point, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make this Z axis hold bottom coordinate lower then cycle return R.

**INT 3070 : WRONG DATA : R=0**

- (1) When using G101~G133, the R radius value=0, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make R radius is not 0.

**INT 3071 : WRONG DATA :  $2R \leq \Phi$** 

- (1) When using G101~G133, check if 2 times of R radius is smaller than tool diameter, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make R radius to be reasonable value.

**INT 3072 : WRONG DATA :  $V > \Phi$** 

- (1) When using G101~G133, check if finish amount is over tool diameter, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make finish amount to be reasonable value.

**INT 3073 : WRONG DATA :  $Q = 0$** 

- (1) When using G101~G133, check if setting total move value, and setting=0, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program to prevent total setting= 0.

**INT 3074 : WRONG DATA :  $V \geq Q$** 

- (1) When using G101~G133, check if finish amount is over total cutting alarm, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make finish amount to be reasonable value.

**INT 3075 : WRONG DATA :  $(\Phi + 2Q) \geq 2R$** 

- (1) When using G101~G133, check if setting diameter+2 times' total cutting amount is over 2 times' R radius, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3076 : WRONG DATA :  $I (J) = 0$** 

- (1) When using G101~G133, check if complete work size I or J=0, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3077 : WRONG DATA :  $I (J) - 2R \leq 0$** 

- (1) When using G101~G133, check if complete work size I or J is smaller than 2 times' R radius, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3078 : WRONG DATA :  $2Q + \Phi \geq I (J)$** 

- (1) When using G101~G133, check if setting diameter+2 times' total cutting amount is over complete work piece I or J, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3079 : WRONG DATA :  $2V+PHI>I$  ( J )**

- (1) When using G101~G133, check if setting diameter+2 times' finish amount is over complete work piece I or J, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3080 : DISTANCE OF TWO CENTER IS 0**

- (1) When using G123~G133, check the runway-shaped command's center distance is less than or equal to 0, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3081 : WRONG DATA :  $2(R-V) < PHI$** 

- (1) When using G123~G133, check the runway-shaped command's R radius and finish amount difference is smaller than tool radius, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3082 : WRONG DATA :  $2(R-V) \leq PHI$** 

- (1) When using G131~G133, check the runway-shaped command's R radius and finish amount difference is smaller than tool radius, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3083 : WRONG DATA :  $2V+PHI \geq I$  ( J )**

- (1) When using G132, check this command's tool radius +2 times finish amount are over complete work piece I or J, Alarm.
- (2) Check program, if has this situation.
- (3) Revise program, make above setting to be reasonable value.

**INT 3090 : T COMMAND ERROR**

- (1) Check if T code has errors.
- (2) Check program, if has this situation.
- (3) Revise program, make change tool command tool number correct.

**INT 3100 : ILLEGAL G CODE**

- ( 1 ) Illegal G code in part program.
- ( 2 ) Delete illegal G code.
- ( 3 ) Press RESET to clear wrong warning message.

**INT 3101 : PFM INITIAL ERROR**

**INT 3102 : TRANS INITIAL ERROR**

**INT 3103 : TABLE INITIAL ERROR**

**INT 3104 : POST INITIAL ERROR**

**INT 3105 : PFM ERROR**

**INT 3106 : NC Program Block Interpretation Fail**

INT3101~INT3106

- (1) System is not normal right now.
- (2) System error, contact LNC.

**INT 3110 : FETCH ERROR**

- (1) System is not normal right now.
- (2) System error, contact LNC.

**INT 3111 : LACK OF FILENAME ( P address is not entered )**

- ( 1 ) Lack of filenames in part program. (P address is not entered. )
- ( 2 ) Increase number of filenames.
- ( 3 ) Press RESET to clear wrong warning message.

**INT 3112 : ILLEGAL FILENAME**

- ( 1 ) Illegal filename in part program.
- ( 2 ) Modify filename.
- ( 3 ) Press RESET to clear wrong warning message.

**INT 3113 : FILE NOT FOUND**

- ( 1 ) File not found in the system.
- ( 2 ) Making/modifying the executing file.
- ( 3 ) Press RESET to clear wrong warning message.

**INT 3114 : END OF FILE**

**INT 3120 : POST ERROR**

**INT 3121 : LACK OF SUB RETURN ( M99 )**

- ( 1 ) No returning to the part program command in the sub-program.
- ( 2 ) Add returning back to the part program command in the sub-program.
- ( 3 ) Press RESET to clear wrong warning message.

**INT 3122 : PROGRAM OVERFLOW ( 8 )**

- ( 1 ) Total numbers of calling Sub-program or MACRO is over the total level limit. ( i.e., 8 )
- ( 2 ) Decrease numbers of calling program level.
- ( 3 ) Press RESET to clear the wrong warning message.

**INT 3123 : MACRO OVERFLOW ( 4 )**

- ( 1 ) Total numbers of calling MACRO is over the total level limit. (i.e., 4).
- ( 2 ) Decrease numbers of calling program level.
- ( 3 ) Press RESET to clear the wrong warning message.

**INT 3124 : MACRO UNDERFLOW ( G67 )**

- (1) Under G67, but give G67 command again.
- (2) Delete G67.
- (3) Press RESET to clear warning.

**INT 3125 : WITHOUT LABEL**

- ( 1 ) NO such LABEL .
- ( 2 ) Please check LABEL name.
- ( 3 ) Press RESET to clear wrong message error.

**INT 3126 : BLOCK NOT FOUND**

- ( 1 ) The designated BLOCK is not found.
- ( 2 ) Check whether or not the designated BLOCK is exist in part program.
- ( 3 ) Press RESET to clear wrong message error.

**INT 3127 : ILLEGAL LABEL****INT 3128 : FEEDRATE OUT OF RANGE , Check G94/G95**

- ( 1 ) Check G94 and G95 are used correctly.
- ( 2 ) Check whether or not the FEEDRAT F value is too big.
- ( 3 ) Press RESET to clear the error and adjust G94, G95 and F value.

**INT 3130 : COORDINATE ERROR****INT 3131 : UNKNOWN PLANE****INT 3132 : ILLEGAL RADIUS**

- ( 1 ) Using G02, G03 arch cutting command, the final coordinate is NOT on the arch
- ( 2 ) Check the position of center point, direction and final point's coordinate value



(3) Press RESET to clear error.

**INT 3135 : Tool Number Assigned Over Range**

- (1) T code range is 0~99.
- (2) Modify T code.

**INT 3136 : Tool Length Comp Number Assigned Over Range**

- (1) Tool length compensation number or tool diameter compensation number over range.
- (2) Tool length or tool diameter range is 0~99.

