# LNC-M515i

# Operator's Manual

2010/8 Ver.：V04.00.000(4408210085)

*Leading Numerical Controller*

LNC Technology Co., Ltd.

# Table of contents

# 1 CNC OPERATION

## 1.1 Operation Devices Type

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

## 1.2 Display Device

You will three types of function keys at the LCD display device：

(1) **Main Function Keys：**

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

(2) **Sub-Function Keys：**

After choosing the main function button, there will be 5 vertical type buttons at the left side of the monitor. It will show the content of sub-function. Press the corresponding function button to select the wanted function.

(3) **Additional function keys：**

CF card support.


Figure 1.2-1 LCD Monitor

According to different functions, there are 2 kinds of keys at the OP：

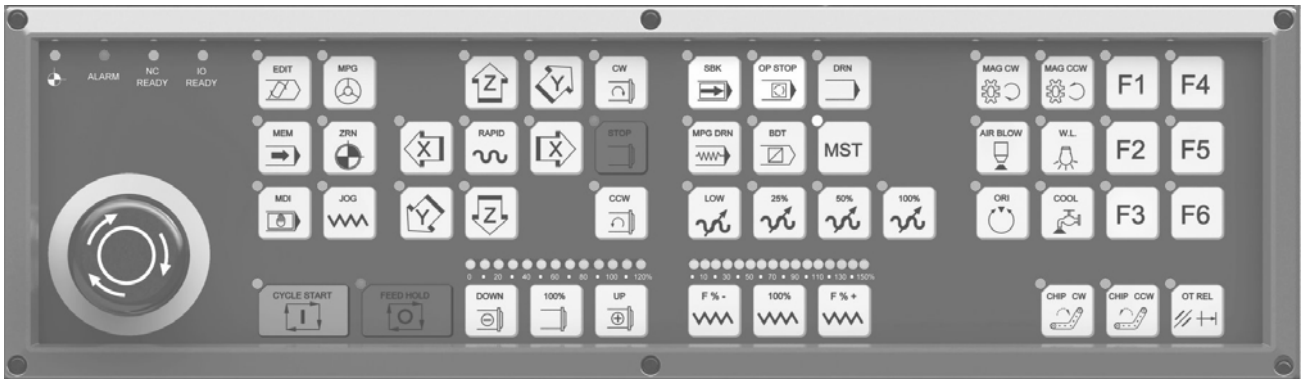
(1) **MDI：**

To Input functions into machine controller.


MDI

(2) **OP：**

To Input functions into machine controller.




OP

(3) **MDI：**

**A. CNC Group key：**

| Name | Group Key | Description |
|---|---|---|
| **<POS>** | POS | 1、 To display positions. |
| **<PROG>** | PROG | 1、 To edit and to display program ( With 1 EDIT mode 2. MEM mode 3. MDI mode ). |
| **<OFFSET>** | OFFSET | 1、 To set and to display tool offsest. |
| **<GRAPH>** | GRAPH | 1、 To draw tool path. |
| **<DGNOS>** | DGNOS | 1、 To display instance messge at DGNOS page. |
| **<PARAM>** | PARAM | 1、 To display parameter screen. |

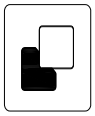
**B. Character & Symbol Keys：**

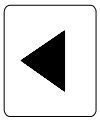
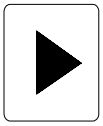
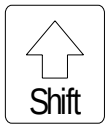
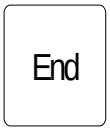
These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.

| Name | Help Key | Description |
|---|---|---|
| **Character** | X A ～ Z B | 1、 Character, from A to Z, total 26 buttons for position command or argument command.. |
| **Numbers** | 0 = ～ 9 ? | 1、 Numbers, 0~9, total 10 keys for value inputing.. |
| **Symbol** | / | 1、 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 | / / Reset / / | 1. To reset system to return default.<br>2. When alarm removing, cancel alarm.<br>3. When start working, to cancel working.<br>4. At edit mode, cursor returns to program head. |
| PAGE↑ |  | 1、 To turn to the previous page. |

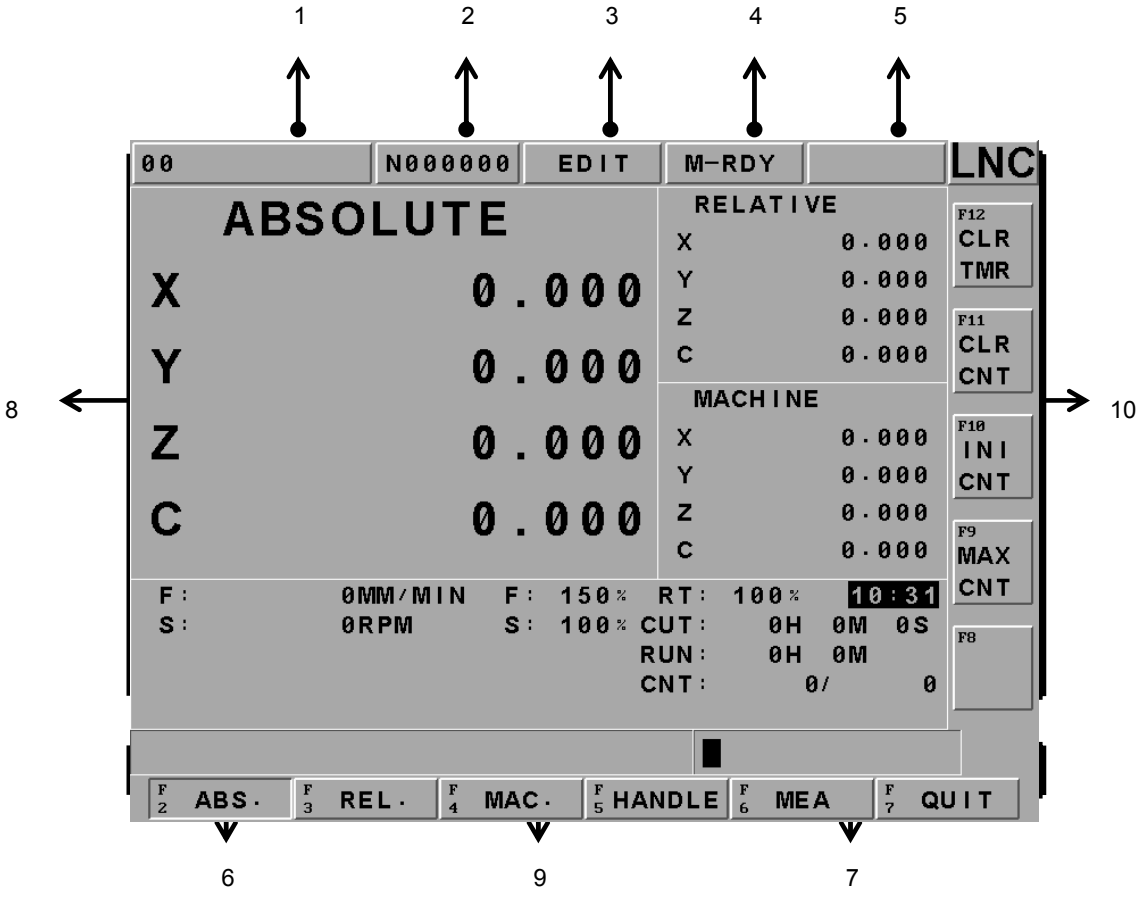
| Name | Help Key | Description |
|---|---|---|
| PAGE↓ |  | To turn to the next page. |
| Enter |  | 1. Input at the data input area, press enter to confirm. The column will be filled with the data.<br>2. At file management, move cursor to the file location, press enter to open the file.<br>3. At edit mode, press enter to insert space line. |
| ↑ |  | Under edit mode, move cursor upper.<br>To move cursor upper. |
| ↓ |  | ● Under edit mode, move cursor down.<br>● To move cursor down. |
| ← |  | ■ Under edit mode, move cursor left.<br>■ To move cursor left. |
| → |  | 2、 Under edit mode, move cursor right.<br>3、 To move cursor right. |
| Space |  | 2、 To key-in into empty space. |
| Shift |  | 2、 To key-in special symbols with use of symbol number keys.<br>Note: you need to press SHIFT and symbol buttons at the same time. |
| Home |  | 2、 To return cursor to the beginning of the sentence when editing program. |
| End |  | 2、 To return cursor to the end of the sentence when editing program. |
| CAN |  | 1、 Cancel before words. |

| Name | Help Key | Description |
|---|---|---|
| ESC | Esc | 1、 Return back<br>Note: this function will be valid under BIOS mode. |

## 1.3 Screen and Function Description

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

1.3.1 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 mode 2.auto mode 3.manual mode 4.MPG 5.quick locate 6.continuous jog 7.return reference point

**4：Machine condition signal**

Note: 1.Preparation unfinished 2.preparation complete 3.start to work 4.machine pause 5.block 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**

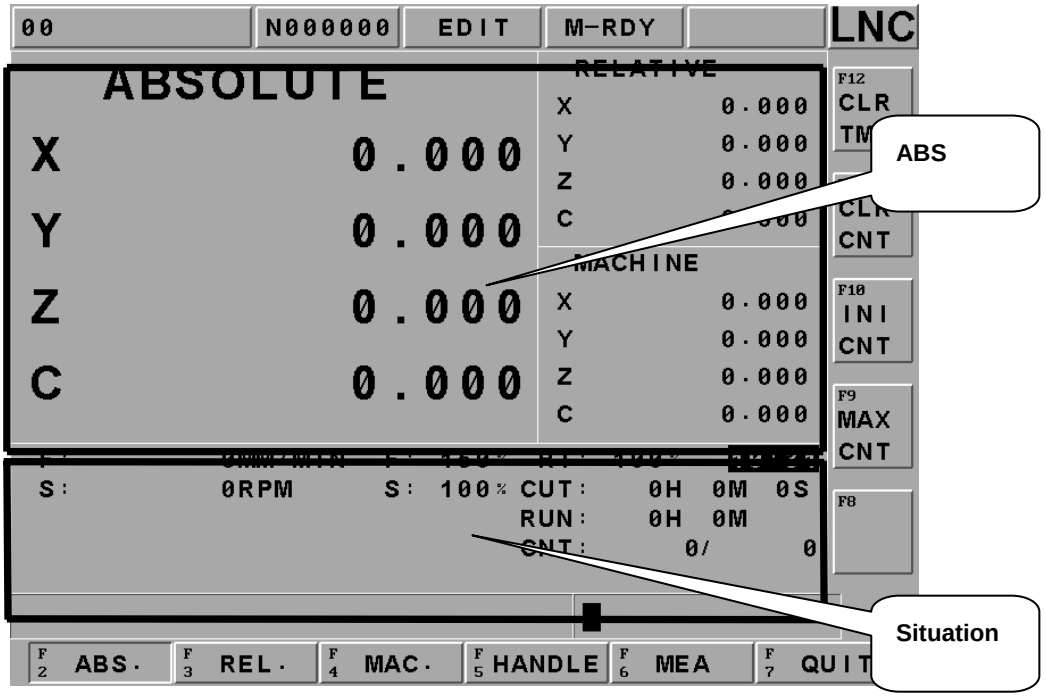
Note: This area is for main function buttons to use.

**10：Sub-function button display**

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

## 1.4 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 is displayed in those screens, which will be described separately as following：




POS Function Display Screen

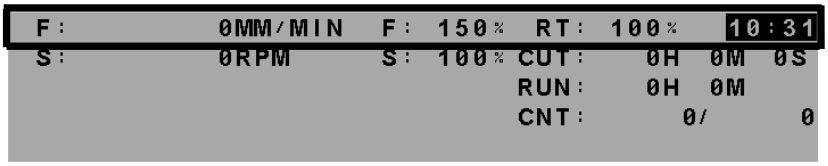
**Actual Feed Rate**

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 federate 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%>.

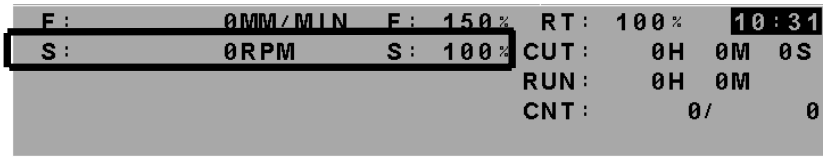
Time：Current time.



**Rotation situation**

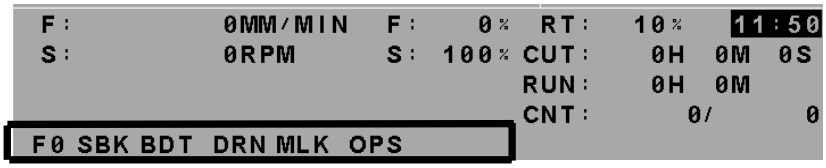
Rotational speed (rpm/min)：Current spindle actual rotation. Unit is rpm/min.

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




**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）

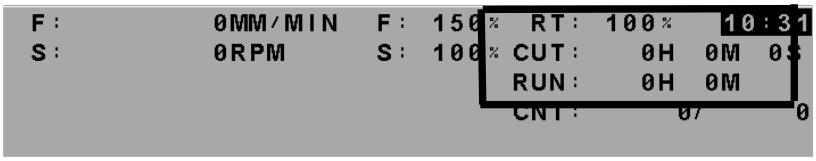



**Situation of CUT, RUN, CNT**

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.

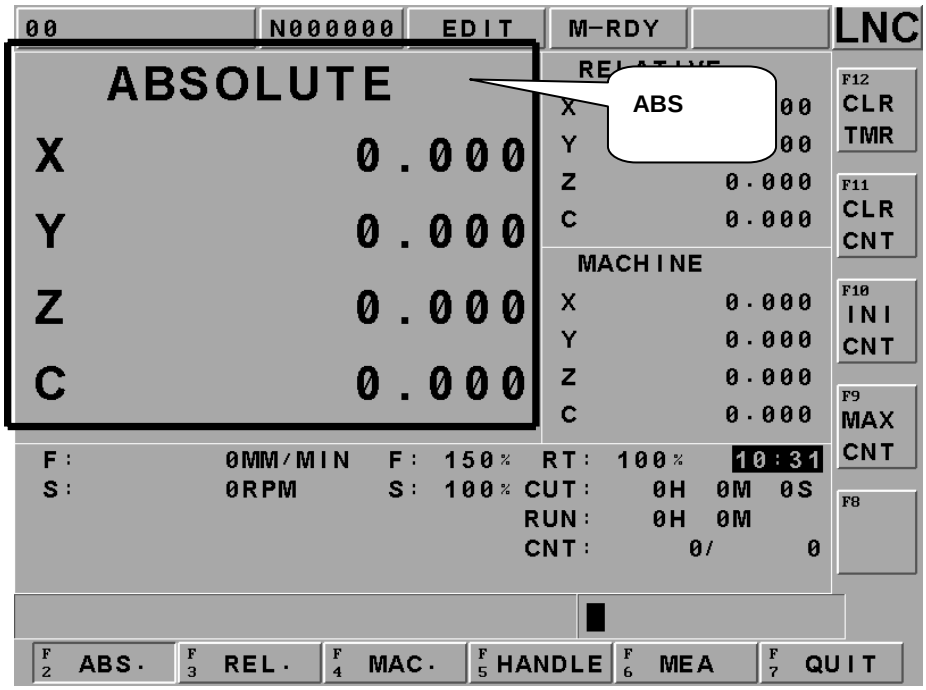
RUN(H"M)：Running time is the total cutting time after every booting, until the system shuts down, the time will be reset to be zero. You can also use【TIME CLEAR】 to make it to be zero.

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.




**【ABSOLUTE COORDINATE】**

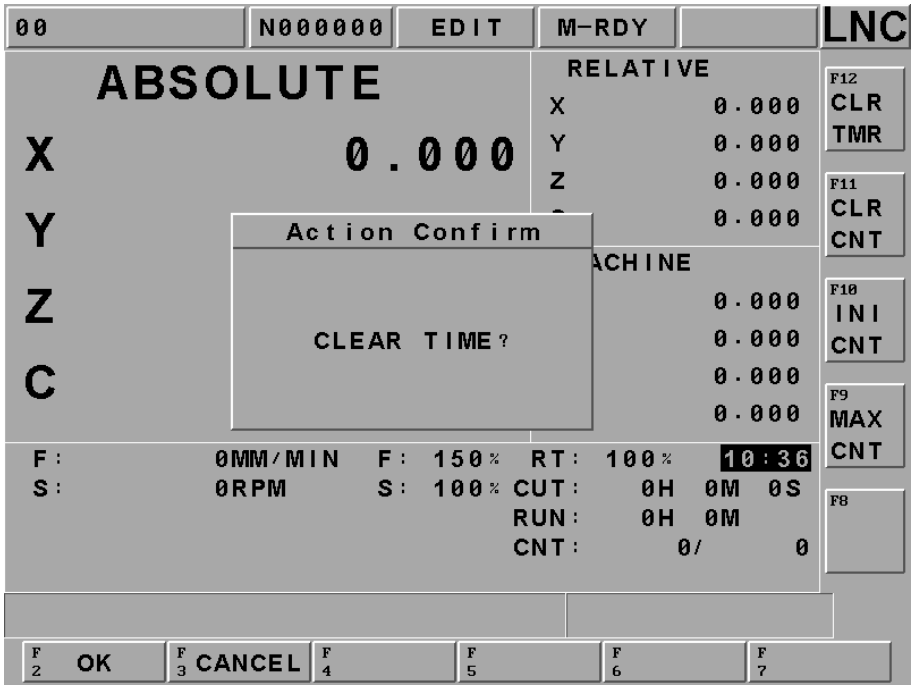
Pressing 【**ABS.**】 to enter into absolute coordinate screen. Details will be shown in bigger words at the upper left side of the screen. Using the program home (means the workpiece coordinate) to show tool's current location, the absolute coordinate will be changed according to the movement of tool.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




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

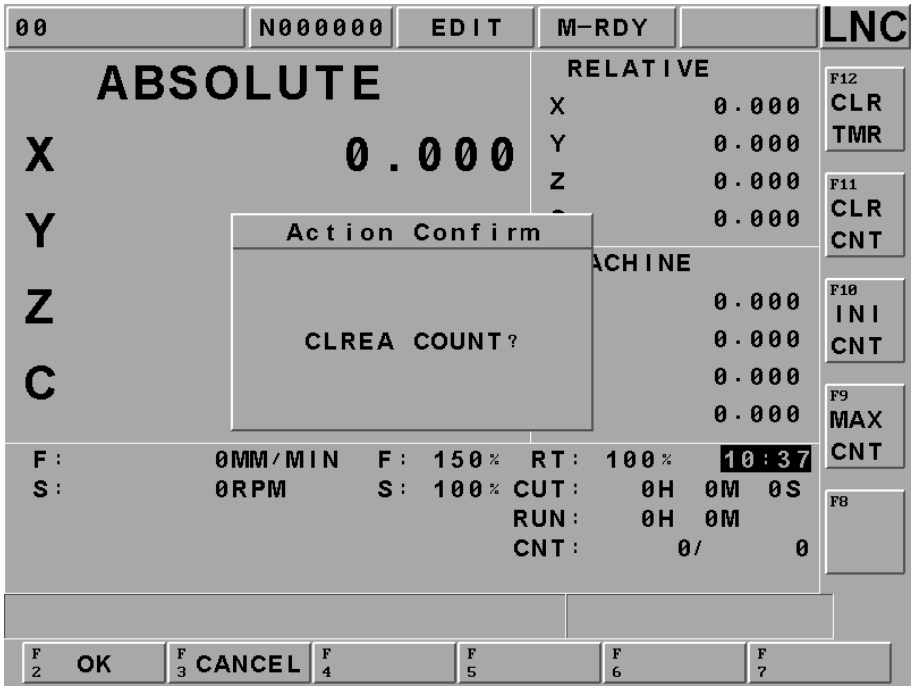
**【Time Clear】**

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



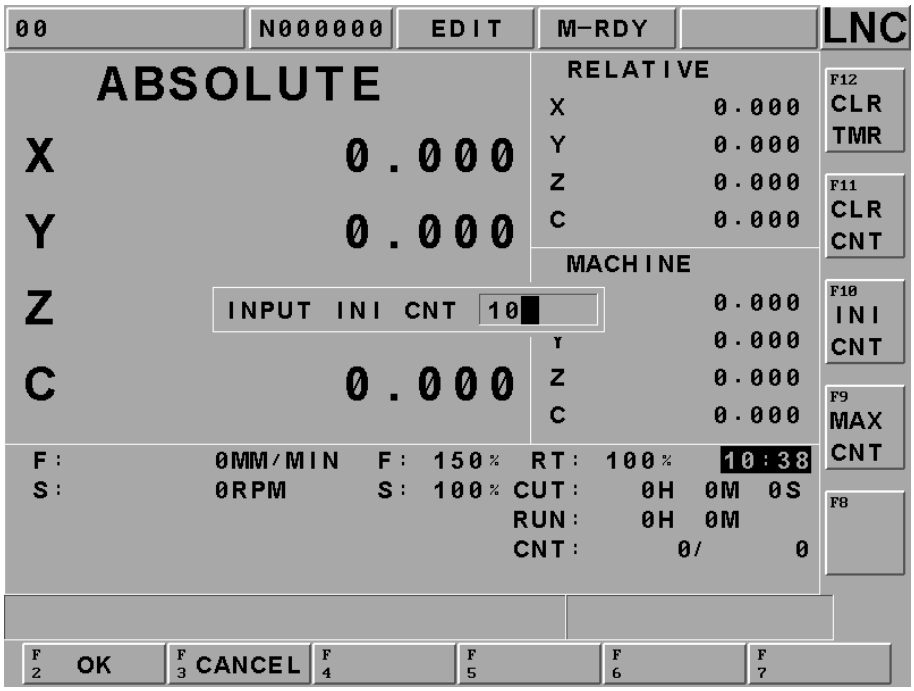
**【Workpiece Clear】**

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



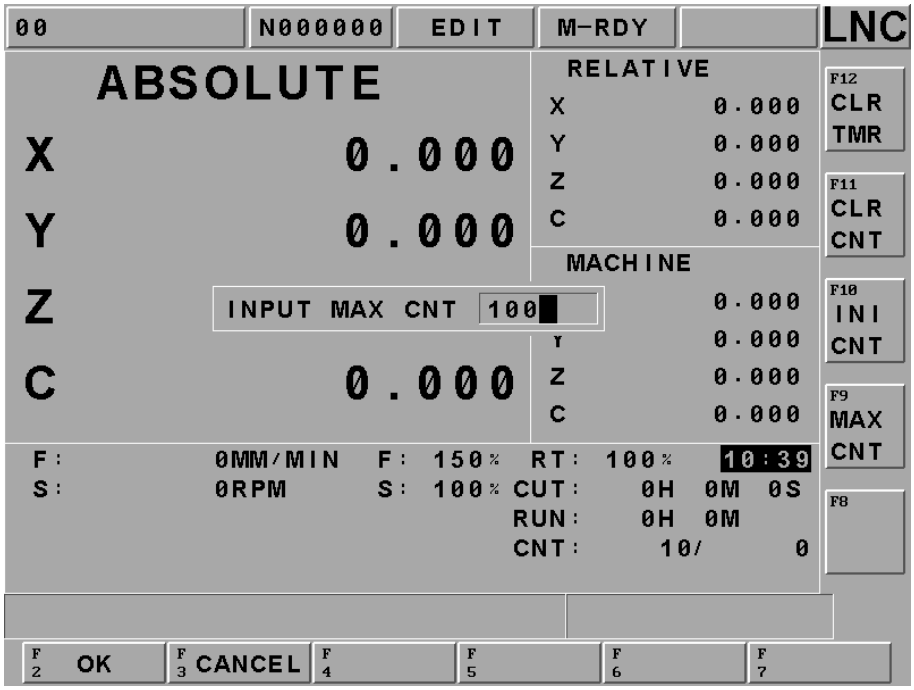

**【Initial Workpiece】**

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



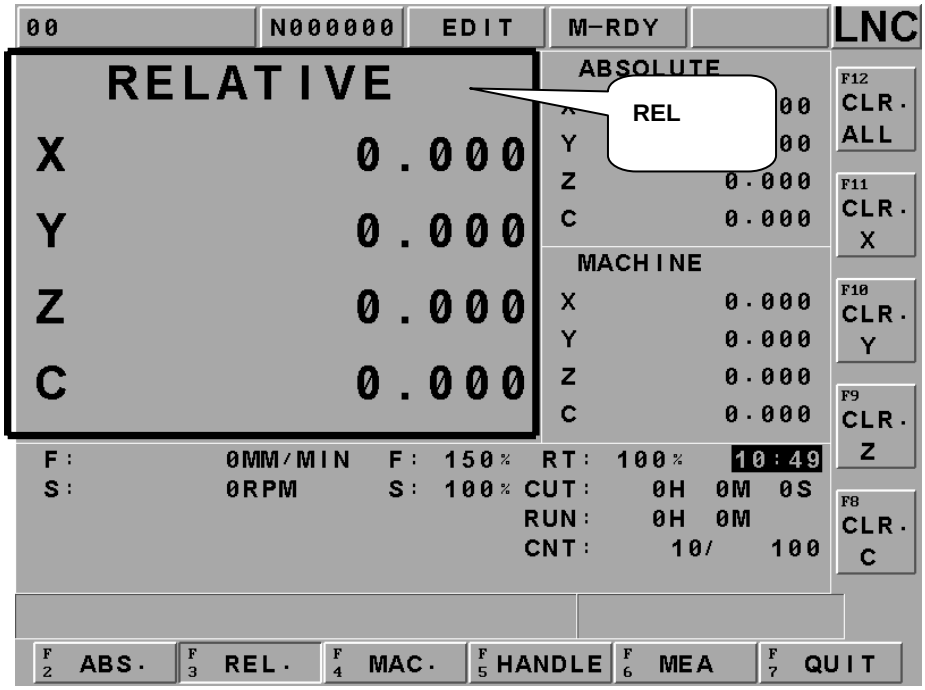
**【Max Workpiece】**

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.




**【RELATIVE COORDINATE】**

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：

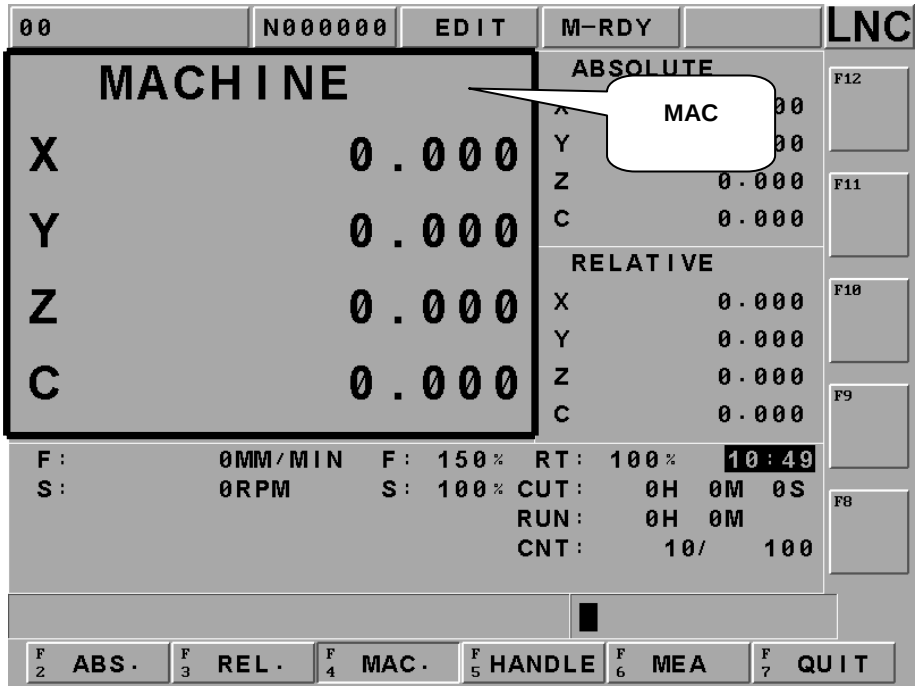
If want to set the 3 axes relative coordinate values to zero separately or at the same time, users only need to press the corresponding sub-function buttons **【CLR. X】**, **【CLR. Y】**, **【CLR. Z】**, **【CLR. ALL】** to execute the corresponding clean.

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.

**【MACHINERY COORDINATE】**

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

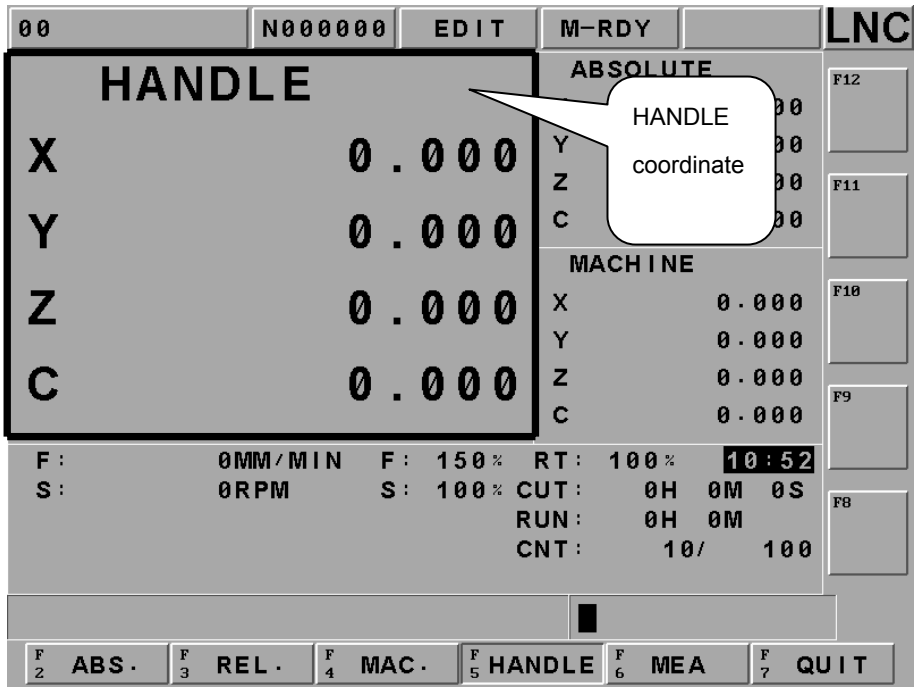



Machine coordinate is the distance of the present position corresponding to the reference point

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

**【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.

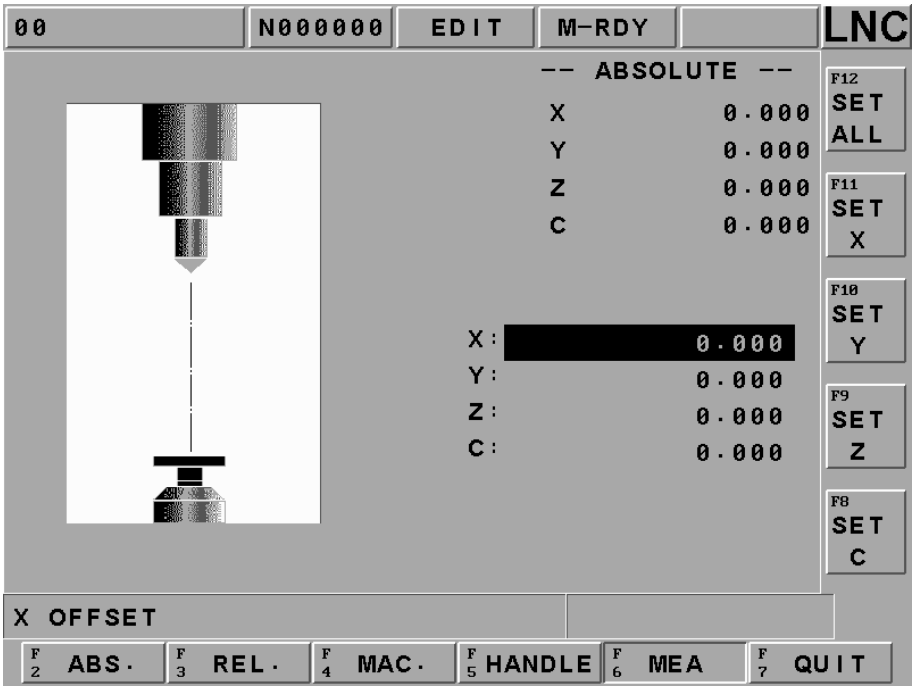



**Note：**Because the absolute coordinate will not be changed by MPG interruption, therefore the tool path later will have certain offset with original program path. This offset need to be modified by manual return zero to clear. And about the offset value, you can refer to MPG coordinate.

**【MEA】**(Measure)

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:**

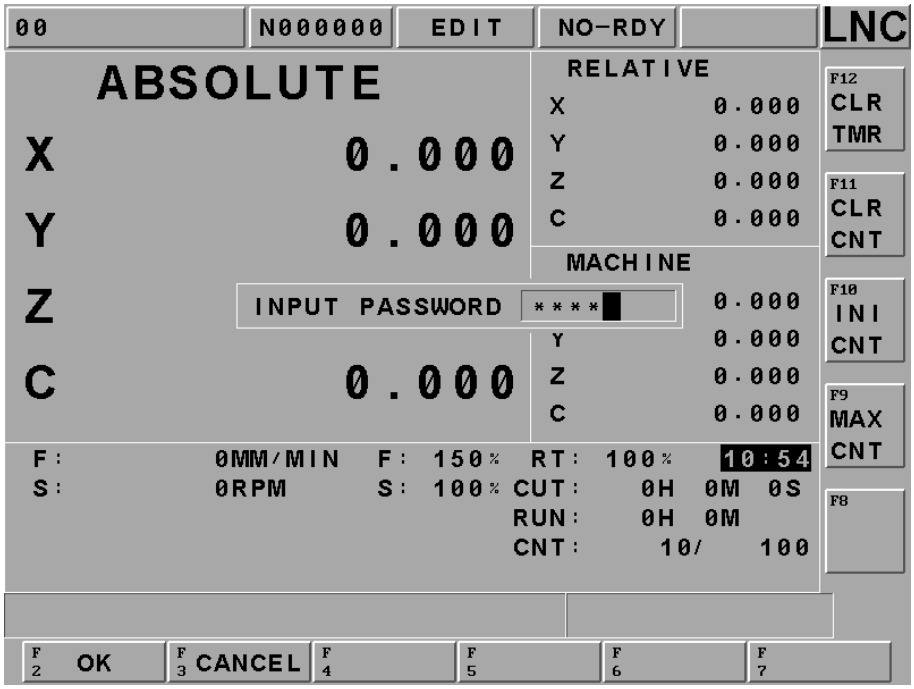
1. Move Z Axis-100, make the tool to touch working part's surface, right now the absolute coordinate is -100, and the machine coordinate is Z Axis-100
2. Pressing 【**SET Z**】, using G92 to set Z axis coordinate to 0. Re-define Z axis reference point of absolute coordinate.