**INT 3137 : System Program Overflow**

- (1) System sub-program +system MACRO are over call layer.
- (2) Decrease calling layers.

**INT 3138 : System Macro Overflow**

- (1) Call layer of system MACRO over limit (8)
- (2) Decrease calling layers.

**INT 3139 : Does Not Permit Set Model Macro Call**

- (1) This is in the model MACRO call, can not be call again.
- (2) Check check model MACRO call,

**INT 3140 : SEND TABLE ERROR**

- (1) System is not normal right now.
- (2) System error, contact LNC.

**INT 3141 : NO FREE VARIABLES**

- (1) System is not normal right now.
- (2) System error, contact LNC.

**INT 3142 : Address Duplication Error**

- (1) There is duplication in program single block, EX : G01 X10 Y20 X30 °
- (2) Modify program.

**INT 3150 : INSUFFICIENT DATA**

- (1) Not enough executing G code data. (Lack of G10's P , R , Z)
- (2) Supply the needed data.
- (3) Press RESET to clear wrong warning message.

**INT 3152 : CC R RETURN**

- (1) Executing G27 , G28 , G29 , G30 in the CANNED CYCLE.
- (2) Cancel the above G codes in CANNED CYCLE before executing.
- (3) Press RESET to clear wrong warning message.

**INT 3153 : NO SUCH R POINT**

- (1) Wrong reference point number in G30.

- (2) Modify the entered reference point number.
- (3) Press RESET to clear wrong warning message.

**INT 3156 : ILLEGAL G31 IN COMPENSATION**

- (1) G31 Skip signal stop SBK can not be used under G41/G42, adjust program to prevent G31 in G41/G42.

**INT 3157 : G10 P CODE OUT OF RANGE**

- (1) P argument is over command legal usage range.
- (2) Modify P argument

**INT 3158 : G10 L/E CODE OUT OF RANGE**

- (1) L argument is over command legal usage range or E argument is over command legal usage range.
- (2) Modify P argument or E argument.

**INT 3160 : DNC : INCORRECT READ SEQUENCE**

- (1) Check whether or not there is incorrect sub-program or jump sequence from the part program.
- (2) Press RESET to clear wrong warning message, and modify part program.

**INT INT 3161 : DNC : LOSS DATA PACKET**

- (1) DNC RS232 info lost when transmission.
- (2) Transmission cable loose or brake.
- (3) Lower Transmission speed parameter
- (4) Press RESET to clear warning or reboot.

**INT 3162 : DNC : PROGRAM BUFFER OVERFLOW**

- (1) Program buffer overflow while DNC RS232 is transmitting program.
- (2) Check whether or not the connecting line is disconnect or fall.
- (3) Press RESET to clear wrong warning message or reboot.

**INT 3163 : G05 R CODE RANGE BETWEEN 1 TO 10**

- (1) G05 R argument range 1-10.
- (2) Revise program to make G05 R argument in legal range.
- (3) Press RESET to clear warning or reboot.

**INT 3165 : Illegal data in G54 P address**

- (3) Press RESET to clear warning or reboot.

**INT 3166 : Illegal data in G10**

- (1) G10 gives L argument, L20 must set up extension coordinate.
- (2) Use G10 to set up extension coordinate, P range=1-300
- (3) Revise program to make G10 at reasonable range.
- (4) Press RESET to clear warning or reboot.

**INT 3167 : G05 P/L CODE MUST APPEAR SIMULTANEOUSLY**

- (1) G05, if set up for individual items, P and L must show together.
- (2) Revise program to make G05 at reasonable range.

- (3) Press RESET to clear warning or reboot.

**INT 3168 : NO RETURN HOME**

- (1) Before run HOME, make sure if return home manually.

**INT 3169 : RETURN HOME FAILURE**

- (1) Current position is not at home.  
(2) Check program.

**INT 3177 : Address Q Not Found**

- (1) G73/G83 did not give cutting feedrate argument, check program to make Q has value, and can not be 0.

**INT 3200 : THE SEQUENCE NO OF GOTO COMMAND ERROR**

- (1) Skip line number is at the current line number, infinitive loop.  
(2) Modify skip serial.

**INT 3201 : COMP UNIT VECTOR 0**

- (1) Compensation unit vector is 0.  
(2) System error, please contact supplier.

- (3) Press RESET to clear wrong warning message

**INT 3202**

**Mill V3: Comp Operation Does Not Permit Change Plane**

- (1) Under G41/G42, can not change surface.  
(2) Revise program and cancel G17/G18/G19.

**Mill V6: COMP START UP ARC**

- (1) Compensating start-up is arch.  
(2) Please start compensating according to G00/G01.  
(3) Press RESET to clear wrong warning message.

**INT 3203**

**Mill V3: Arc Line Not Found INTERsect At Comp Start Up Arc**

- (1) Compensation starting is arc.  
(2) Use linear as compensation starting.  
(3) Press RESET to clear warning.

**Mill V6: COMP CANCEL ARC**

- (1) Compensating cancel is arch.  
(2) Please cancel compensation according to G00/G01.  
(3) Press RESET to clear wrong warning message.

**INT 3204 : Arc Line Not Found INTERsect At Comp Cancel Arc**

- (1) Compensation cancel is arc.  
(2) Use linear to cancel compensation.  
(3) Press RESET to clear warning.

**INT 3205 : COMP VECTOR LENGTH 0**

- (1) When compensation calculating, there is no moving SBK.
- (2) Check program.

**INT 3206 : Arc Line INTERsect Vector Over Comp Range**

- (1) Not able to find the intersect points while calculating compensation value.
- (2) Decrease compensation radius or modify part program pathway.
- (3) Press RESET to clear wrong warning message.

**INT 3207 : Arc Line INTERsect Vector Length Over Arc Move Vector**

- (1) Not able to find the intersect points while calculating compensation value.
- (2) Decrease compensation radius or modify part program pathway.
- (3) Press RESET to clear wrong warning message.

**INT 3223 : COMP OPERATION DOES NOT PERMIT CHANGE TOOL RADIUS OFFSET NO.**

- (1) Under G41/G42, can not change tool diameter compensation number at arc command SBK.
- (2) Revise program to cancel tool diameter compensation number at radius compensation mode.

**INT 3301 : THIS AXIS IS SLAVE AXIS IN SYNCHRONOUS CONTROL**

- (1) You can not give command to slave axis, because the control is from master axis.
- (2) Revise program to delete slave axis program command.
- (3) Press RESET to clear warning.

**INT 3303 : THIS REPLACED AXIS MOVE REPEAT IN AXIS NAME CHANGE MODE**

- (1) You can not give command to replace axis, because the control is from this SBK axis.
- (2) Revise program to cancel this SBK.
- (3) Press RESET to clear warning.

## **5.4 MOT MOTION RELATED ALARM**

### **MOT 4001 : X-AXIS ERROR COUNTER OVERFLOW**

- (1) Motion board X- Axis ERROR COUNTER overflow ( 16-BIT ) .
- (2) Check or not the commanding speed is too fast.
- (3) Check whether or not servo motor is working normally.
- (4) Check whether or not the machine is running normally.
- (5) Check whether or not the board is normal.

### **MOT 4002 : Y-AXIS ERROR COUNTER OVERFLOW**

- (1) Motion board Y-AXIS ERROR COUNTER overflow ( 16-BIT )
- (2) Check or not the commanding speed is too fast.
- (3) Check whether or not servo motor is working normally.
- (4) Check whether or not the machine is running normally.
- (5) Check whether or not the board is normal.

### **MOT 4003 : Z-AXIS ERROR COUNTER OVERFLOW**

- (1) Motion board Z-AXIS ERROR COUNTER overflow ( 16-BIT )
- (2) Check or not the commanding speed is too fast.
- (3) Check whether or not servo motor is working normally.
- (4) Check whether or not the machine is running normally.
- (5) Check whether or not the board is normal.

### **MOT 4004 : 4TH-AXIS ERROR COUNTER OVERFLOW**

- (1) Motion board 4th-AXIS ERROR COUNTER overflow ( 16-BIT )
- (2) Check or not the commanding speed is too fast.
- (3) Check whether or not servo motor is working normally.
- (4) Check whether or not the machine is running normally.
- (5) Check whether or not the board is normal.

### **MOT 4005 : SET FIRST SOFT LIMIT ERROR**

- (1) Error setting of 1st soft limit parameter (i.e., +ive soft limit is smaller than -ive soft limit) Please check parameter numbers 1006~1013.
- (2) Clicking RESET to set new parameter.
- (3) After changing the parameter, please reboot.

**MOT 4006 : X AXIS SERVO LAG OVERFLOW**

- (1) X Axis servo lag over parameter 0002 setting value.
- (2) Check whether the setting speed is too fast or the parameter 0002 setting value is too small.
- (3) Clicking RESET to continue operating.
- (4) If reset parameter, must reboot.

**MOT 4007 : Y AXIS SERVO LAG OVERFLOW**

- (1) Y Axis servo lag over parameter 0003 setting value.
- (2) Check whether the setting speed is too fast or the parameter 0003 setting value is too small.
- (3) Clicking RESET to continue operating.
- (4) If reset parameter, must reboot.

**MOT 4008 : Z AXIS SERVO LAG OVERFLOW**

- (1) Z Axis servo lag over parameter 0004 setting value.
- (2) Check whether the setting speed is too fast or the parameter 0004 setting value is too small.
- (3) Clicking RESET to continue operating.
- (4) If reset parameter, must reboot.

**MOT 4009 : 4TH AXIS SERVO LAG OVERFLOW**

- (1) 4th Axis servo lag over parameter 0005 setting value.
- (2) Check whether the setting speed is too fast or the parameter 0005 setting value is too small.
- (3) Clicking RESET to continue operating.
- (4) If reset parameter, must reboot.

**MOT 4012 : SBK COMMAND ERROR**

- (1) Check program.

**MOT 4014 : X AXIS ON HOME DOG**

- (1) X Axis on HOME DOG.
- (2) Clicking RESET, use JOG to take the machine away from HOME DOG then will be able to return to the reference point.

**MOT 4015 : Y AXIS ON HOME DOG**

- (1) Y Axis on HOME DOG.
- (2) Clicking RESET, use JOG to take the machine away from HOME DOG then will be able to return to the reference point.

**MOT 4016 : Z AXIS ON HOME DOG**

- (1) Z Axis on HOME DOG.
- (2) Clicking RESET, use JOG to take the machine away from HOME DOG then will be able to return to the reference point.

**MOT 4017 : 4TH AXIS ON HOME DOG**

- (1) 4th Axis on HOME DOG.
- (2) Clicking RESET, use JOG to take the machine away from HOME DOG then will be able to return to the reference point.

**MOT 4018 : NO RETURN HOME**

- (1) Not return to the reference point after rebooting.
- (2) Clicking RESET and return to the reference point first.

**MOT 4023 : SET SECOND SOFT LIMIT ERROR**

- (1) Maximum parameter value of the 2<sup>nd</sup> soft limit is smaller than the smallest value.
- (2) Checking parameters 1034~1041.
- (3) Reset parameter and then reboot.

**MOT 4025 : G10 P RANGE ERROR**

- (1) P value is over range.
- (2) Please checking the part program.

**MOT 4026 : ENCODER A , B PHASE ERROR**

- (1) ENCODER disconnect or 5V power problem
- (2) Checking whether or not the ENCODER or power port is connected.

**MOT 4027 : HOME DOG TOO SHORT**

- (1) Check whether or not HOME DOG is too short or the speed is too fast.
- (2) Extend DOG or slow down the zero return speed.

**MOT 4035 : SET CMR ERROR**

- (1) Setting CMR error
- (2) Checking Para. # 0053 ~ 0056 , 0067 ~ 0070 , 0072 , 0100 ~ 0107 , 1112 ~ 1115.
- (3) Re-setting parameter and re-booting.

**MOT 4037 : SET DMR ERROR**

- (1) DMR setting error.
- (2) Checking whether or not the setting value is over setting parameter 53~57 range.
- (3) Re-setting parameter and re-booting.

**MOT 4039 : HOME LOW SPEED ERROR**

- (1) 2<sup>nd</sup> home speed setting error.
- (2) Checking whether or not the setting value is over setting parameter 1108-1111 range.
- (3) Re-setting parameter and re-booting.

**MOT 4040 : X CMP NO. ERROR**

- (1) Pitch compensation section setting error.
- (2) Checking Parameter # 0112 range.
- (3) Re-setting parameter and re-booting.

**MOT 4041 : Y CMP NO. ERROR**

- (1) Pitch compensation section setting error.
- (2) Checking Parameter # 0113 range.
- (3) Re-setting parameter and re-booting.

**MOT 4042 : Z CMP NO. ERROR**

- (1) Pitch compensation section setting error.
- (2) Checking Parameter # 0114 range.
- (3) Re-set parameter and re-boot.

**MOT 4043 : 4TH CMP NO. ERROR**

- (1) Pitch compensation section setting error.
- (2) Checking Parameter # 0115 range.
- (3) Re-setting parameter and re-booting.