**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.

**【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.




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

**（PROG）**

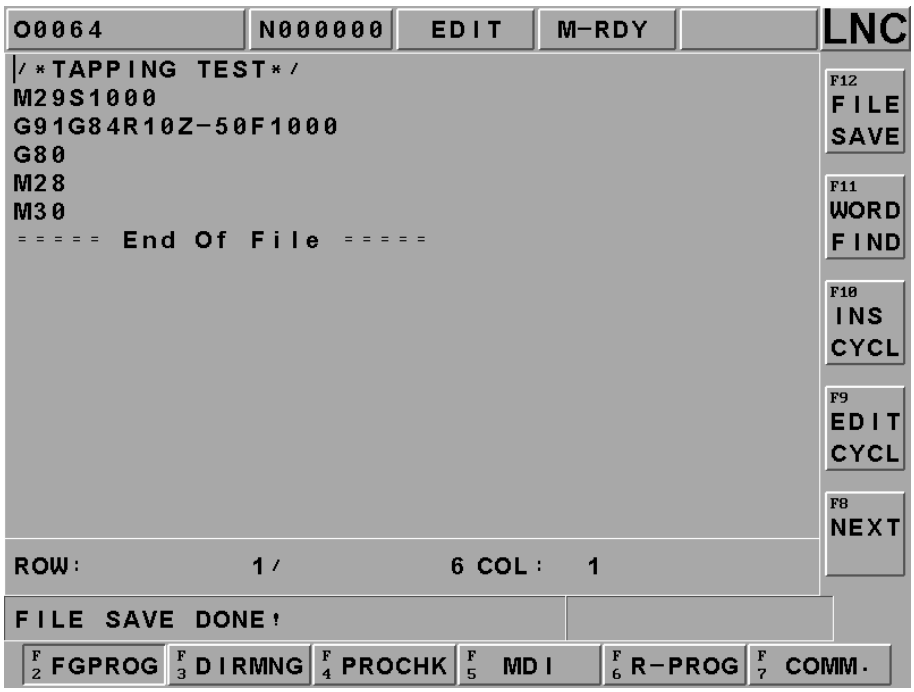
Pressing **<PROG>** to enter into program function group screen. This function group provides part program editing, file managing, checking, rebooting, RS232 transmitting and other related functions.

**【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】**

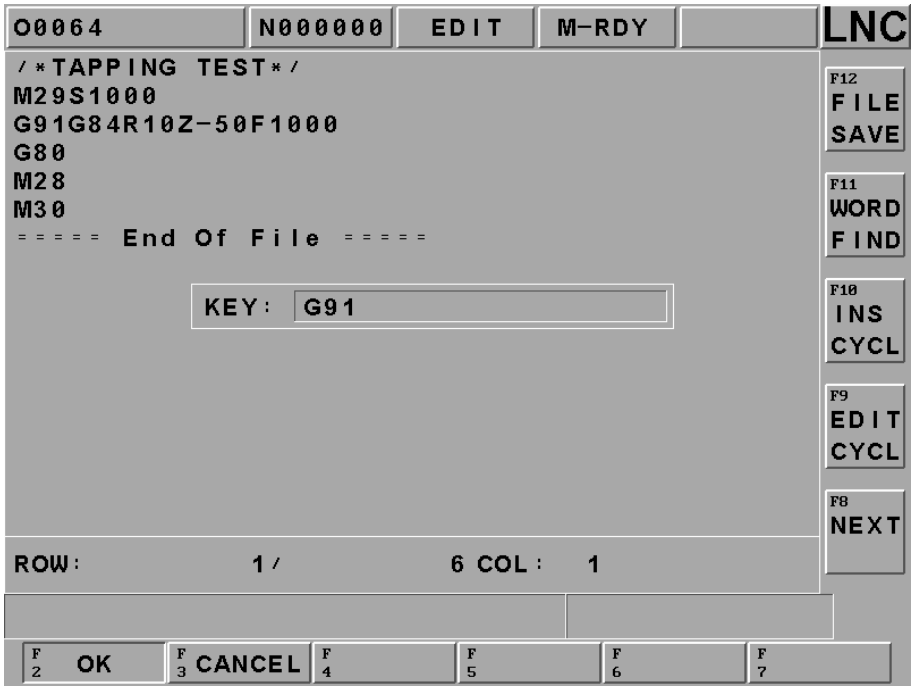
  Press right side【FILE SAVE】function key to save current editing program or the file will save automatically when group page changing.




- **【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.



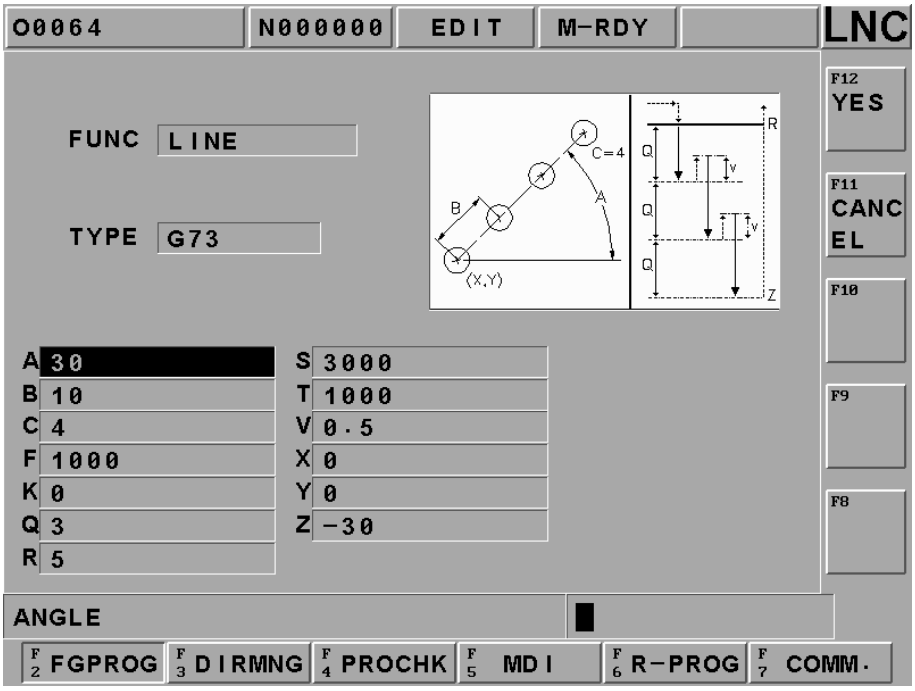

- **【INS CYCL】**

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

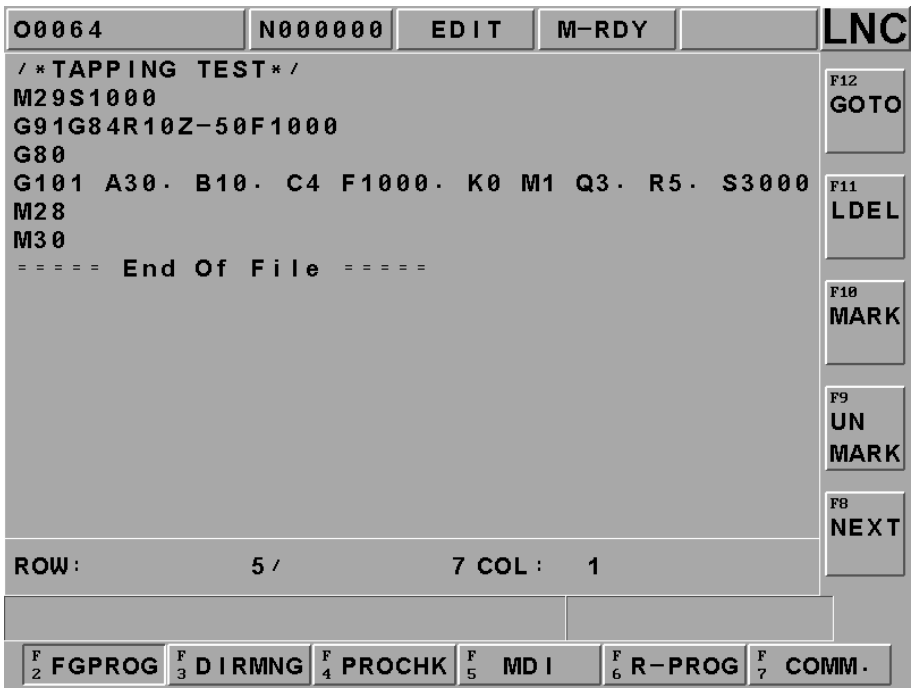
After pressing **【INS CYCL】**, the screen will be as below：

Pressing “Input” after selecting working method：

To press “Input” key to enter into the following Figure page. To move the highlighter to each parameter column and then press <INPUT> key




Program code page is as following after pressing 【OK】

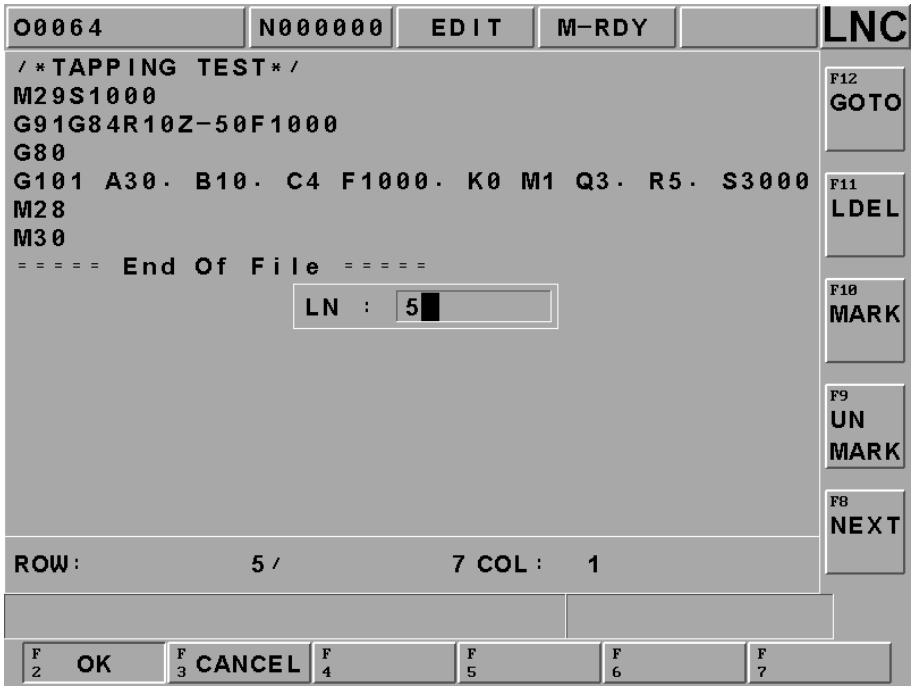


- **【EDIT CYCL】**

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.

- **【LPOSITION】**

After pressing right side【LPOSITION】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:




（Note）Users can not do cycle start under edit mode.

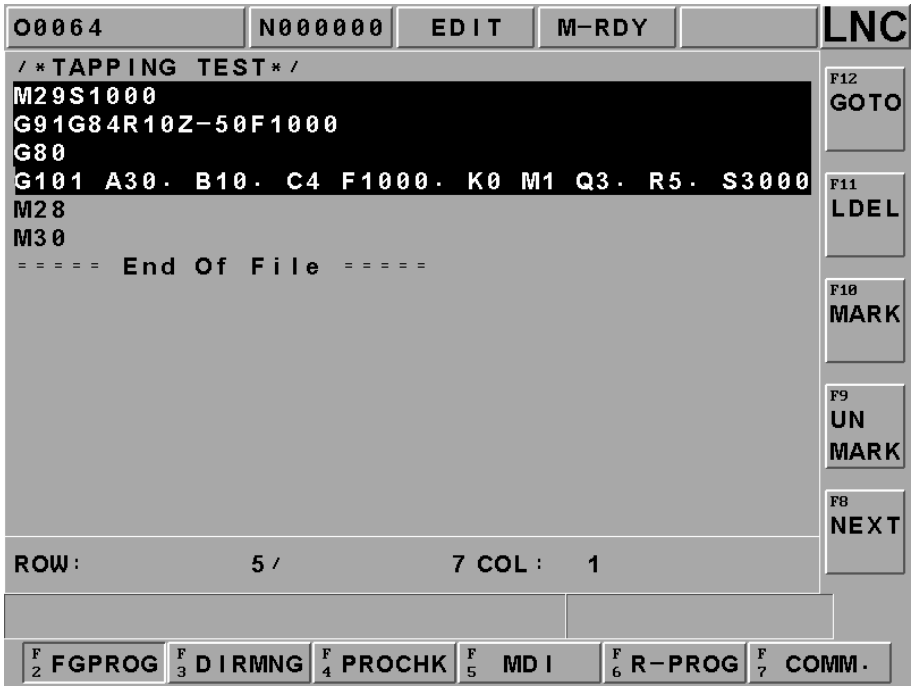
- 【LDEL】

  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 LDELET to do multi-line program editing.



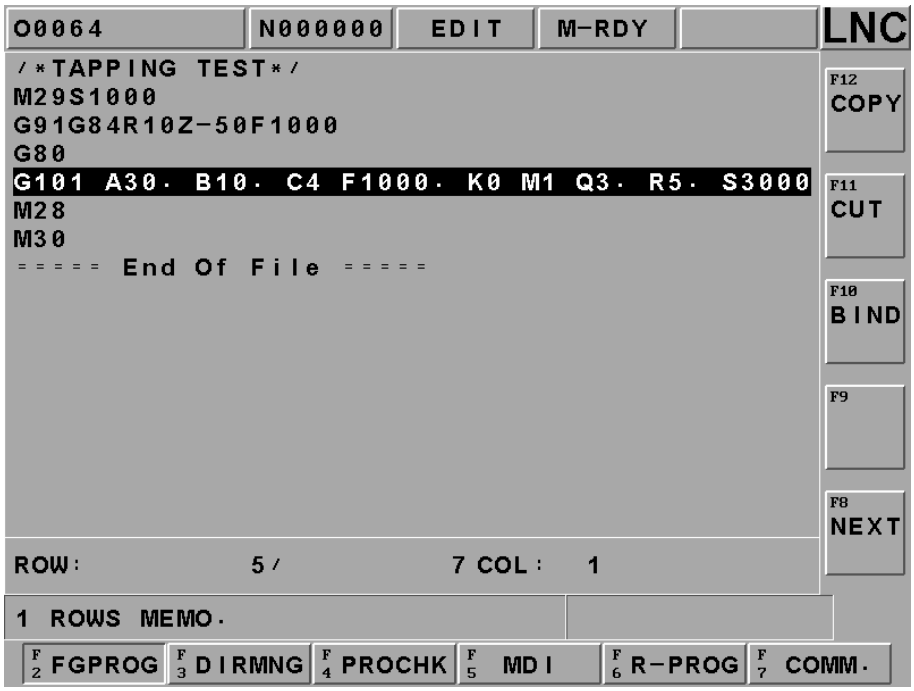
- **【UNMARK】**

  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】**

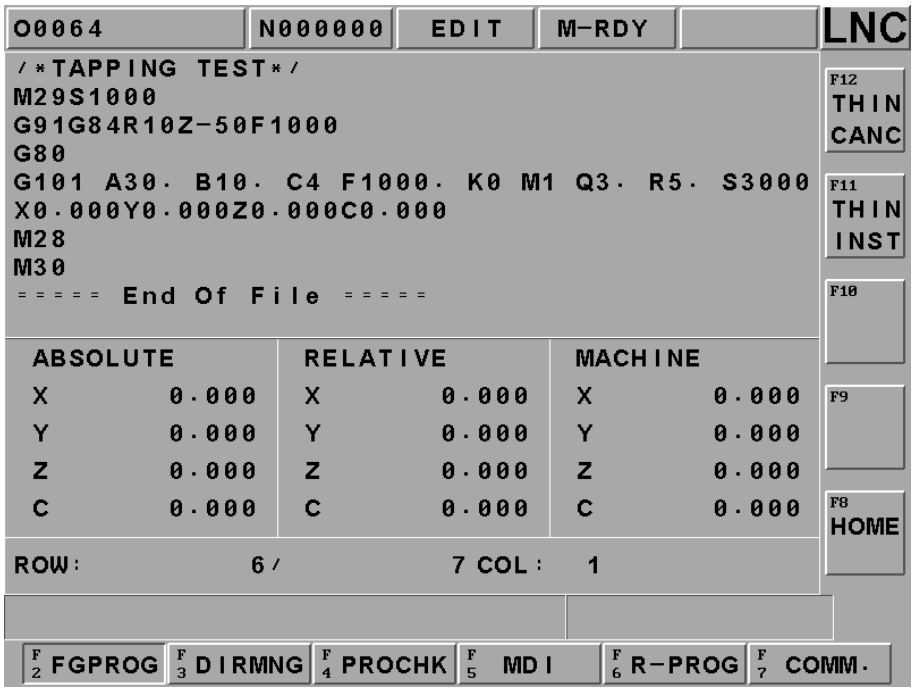
  To cut down the marked program in local range.

- **【BIND】**

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

- **【Teach Mode】/【Delete Teach】**

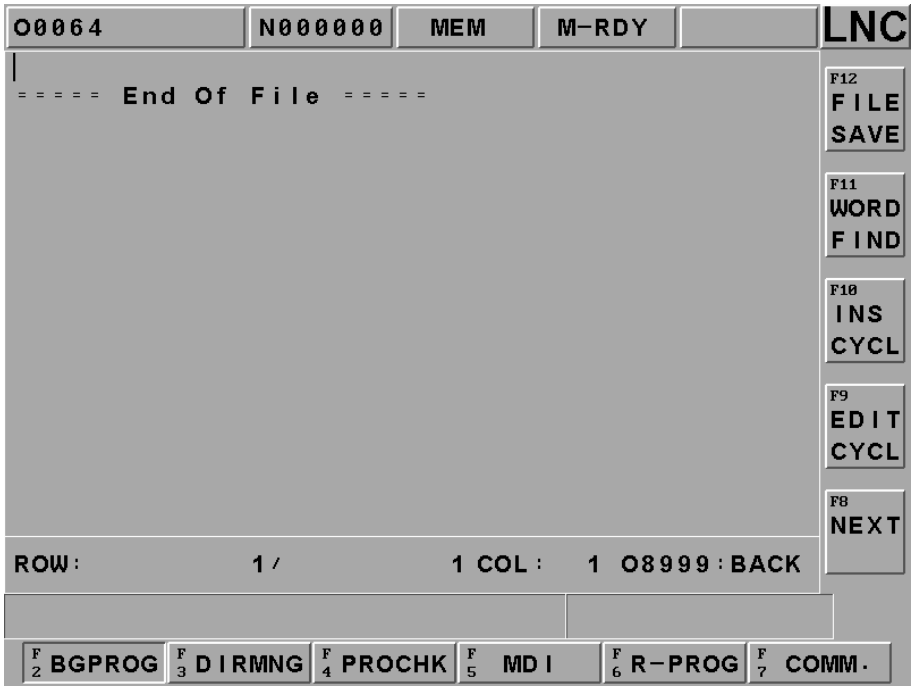
Press this button to show ABS/REL/MAC at the down side for reference. At the right side of sub-function keys will show 【INSERT COORDINATE】for users to insert current location's absolute coordinate quickly, detail is as below:



**BGPROG** (Background Program)

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

Note: Background file is 08999 permanently.



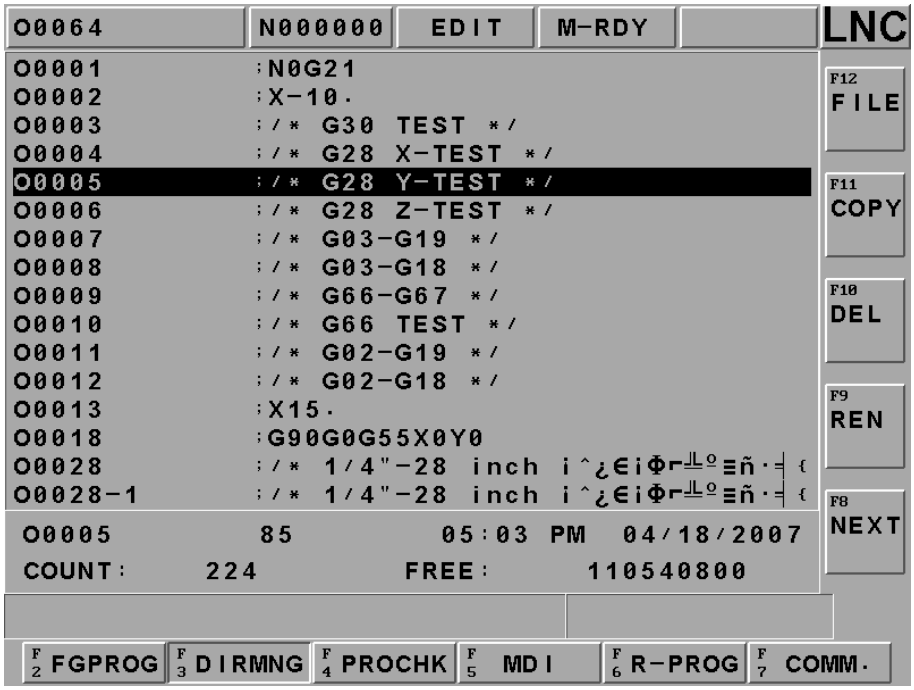
### 1.4.1 DIRMNG (File Management)

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

- **【FILE】**

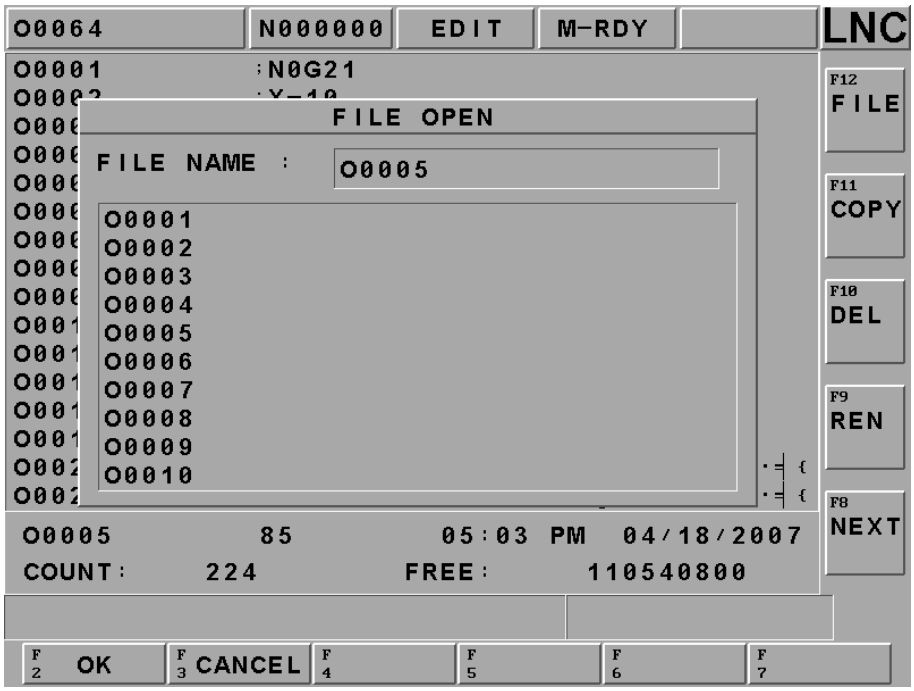
Solution 1

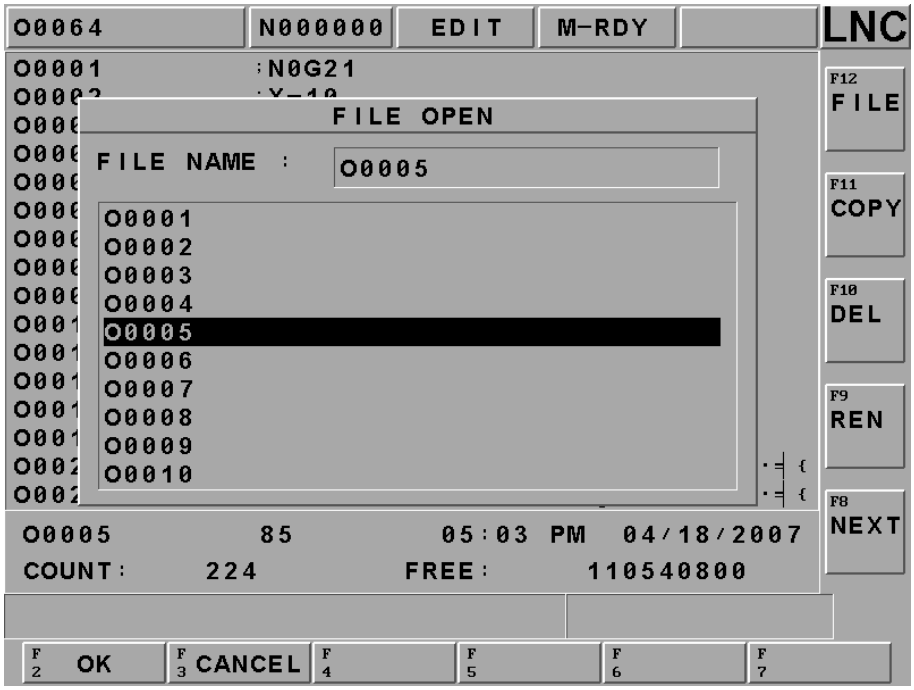
Able to use direction key to choose the wanted open file after entering into the file management screen. Also, users are able to open a file after pressing <INPUT>.



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

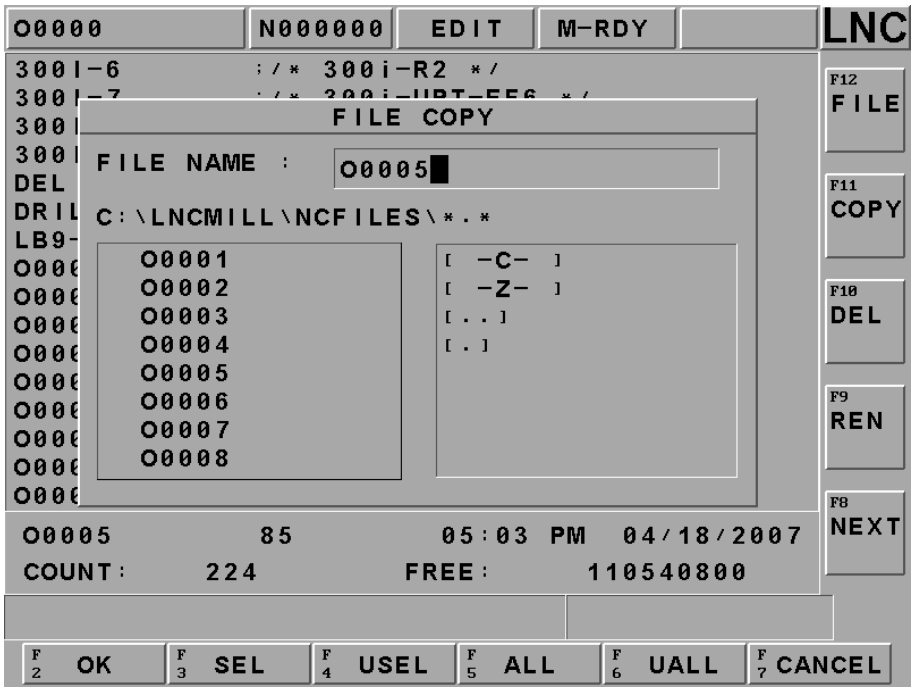


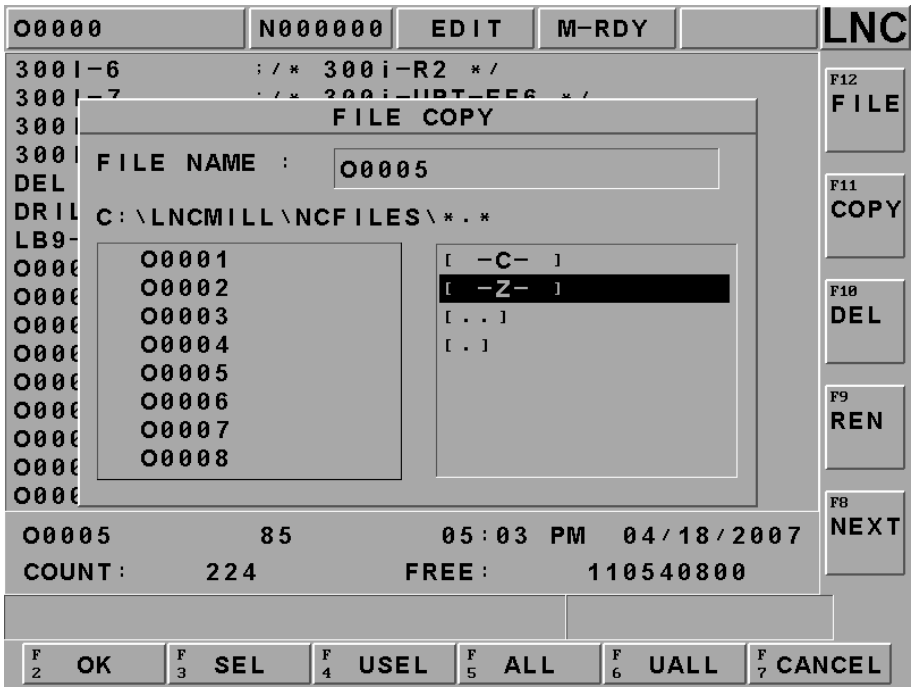

Note：The opened file is a background program under auto mode. Automatically setting the opened file to the present program under editing mode means NOT able to open the file under other modes.

- 【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:

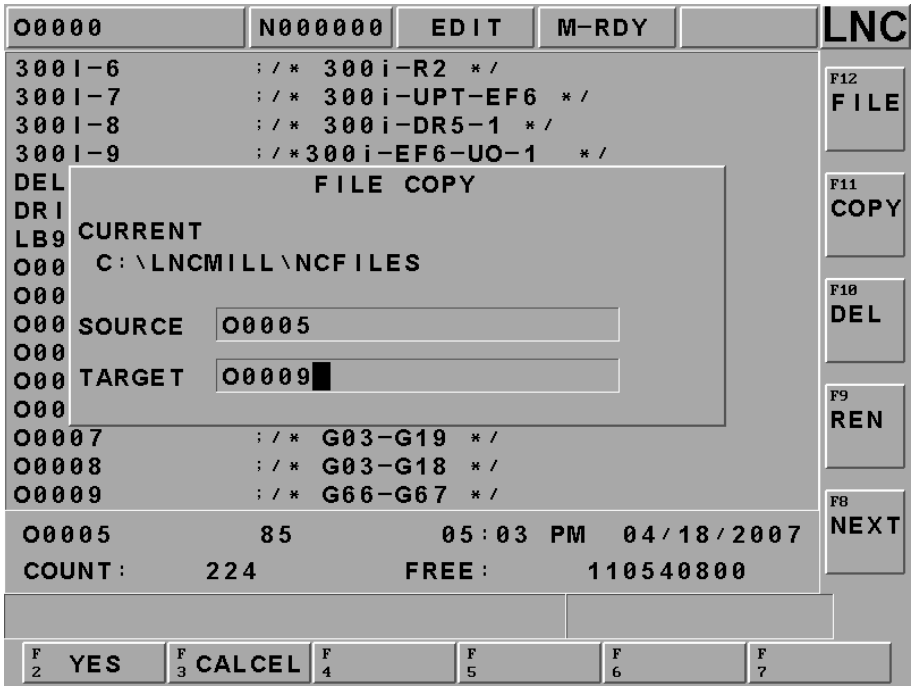



Figure 1.4-1 Screen of Key-In Destination File Name

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

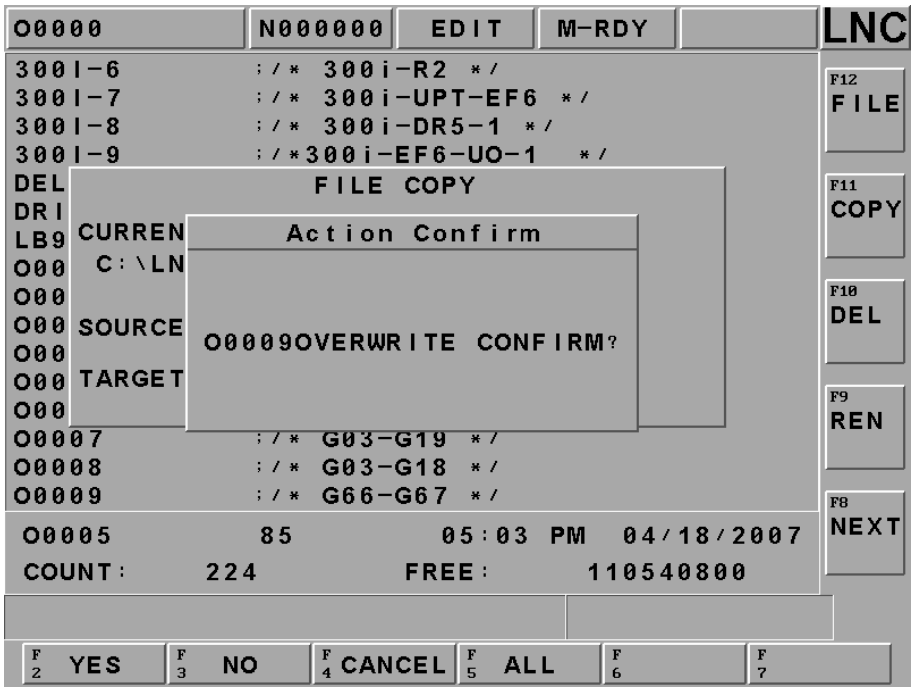


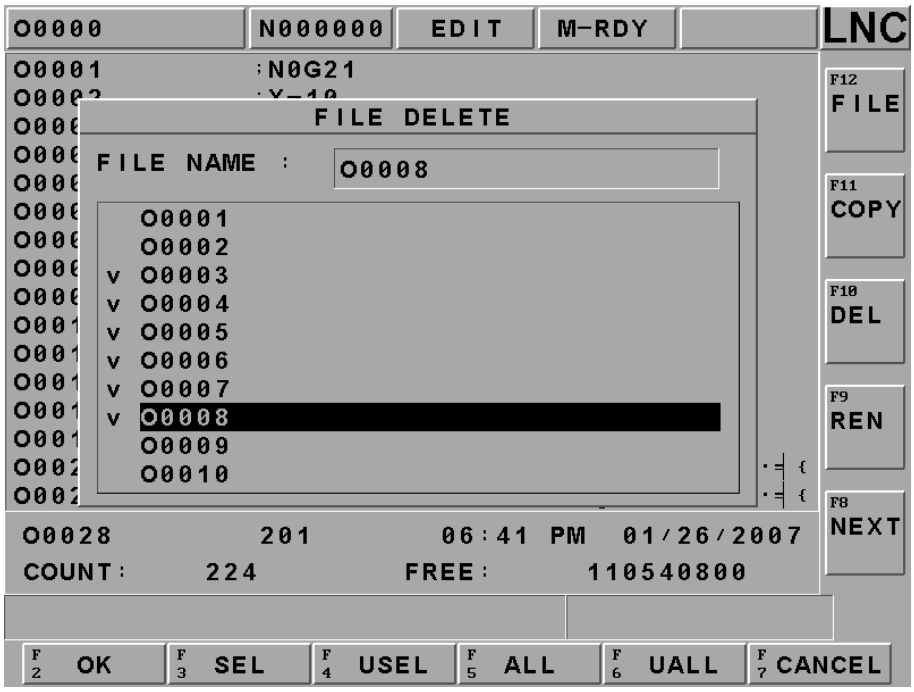

Figure 1.4-2 Overwriting

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【no select】【select all】【no select all 】to manage file.