**MOT 4044 : CMP INTERVAL ERROR**

- (1) Pitch compensation section setting error.
- (2) Checking Parameter # 0114 range.
- (3) Re-setting parameter and re-booting.

**MOT 4045 : NO INDEX INTERRUPT**

- (1) Reference point index disconnect signal error or HOME DOG too short.
- (2) Checking connection for motor to servo.

**MOT 4046 : RETURN HOME FAILURE**

- (1) Checking whether nor not machine lock or other machine problems.
- (2) Press RESET to clear the condition.



#### **MOT 4047 : I/O COMMUNICATION ERROR**

- (1) Checking I/O board.
- (2) Checking all connectors on I/O board.

#### **MOT 4048 : SPINDLE ORIENTATION SIGNAL NOT RELEASE**

Assumed executing machine adjustment and orientation. If initially, the spindle is positioned on the sensor, the spindle will move away automatically. After the orientation signal is off, user will be able to execute machine adjustment or orientation. However, if the spindle already rotates one cycle but the orientation signal has not been released, then this alarm message will occur.

- (1) Checking whether or not there is connection problem on the spindle orientation sensor.
- (2) Checking whether or not the spindle orientation sensor signal and type setting is correct (Parameter # 184).
- (3) Checking whether or not the spindle orientation sensor is broken.

#### **MOT 4049 : SPINDLE ORIENTATION FLAUT**

When executing spindle orientation, the spindle is NOT able to reach the correct orientation point.

- (1) Please go to DGNOS page. If the data system NO. 10 is changing, but the spindle is not rotating. Then, it means the spindle motor ENCDER signal has external distribution that makes the system misunderstood.
- (2) Please check whether or not the setting orientation rate is too high by parameter NO. 21 that makes spindle motor has missing step in the Pulse Mode.

#### **MOT 4050 : SPINDLE ORIENTATION SIGNAL NOT FOUND**

Assumed the spindle has rotated one time already but it hasn't found the orientation position sensor while executing machine adjustment or while orientation program is running. The following alarm messages will occur.

- (1) Checking whether or not there is connection problem on the spindle orientation sensor
- (2) Checking whether or not the spindle orientation sensor signal and type setting is correct (Parameter # 184).
- (3) Checking whether or not the spindle orientation sensor is broken.

#### **MOT 4051 : NO SPINDLE SPEED DEFINE**

No spindle speed definition. So please define rotary command in rigid tapping. Please key-in rotational speed command S before tapping command.

**MOT 4052 : MOVE ERR OVER LIMIT IN RIGTAP**

- (1) Please check whether or not the procedure of rigid machine adjustment is completed.
- (2) Please check whether not the setting value of Parameter 1058 is too small.

**MOT 4053 : SPD SPEED WILL OVER LIMIT**

- (1) Please check whether or not the setting spindle rotational speed is over the maximum speed that this gear spindle can handle.
- (2) Please check whether or not the setting of Parameter NO. 1060 for the spindle return accelerating speed is too big under rigid tapping. Unless necessary, recommend setting to 100.

**MOT 4054 : Z AXIS FEEDRATE WILL OVER LIMIT**

Z axis cutting feedrate speed of rigid tapping is larger than the maximum cutting speed that is set by Parameter 1004.

- (1) Checking whether or not Parameter NO. 1060 has set the spindle returning accelerating speed too large under rigid tapping. If speed starts to accelerate while returning (i.e., P1060 > 1000), then speed of cutting spindle will accelerate too. Unless necessary, recommend to set 100.

**MOT 4055 : SPD SERROLAG OVER LIMIT IN RT**

- (1) Please check whether or not ladder has error.
- (2) Please check whether or to the spindle is rotating in rigid tapping procedure. If not, please check (3) whether or not there is problem for the spindle driver setting.
- (4) Please check whether or not the spindle Encoder line is discount or fall.
- (5) Please check whether or not the setting of Parameter 1075 is too small.

**MOT 4057 : Z TRAVEL IS OVER 2 HOME LIMIT**

- (1) When Z is at second home, stop move down.

**MOT 4058 : OVER SOFTLIMIT**

- (2) RESET system.
- (3) Move axis out of limit manually.

**MOT 4059 : SP ERROR COUNTER OVERFLOW**

- (1) SP ERROR COUNTER overflow.
- (2) Check if command speed too big.
- (3) Check if driver is ok.
- (4) Check if machine is ok.
- (5) Check if board is ok.

**MOT 4062 : X-AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

After X axis re-finds home, NC will ask 4th absolute encoder to do returning zero. After complete, NC re-read encoder data to make sure it has return zero. But if the data is over the Pr.1098 setting, Alarm. (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on, during thie time, NC will follow 4<sup>th</sup> encoder to update coordinate. Later's comparison also use update coordinate.)

- (1) Check if Pr.1098 is correct or not.
- (2) Check if X axis absolute encoder completes the action of returning zero.

#### **MOT 4063 : X AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

When NC is doing X axis absolute encoder reading, if it didn't finish the reading action under Pr. 0834 setting time, Alarm

- (1) Check if servo drives set to be absolute encoder.
- (2) Check hardware cable connections.
- (3) Check if X axis Ladder correct (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on. Check if CS bit is correct.)
- (4) Check if Pr.0834 setting time is enough to complete absolute encoder reading action.

#### **MOT 4064 : X AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

X axis reading-absolute encoder has CHECK SMM error.

- (1) Check hardware cable connections.
- (2) Please check whether or not X Axis ladder has error

#### **MOT 4065 : Y-AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

After Y axis re-finds home, NC will ask 4th absolute encoder to do returning zero. After complete, NC re-read encoder data to make sure it has return zero. But if the data is over the Pr.1098 setting, Alarm. (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on, during thie time, NC will follow 4<sup>th</sup> encoder to update coordinate. Later's comparison also use update coordinate.)

- (1) Check if Pr.1098 is correct or not.
- (2) Check if Y axis absolute encoder completes the action of returning zero.

#### **MOT 4066 : Y AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

When NC is doing Y axis absolute encoder reading, if it didn't finish the reading action under Pr. 0834 setting time, Alarm.

- (1) Check if servo drives set to be absolute encoder.
- (2) Check hardware cable connections.
- (3) Check if Y axis Ladder correct (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on. Check if CS bit is correct.)
- (4) Check if Pr.0834 setting time is enough to complete absolute encoder reading action.

#### **MOT 4067 : Y AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

Y axis reading-absolute encoder has CHECK SMM error.

- (1) Check hardware cable connections.

- (2) Please check whether or not Y Axis ladder has error

#### **MOT 4068 : Z-AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

After Z axis re-finds home, NC will ask 4th absolute encoder to do returning zero. After complete, NC re-read encoder data to make sure it has return zero. But if the data is over the Pr.1098 setting, Alarm. (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on, during thie time, NC will follow 4<sup>th</sup> encoder to update coordinate. Later's comparison also use update coordinate.)