O0000
N000000
EDIT
M-RDY
LNC
O0001 ;N0G21
FILE DELETE
FILE NAME : O0008
O0001
O0002
v O0003
v O0004
v O0005
v O0006
v O0007
v O0008
O0009
O0010
F12 FILE
F11 COPY
F10 DEL
F9 REN
F8 NEXT
O0028 201 06:41 PM 01/26/2007
COUNT: 224 FREE: 110540800
F2 OK
F3 SEL
F4 USEL
F5 ALL
F6 UALL
F7 CANCEL

- 【DEL】

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

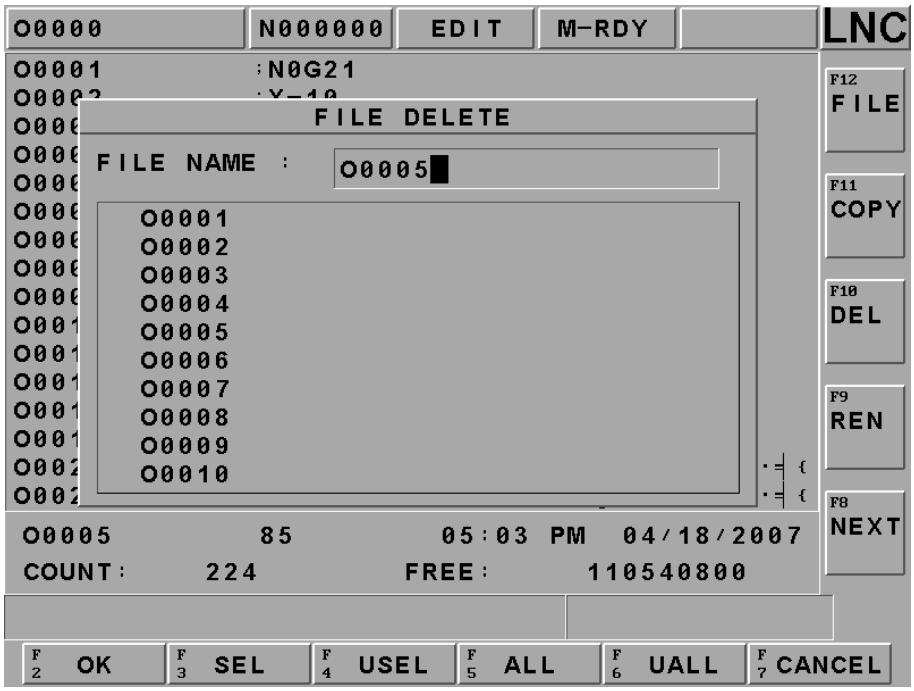
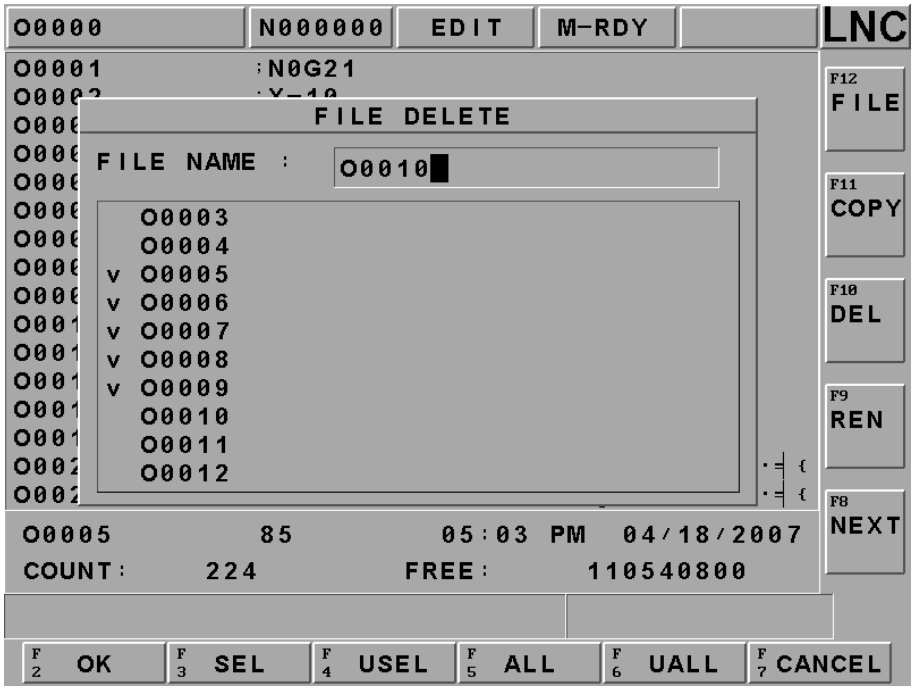



Figure 1.4-3 【DEL】screen

- **【COPY】MULTI-DELETE**

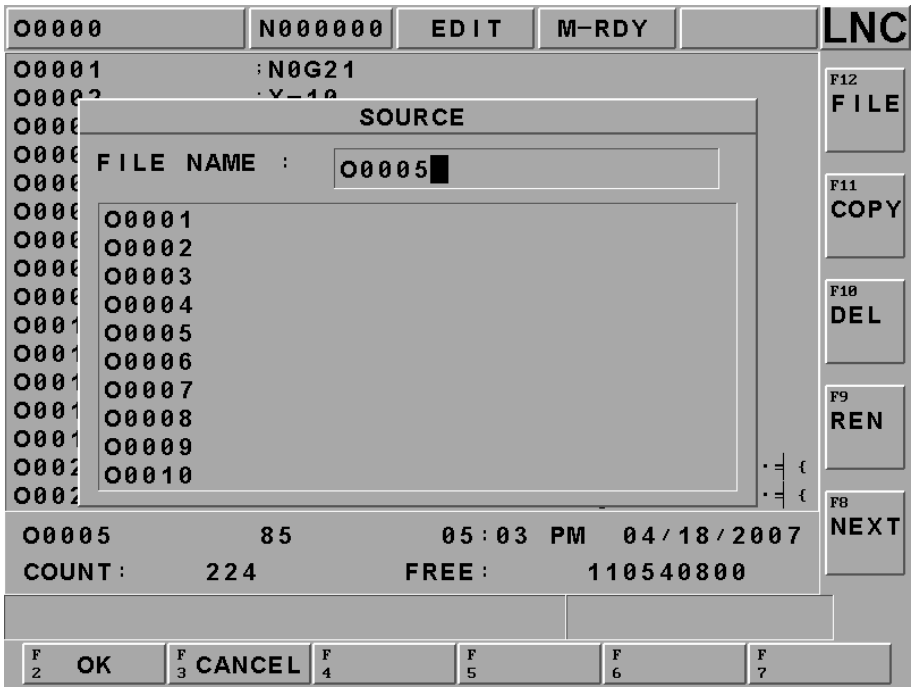
Except single file delete, our controller also offer multi-file delete function. In the dialogue box mode, you can press left or right button to move cursor to switch line to the left side. Press select to choose check box and press confirm to delete. After pressing confirm, the system will pop up hint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

You can also use【no select】【select all】【no select all 】to manage file.

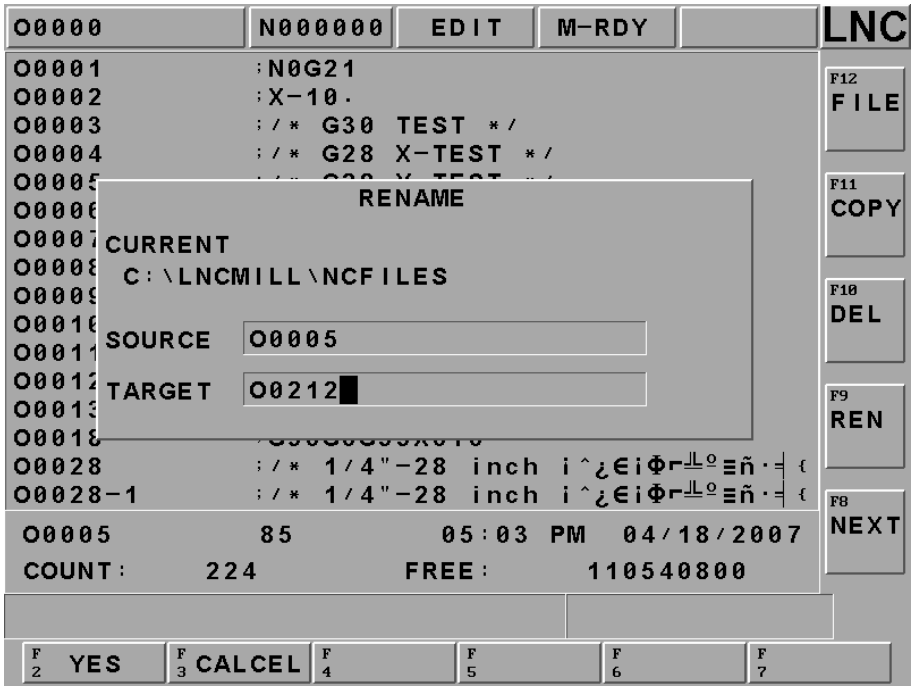


- 【REN】

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 confirm 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.



- 【DIR SET】

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

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

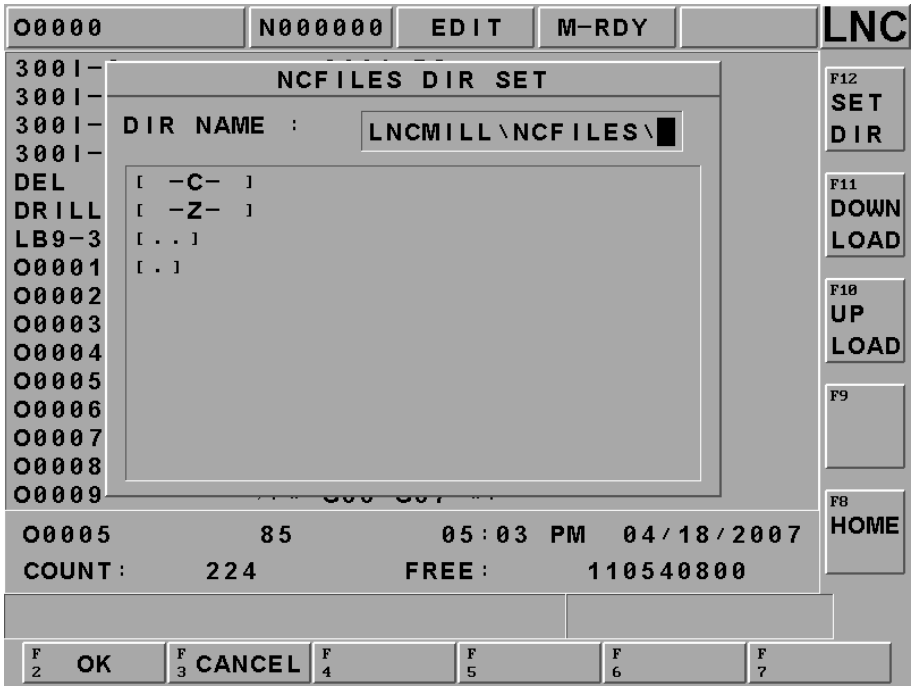
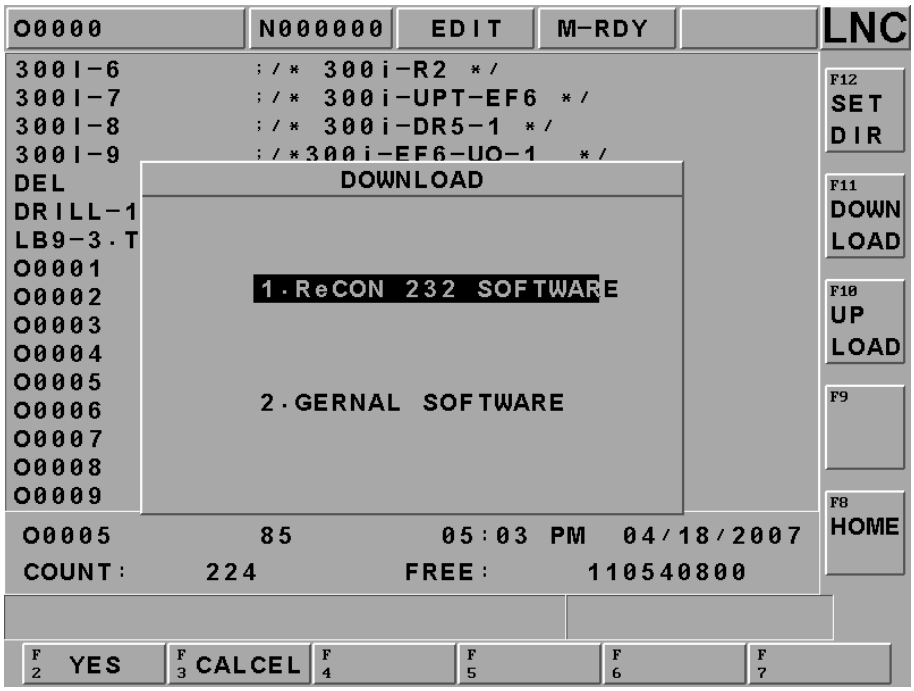



Figure 1.4-4 Screen when Setting Part Program Menu

- 【FILE 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.

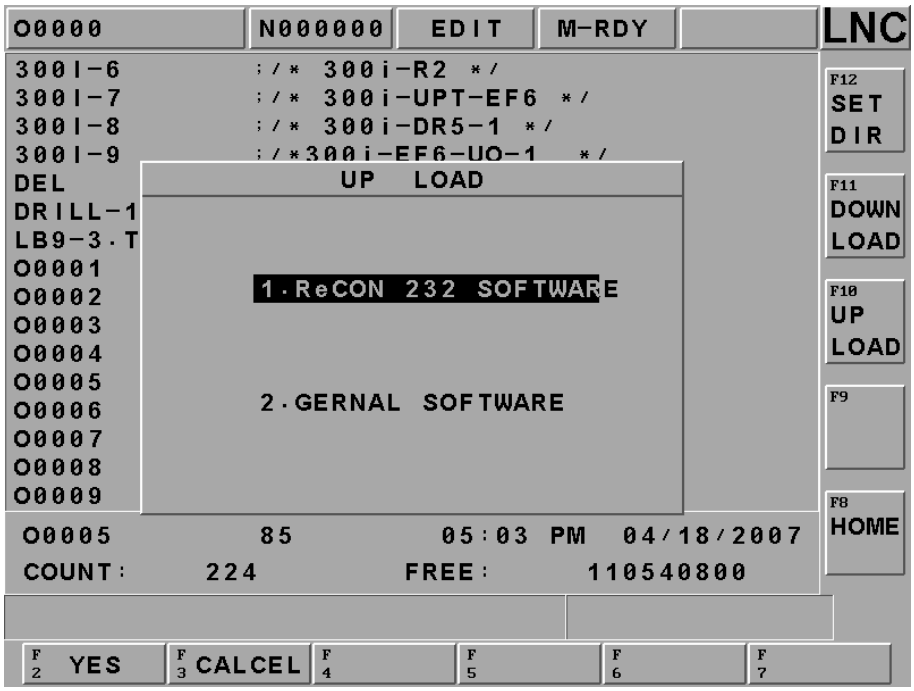
Note：As for detail information, please refer to RS232 communication software chapter.



- 【FILE 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.

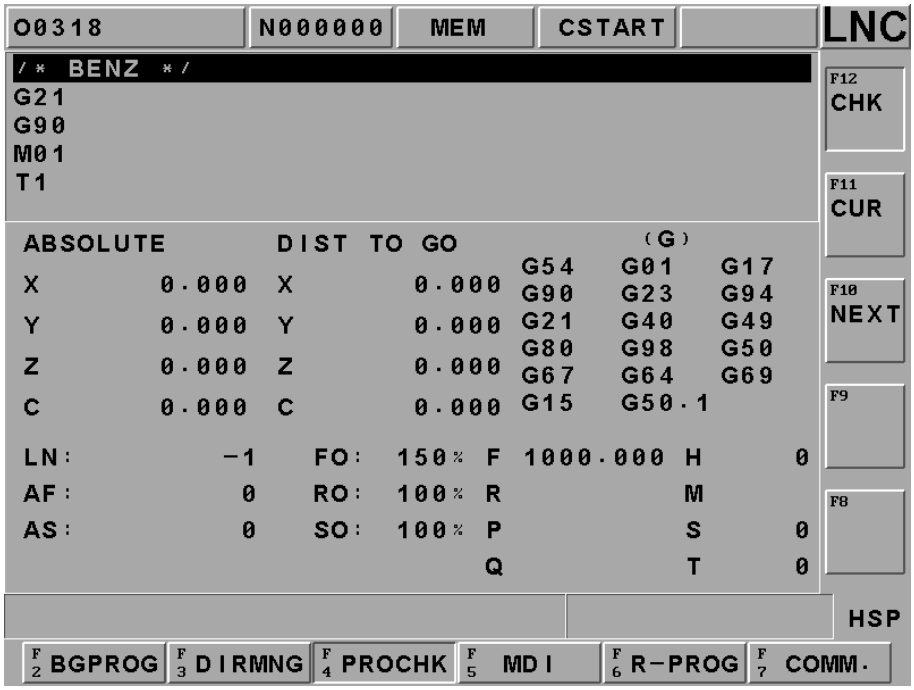


### 1.4.2 PROCHK (Program Checking)

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

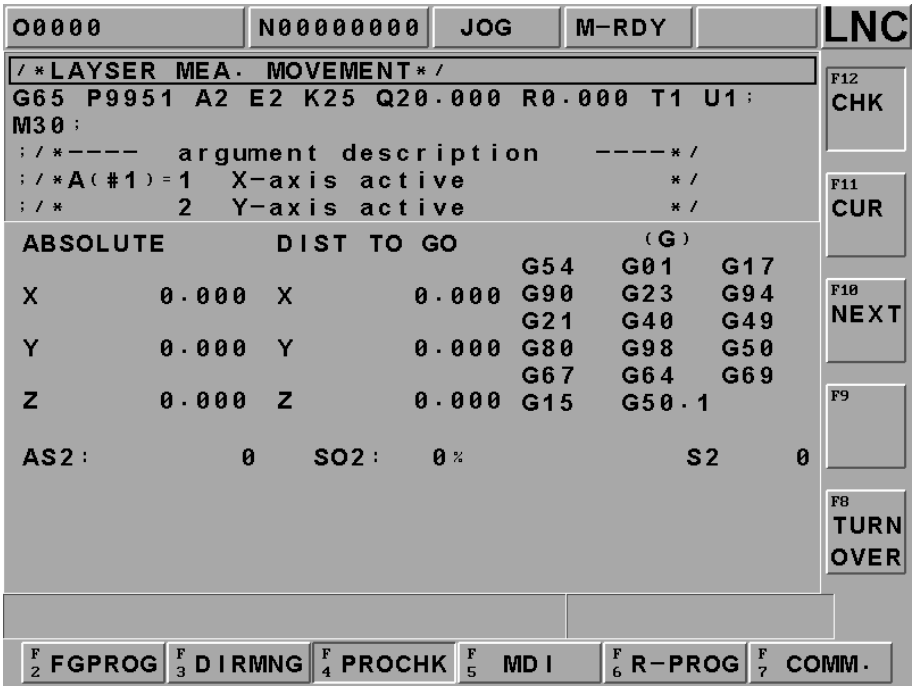
- 【**SINGLE CHECK**】

  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.



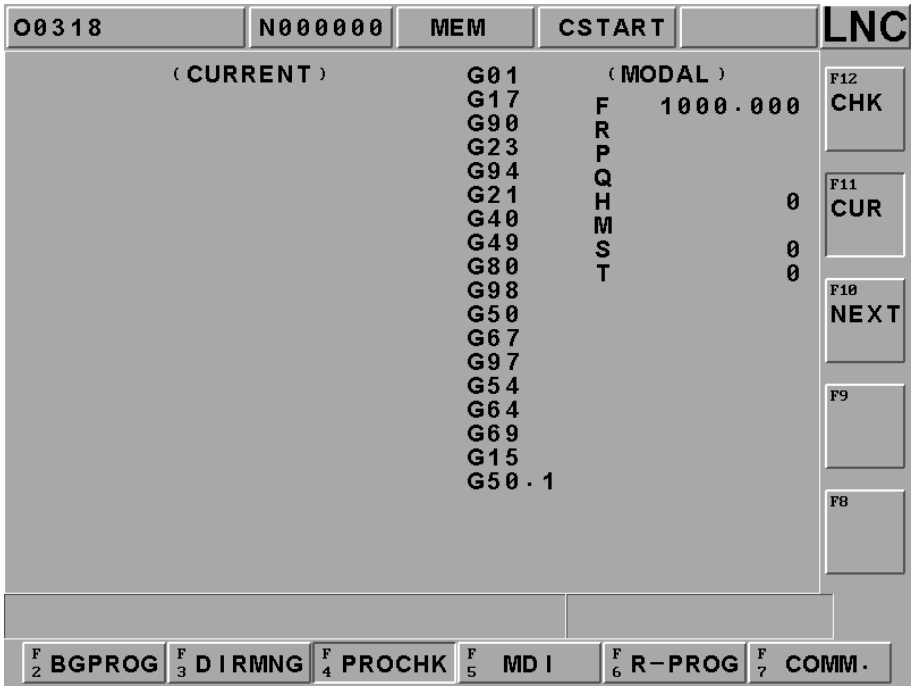
- 【Trun Over】

When there is still some info at the below area, users can press【**turn over**】**to see other detailed info.**



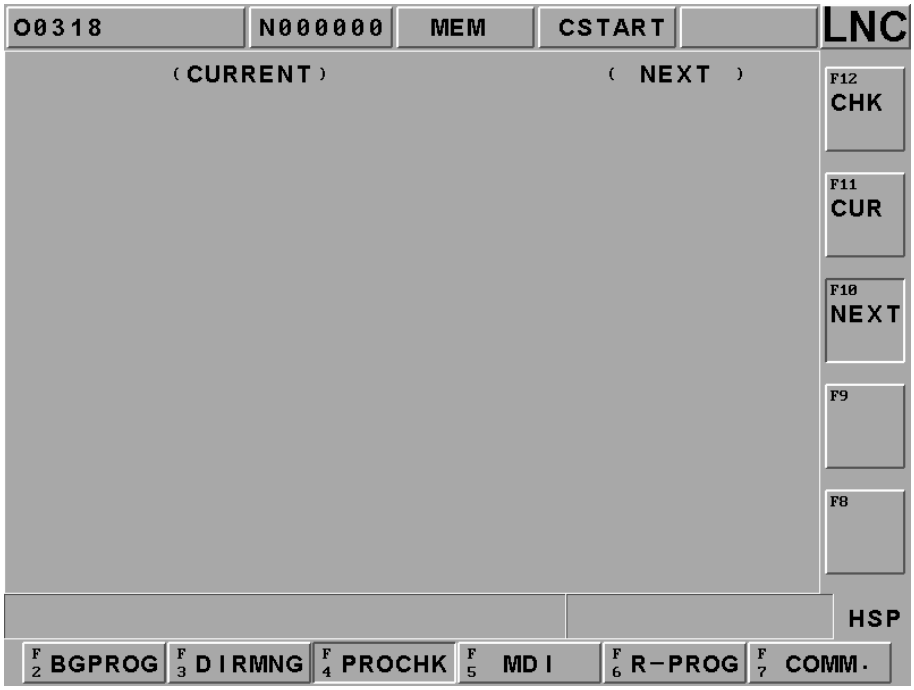
- 【Current Single Block】

To display the current single block.




- 【Next Single Block】

To show the next single block.

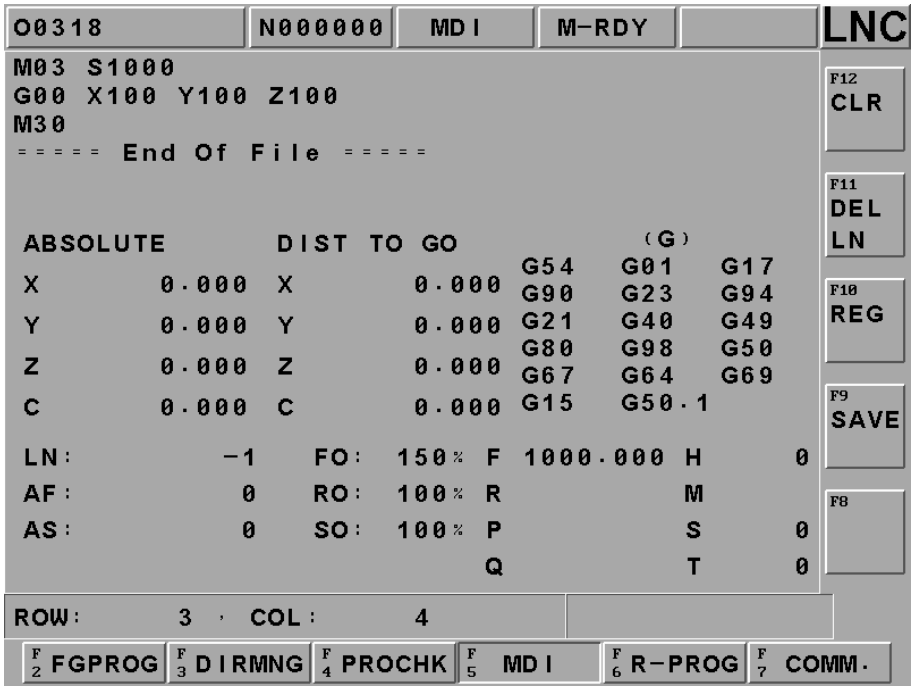


### 1.4.3 MDI

Pressing【MDI】key to entry into MDI page under MDI mode. You can see MDI page as below: At this page, user can execute directly the single block program that just input. Systems also allow multi-line programs, so called multi-line MDI.

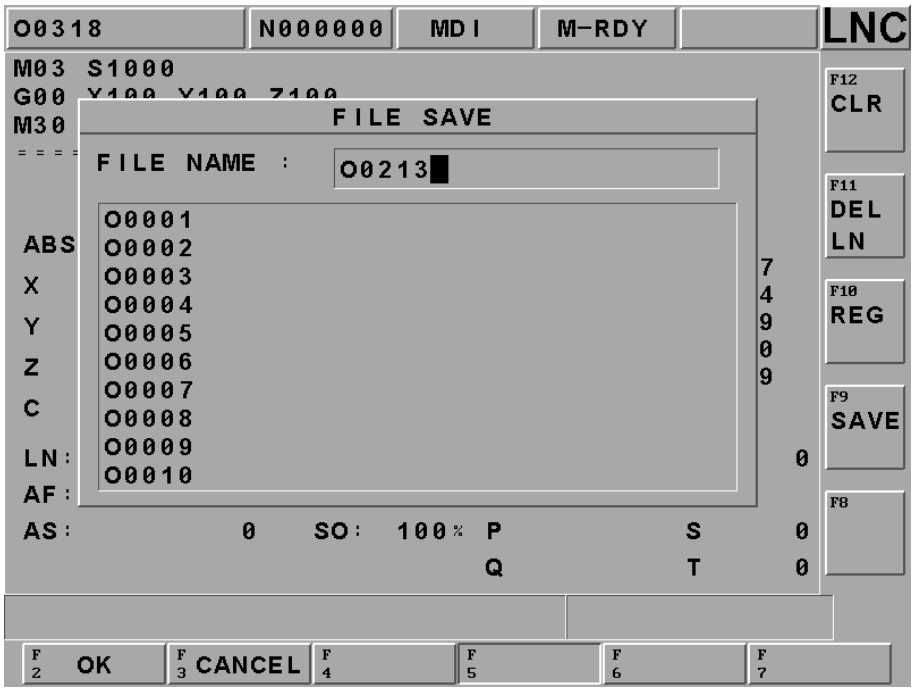
The operation is as below: Users key-in command and then press <INPUT>. Then, the command will occur at the left section of the main page. Then, pressing Cycle Start key and the command will be executed and related data will be displayed. Using this method to do test running is much safer and more time-efficiency.

Also systems provide【CLR】【DELLN】to help users.



- 【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.



**PROGRAM RESTART**

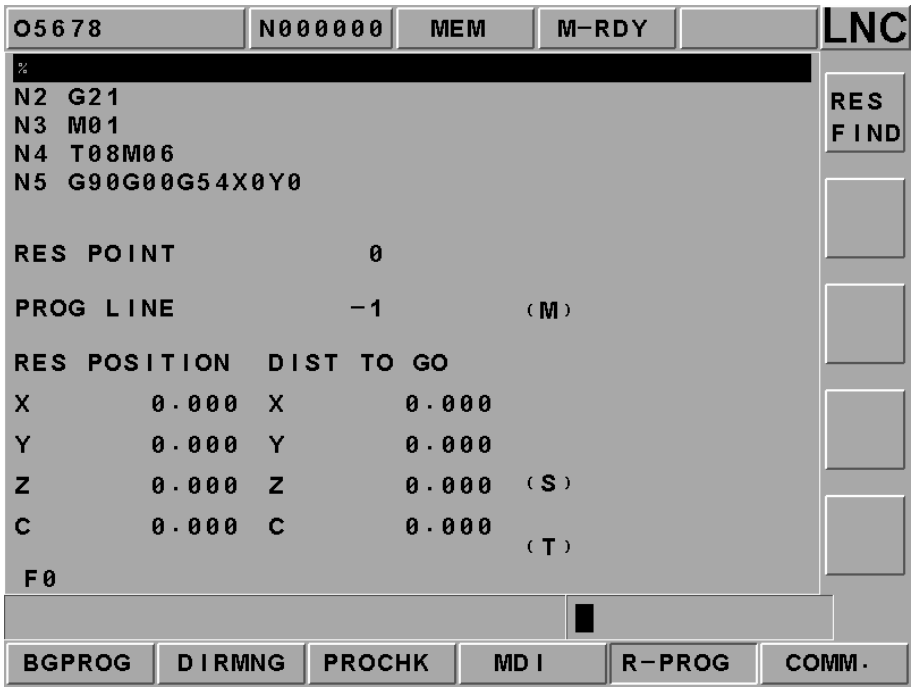
During program working, if the tool brakes, emergency stop happens, or other situations to make the working process stop, this function can search the interruption point (single block series or program code) to restart the processing. It can also be a quick exam function.

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：**

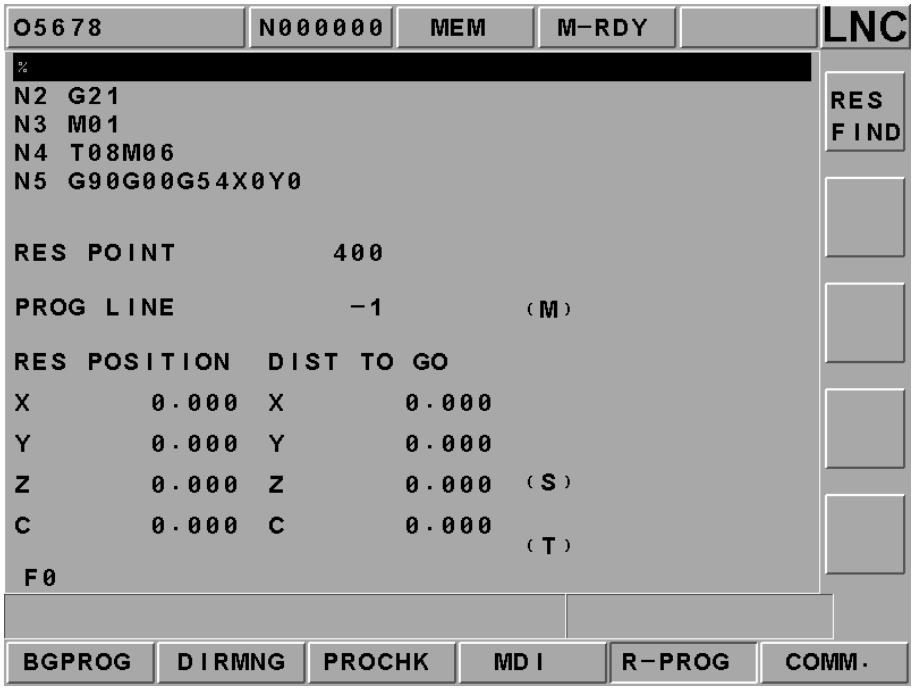
- **Input Allowed：**

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

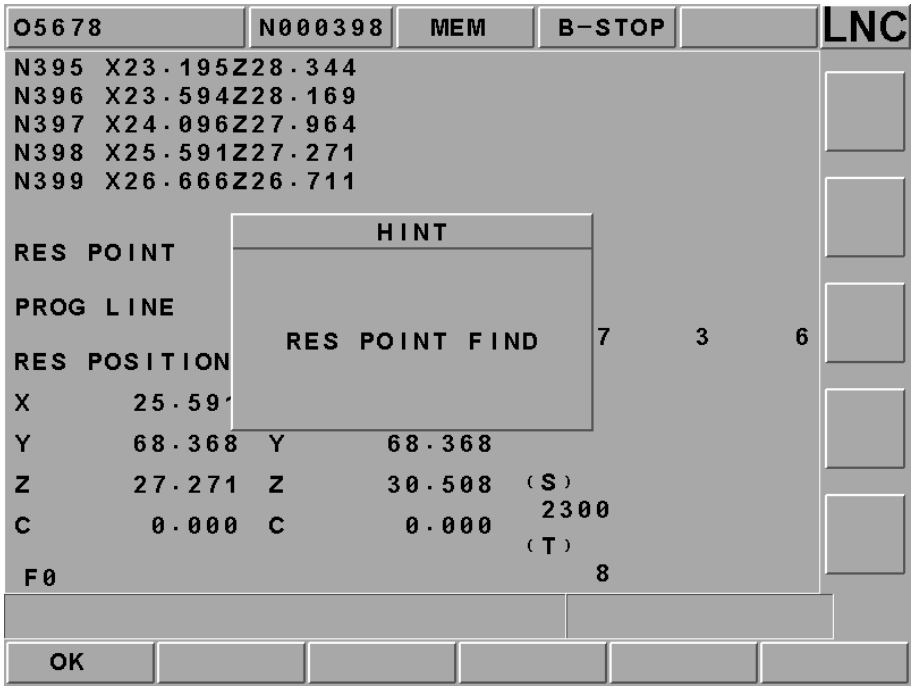


**Program Code Search：**

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



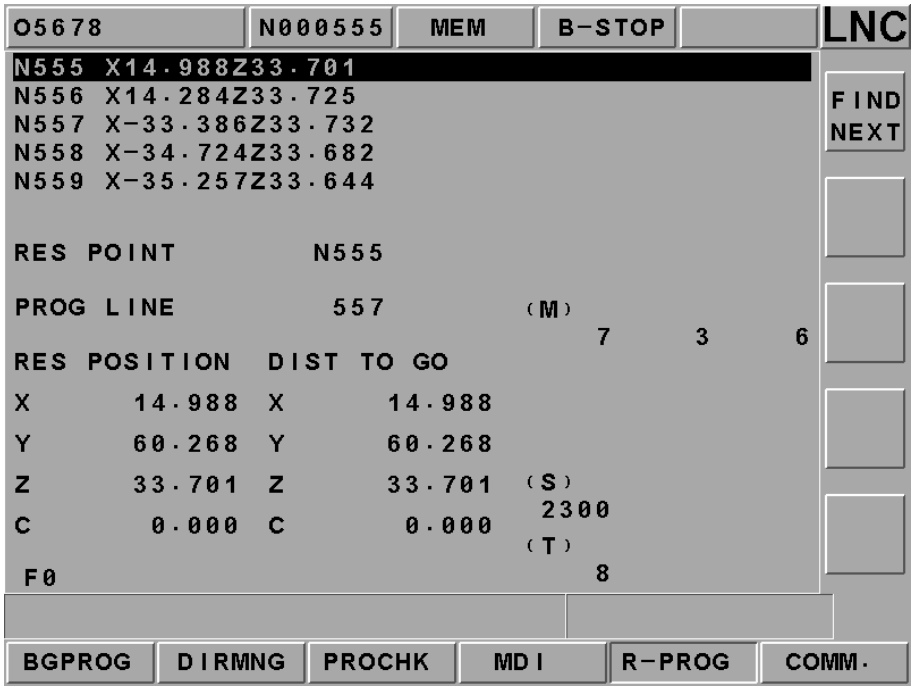

NC will start to search this processing file from the first line, when “400” window pops out at the screen, it means the search is completed.