- (1) Check if Pr.1098 is correct or not.
- (2) Check if Z axis absolute encoder completes the action of returning zero.

#### **MOT 4069 : Z AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

When NC is doing Z axis absolute encoder reading, if it didn't finish the reading action under Pr. 0834 setting time, Alarm.

- (1) Check if servo drives set to be absolute encoder.
- (2) Check hardware cable connections.
- (3) Check if Z axis Ladder correct (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on. Check if CS bit is correct.)
- (4) Check if Pr.0834 setting time is enough to complete absolute encoder reading action.

#### **MOT 4070 : Z AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

Z axis reading-absolute encoder has CHECK SMM error.

- (1) Check hardware cable connections.
- (2) Please check whether or not Z Axis ladder has error

#### **MOT 4071 : 4TH-AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

After 4th axis re-finds home, NC will ask 4th absolute encoder to do returning zero. After complete, NC re-read encoder data to make sure it has return zero. But if the data is over the Pr.1098 setting, Alarm. (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on, during thie time, NC will follow 4<sup>th</sup> encoder to update coordinate. Later's comparison also use update coordinate.)

- (1) Check if Pr.1098 is correct or not.
- (2) Check if 4th axis absolute encoder completes the action of returning zero.

#### **MOT 4072 : 4TH AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

When NC is doing 4th axis absolute encoder reading, if it didn't finish the reading action under Pr. 0834 setting time, Alarm

- (1) Check if servo drives set to be absolute encoder.
- (2) Check hardware cable connections.
- (3) Check if 4th axis Ladder correct (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on. Check if CS bit is correct.)
- (4) Check if Pr.0834 setting time is enough to complete absolute encoder reading action.

### **MOT 4073 : 4TH AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

4th axis reading-absolute encoder has CHECK SMM error.

(1) Check hardware cable connection.

(2) Please check whether or not 4th Axis ladder has error.

### **MOT 4079 : 5TH AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

(1) When NC reads 5 axis ABS encoder, if not finish at P.0834 setting time, Alarm.

(2) Check servo drive is ABS setting.

(3) Check hardware wiring see if there is any problem.

(4) Check 4 axis Ladder(If you want to read MITSUBISHI ABS encoder, you will need to make servo motor servo off, wait 20ms and above time and servo on, check if corresponding CS bit is correct.)

(5) Check if P.0834 setting too fast.

### **MOT 4080 : 5th AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

(1) 5 read ABS encoder has CHECK SUM error.

(2) Check hardware wiring see if there is any problem.

(3) Check if 5 Ladder is correct.

### **MOT 4081 : 6th AXIS ABSOLUTE ENCODER BIG DIFFERENCE ERROR**

(1) When 6 axis research home, NC will ask 6 axis ABS encoder to do zero ; after completing, NC will read again ABS encoder to make sure this value is zero. If reading value and NC inner value is over P.1098 setting, Alarm. (If you want to read MITSUBISHI ABS encoder, you will need to make servo motor servo off, wait 20ms and above time and servo on. During this time, NC will update coordinate by 6 axis encoder, later will use new coordinate.)

(2) Check P.1098 setting.

(3) Check 6 axis ABS encoder if complete zero.

### **MOT 4082 : 6TH AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

When NC is doing servo axis absolute encoder reading, if it didn't finish the reading action under Pr. 0834 setting time, Alarm.

(1) Check if servo drives set to be absolute encoder.

(2) Check hardware cable connections.

(3) Check if servo axis Ladder correct (MITSUBISHI servo motor-when you want to re-read absolute encoder, you need to servo off, wait around 20ms and servo on. Check if CS bit is correct.)

(4) Check if Pr.0834 setting time is enough to complete absolute encoder reading action.

### **MOT 4083 : 6th AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

Servo axis reading-absolute encoder has CHECK SMM error.

(1) Check hardware cable connections.

(2) Please check whether or not servo axis ladder has error.

**MOT 4084 : 5 th AXIS ABSOLUTE ENCODER BIG DIFFERENCE ERROR**

- (1) When 5 axis research home, NC will ask 6 axis ABS encoder to do zero ; after completing, NC will read again ABS encoder to make sure this value is zero. If reading value and NC inner value is over P.1098 setting, Alarm. (If you want to read MITSUBISHI ABS encoder, you will need to make servo motor servo off, wait 20ms and above time and servo on. During this time, NC will update coordinate by 6 axis encoder, later will use new coordinate.)
- (2) Check P.1098 setting.
- (3) Check 5 axis ABS encoder if complete zero.

**MOT 4087 : Y MOTOR ENCODER WIRING ERROR**

- (1) ENCODER brake or 5V power error.
- (2) Stop machine and check wiring or connector.

**MOT 4090 : X AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 416**

X servo lag too big when stop, check wiring or P.416 setting too small.

**MOT 4091 : Y AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 417**

Y servo lag too big when stop, check wiring or P.417 setting too small.

**MOT 4092 : Z AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 418**

Z servo lag too big when stop, check wiring or P.418 setting too small.

**MOT 4093 : 4TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 419**

4 servo lag too big when stop, check wiring or P.419 setting too small.

**MOT 4094 : 5TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 420**

5 servo lag too big when stop, check wiring or P.420 setting too small.

**MOT 4095 : 6TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 421**

6 servo lag too big when stop, check wiring or P.421 setting too small.

**MOT 4096 : Z MOTOR ENCODER WIRING ERROR**

- (1) ENCODER BRAKE OR 5V POWER ERROR.
- (2) Stop machine and check wiring or connector.

**MOT 4097 : 4 MOTOR ENCODER WIRING ERROR**

- (1) ENCODER BRAKE OR 5V POWER ERROR.
- (2) Stop machine and check wiring or connector.

**MOT 4098 : 5 MOTOR ENCODER WIRING ERROR**

- (1) ENCODER BRAKE OR 5V POWER ERROR.
- (2) Stop machine and check wiring or connector.

**MOT 4099 : 6 MOTOR ENCODER WIRING ERROR**

- (1) ENCODER BRAKE OR 5V POWER ERROR.
- (2) Stop machine and check wiring or connector.

**MOT 4100 : COM REPEAT CHECK PR.1810 AND 1811**

- (1) Check if P.1810 & 1811 is the same.
- (2) Press RESET to clear alarm.