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.




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.

### 1.4.4 COMM (Communication of Files（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.

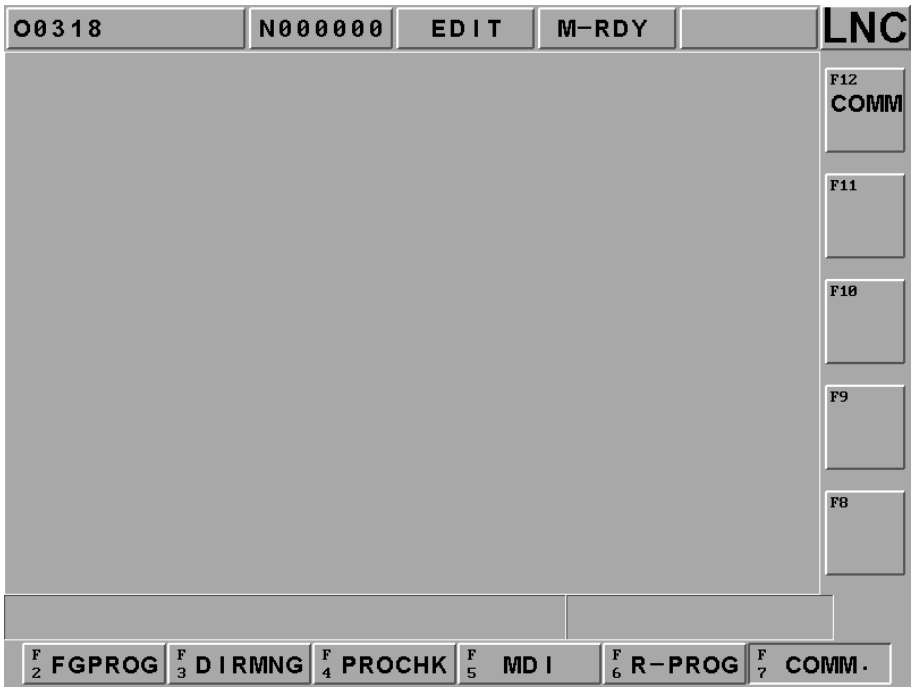



Figure 1.4-5【**COMM**】 screen

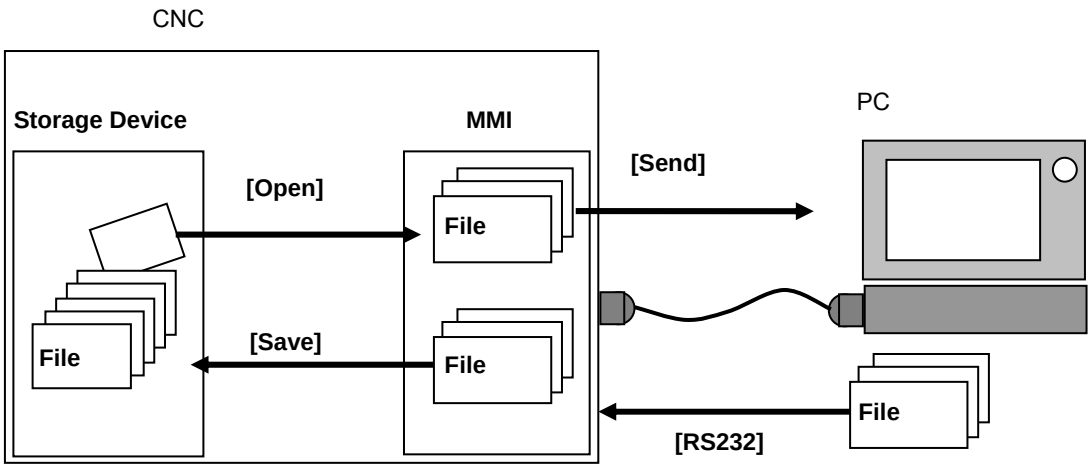



Figure 1.4-6 Files Transmitting/Receiving Figure between Controller and Other PCs

- 【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.

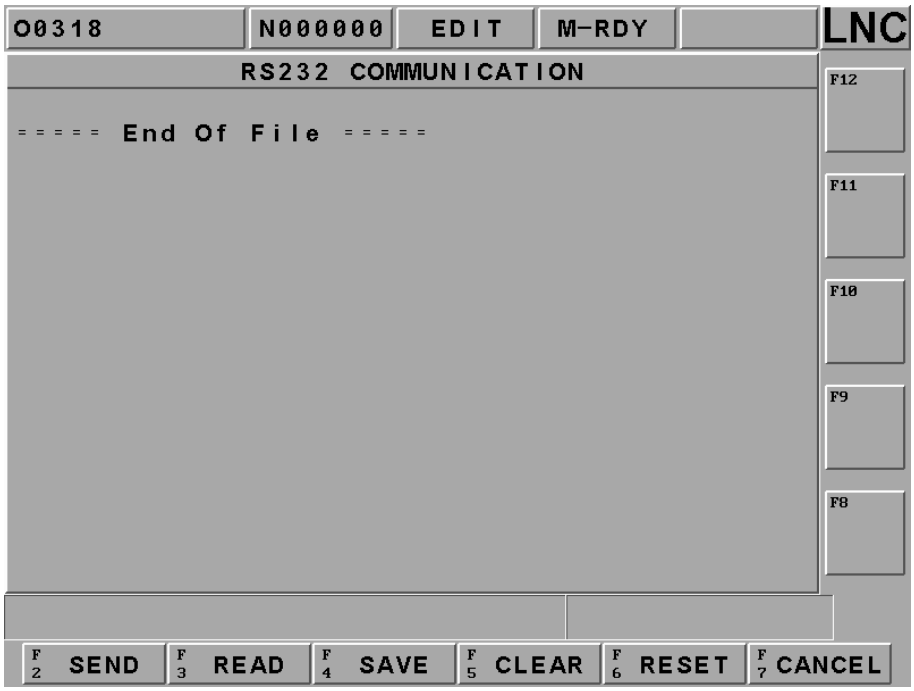



Figure 1.4-7 send main screen

【**SEND**】Before pressing send button, the file select window will be pop up. After choosing the file you want to send, press confirm 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**】Giving up and clearing 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

a. 99 sets of setting for Tool Offset. To switch the setting screen by using <PAGE ↓ > and <PAGE ↑ > key

4. 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

5. 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）

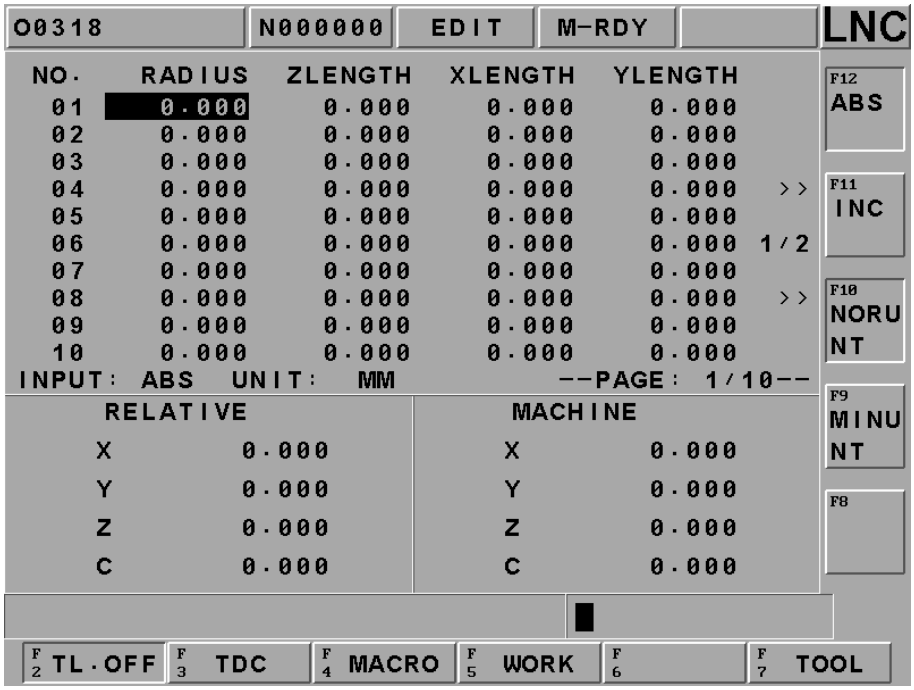



Figure 1.5-1 Tool OFFSET Screen

【**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.

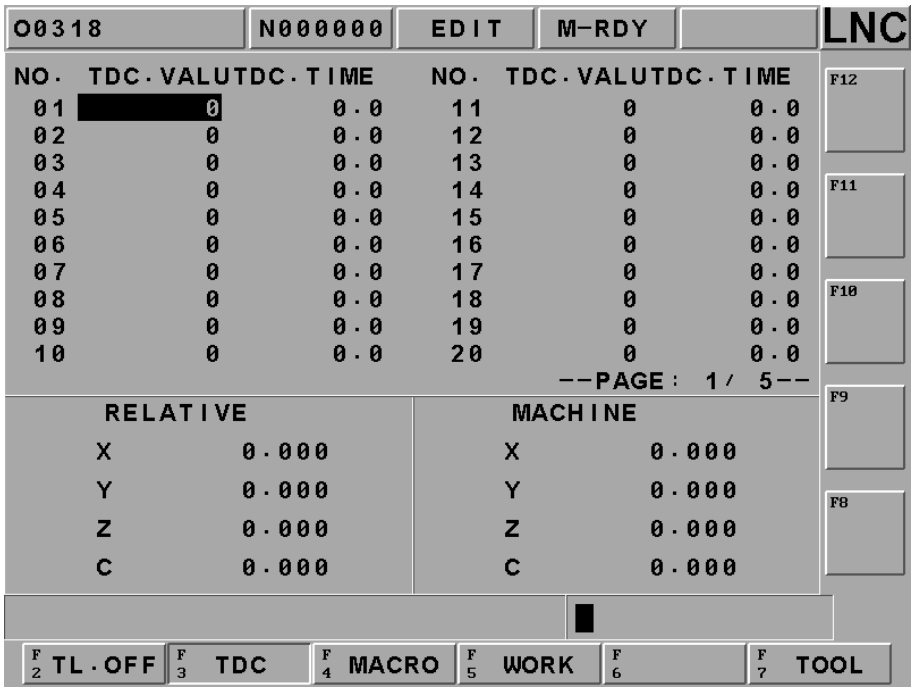
【**NORUNT**】Input by normal unit,ex: value 0.500, input 1, and it will show 1.000.

【**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




（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.

### 1.5.3 MACRO

Pressing【**MACRO**】button to enter into the MACRO variable screen. Detail information, please refer to MACRO Chapter in Program manual.

1. MACRO Variables: the variables which start with＃ is local variables, offer 49 sets of setting. The variables which start with@ are common variables, offer 999 sets of setting. To switch the setting screen by using <PAGE ↓ > and <PAGE ↑ > key.
2. Or input @100 at input line and press enter, it will start to search @100 automatically.
3. In this function, variables can be entered or modified. The modification method is to move the highlighter to the wanted modify position and then to press <INPUT> key after entering the wanted value.

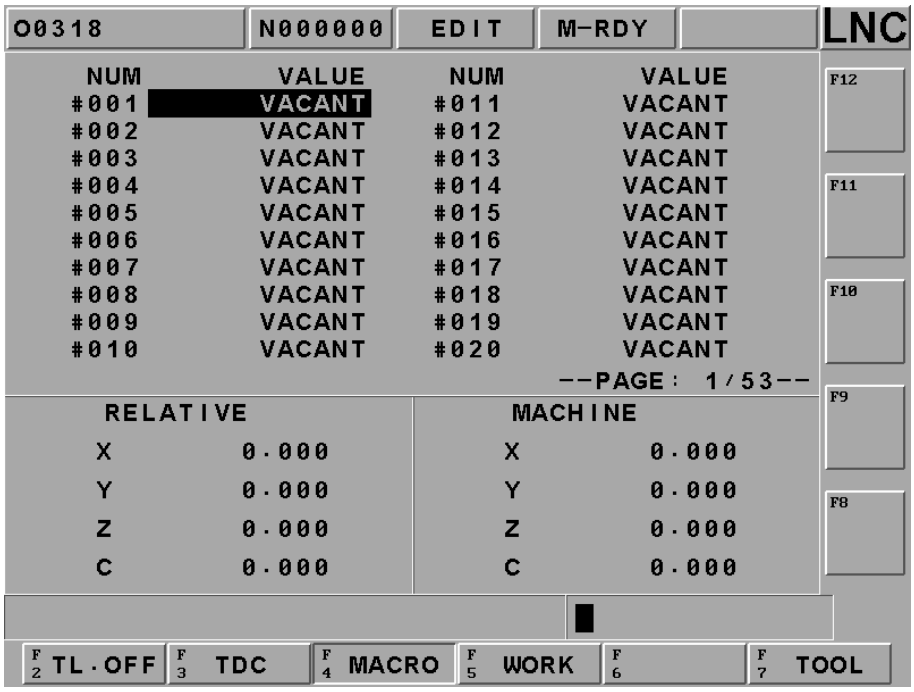



Figure 1.5-2 Local Variables of MACRO Variables

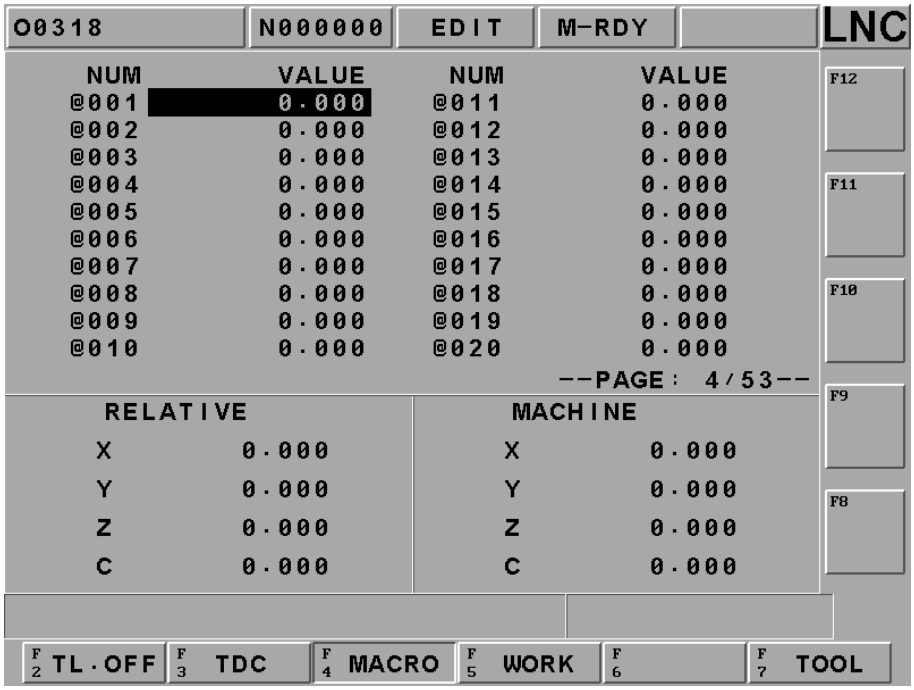
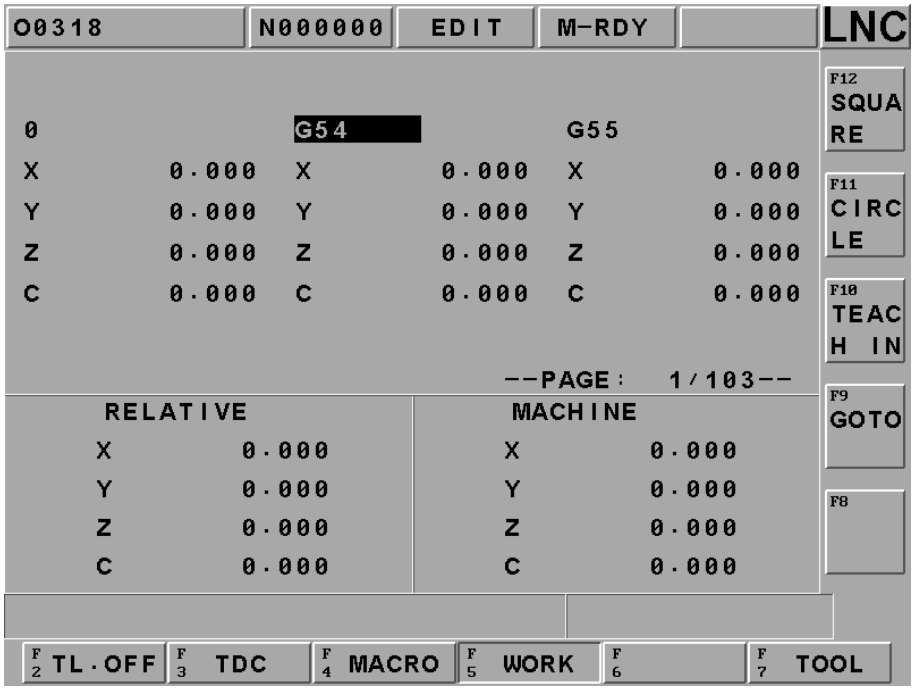



Figure 1.5-3 Common Variables of MACRO Variables

1.5.4 WORK

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

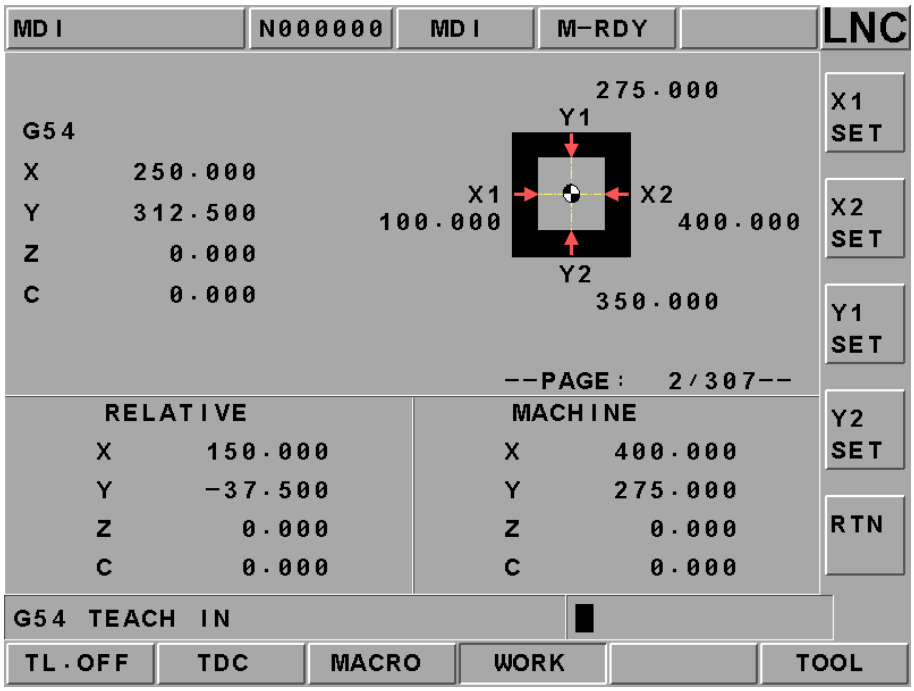
1. The controller provides working coordinate system for users. Users can execute the following settings such as G0、G54、G55、G56、G57、G58、G59、G54P1~P300. Also, using <PAGE ↓ > and <PAGE ↓ > key to switch screen
2. Using direction keys to move cursor to the wanted change column and to key-in the setting 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.



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.

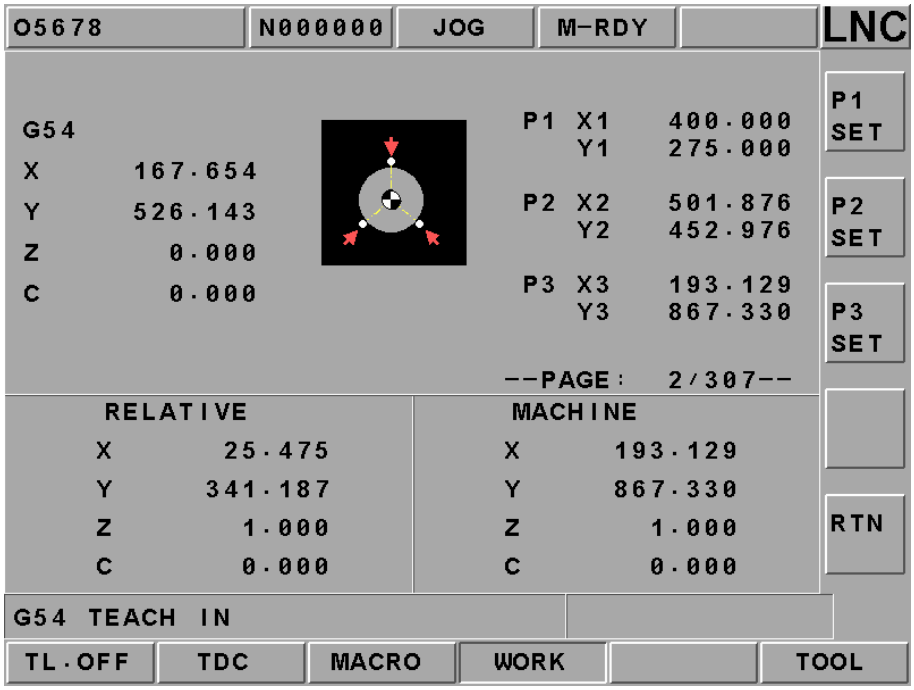
**X2 SET** ：Set square 2 point X2 Axis's machine coordinates value into X2 working coordinates.

**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.



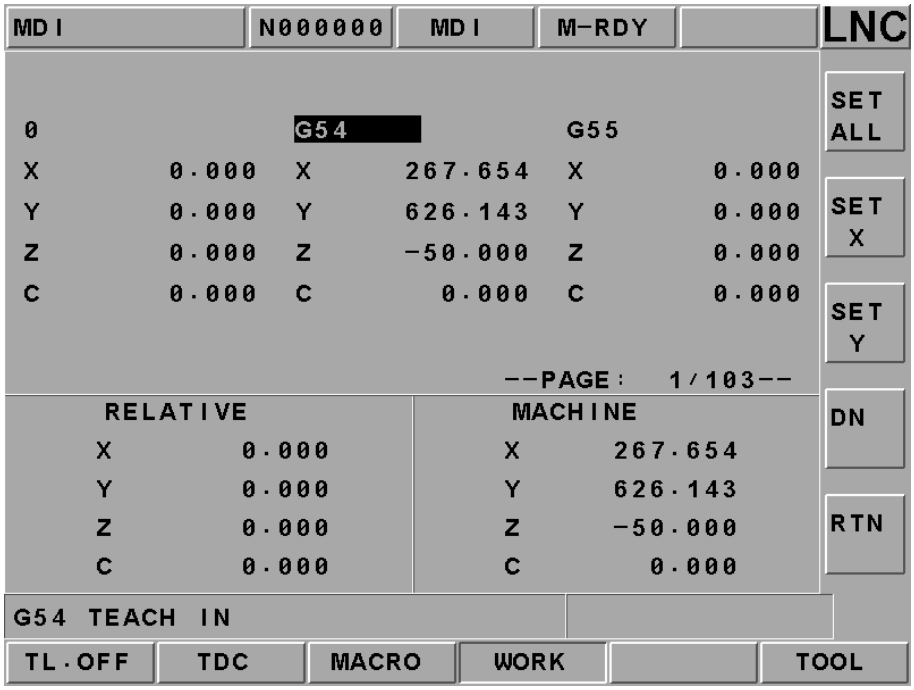

**P1 SET**：Set circle point 1 X, Y Axis's machine coordinates value into P1 X1,Y1 working coordinates.

**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.




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

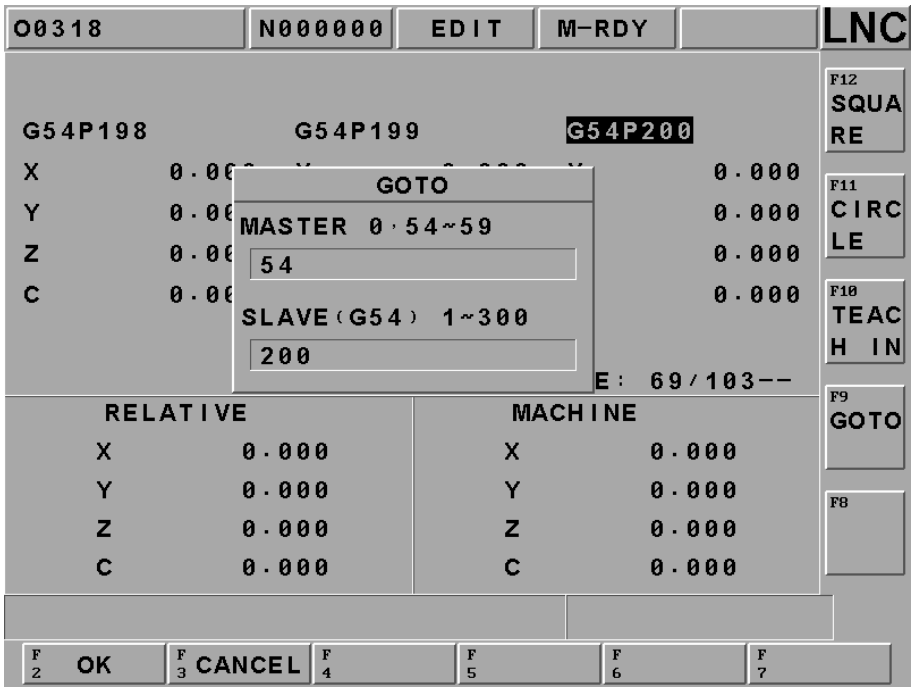
**X SET** ：Set X Axis' machine coordinates value into working coordinates which the cursor stops.

**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.

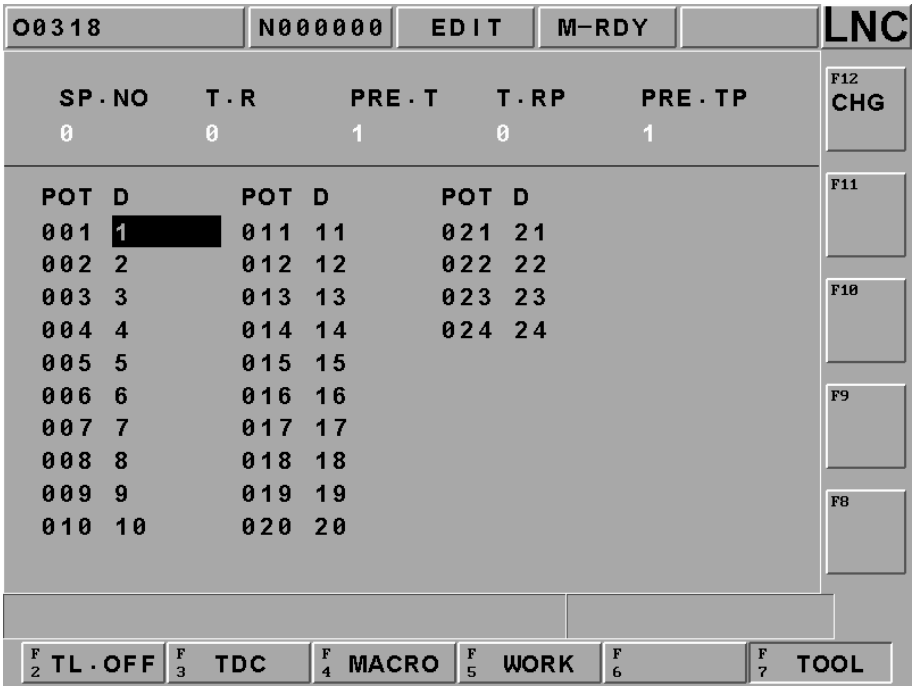
Press【**GO TO**】 to enter work page for users to search quickly, sub-coordinate only G54 can be used.



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.




**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 GRAPH

### 1.6.1 Function Introduction

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.

### 1.6.2 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

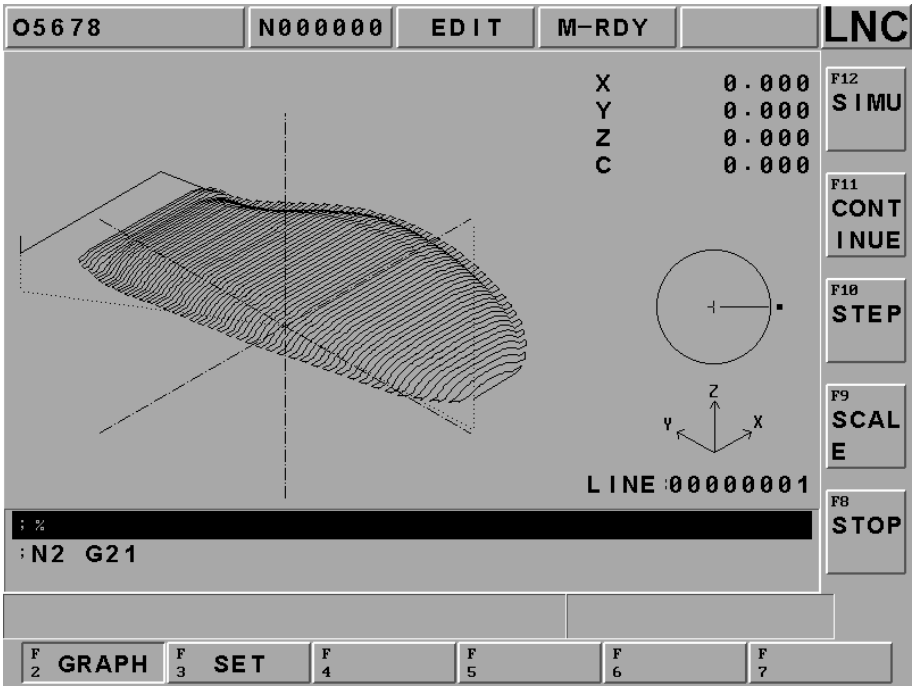



Figure 1.6-1 GRAPH page

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

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

【**STEP**】: 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 <INPUT> 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.

1.6.3 SET

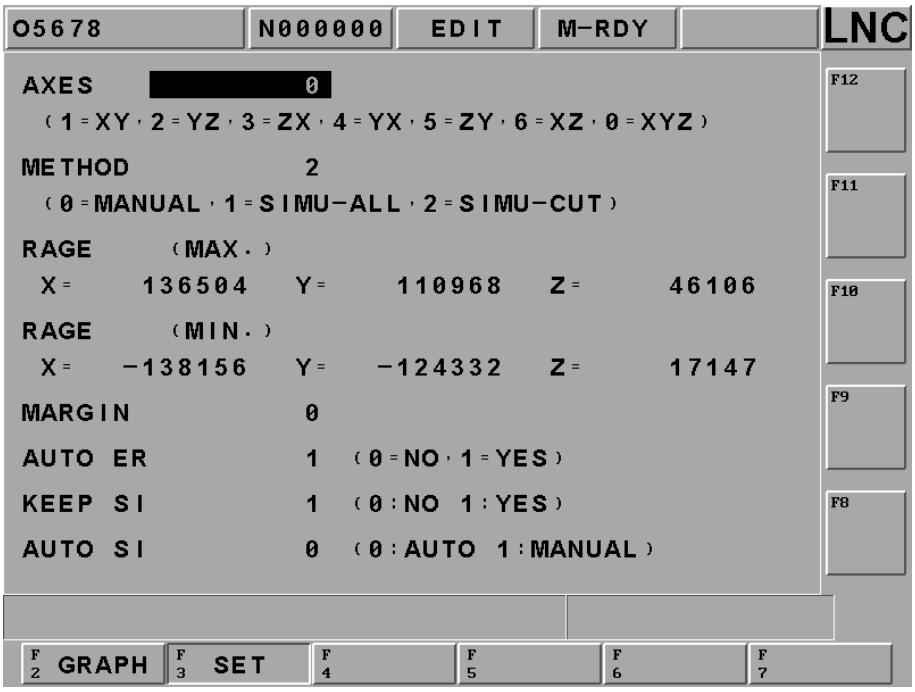
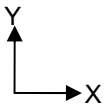
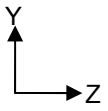
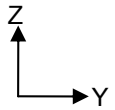
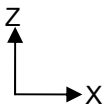
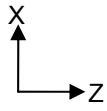
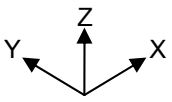
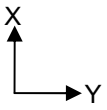



Figure 1.6-2 Main Screen of the Windows Definition

- **Drawing Surface**：the using coordinate view angles are （1=XY, 2=YZ, 3=ZX, 4=YZ, 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 preview result → full traveling**（**including moving path**）：Preview the drawing range as the reading part program path's max/min values, the smallest value.

  **2 preview result → 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.
- **Preserve Border**：Setting the preservation value for【**SET**】screen border.
- **Auto Delete**：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）
- **Preview reserve**：At auto mode, you can set if you want to use previewed graph or to preserve preview.

  0 NO：show exact working path.

  1YES：show current preview path.
- **Auto Preview**：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.

## 1.7 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

### 1.7.1 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.

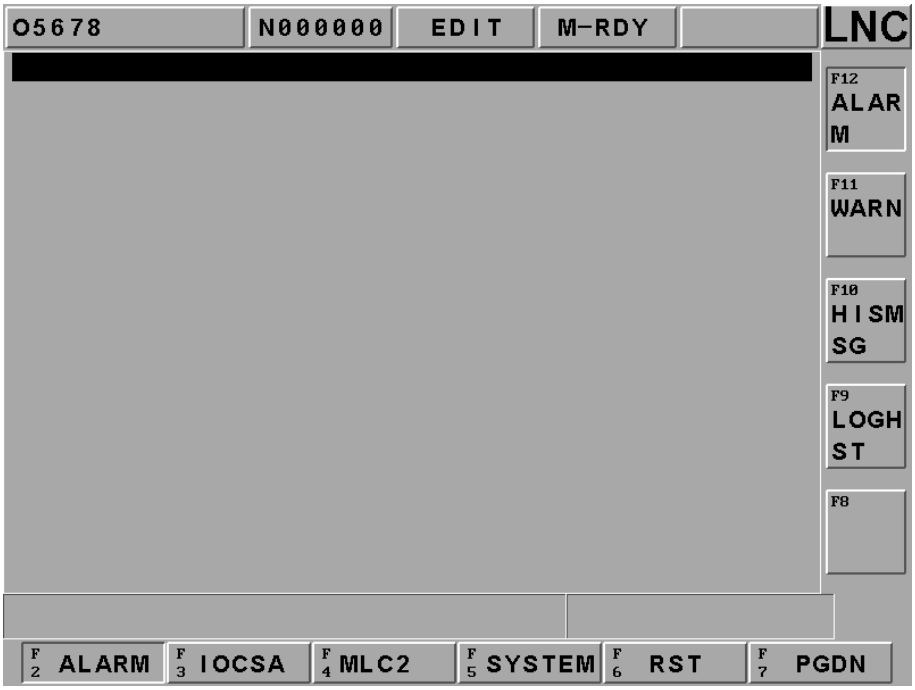
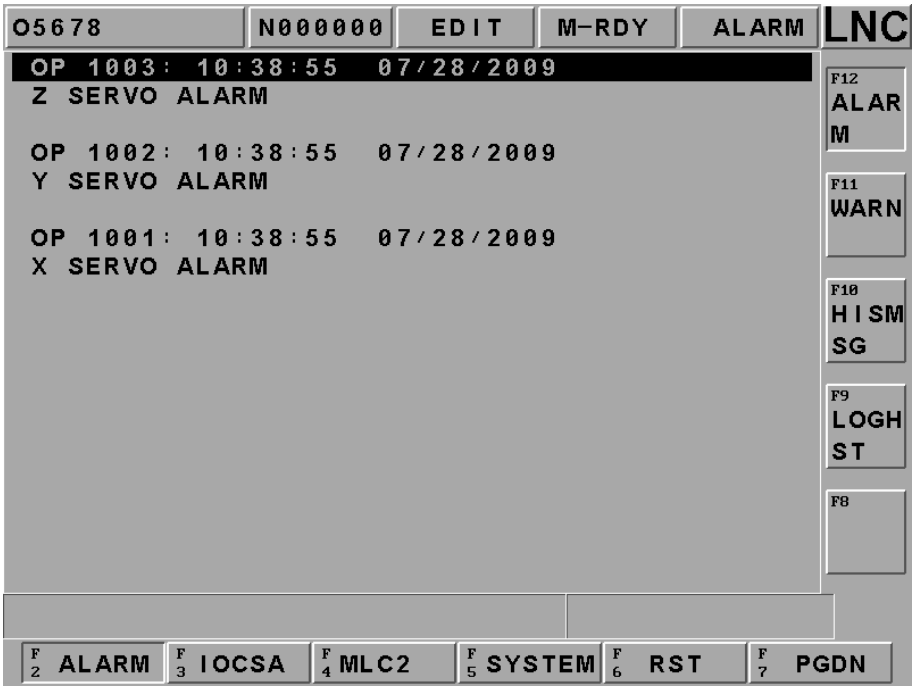
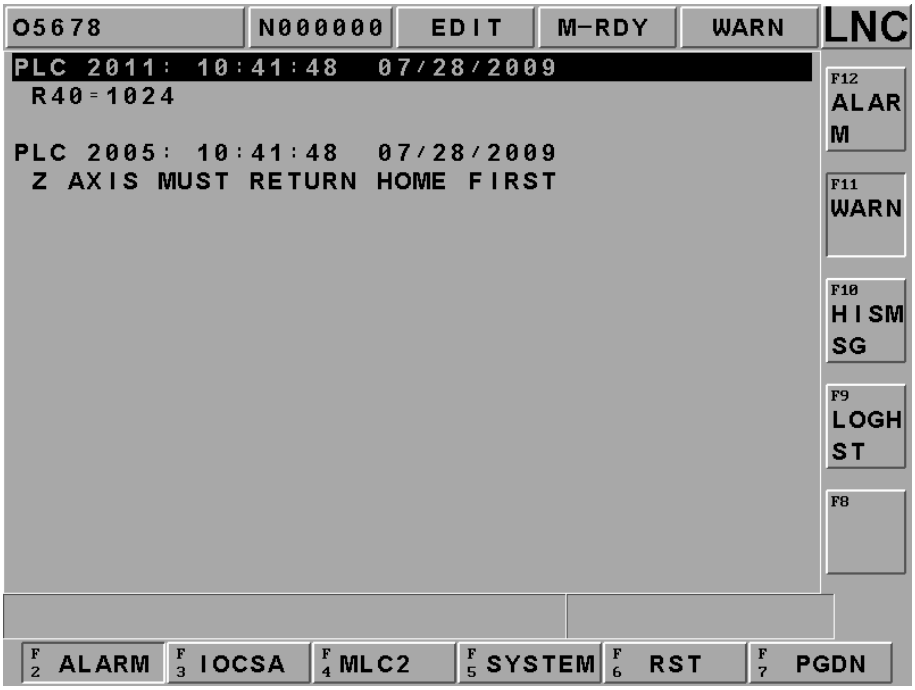



Figure 1.7-1 ALARM screen

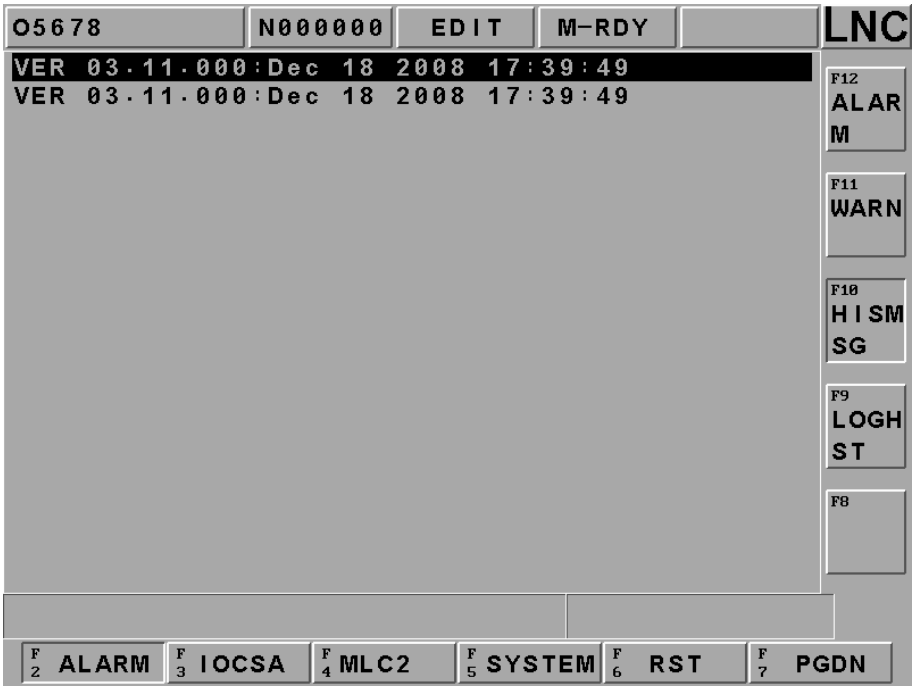
【**ALARM**】Alarm message will occur when there is any problem in system running. This will make the system stop running and also will display the alarm message on the screen. When the problem is solved, must press **<RESET>** in order to release the situation.




【**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.


O5678
N000000
EDIT
M-RDY
WARN
LNC
PLC 2011: 10:41:48 07/28/2009
R40=1024
PLC 2005: 10:41:48 07/28/2009
Z AXIS MUST RETURN HOME FIRST
F12 ALARM
F11 WARN
F10 HISMSG
F9 LOGHST
F8
F2 ALARM
F3 IOCSA
F4 MLC2
F5 SYSTEM
F6 RST
F7 PGDN

【**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.

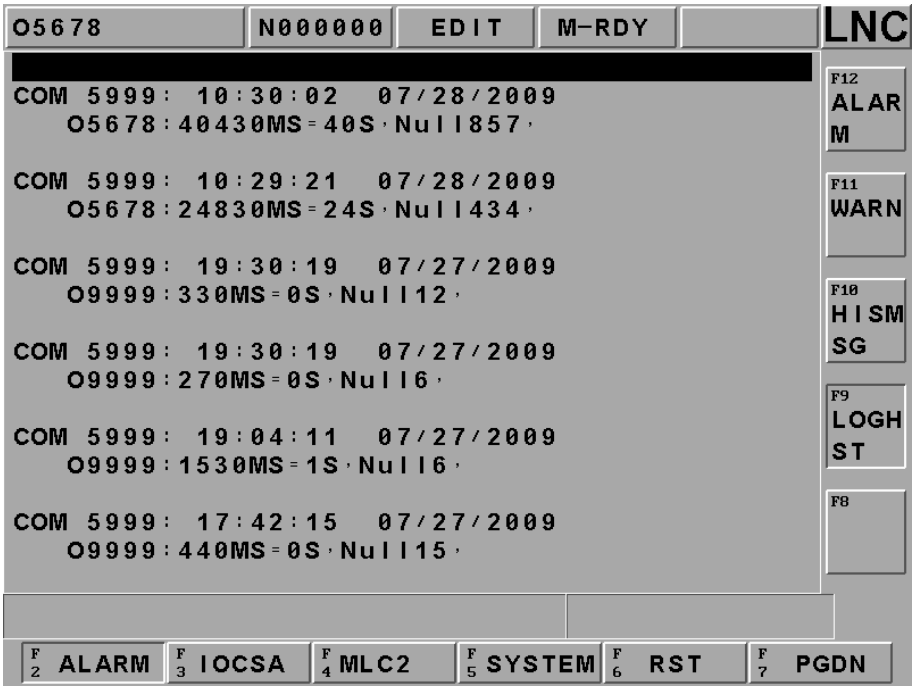



Figure 1.7-2 ALARM Message Screen

1.7.2 IOCSA

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

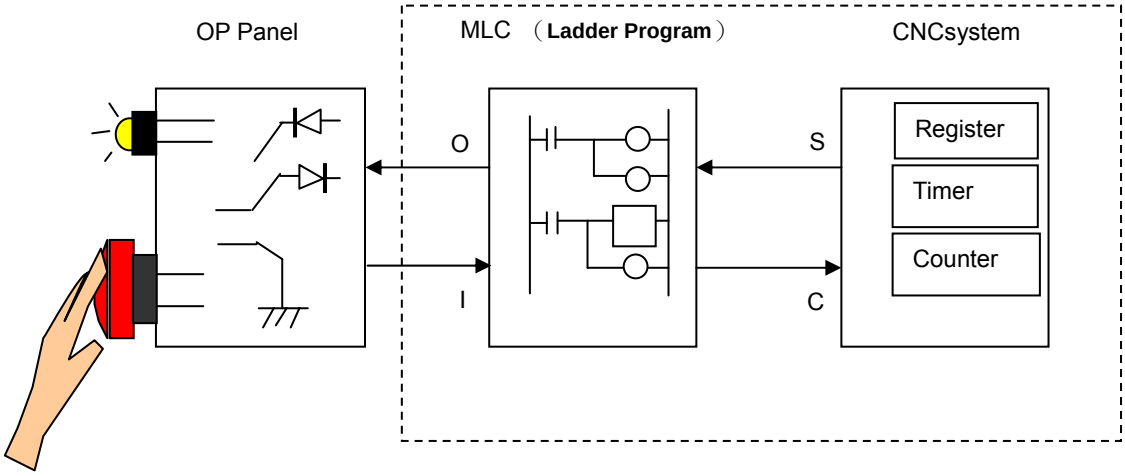



Figure 1.7-3 I/O and system internal condition

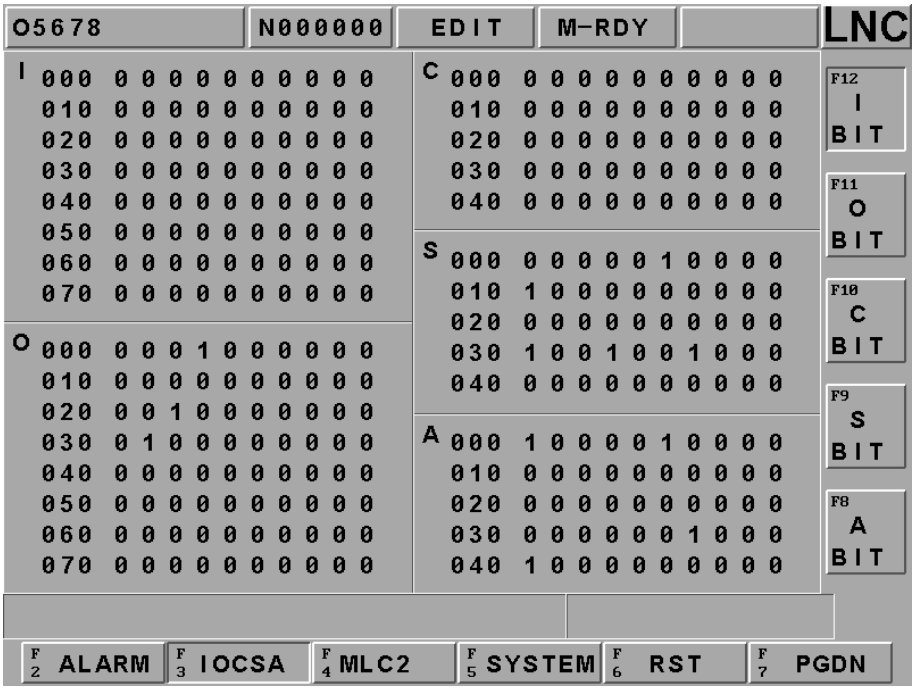



Figure 1.7-4 IOCSA Main Screen

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

### 1.7.3 MLC2

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

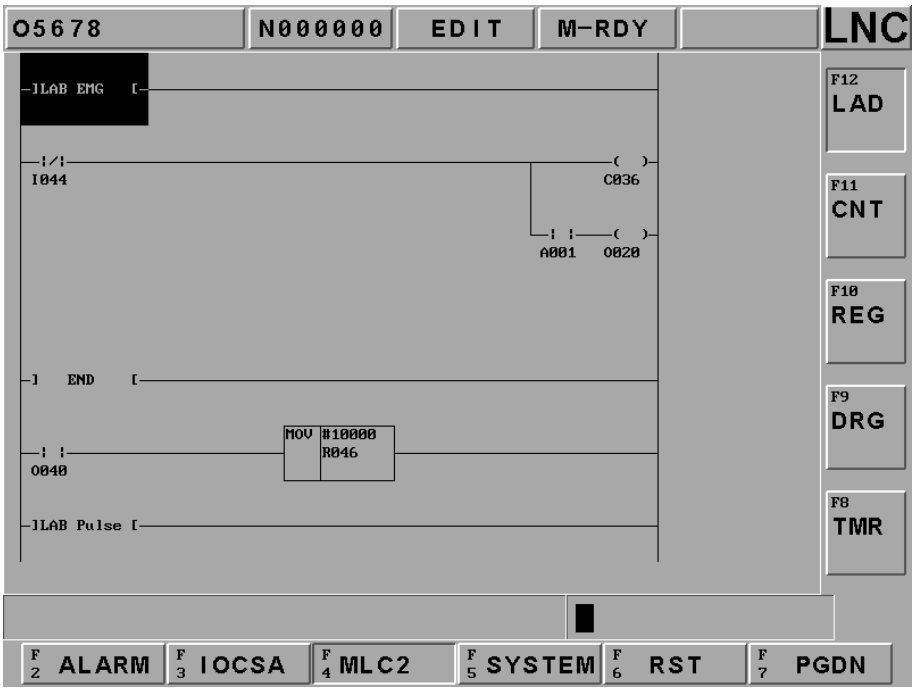
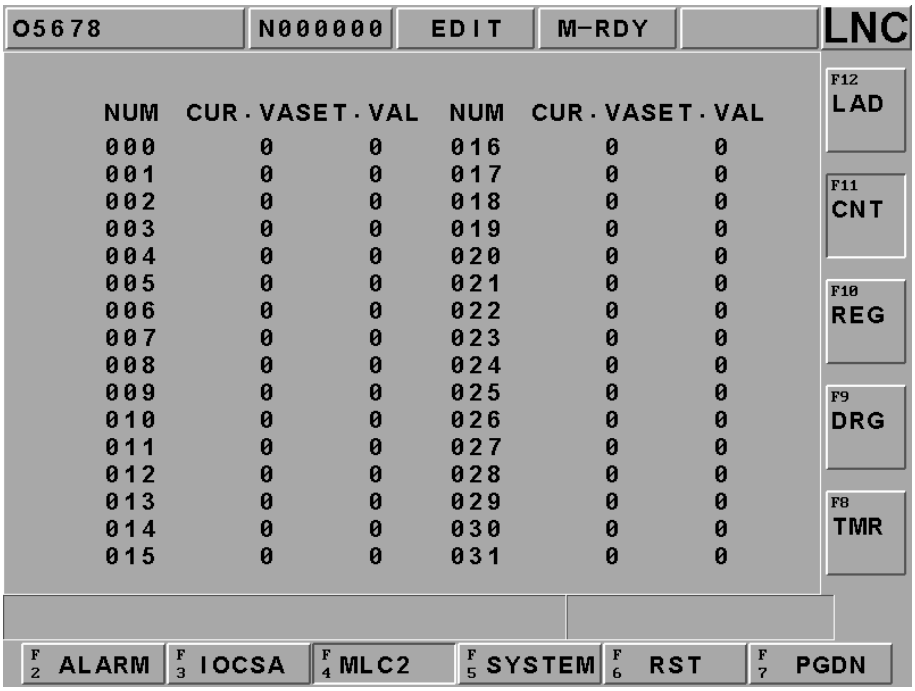


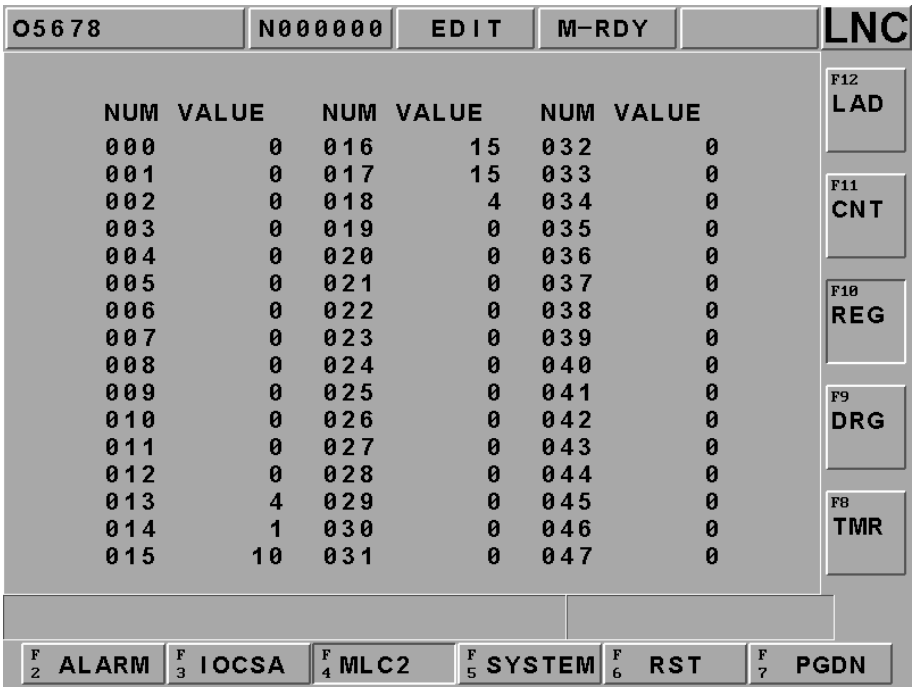

Figure 1.7-5 LADDER Figure

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.

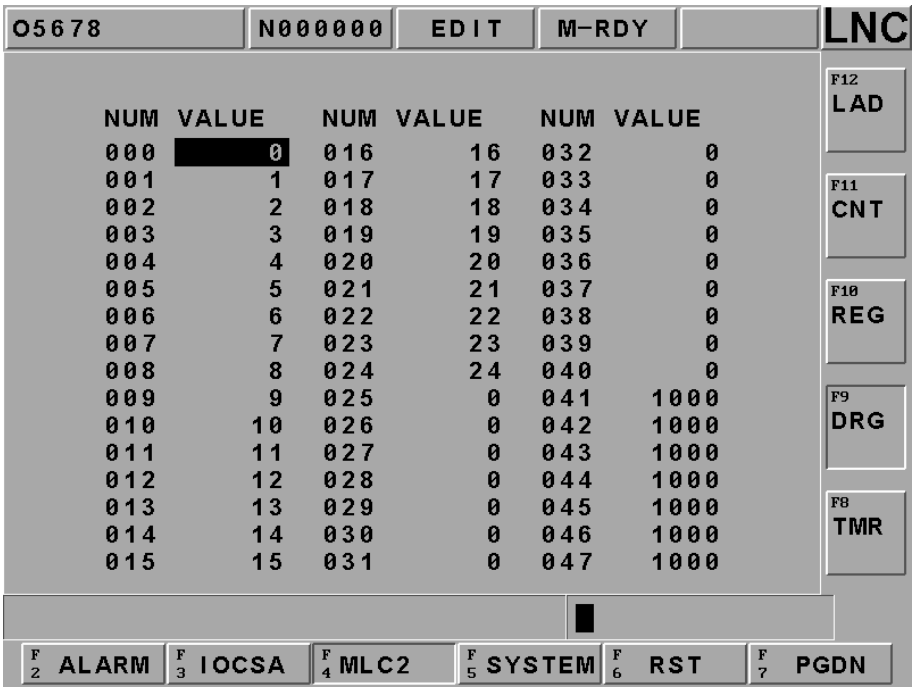
Press【CNT】to check data as below.




Press【REG】to check data as below.

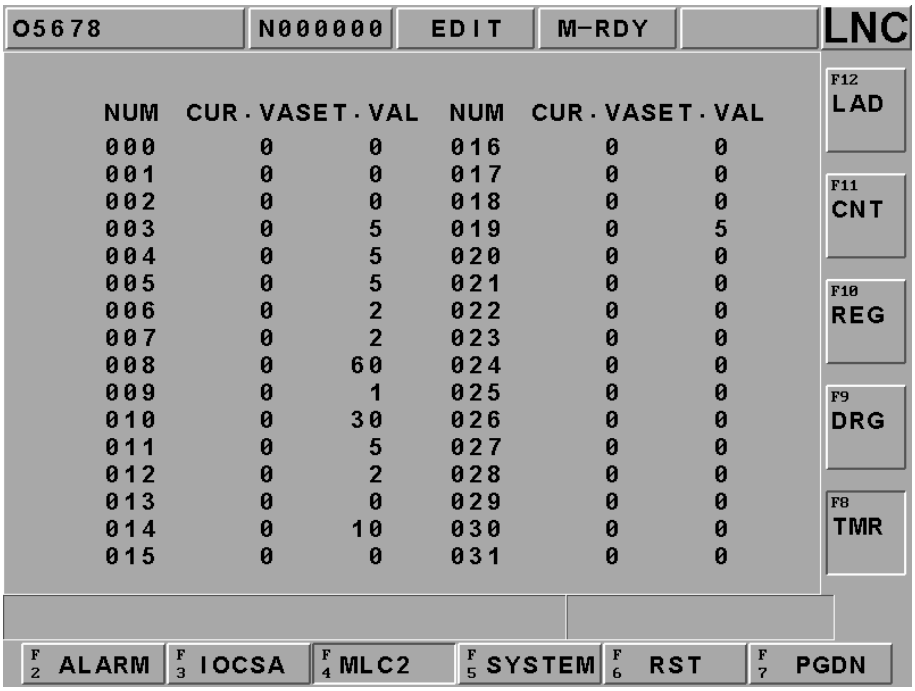


Press【TMR】、【REG】、【DRG】、【TMR】to see data as below.




| 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 |

Press【TMR】to check data as below.




| 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 | |

### 1.7.4 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.)

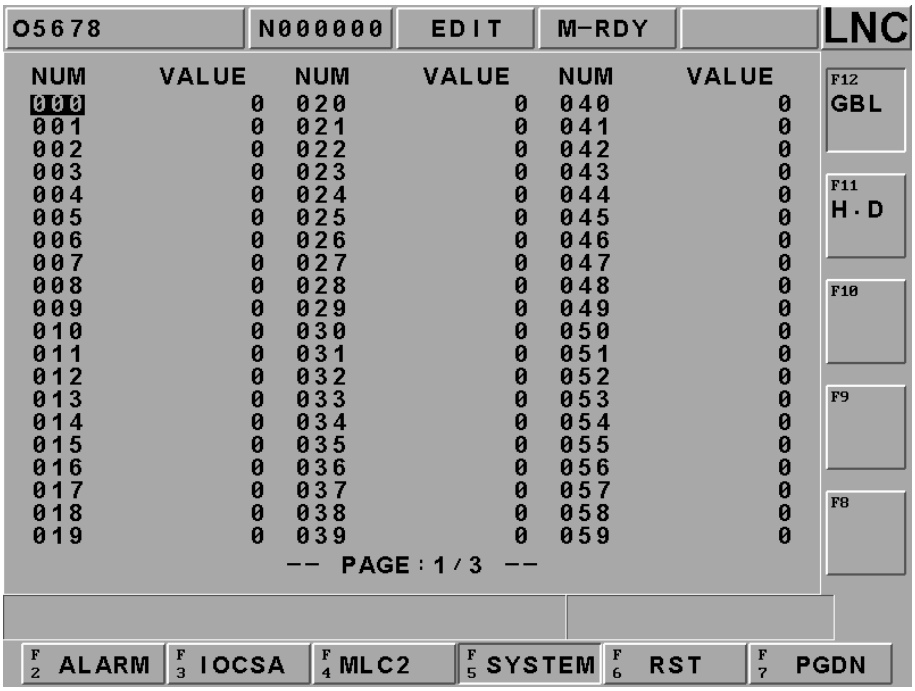



Figure 1.7-6 System Data\System

**【H.D.】**

There are 9 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 position 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.

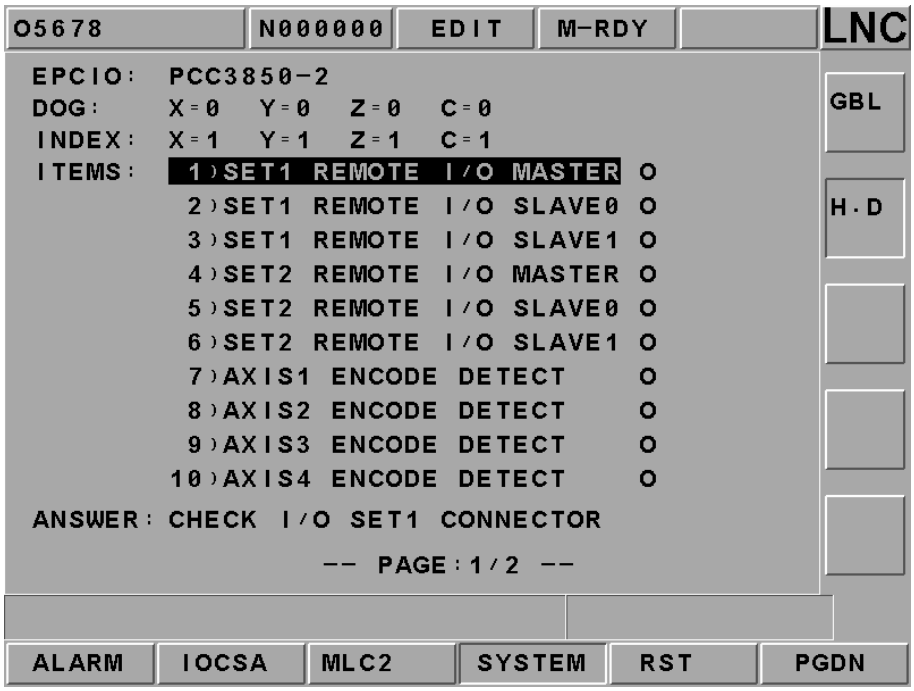



Figure 1.7-7 SYSTEM\H.D.

### 1.7.5 Working Parameter Page

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.

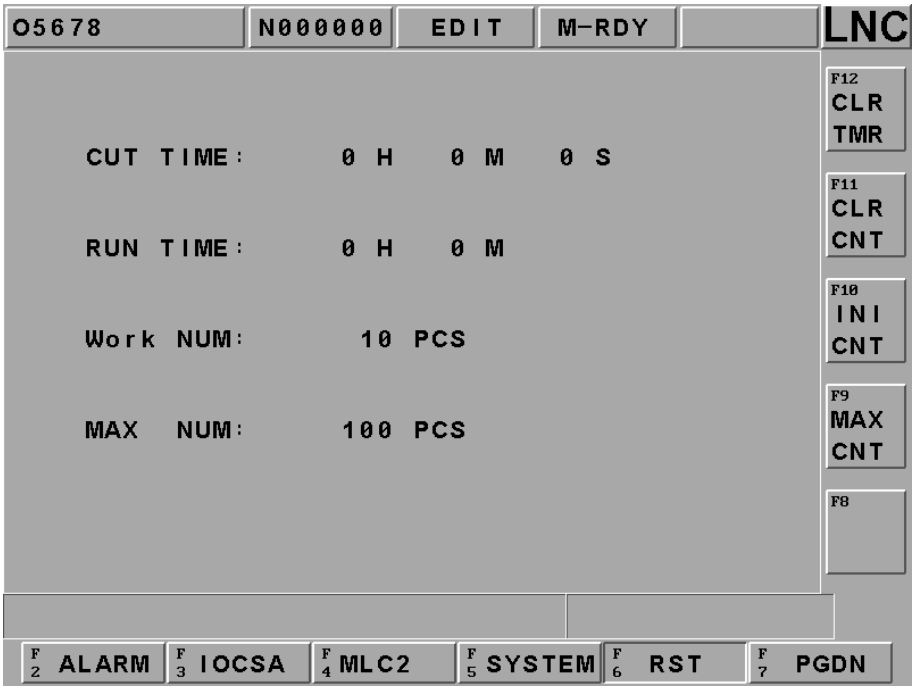



Figure 1.7-8 Working Parameter Page

### 1.7.6 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:

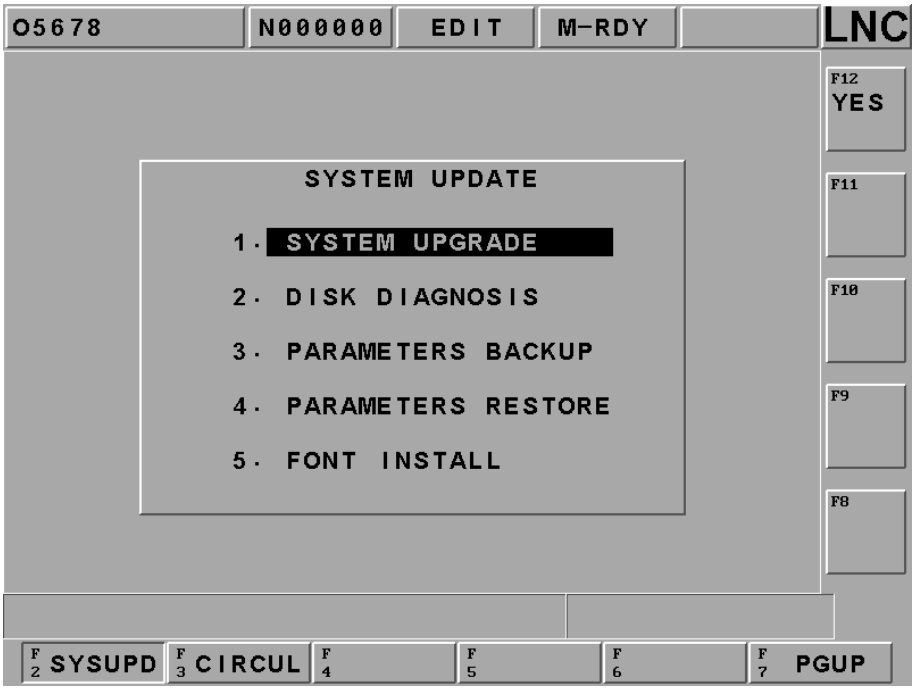



Figure 1.7-9 System Upgrade Screen

- SYSTEM 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.

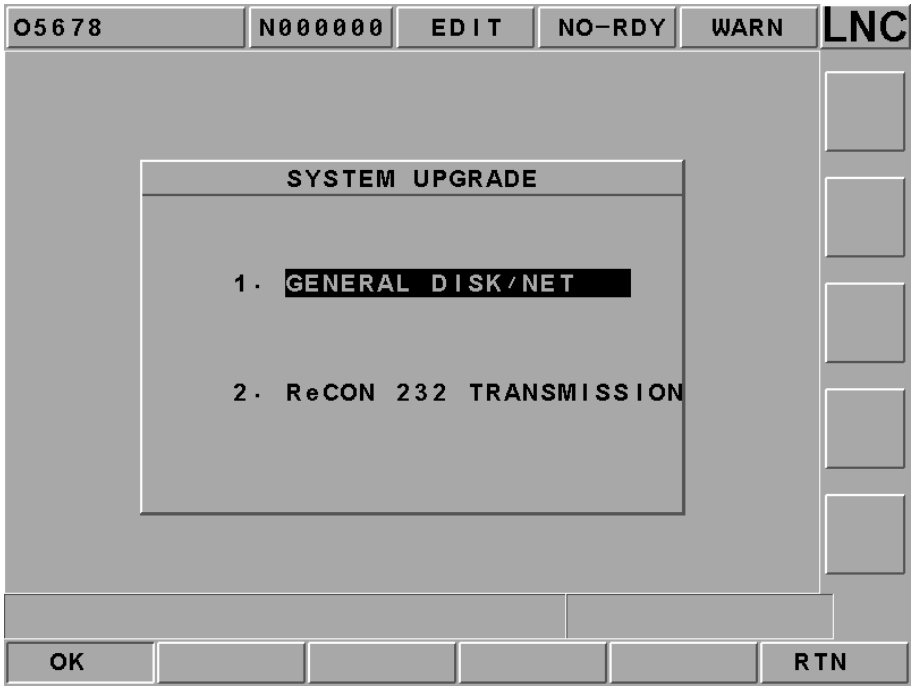
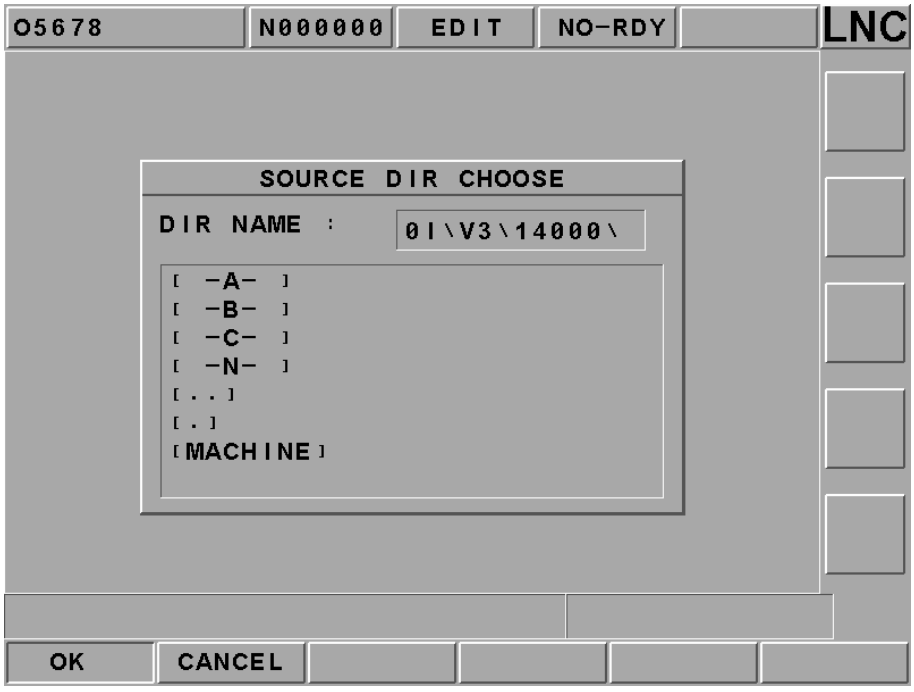



Figure 1.7-10 SYSTEM UPGRADE

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]?
```

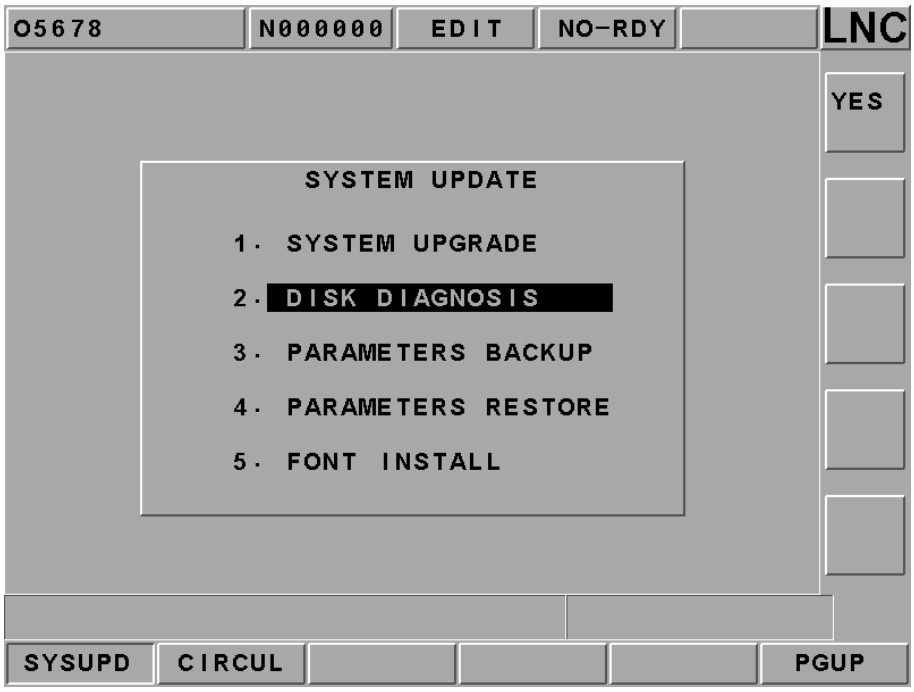
Please wait, file preparing.