**MOT 4104 : X PMC/INT OUTPUT ERROR**
**MOT 4105 : Y PMC/INT OUTPUT ERROR**
**MOT 4106 : Z PMC/INT OUTPUT ERROR**
**MOT 4107 : 4 PMC/INT OUTPUT ERROR**
**MOT 4108 : 5 PMC/INT OUTPUT ERROR**
**MOT 4109 : 6 PMC/INT OUTPUT ERROR**

MOT4104~MOT4109: modify command to make it ok for PMC axis.

**MOT 4121 : X AXIS COMMANDED UNDER DETACHED**

- (1) RESET SYSTEM TO CLEAR ALARM.
- (2) Check if PLC give axis C bit 130 and clear.

**MOT 4122 : Y AXIS COMMANDED UNDER DETACHED**

- (1) Check if PLC give axis C bit 131 and clear.

**MOT 4123 : Z AXIS COMMANDED UNDER DETACHED**

- (1) RESET SYSTEM TO CLEAR ALARM.
- (2) Check if PLC give axis C bit 132 and clear.

**MOT 4124 : 4th AXIS COMMANDED UNDER DETACHED**

- (1) RESET SYSTEM TO CLEAR ALARM.
- (2) Check if PLC give axis C bit 133 and clear.

**MOT 4126 : MOVE COMMAND NOT RESTART**

Under MDI, you can not give move command at program restart.

**MOT 4127 : 5th AXIS COMMANDED UNDER DETACHED**

- (1) RESET SYSTEM TO CLEAR ALARM.
- (2) Check if PLC give axis C bit 168 and release.

**MOT 4128 : 6th AXIS COMMANDED UNDER DETACHED**

- (1) RESET SYSTEM TO CLEAR ALARM.
- (2) Check if PLC give axis C bit 169 and release.

**MOT 4129 : LIMIT AND INDEX HAS CERTAIN SAFE DISTANCE, USE GRID TO PROTECT**

- (1) The distance from index to limit sensor is smaller than encoder 1/5 or bigger than 4/5 revolution, then open grid protection function, if no, then close grid protection function.
- (2) If confirming that the distance is smaller than 1/5 or bigger than 4/5 revolution, but this alarm still exist, check if home limit sensor position error.

**MOT 4130 : X AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) X multi-home linear scale home interval setting error.(P.1171 、 1172 small or equal to 0.

- (2) Refer to linear scale manual and system P.1171 、 1172, reset.

**MOT 4131 : Y AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) Y multi-home linear scale home interval setting error.(P.1173 、 1174 small or equal to 0.  
(2) Refer to linear scale manual and system P.1173 、 1174, reset.

**MOT 4132 : Z AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) Z multi-home linear scale home interval setting error.(P.1175 、 1176 small or equal to 0.  
(2) Refer to linear scale manual and system P.1175 、 1176, reset.

**MOT 4133 : 4TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) 第 4 multi-home linear scale home interval setting error.(P.1177 、 1178 small or equal to 0.  
(2) Refer to linear scale manual and system P.1177 、 1178, reset.

**MOT 4134 : 5TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) 第 5 multi-home linear scale home interval setting error.(P.1179 、 1180 small or equal to 0.  
(2) Refer to linear scale manual and system P.1179 、 1180, reset.

**MOT 4135 : 6TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

- (1) 第 6 multi-home linear scale home interval setting error.(P.1181 、 1182 small or equal to 0.  
(2) Refer to linear scale manual and system P.1181 、 1182, reset.

**MOT 4136 : 5 AXIS PITCH COMPENSATION SEGMENT ERROR**

- (1) Pitch compensation segment errors.  
(2) Check P.0280  
(1) Reset parameter and reboot.

**MOT 4137 : 6 AXIS PITCH COMPENSATION SEGMENT ERROR**

- (2) Pitch compensation segment errors.  
(3) Check P.0281  
(4) Reset parameter and reboot.

**MOT 4138 : FOLLOW SP NAME ERROR**

Follow master axis number setting error, check if P. 500~505 setting error.

**MOT 4139 : ANALOG VOLTAGE CHECKING CARD DO NOT EXIST**

This controller did not have DAQ3718 analog voltage check card, make this to be 0, no use.

**MOT 4140 : ANALOG VOLTAGE CHECKING MAX AND MIN SETTING ERROR**

Maximum setting can not smaller than minimum setting.

**MOT 4141 : EXECUTE RETURN HOME WHEN AT PREPARE COMPLETE**

Under NC-ready can return home.



**MOT 4142 : STOP INT M CODE, M CODE REPEAT, CHECK PR.211~220**

Check P.211~220 if using system M code (EX:M0, M1, M2, M30, M98, M99)

**MOT 4143 : X AXIS ABS ENCODER COMMUNICATION ERROR****MOT 4144 : Y AXIS ABS ENCODER COMMUNICATION ERROR****MOT 4145 : Z AXIS ABS ENCODER COMMUNICATION ERROR****MOT 4146 : 4TH AXIS ABS ENCODER COMMUNICATION ERROR****MOT 4147 : 5TH AXIS ABS ENCODER COMMUNICATION ERROR****MOT 4148 : 6TH AXIS ABS ENCODER COMMUNICATION ERROR**

MOT4143~MOT4148:

- (1) Read this axis ABS encoder fail.
- (2) Check serial communication wiring.
- (3) Check if servo motor serial communication setting correct.
- (4) If PLC editing correct.
- (5) Press RESET to clear alarm.

**MOT 4149 : X AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.845**

RS-422ABS encoder can not use at rotating axis, check P.845 if set up X to be rotating axis.

**MOT 4150 : Y AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.846**

RS-422ABS encoder can not use at rotating axis, check P.846 if set up Y to be rotating axis.

**MOT 4151 : Z AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.847**

RS-422ABS encoder can not use at rotating axis, check P.847 if set up Z to be rotating axis.

**MOT 4152 : 4TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.66**

RS-422ABS encoder can not use at rotating axis, check P.66 if set up 4 axis to be rotating axis.

**MOT 4153 : 5TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.260**

RS-422ABS encoder can not use at rotating axis, check P.260 if set up 5 axis to be rotating axis.

**MOT 4154 : 6TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.261**

RS-422ABS encoder can not use at rotating axis, check P.261 if set up 6 axis to be rotating axis.

**MOT 4155 : PMC G00 ACC/DEC(PR.398)ERROR**

Check P.398setting error.

**MOT 4156 : PMC G01 ACC/DEC (PR.399) ERROR**

Check P.399setting error.

**MOT 4157 : PMC G00 POSITION CHECK RANGE(PR.1831~1836)ERROR**

Check P.1831~1836setting error.

**MOT 4158 : PMC G01 POSITION CHECK RANGE (PR.1837~1842) ERROR**

Check P.1837~1842setting error.