```
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 (DISK DIAGNOSIS)




Selecting this function, a confirm dialog box will occur.

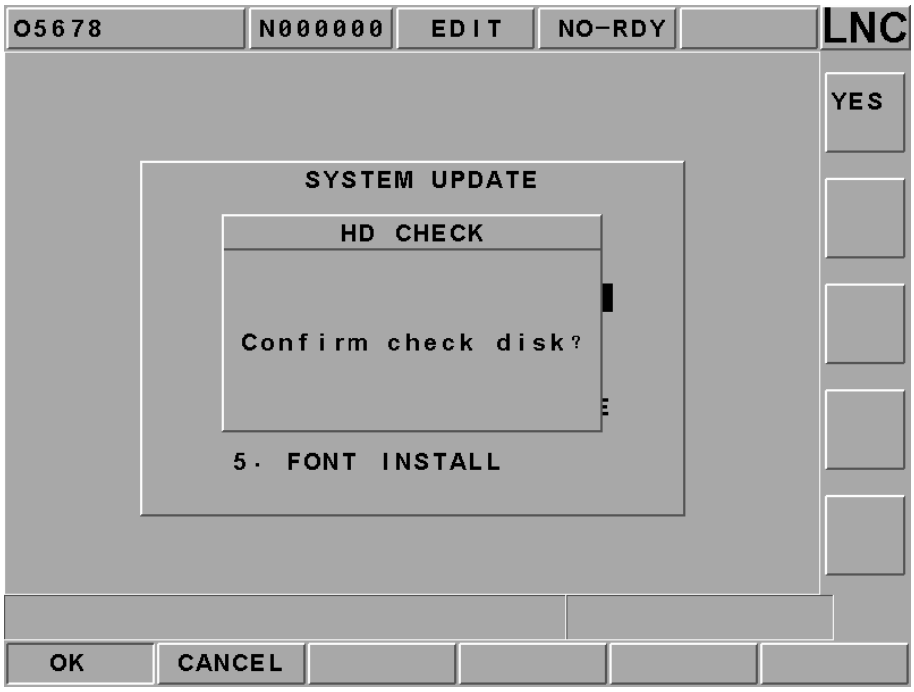



Figure 1.7-11 Confirm Whether or Not to Do the Hardware Checking

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 2 types of working items for users to choose:

（1）anti-virus（2）hard drive scanning（0）exit, as below：

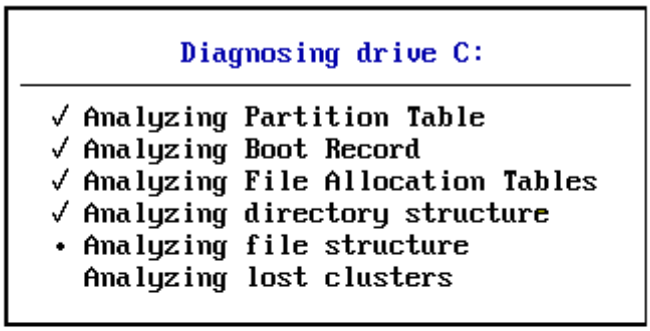
```
1.PCscan
  Virus checking
2.DISK doctor
  disk diagnostic and errer-fix

0.Quit

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

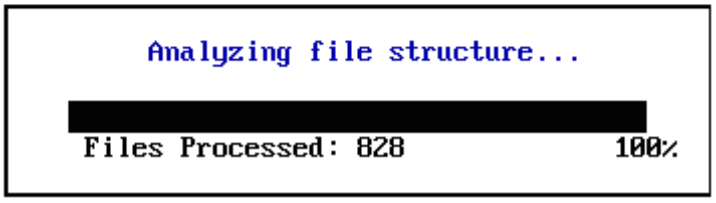
Figure 1.7-12　Hardware Checking Function Selection

**（1）PC scan ,** as below：

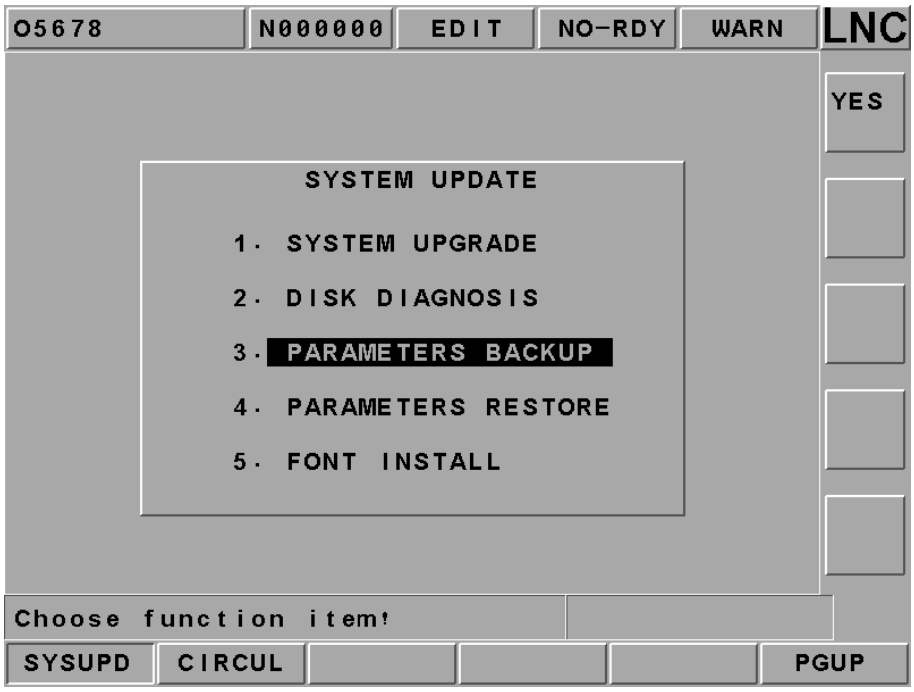
```
DOS/4GW Protected Mode Run-time  Version 2.01a
Copyright (c) Tenberry Software, Inc. 1996
Reading virus pattern file...

                        +-----------------------------------+
                        |        PCSCAN.EXE- V7.50          |
                        | Copyright 1997-2002 TREND MICRO   |
                        +-----------------------------------+

VSAPI version 6.510-1002
Pattern version 837

Scanning for viruses.
No viruses in OS memory.
No viruses in UMB memory.
No viruses in near 640K memory.

Scanning disk partition C:

Scanning boot sector of drive C:

Scanning C:\DOS\LOADSYS.EXE
```

（**2**）**DISK doctor**, as below：



- PARAM BACKUP：




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

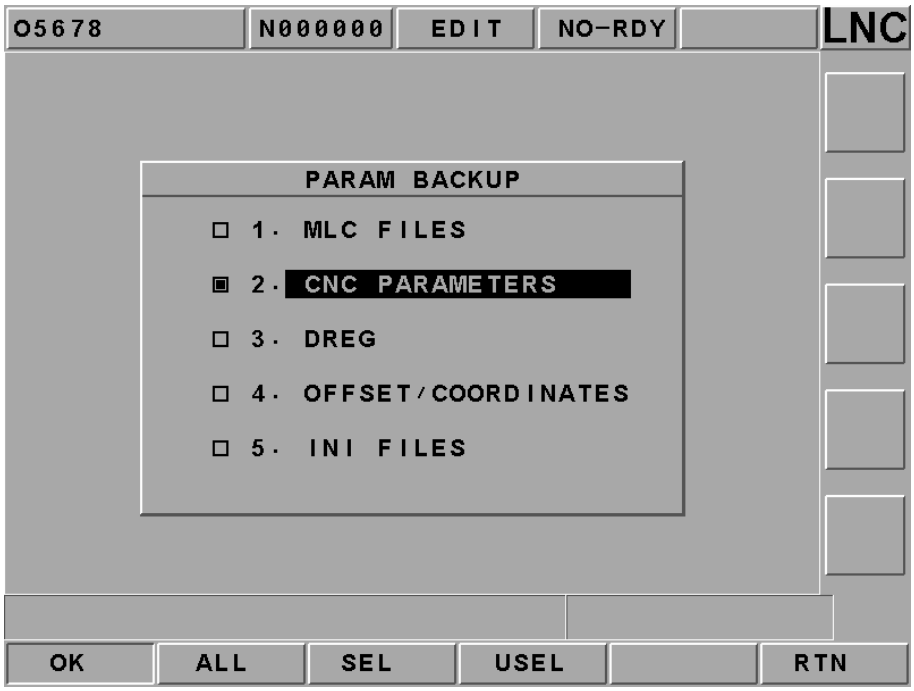
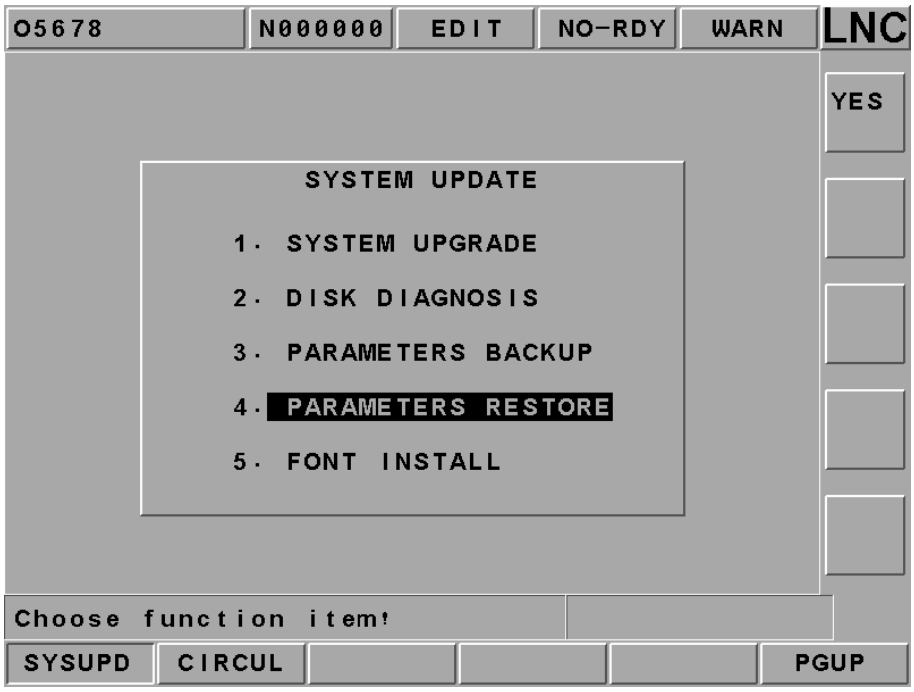



Figure 1.7-13 PARAM BACKUP Selection Screen

- PARAM LEAD-IN：




By choosing this function, a window dialog box will occur to remind users that needs to reboot program：

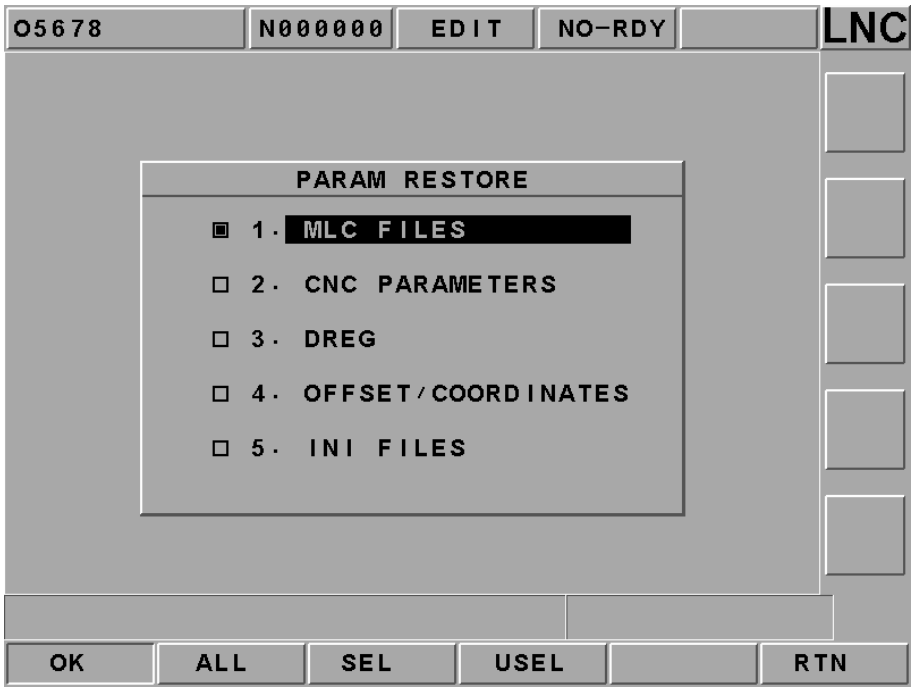



Figure 1.7-14 PARAM LEAD-IN SCREEN

If executing this function. Selecting the “renew parameter” item from the popup dialog box after pressing confirm button. A dialog box of key-in parameter renew source path occurs after pressing confirm button.

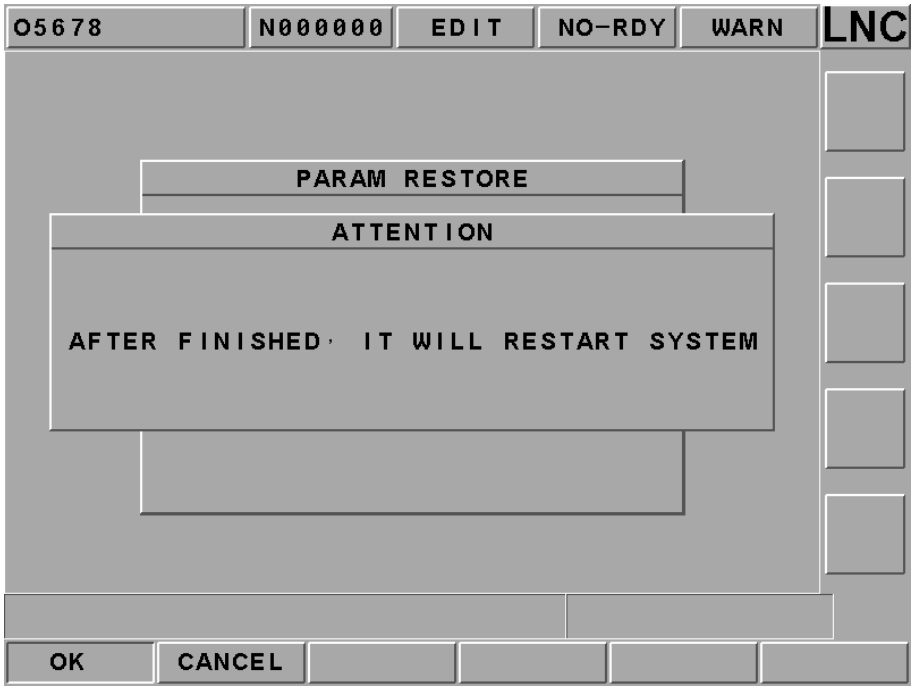
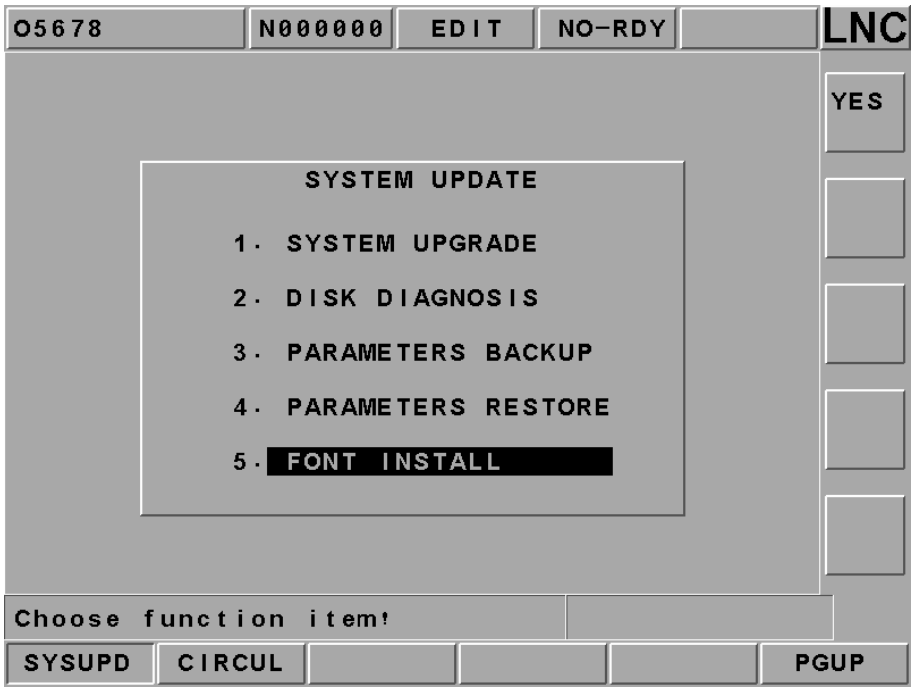



Figure 1.7-15 Reminding Users to Reboot after Key-in Parameter

- **FONT：**




After user selects a designated text font source, please pressing【OK】.

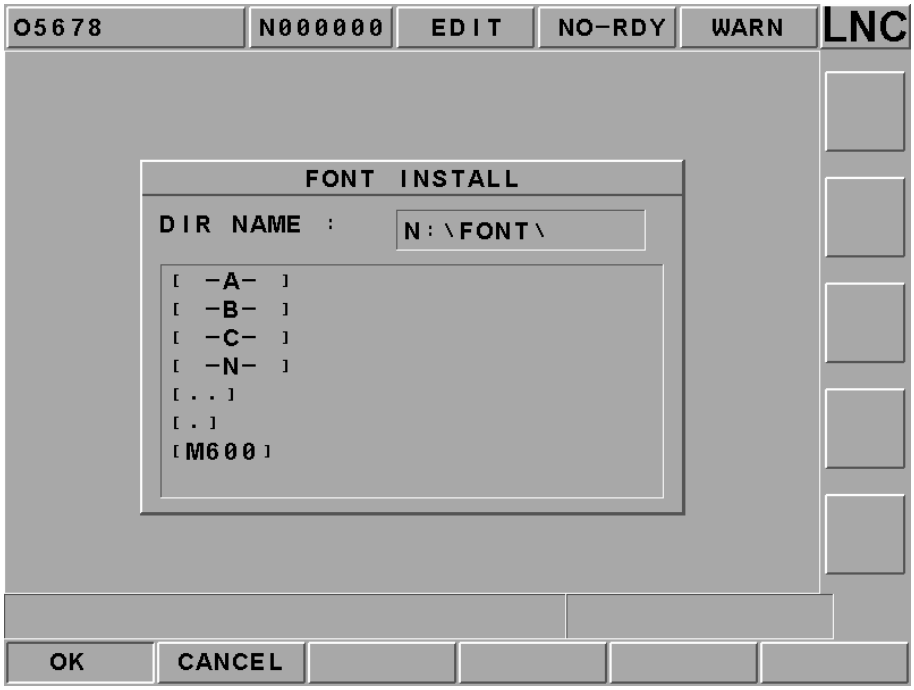



Figure 1.7-16 Text Font Installation Source Page

1.7.7 CIRCUL

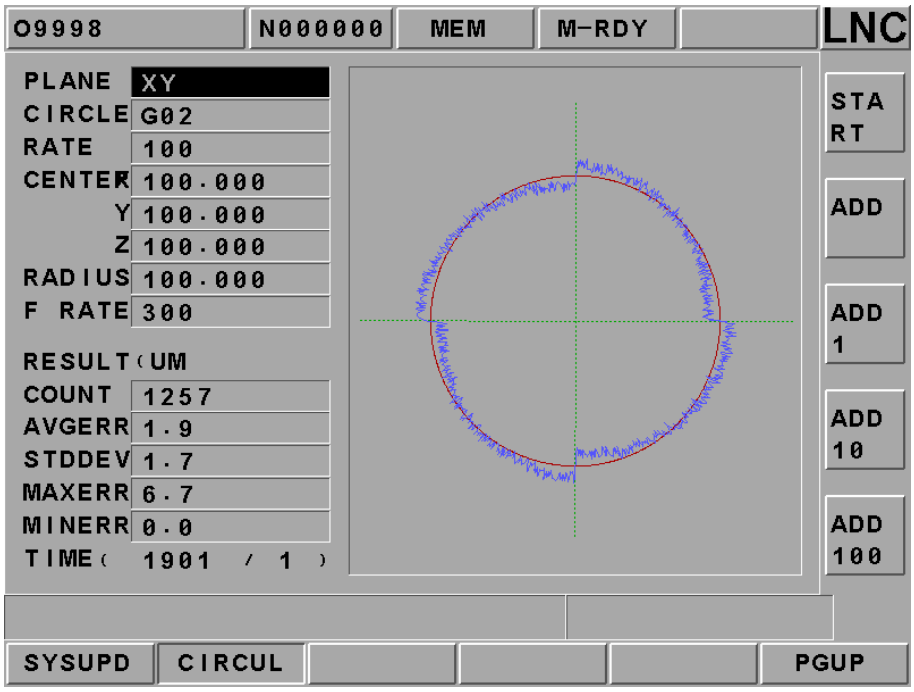



Figure 1.7-17 Ball Bar Test Screen

**【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.

**【Center Point XYZ】** Absolute coordinates（X、Y、Z） of the entered center point.

**【Radius】** Setting radius of circular（mm）.

**【Feedrate】** Setting feedrate of circular（mm/min）.

**【Sample Point #s】** Total numbers of the sample estimation: Sample point numbers= （2*PI*circular radius*60*1000）／（feedrate * sample interval）.

**Note：**PI= 3.1415926

**【Average Error】** Display average error value.

Average error =total/sample point numbers

**【Standard Error】** The difference between the data information and the balance number.

**【Max Error】** The largest circular error value.

**【Min Error】** The smallest circular error value.

**【Ratio】** Display the circular error's zoom-in ratio（N / 1），N is the ratio.

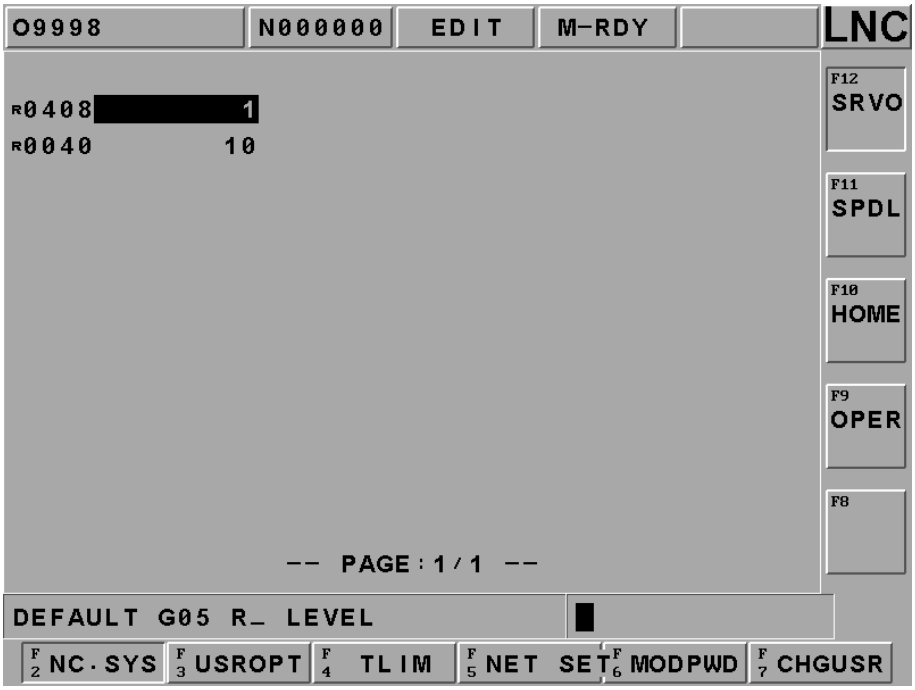
**Note:** the canned cycle error is the distance between each simple point and the center point and also the different value between each simple point and cycle radius.

| | |
|---|---|
| **【Ratio +/-】** | To set the error zoom-in ratio of the sub-function operation button.<br>Pressing this key, this button will be sagged and 【Ratio -】will occur. Pressing the key again, this button will be raising and 【Ratio +】will occur. |
| **【1】** | When the upper key displays 【Ratio +】, the ratio will increase by 1 when pressing this button.<br>When the upper button display 【Ratio -】, the ratio will decrease by 1 when pressing this button. |
| **【10】** | when the upper button displays 【Ratio +】, the ratio will increase by 10 when pressing this button.<br>When the upper button display 【Ratio -】, the ratio will decrease by 10 when pressing this button. |
| **【100】** | when the upper button displays 【Ratio +】, the ratio will increase by 100 when pressing this button.<br>When the upper button display 【Ratio -】, the ratio will decrease by 100 when pressing this button. |
| **【Starting】** | 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. |

## 1.8 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**】

1.8.1 PARAM




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. ㄣ（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.

### 1.8.2 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.

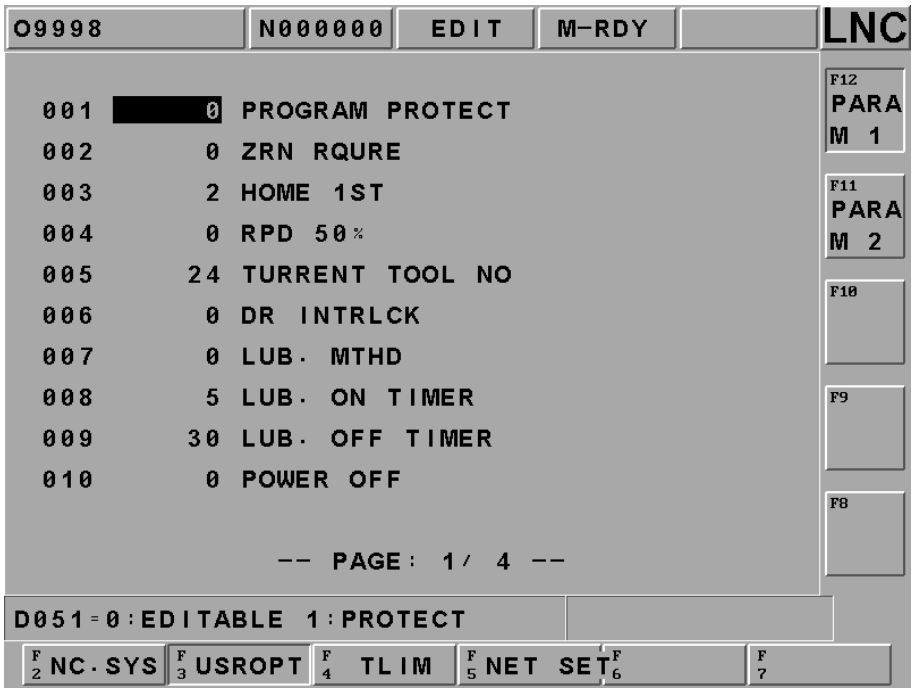
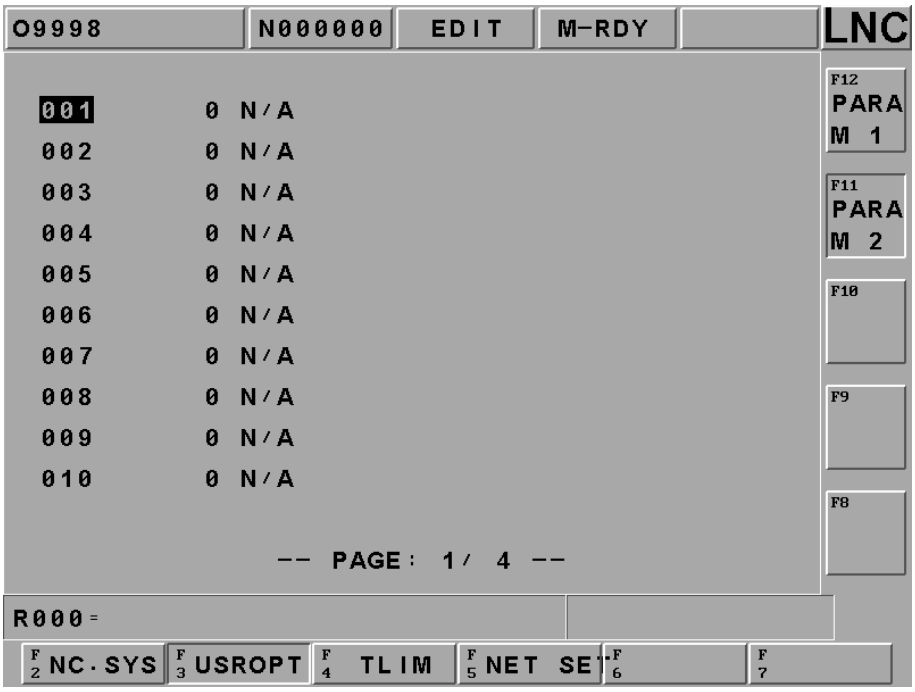



Figure 1.8-1 USROPT

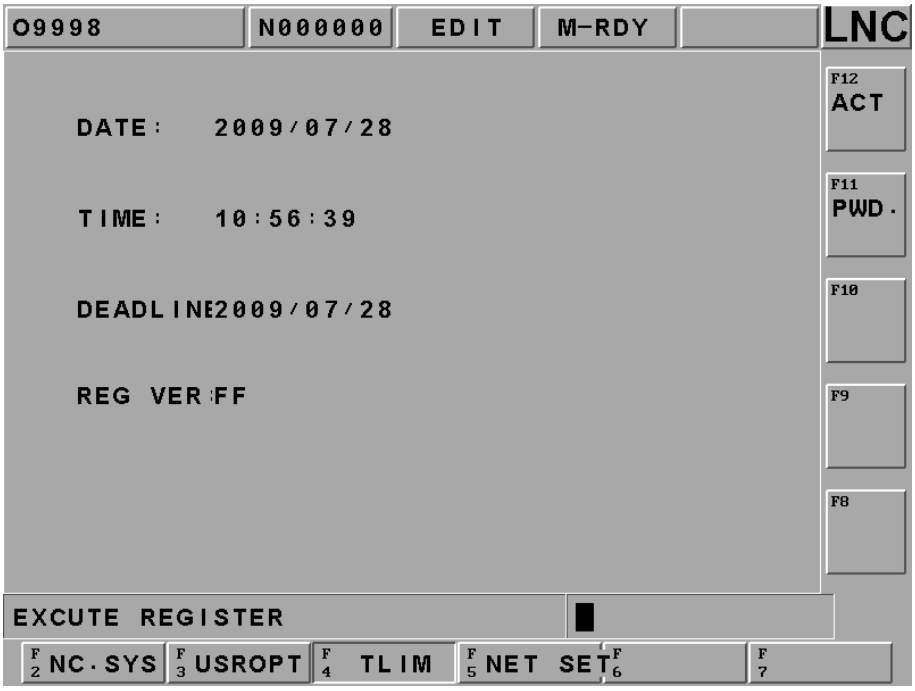
Press【PARAM 2】page, there are 40 sets of parameters. The way to change the parameter is the same to system parameter, please refer to last chapter.


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

1.8.3 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.




- 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.】

1.8.4 NET SET

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

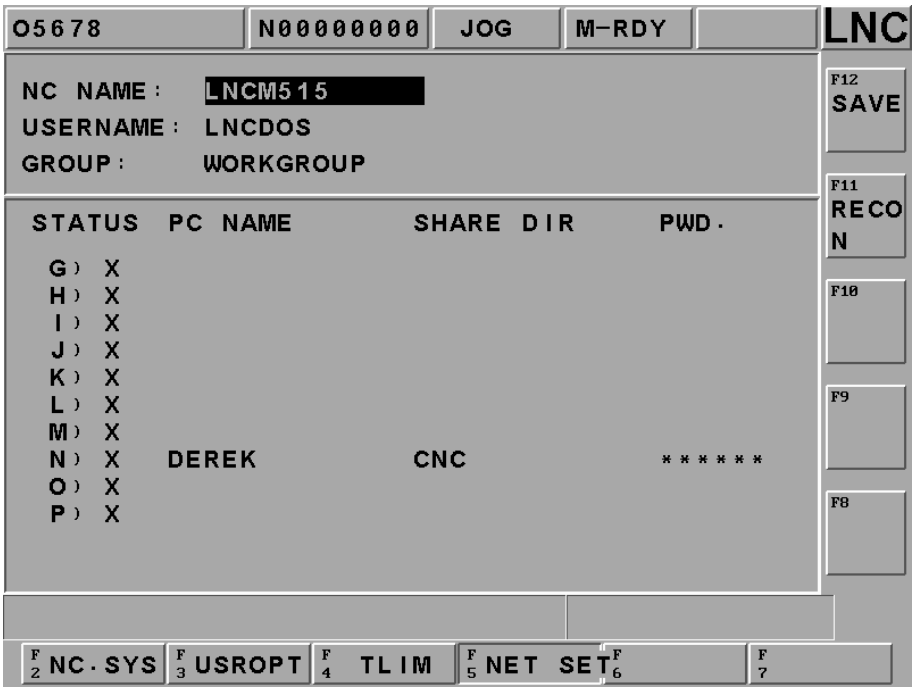
The operation method is as below：

Change to <PARAM>, press【CHGUSR】and change to machine maker level and input P651 and press INPUT, system will search P651and set this parameter to be as below（0 means NET method；1 means ReCON method）, after setting, please reboot the system.

**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 press【ReCONNECT】, the system will reconnect again.




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.

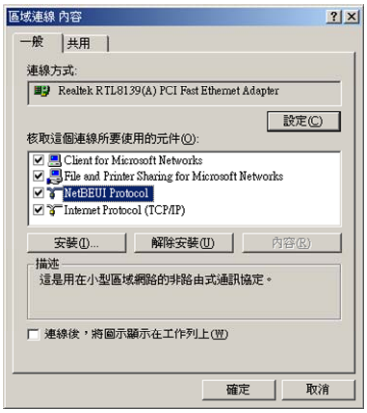
**Share File**：The share files on the PC

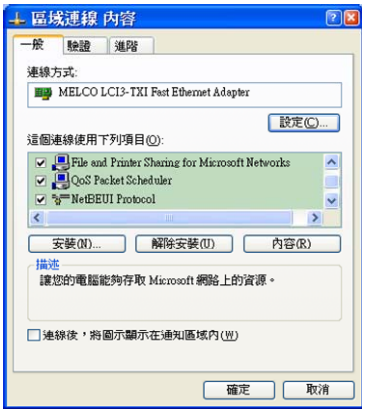
**Password**：This is the password for controller to log into PC. Also means PC end LNCDOS user's password.

**PC end NET Connetion 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.

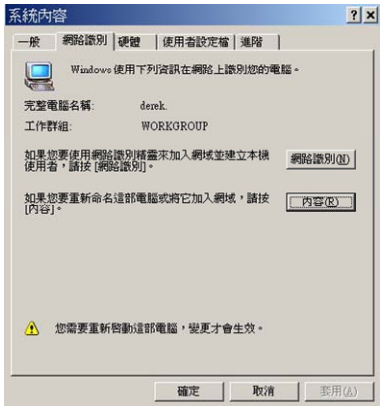
**Note**：Nwlink NetBIOS, this protocol can't be drove, otherwise the connection will fail.

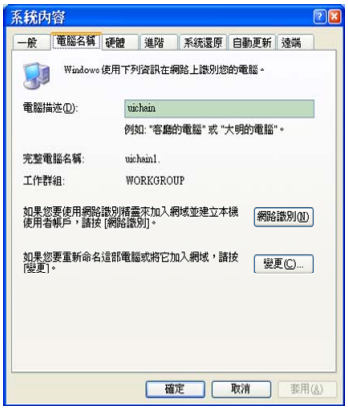




**WINDOWS-98** **WINDOWS-2000** **WINDOWS-XP**

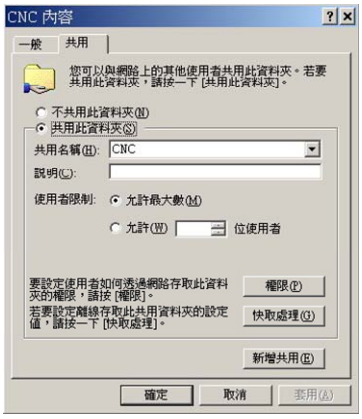
B.Confirm [PC name] and [Work Group] is correct setting.（Controller side's PC name and work group name should be the same, and vice versa）


**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.

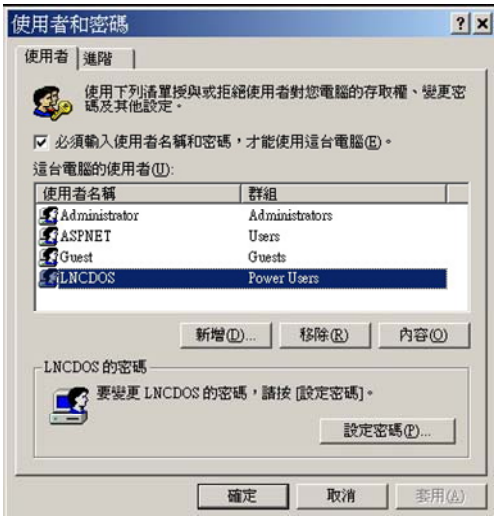



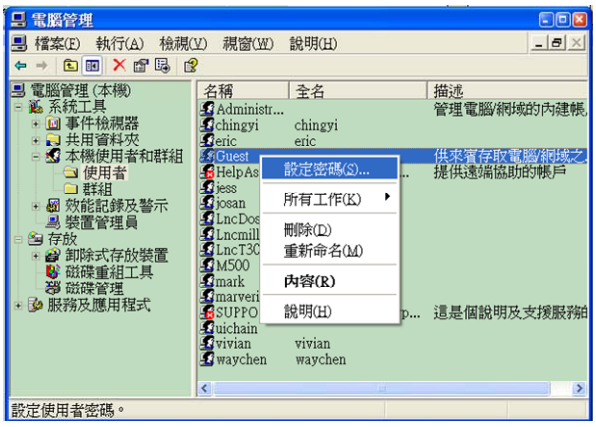


**WINDOWS-98** **WINDOWS-2000** **WINDOWS-XP**

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

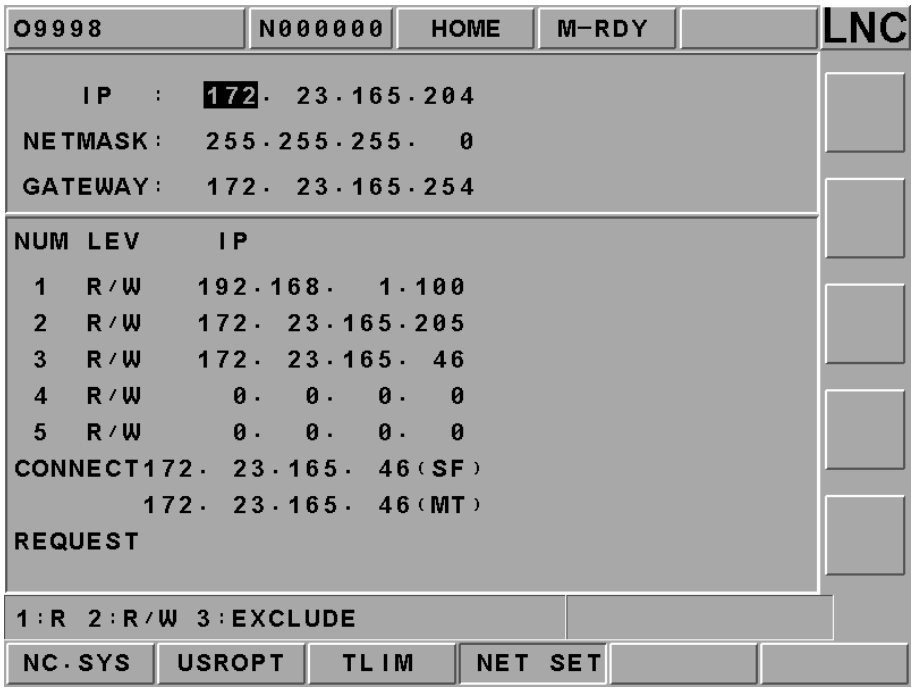
（Window98：If you have files to share, no need to add new users.）




**WINDOWS-2000** **WINDOWS-XP**

**Use【ReCON】to connect and the setting of PC end：**




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.

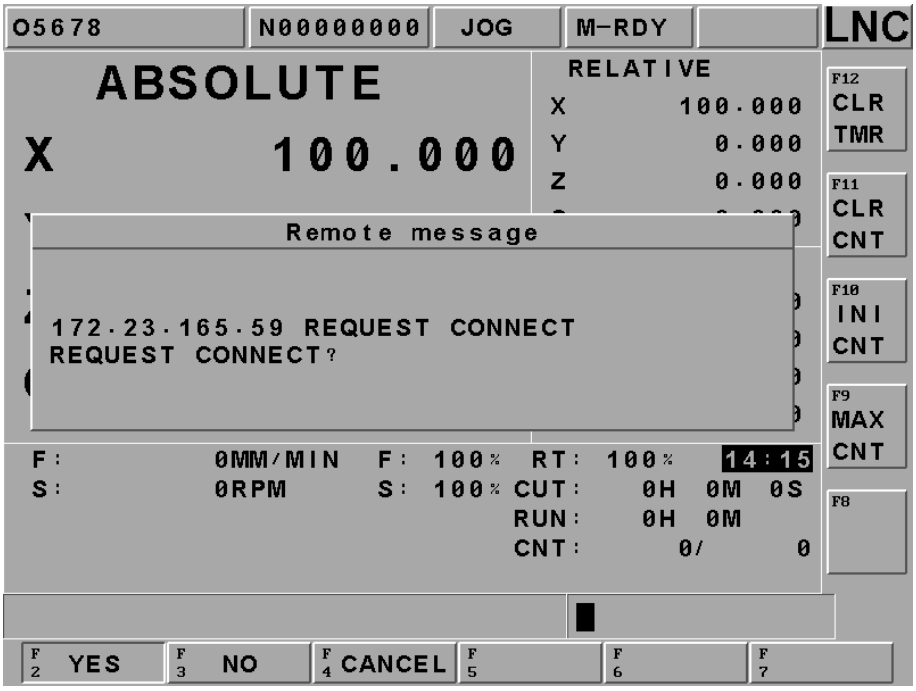
**3. GATEWAY**：Please set this gateway the same with PC end.

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

## 1.9 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**

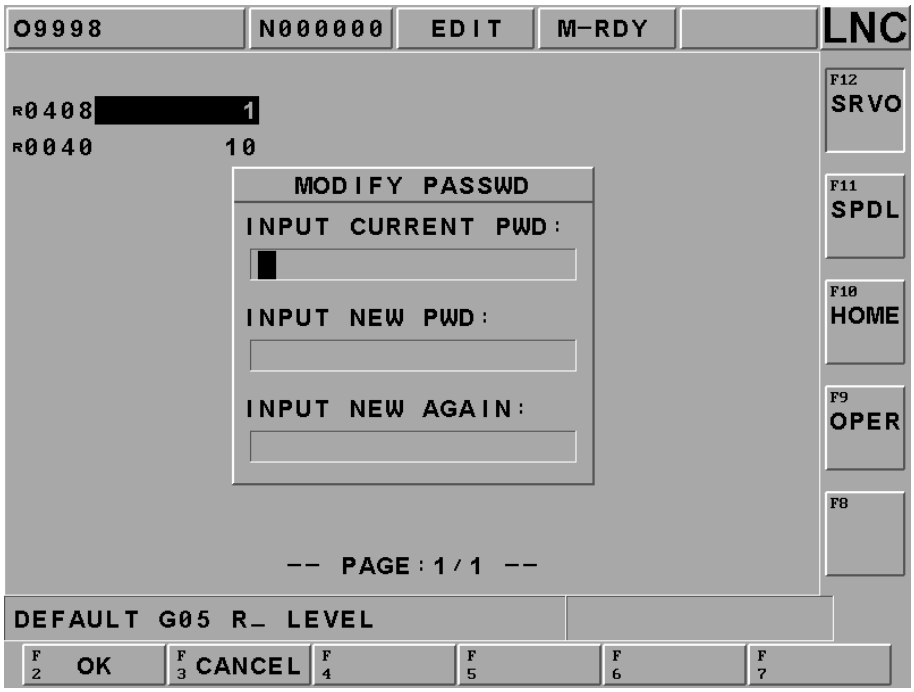
If you want to cancel the Ip address, you can move cursor to this are and press INPUT to make the data to be 0. Next time when you connect to PC, it will show a dialogue again to request you fill up the IP address.

**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 onit, 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.

### 1.8.5 【Change password】

Press【**CHGUSR**】and the screen will show changing password dialogue. User can change password of modifying parameter function. However, password of exit is not to be changed here.




## ■ 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.

- **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.

### 1.8.6 【Switch user】

In order to prevent end-user to change some parameters and make the machine error, our controller provides two identity for using, End-User and Machine Makers. With different identity, there will also have different sub-functions keys.

- **End-Users**

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

When changing to End-User, you don't have to input password and you can change to End-User. But if you want to change parameter setting, you need to input password to confirm, the default password is『0000』

- **Machine Makers**

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

  When changing to Machine Maker, you will need to input default password 『0000』 while log in and change parameters.

# 2 OP Panel Operation

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 8 sections according to different functions：

(1) LED SIGNAL

(2) MODE SELECT

(3) AXIS SELECTION

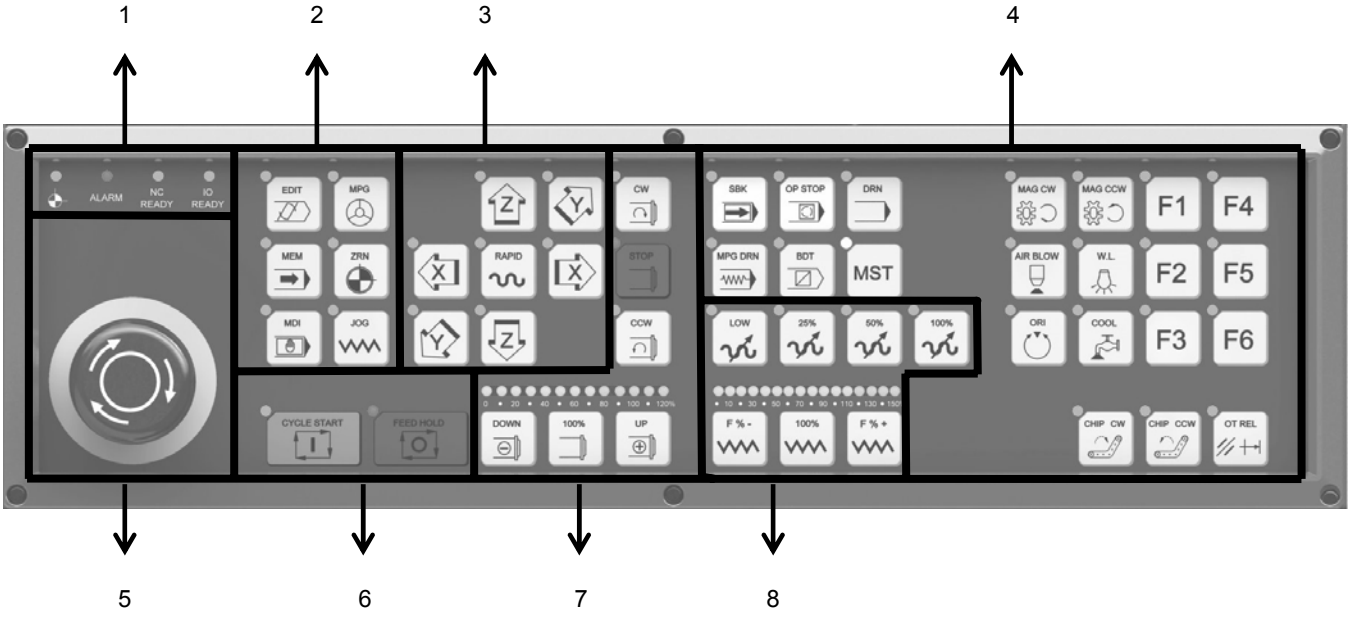
(4) AUXILIARY

(5) EMERGENCY STOP

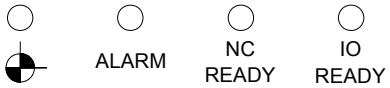
(6) CYCLE START & FEED HOLD

(7) SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH

(8) FEEDRATE ADJUSTMENT



## 2.2 LED SIGNAL



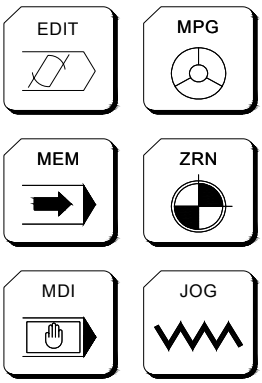

1：AXIS ZRN

2：ALARM, LED ON

3：POWER ON, LED ON

4：I/O MOTION CARD SELF HARDWARE TESTING, CONFIRM OK, LED ON

## 2.3 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**）

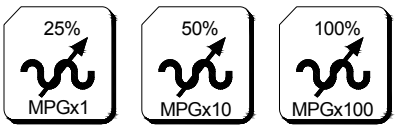
Users can do auto program executing.

(3). （**MDI**）

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

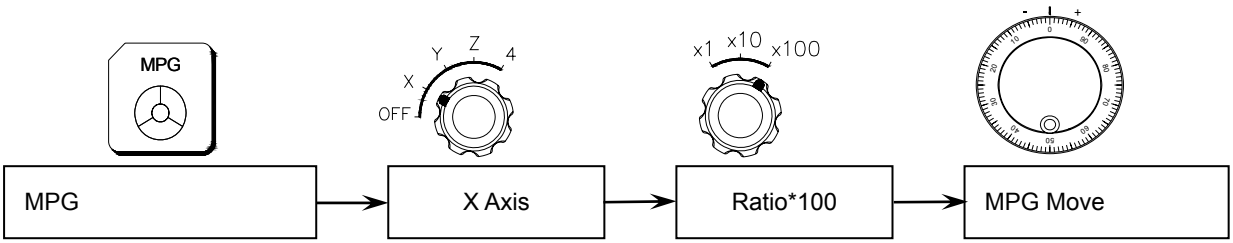
(4). （**MPG**）

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.




RAPID and MPG ratio mutual key

**MPG Flow：Move X axis**




(5). **ZRN**

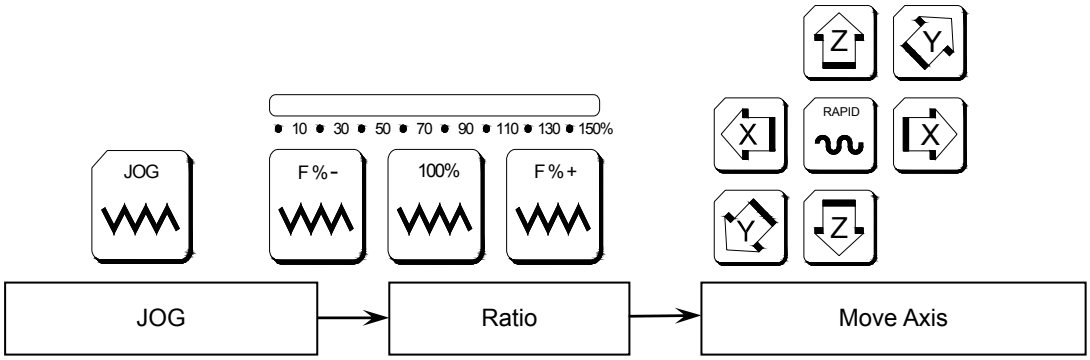
Operating home return of each axis. When changing to this mode, pressing home return direction button of each axis (the same button as JOG). This axis will start home return procedure according to the setting speed of the parameter until reaching DOG. Also, servo axis starts searching for the home position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