**MOT 4159 : 1 SOFTWARE PROHIBIT ERROR**

- (1) P.1900~19111 soft limit protection (-)value do not bigger than (+)value.
- (2) Modify this parameter, press RESET.

**MOT 4160 : 2 SOFTWARE PROHIBIT ERROR**

- (1) P.1912~19231 soft limit protection (-)value do not bigger than (+)value.
- (2) Modify this parameter, press RESET.

**MOT 4161 : I/O CARD COMMUNICATION ERROR (RIO2)**

- (1) Check I/O board.
- (2) Check I/O all wiring.
- (3) Check PLCIO.CFGsetting error.

MOT 4162

**Mill V3 : OUT OF X AXIS HOME DOG, CHECK PR.1890 OR DOG SIGNAL**

Check P.1890 if set up too small or check X home dog if fail or error.

**Mill V6 : IO COMMUNICATION CABLE BRAKE**

RS232 communication brake, check wiring and reboot.

**MOT 4163****Mill V3 : OUT OF Y AXIS HOME DOG, CHECK PR.1891 OR DOG SIGNAL**

Check P.1891 if set up too small or check Y home dog if fail or error.

**Mill V6 : OUT OF X AXIS HOME DOG, CHECK PR.1890 OR DOG SIGNAL**

Check P.1890 if set up too small or check X home dog if fail or error.

**MOT 4164****Mill V3 : OUT OF Z AXIS HOME DOG, CHECK PR.1892 OR DOG SIGNAL**

Check P.1892 if set up too small or check Z home dog if fail or error.

**Mill V6 : OUT OF Y AXIS HOME DOG, CHECK PR.1891 OR DOG SIGNAL**

Check P.1891 if set up too small or check Y home dog if fail or error.

**MOT 4165****Mill V3 : OUT OF C AXIS HOME DOG, CHECK PR.1893 OR DOG SIGNAL**

Check P.1893 if set up too small or check 4 home dog if fail or error.

**Mill V6 : OUT OF Z AXIS HOME DOG, CHECK PR.1892 OR DOG SIGNAL**

Check P.1892 if set up too small or check Z home dog if fail or error.

**MOT 4166**

**Mill V3** : OUT OF 5TH AXIS HOME DOG, CHECK PR.1894 OR DOG SIGNAL

Check P.1894 if set up too small or check 5 home dog if fail or error.

**Mill V6** : OUT OF C AXIS HOME DOG, CHECK PR.1893 OR DOG SIGNAL

Check P.1893 if set up too small or check 4 home dog if fail or error.

**MOT 4167**

**Mill V3** : OUT OF 6TH AXIS HOME DOG, CHECK PR.1895 OR DOG SIGNAL

Check P.1895 if set up too small or check 6 home dog if fail or error.

**Mill V6** : OUT OF 5TH AXIS HOME DOG, CHECK PR.1894 OR DOG SIGNAL

Check P.1894 if set up too small or check 5 home dog if fail or error.

**MOT 4168**

**Mill V3** : TWO SETS OF INNER M CODER CAN NOT EXIT TOGETHER

Check there are 2 or more CNC inner M code in program, include : M00 、 M01 、 M02 、 M30 、 M98 、 M99.

**Mill V6** : OUT OF 6TH AXIS HOME DOG, CHECK PR.1895 OR DOG SIGNAL

Check P.1895 if set up too small or check 6 home dog if fail or error.

**MOT 4169** : X AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR. 1980 TOO SMALL

**MOT 4170** : Y AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1981 TOO SMALL

**MOT 4171** : Z AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR. 1982 TOO SMALL

**MOT 4172** : 4TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1983 TOO SMALL

**MOT 4173** : 5TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1984 TOO SMALL

**MOT 4174** : 6TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1985 TOO SMALL

**MOT 4169~MOT4174**

- (1) Check if feedback signal was interfered.
- (2) Check if have machine interference
- (3) Reset corresponding 2nd encoder feedback exam tolerance Pr.1980~1985 value, and press reset to continue.
- (4) Enlarge Pr. .1980~1985, press reset, clear alarm, press ZRN to make value to be 0.
- (5) Press reboot to make value to be 0.

**MOT 4175** : X AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1986 TOO SMALL

**MOT 4176 : Y AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1987 TOO SMALL**

**MOT 4177 : Z AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1988 TOO SMALL**

**MOT 4178 : 4TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1989 TOO SMALL**

**MOT 4179 : 5TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1990 TOO SMALL**

**MOT 4180 : 6TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1991 TOO SMALL**

MOT 4175~MOT 4180

- (1) Reboot system.
- (2) Make Pr.1960~1965 bigger, press reset
- (3) Check if have machine interference.

**MOT 4181 : IO COMMUNICATION BRAKE**

RS232 communication brake, check wiring and reboot.

**MOT 4182 : SP POSITIONING ERROR, CHECK P.184 OR WIRING**

- (1) Check P.184 setting.
- (2) Check SP positioning signal wiring.

**MOT 4190 : PROGRAM RESTART, BUT M02.M30/M99 AND END.**

Program restart, but there is M02/M30, check if confirming lines too much.

**MOT 4191 : DO NOT SUPPORT RIGID TAPPING**

Mill V6 did not support rigid tapping, check if there is M29.

**MOT 4202 : 5 ERROR COUNTER OVERFLOW**

- (1) 5 axis ERROR COUNTER overflow ( 16-BIT )
- (2) Check if command speed is too big.
- (3) Check if servo driver is ok.
- (4) Check if machine is moving ok.
- (5) Check if board is ok.

**MOT 4203 : 6 ERROR COUNTER OVERFLOW**

- (1) 6 axis ERROR COUNTER overflow ( 16-BIT )
- (2) Check if command speed is too big.
- (3) Check if servo driver is ok.
- (4) Check if machine is moving ok.
- (5) Check if board is ok.

**MOT 4204 : 5 SERVO LAG OVER P.250**

- (1) 5 axis servo lag is over P.0250.
- (2) Check if speed too big or P.0250 too small.
- (3) Press RESET to continue operation.

(4) If reset parameter, need to reboot.

**MOT 4205 : 6 SERVO LAG OVER P.251**

- (1) 6 axis servo lag is over P.0251.
- (2) Check if speed too big or P.0251 too small.
- (3) Press RESET to continue operation.
- (4) If reset parameter, need to reboot.

**MOT 4206 : 5 AXIS IS ON HOME DOG**

- (1) 5 axis is on HOME DOG.
- (2) Press RESET and use JOG to move axis out of HOME DOG.

**MOT 4207 : 6 AXIS IS ON HOME DOG**

- (1) 6 axis is on HOME DOG.
- (2) Press RESET and use JOG to move axis out of HOME DOG.

**MOT 4208 : SERVO AXIS NO. REPEAT**

Check P.24~27, 256~257 is double setting, revise and reboot to clear alarm. If only using reset to clear this alarm, then input port is invalid.

**MOT 4209 : 2ND FEEDBACK EXAM INPUT PORT REPEAT, CHECK PR.1950~1955**

- (1) Check if Pr.1950~1955 setting repeats. if so, modify and reboot to clear alarm. If only using reset to clear this alarm, this input port will be invalid.

**MOT 4210 : M-II INITIAL FAIL**

Check MII axis setting or controller MII hardware error.

**MOT 4211 : M-II INTERRUPT FAIL**

Check MII axis setting or controller MII hardware error.

**MOT 4212 : M-II X Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C281 OFF to trigger X motor.
- (2) Press RESET to clear alarm.

**MOT 4213 : M-II Y Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C282 OFF to trigger Y motor.
- (2) Press RESET to clear alarm.

**MOT 4214 : M-II Z Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C283 OFF to trigger Zmotor.
- (2) Press RESET to clear alarm.

**MOT 4215 : M-II C Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C284 OFF to trigger Cmotor.
- (2) Press RESET to clear alarm.

**MOT 4216 : M-II 5 Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C285 OFF to trigger 5 axis motor.
- (2) Press RESET to clear alarm.

**MOT 4217 : M-II 6 Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C286 OFF to trigger 6 axis motor.
- (2) Press RESET to clear alarm.

**MOT 4218 : M-II 1 SP Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C287 OFF to trigger 1 SP.
- (2) Press RESET to clear alarm.

**MOT 4219 : M-II 2 SP Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C288 OFF to trigger 2 SP.
- (2) Press RESET to clear alarm.

**MOT 4220 : M-II 3 SP Servo On/Off COMMAND & STATUS NOT THE SAME.**

- (1) Re-ON C289 OFF to trigger 2 SP.
- (2) Press RESET to clear alarm.

**MOT 4221 : M-II COMMUNICATION READ INFO FAIL**

- (1) M-II communication cable brake or disturbed, Re-ON communication cable.
- (2) Press RESET to clear alarm.

**MOT 4222 : M-II COMMUNICATION WRITE INFO FAIL**

- (1) M-II communication cable brake or disturbed, Re-ON communication cable.
- (2) Press RESET to clear alarm.

**MOT 4300 : 1 SP COMMUNICATION ERROR**

Check hardware wiring, BIOS parameter, controller and inverter parameter, press RESET to make it reconnect and clear alarm.

**MOT 4301 ~ MOT 4340 : ALARM**

- (1) This is inverter alarm, refer to inverter manual.
- (2) Press RESET to clear.

**MOT 4341 : 1 SP POLES SETTING CONFLICT, CHECK SYSTEM P.681& INVERTER P.4-04**

- (1) Check controller P.681 & inverter P.4-04, check if both setting are the same.
- (2) After setting, press RESET to make parameter valid and clear alarm.

**MOT 4342 : 1 SP FREQUENCY GAIN SETTING CONFLICT, CHECK SYSTEM P.685& INVERTER P.0-23**

- (1) Check controller P.685 & inverter P.0-23 , check if both setting are reasonable.
- (2) After setting, press RESET to make parameter valid and clear alarm.

**MOT 4343 : 1 SP MAX. FREQUENCY SETTING CONFLICT, CHECK SYSTEM P.682& INVERTER P.1-00**

- (1) Check controller P.682 & inverter P.1-00 , check if both setting are reasonable.
- (2) After setting, press RESET to make parameter valid and clear alarm.

**MOT 4344 : 1 SP UP LIMIT OUTPUT FREQUENCY SETTING CONFLICT, CHECK SYSTEM P.683& INVERTER P.1-21**

- (1) Check controller P.683 & inverter P.1-21 , check if both setting are reasonable.
- (2) After setting, press RESET to make parameter valid and clear alarm.

**MOT 4345 : 1 SP DOWN LIMIT OUTPUT FREQUENCY SETTING CONFLICT, CHECK SYSTEM P.684& INVERTER P.1-22**

- (3) Check controller P.684 & inverter P.1-22 , check if both setting are reasonable.
- (4) After setting, press RESET to make parameter valid and clear alarm.

MOT 4950 : SYSTEM ALARM

Contact LNC.

**MOT 4999 : SP POSITION DIFFERENCE ANGLE OVER SETTING**

- (1) Make P.1999 bigger.
- (2) Reboot system.
- (3) Contact LNC.

## 5.5 MOT

**MOT 9001 : X AXIS OVER SOFTLIMIT (+)**

**MOT 9002 : X AXIS OVER SOFTLIMIT (-)**

**MOT 9003 : Y AXIS OVER SOFTLIMIT (+)**

**MOT 9004 : Y AXIS OVER SOFTLIMIT (-)**

**MOT 9005 : Z AXIS OVER SOFTLIMIT (+)**

**MOT 9006 : Z AXIS OVER SOFTLIMIT (-)**

**MOT 9007 : 4TH AXIS OVER SOFTLIMIT (+)**

**MOT 9008 : 4TH AXIS OVER SOFTLIMIT (-)**

MOT9001~MOT9008: Move this warning servo axis back to safe area, it can clear warning

**MOT 9009 : X AXIS OVER G22 SOFTLIMIT (+)**

**MOT 9010 : X AXIS OVER G22 SOFTLIMIT (-)**

**MOT 9011 : Y AXIS OVER G22 SOFTLIMIT (+)**

**MOT 9012 : Y AXIS OVER G22 SOFTLIMIT (-)**

**MOT 9013 : Z AXIS OVER G22 SOFTLIMIT (+)**

**MOT 9014 : Z AXIS OVER G22 SOFTLIMIT (-)**

MOT9009~MOT9014: Move this warning servo axis back to safe area that G22 has set up , it can clear warning  
Check G22 each axis travel limit and see if need to be adjusted, or modify program command location.

**MOT 9016 : 5 OVER (-) SOFTWARE LIMIT**

**MOT 9017 : 6 OVER (+) SOFTWARE LIMIT**

**MOT 9018 : 6 OVER (-) SOFTWARE LIMIT**

**MOT 9019 : 5 OVER (+) SOFTWARE LIMIT**

MOT9016~MOT9019: Move this warning servo axis back to safe area, it can clear warning

**MOT 4801~MOT 4876 : AS ALARM SHOW**

**MOT 9033~MOT 9051 : AS ALARM SHOW**

- (1) This is M-II drive alarm and warning
- (2) Refer to M-II drive manual to clear this alarm/warning.
- (3) Reboot.