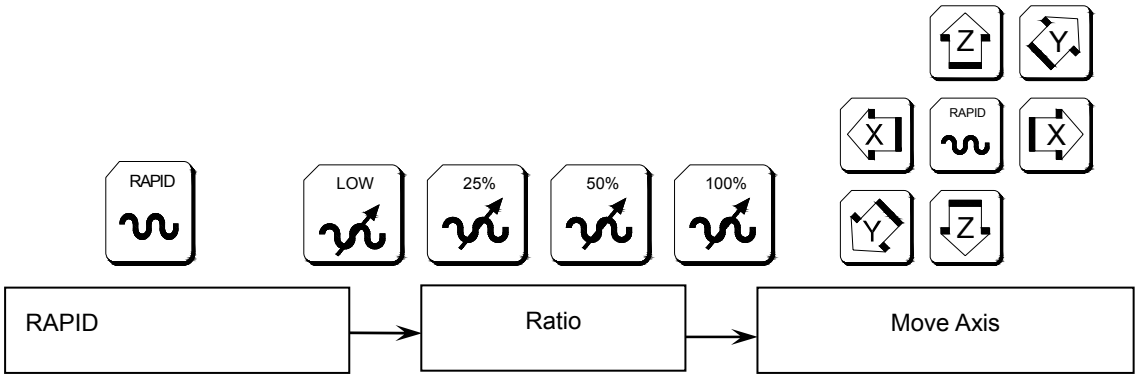
**JOG Flow：Move axis**



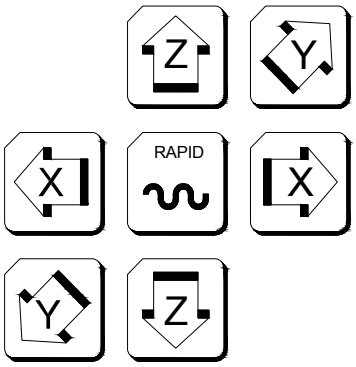
(7). **RAPID**

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.

**RAPID Flow：Move Axis**



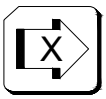
## 2.4 AXIS SELECTION




- These buttons are used to command moving axis direction under JOG, RAPID and ZRN modes：
- For example, pressing 

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 

and 

under RAPID mode, X axis will go rapidly to plus direction, release this key and the X axis wil stop. Other axis has the same functions.
- Pressing 

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

## 2.5 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：

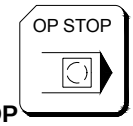

**SBK**

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.


**MPG DRN**

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

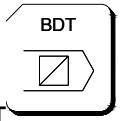
When system cycle start, MPG can control the operation and change the program coordinate to move with servo axis. When MPG runs faster, the program will be executed faster, but it will be faster then the program federate command. When MPG stops, the program will stop.




**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.



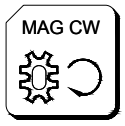

**BDT**

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


**MST**

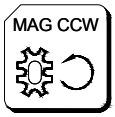
**MST**, press this button to let **MST ignore** function ON, press again to release this function**.**

Part program with " M " " S " " T " code will be ignored and will not be executed.




**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)

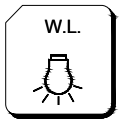



**MAG CCW**

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

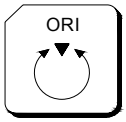

**BLOW**

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



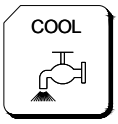

**W.L**

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



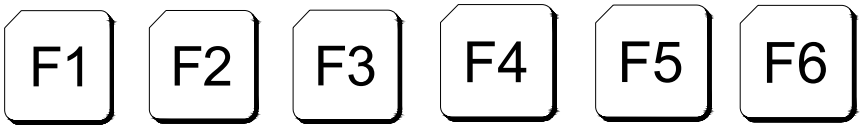

**ORI**

Press this button to do orientation, press reset to release this function.



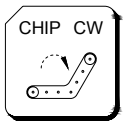

**COOL**

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



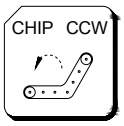

**F1 ~ F6**

F1 ~ F6：These buttons are for machine makers to define.



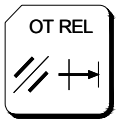

**CHIP CW**

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




**CHIP CCW**

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

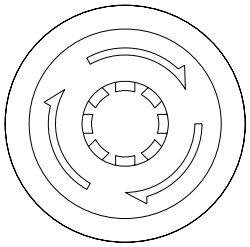



**OT REL**

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.

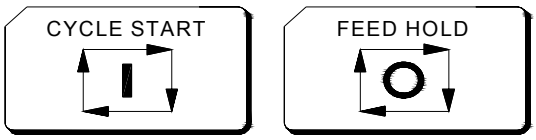
## 2.6 EMG-STOP



Using this butting 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.

## 2.7 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**

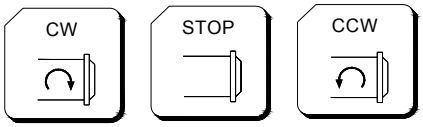
Users can key-in a block of program commanding such as G91 G01 X100. Z100 in MDI mode, then press “CYCLE START” button to execute this single block command. The intension of this executing mode is different than that of MEM mode, which is always used in testing some motion. The light will remain ON during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

## 2.8 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.

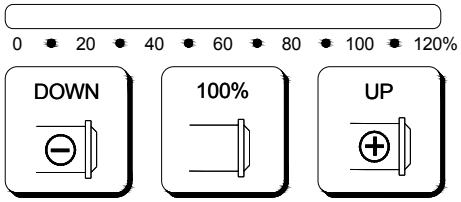
CW ：Rotate in CW direction.

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**



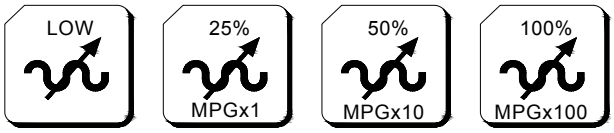

When spindle is under MEM or manual mode, user can use M3（or M4）Sxx…..to turn on spindle. Rotation speed of spindle can be tuned by UP＋、DOWN－ and 0% ~ 120%.

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.

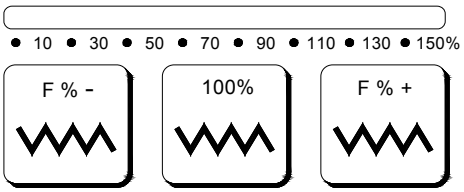
## 2.9 Feedrate & JOG Turn Key

**Feedrate turn key**




These are the turn key of ratio speed for G00, RAPID and ZRN. There are 4 sections of speed, LOW、25%、50% and 100 %. LOW speed will be determined by Pr.40.

**JOG turn key**




**FEEDRATE：**

Under MEM or MDI mode, use G01…F… command to move servo axes. Its exact federate can be tuned by this turn key from 0% to 150%.

For example, F100=100mm/min, but if changing turn key to be 50%, its exact federate is 50mm/min.
At most machines, this turn key is effective to JOG mode.
For example, current setting is 10%, 10mm/min and press 100%, the exact federate will be changed to 100mm/min.

# 3 RS232 COMMUNICATION SOFTWARE

## 3.1 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

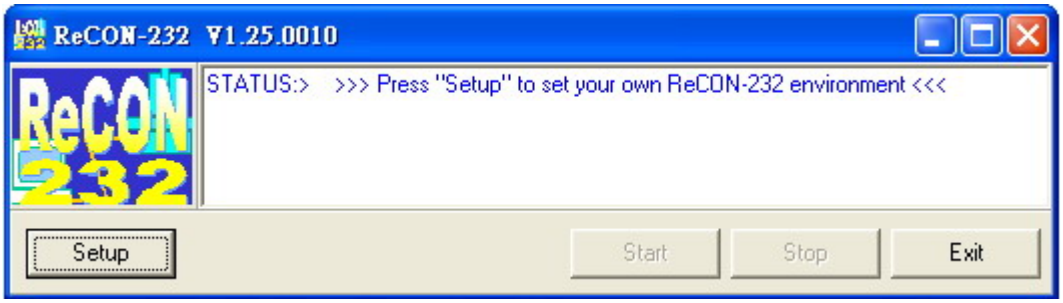
| NC(DB9PIN Jack) | PC end(DB25PIN Jack) |
|---|---|
| pin2(RD) ---------- | pin3(TD) |
| pin3(TD) ---------- | pin2(RD) |
| pin5(SG) ---------- | pin7(SG) |

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

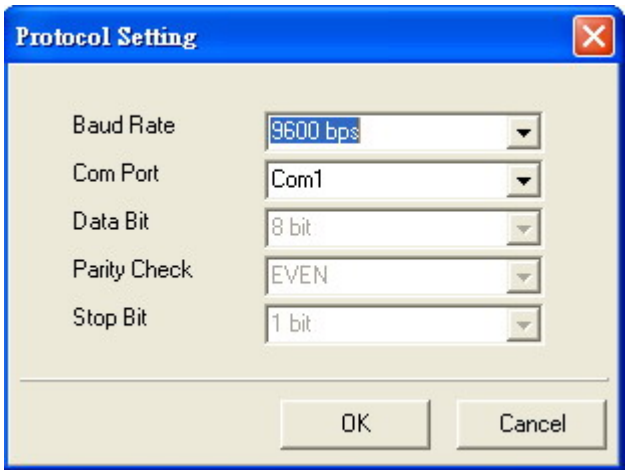
5. Clicks CD> Software\ ReCON-232 V1.25.0010, or double click setup.exe icon, the driver will start to install automatically.
6. According to Wizard's direction to follow next step or click the items to continue.
7. After installation, the desktop will have an icon of ReCON – 232, also at the start>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

**PC end**

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




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




4. 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.


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

## 3.3 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)<br>0:2400<br>1:4800<br>2:9600<br>3:19200<br>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.

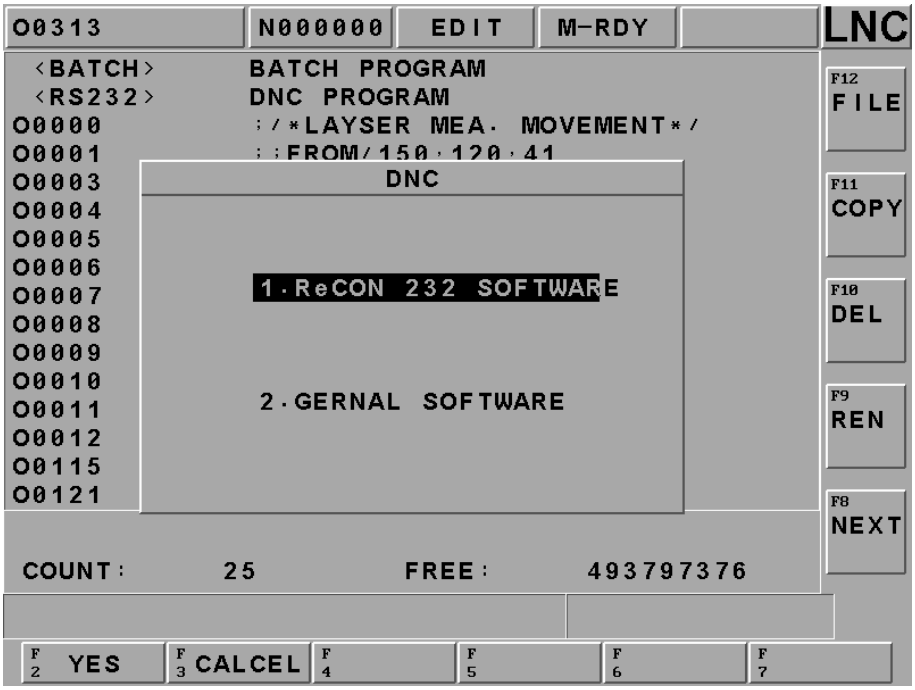
Steps are as below:

1.Edit RS232 setting at controller and PC end, and make sure they are coordinate.

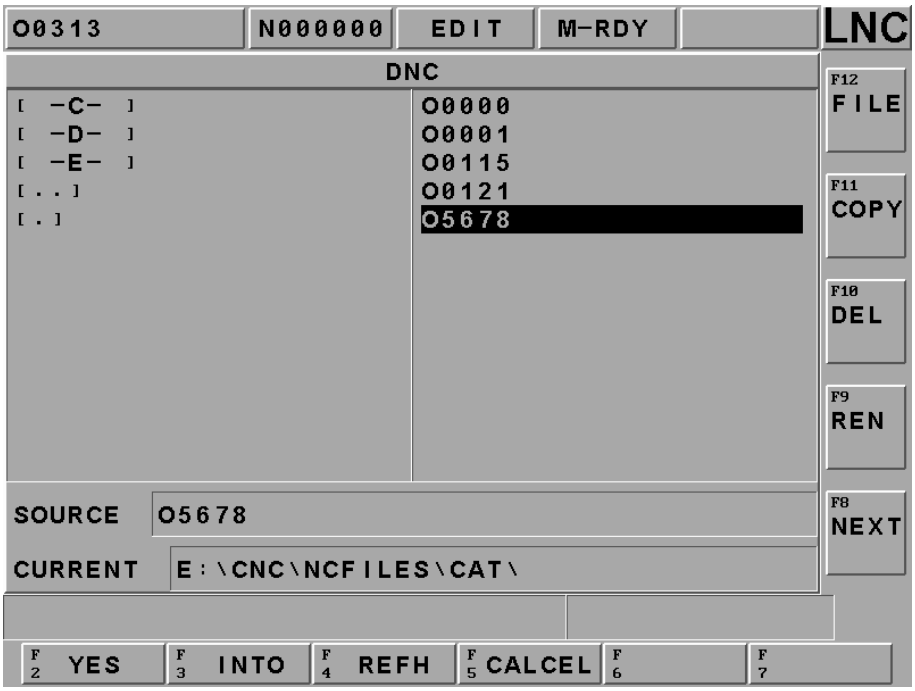
2.Use PROG group, change to EDIT mode.

3.Go to 【DIRMNG】, select 《RS232》DNC PROGRAM and press OK.

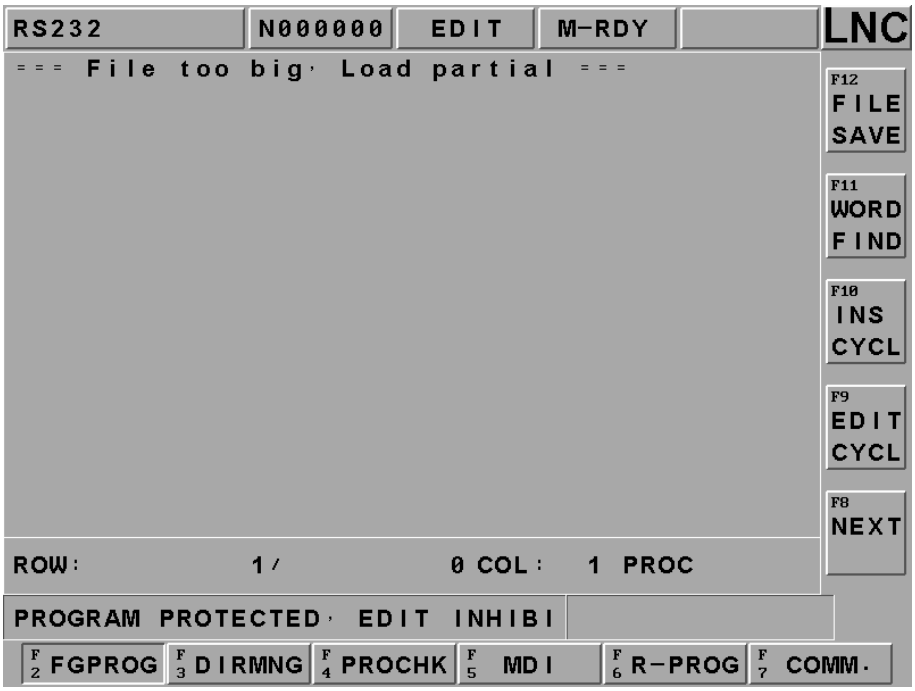

4.Select 1. LNC ReCON 232 software to upload.


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

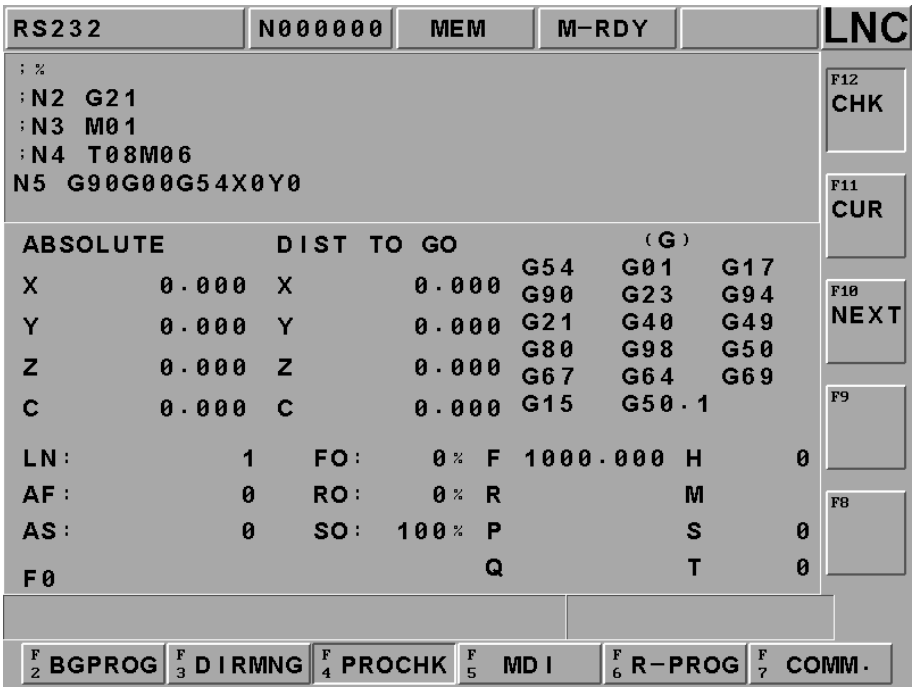
5.Go to PC end disk and select DNC file, press【YES】



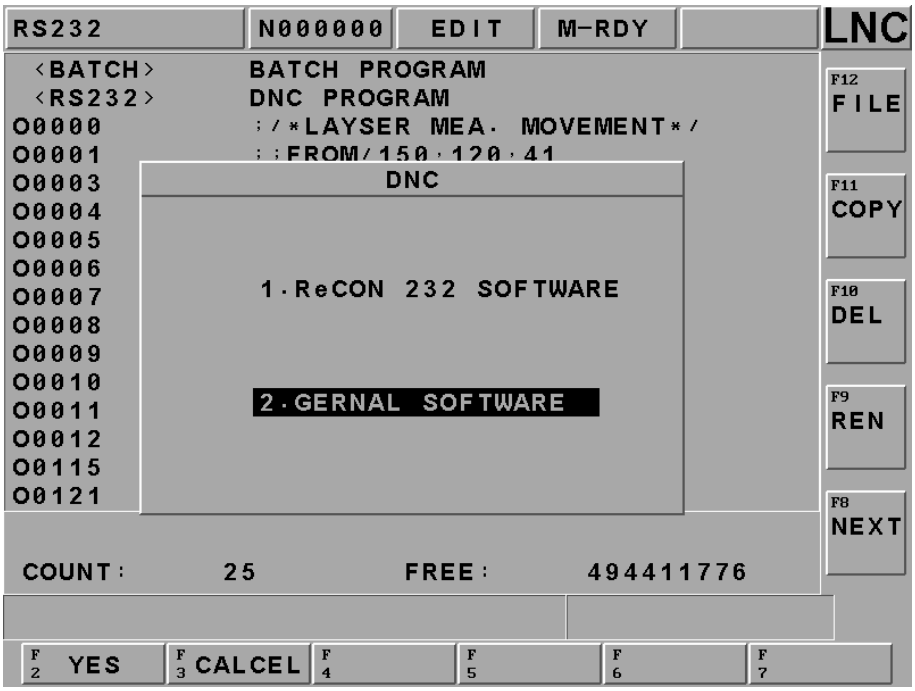

6.Now the file will turn to be RS232 and change mode to MEM.



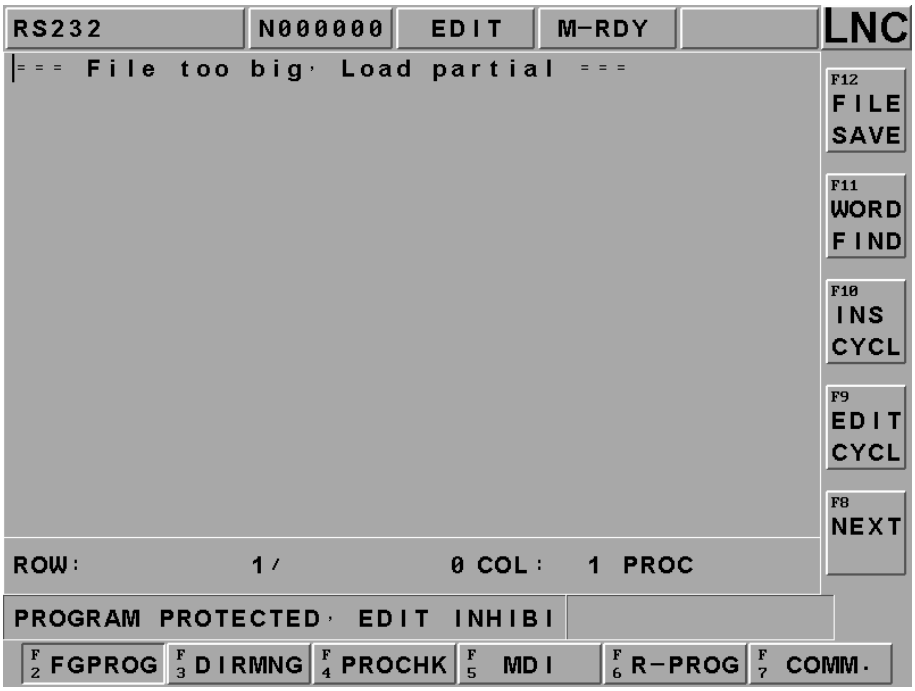
7.Press【CYCLE START】, start to do DNC.



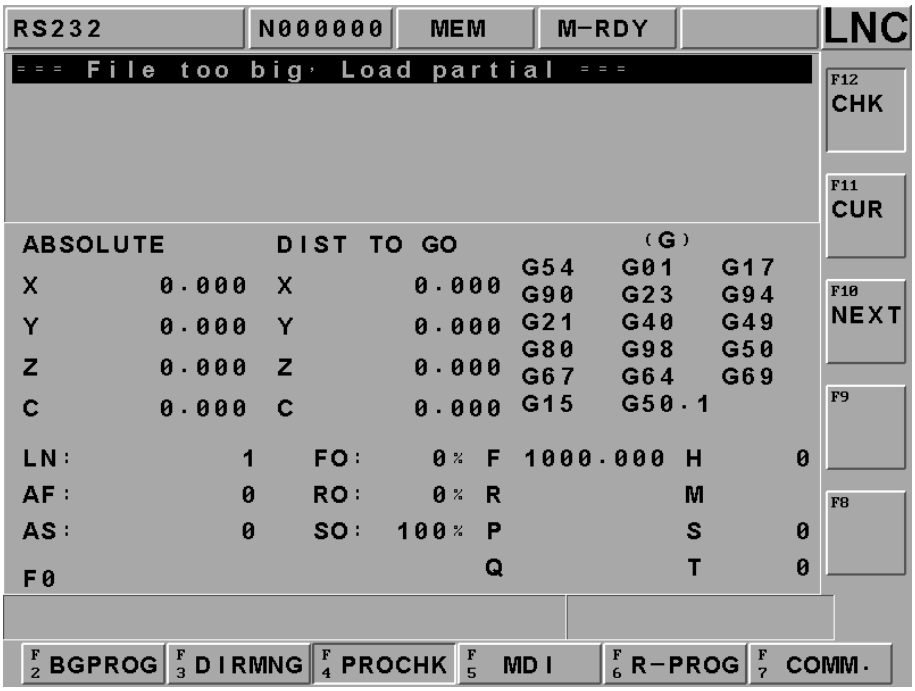

8.Select 2. General software to upload. (Use SM8 sample)



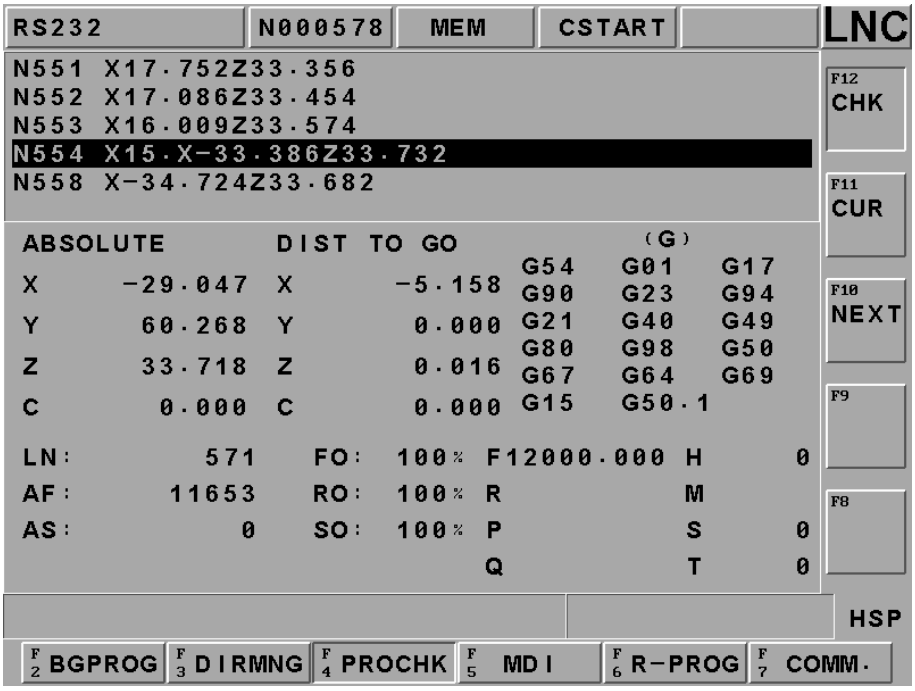
9.Now the file will turn to be RS232 and change mode to MEM.




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

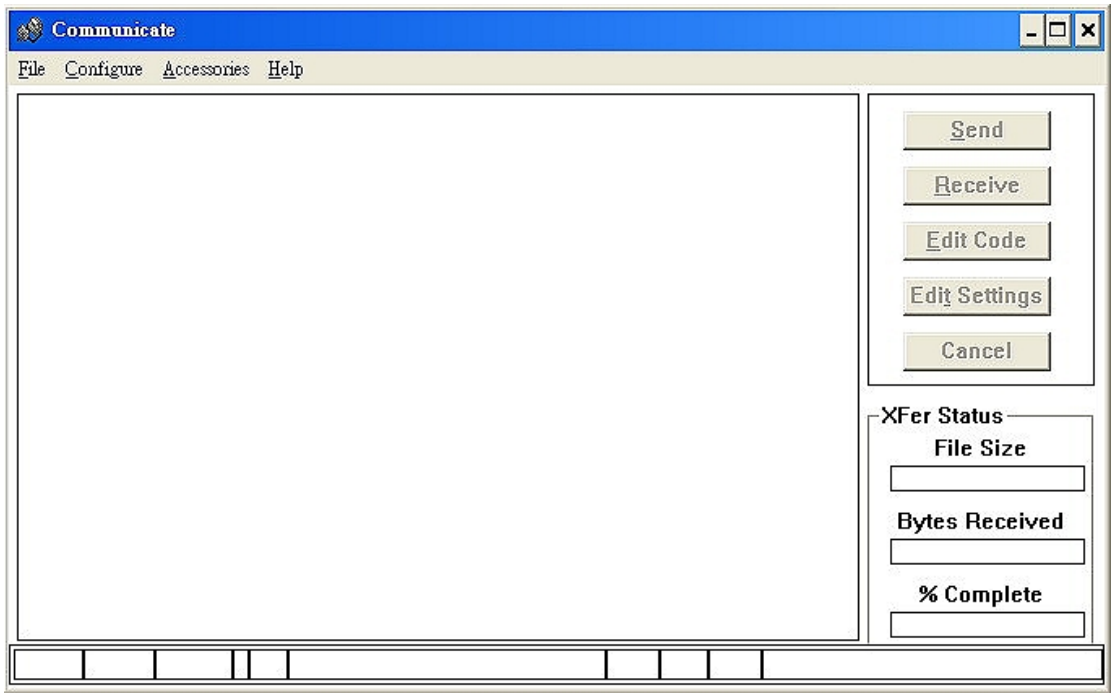


11.Before press Send button at PC end, please make sure NC end is at stand by>Cycle-Start mode. In other words, receiver must be prepared, so the sender can send out files, or after sending, press Cycle-Start to do DNC.




**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.

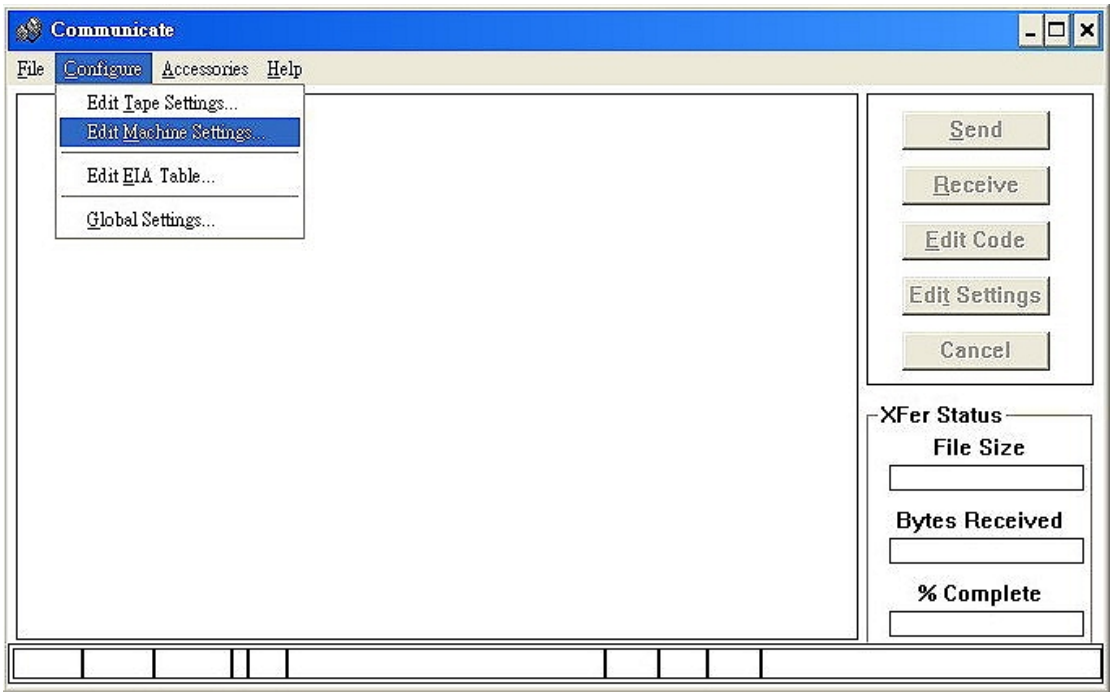
Select the file you want to transfer.

Press Send button, NC end will start to receive.

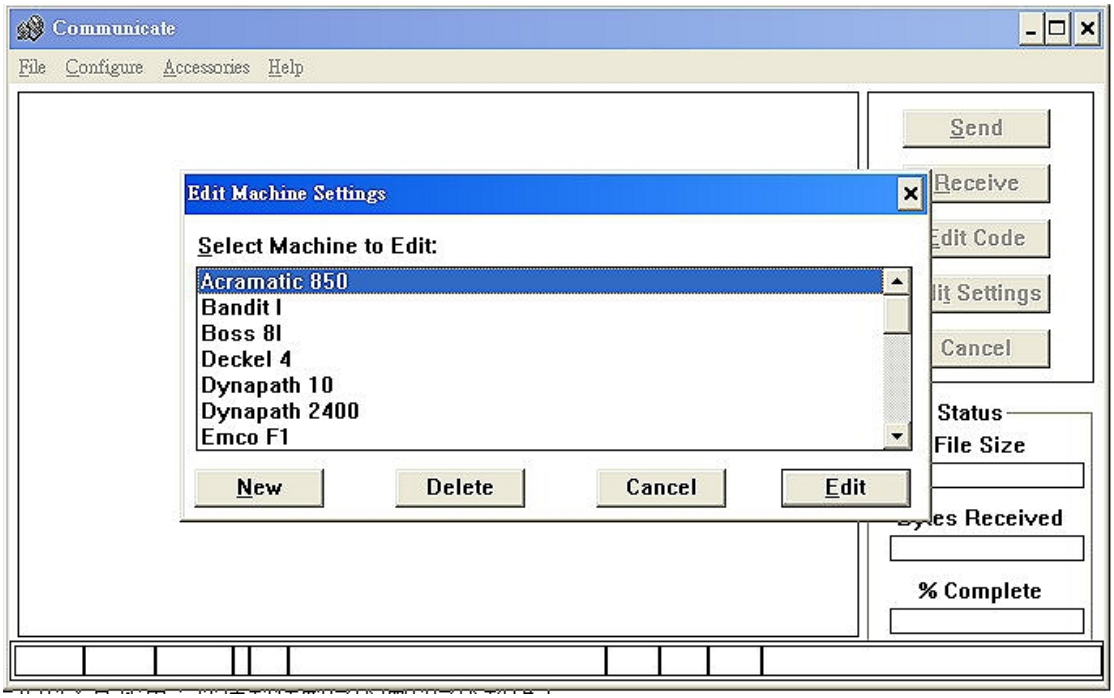
(For safety, we recommend that you make single block enable and minimize override at NC end)

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

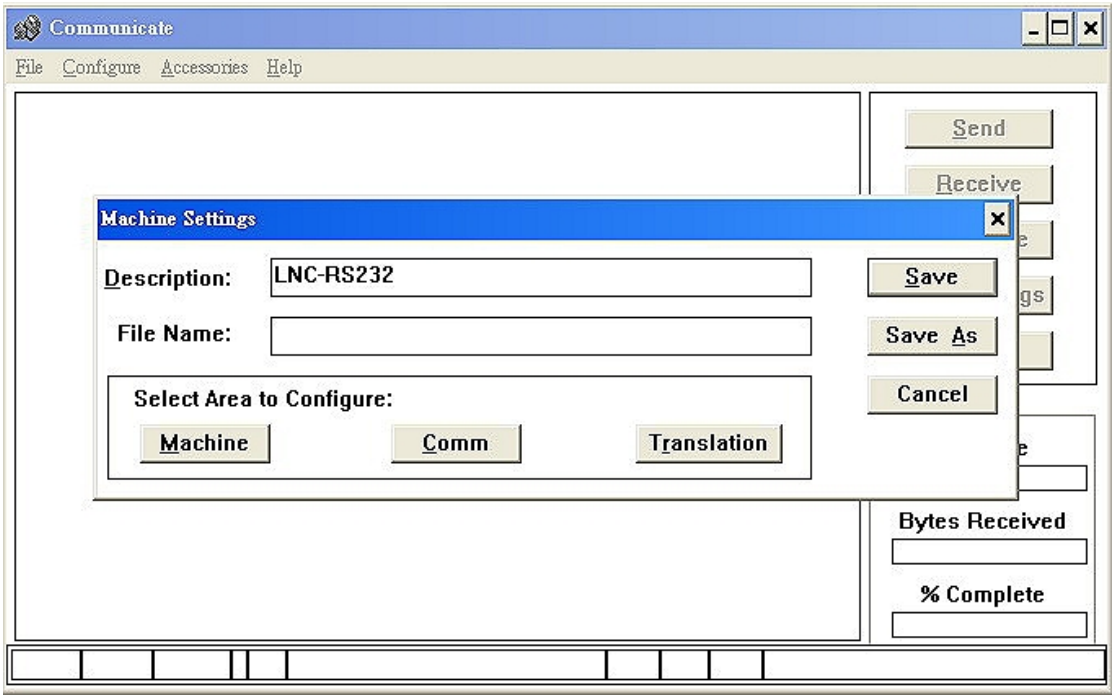
Below is the window of PC end’s communication setting page.Go to [Configure]> [Edit Machine Settings] choose a correct setting to connect with LNC.



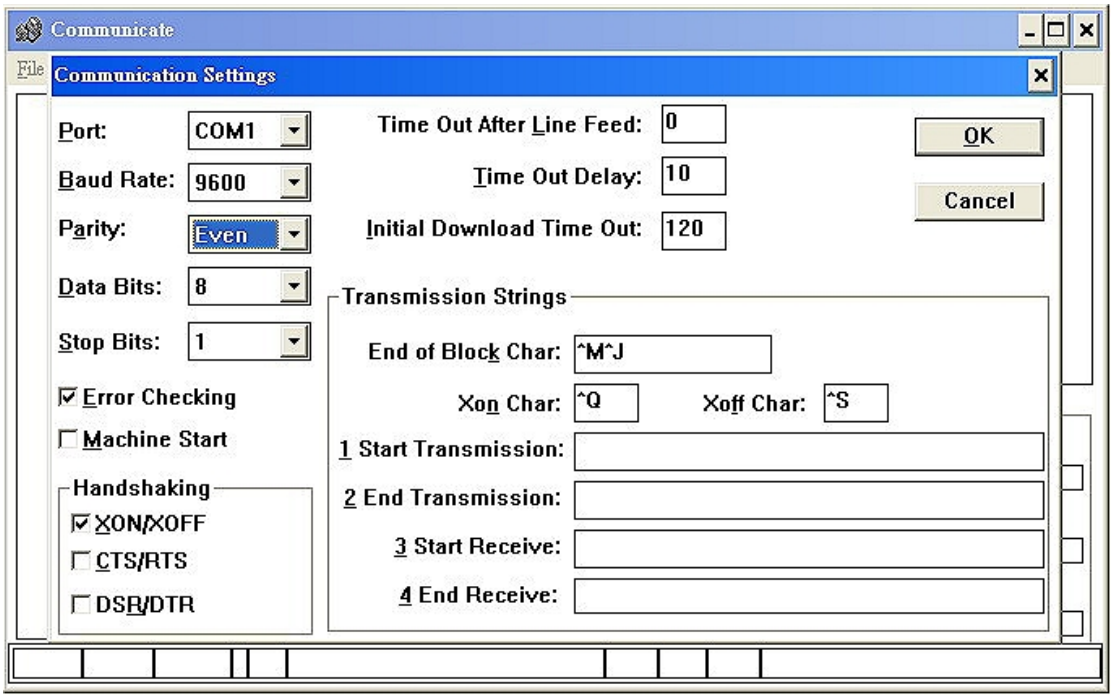
Press [NEW]




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



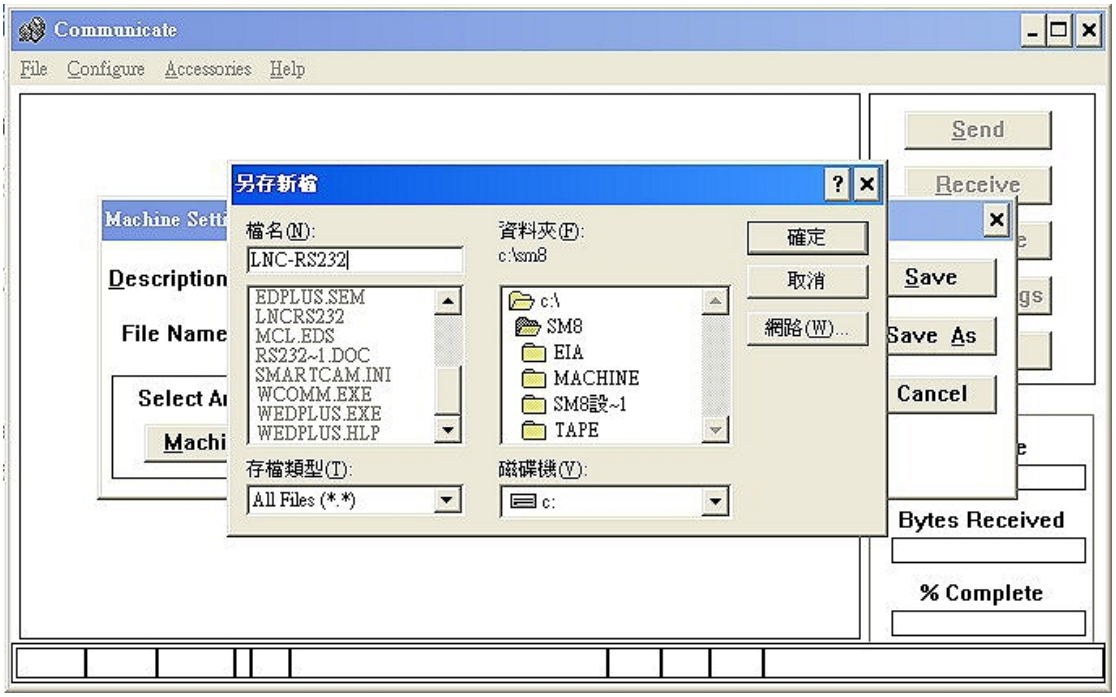
Go to [Comm]>Communication setting, other tables can be the same.



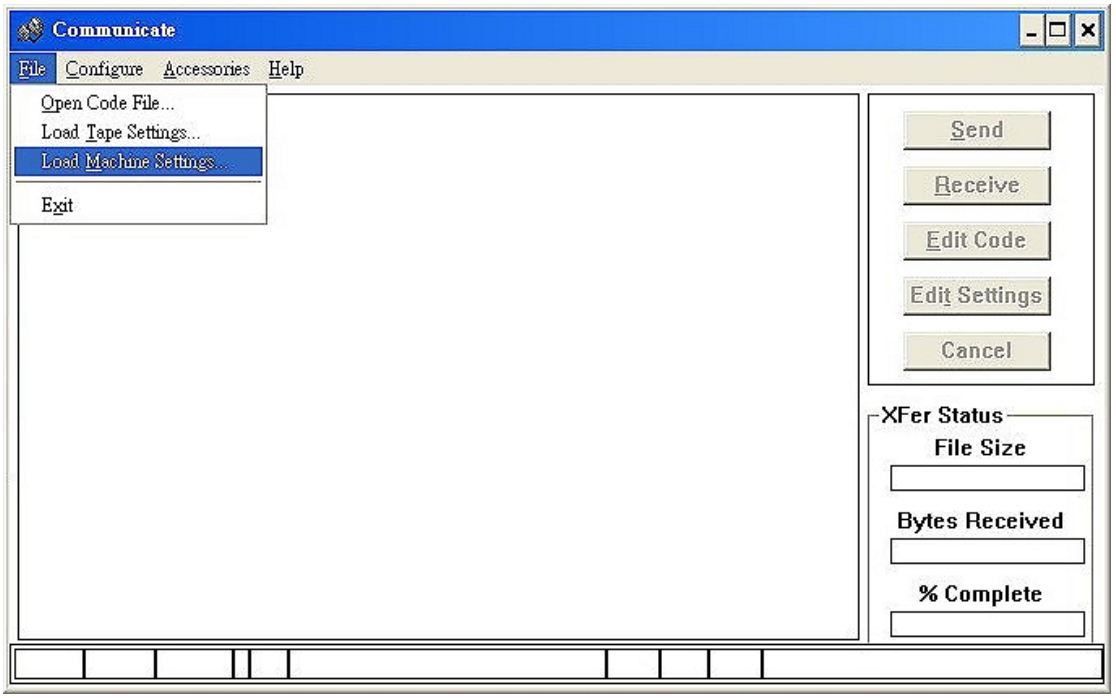

Please click XON/XOFF at HandShaking line; this is a transmission control method which conducts bits by software. If receiver wants to stop transmission from sender, receiver will send a Xon Char (ASCII No.19 bit： ^Q) to sender and stop. But if receiver wants to recover transmission, he will send a Xoff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

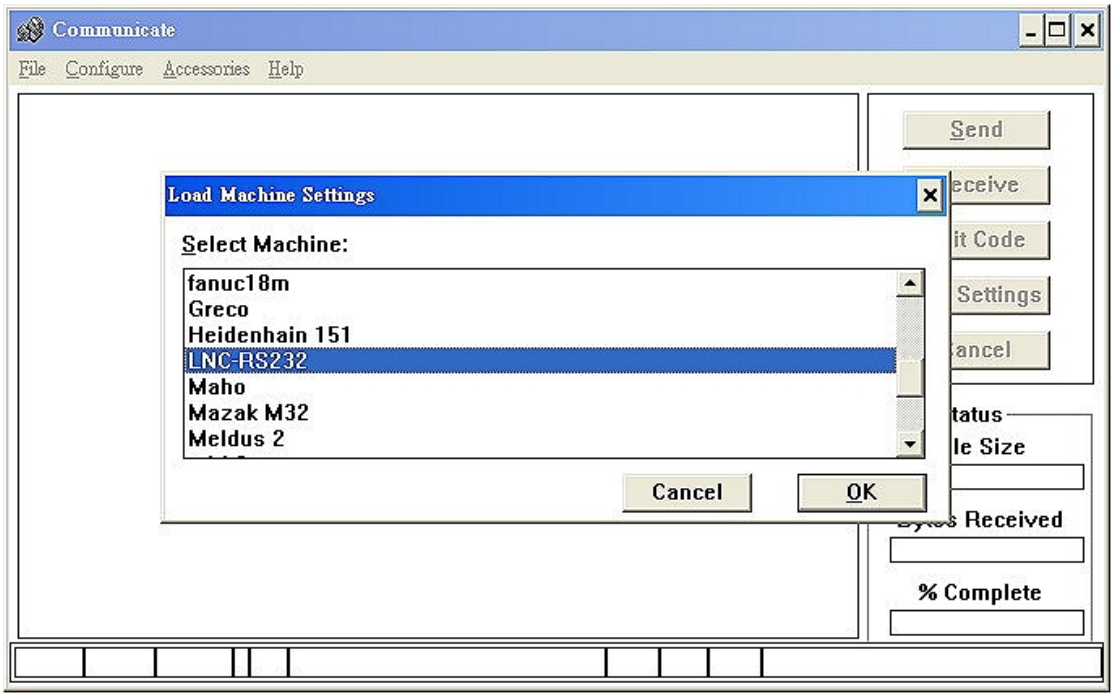
You only need to edit machine setting at the first time, after that all you need to do is to select the same setting.



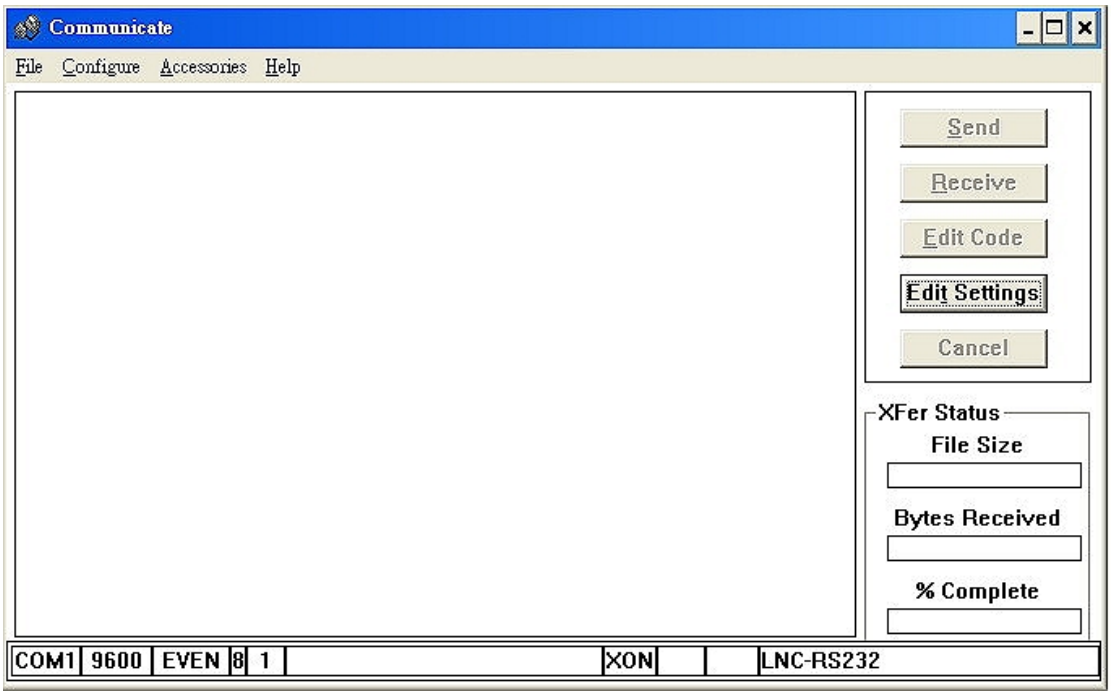

Go to [File]> [Edit Machine Settings]



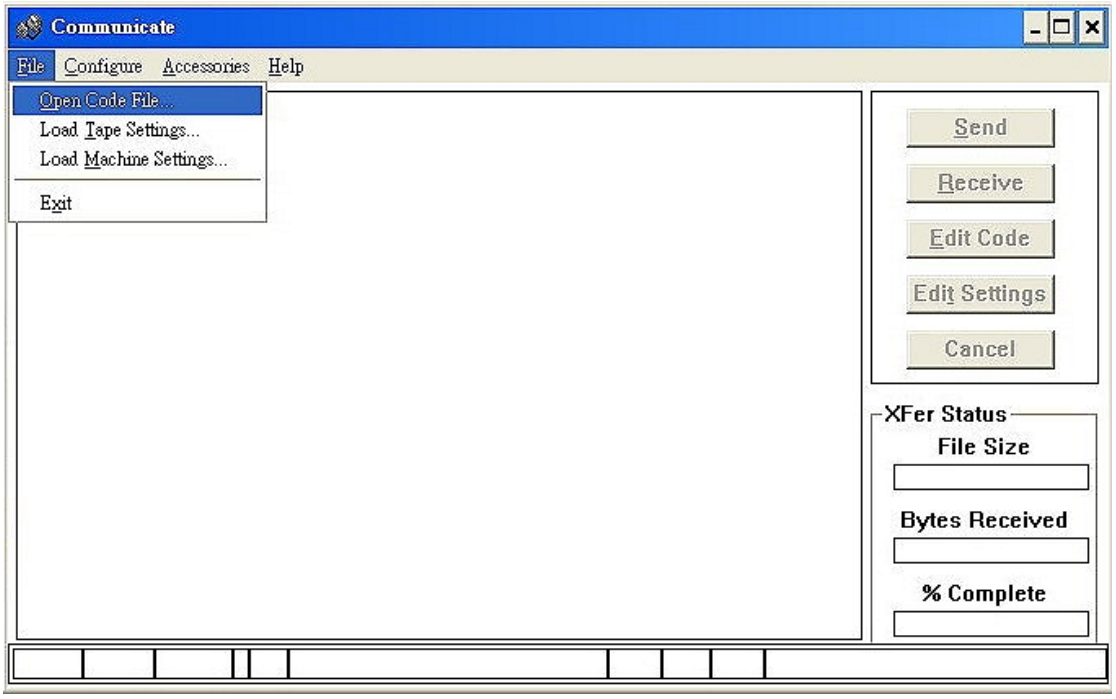
Select the setting of [LNC-RS232] that you just made.



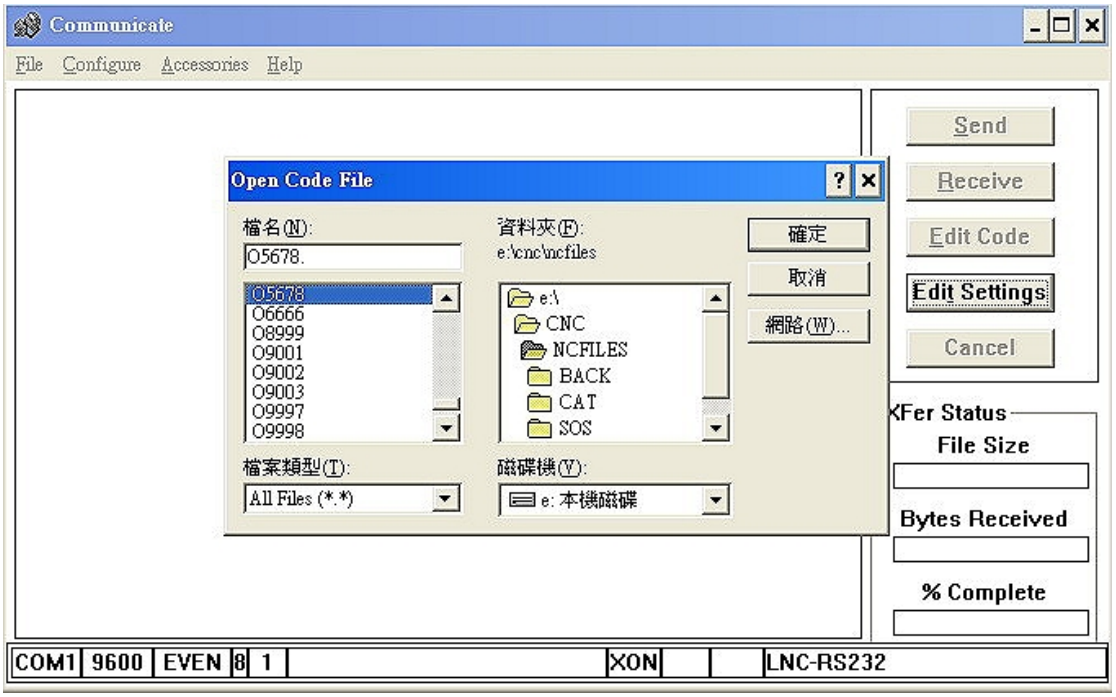

At this time, [Edit Settings] button will pop out, you will see COM1，9600，EVEN，8，1 at down side.



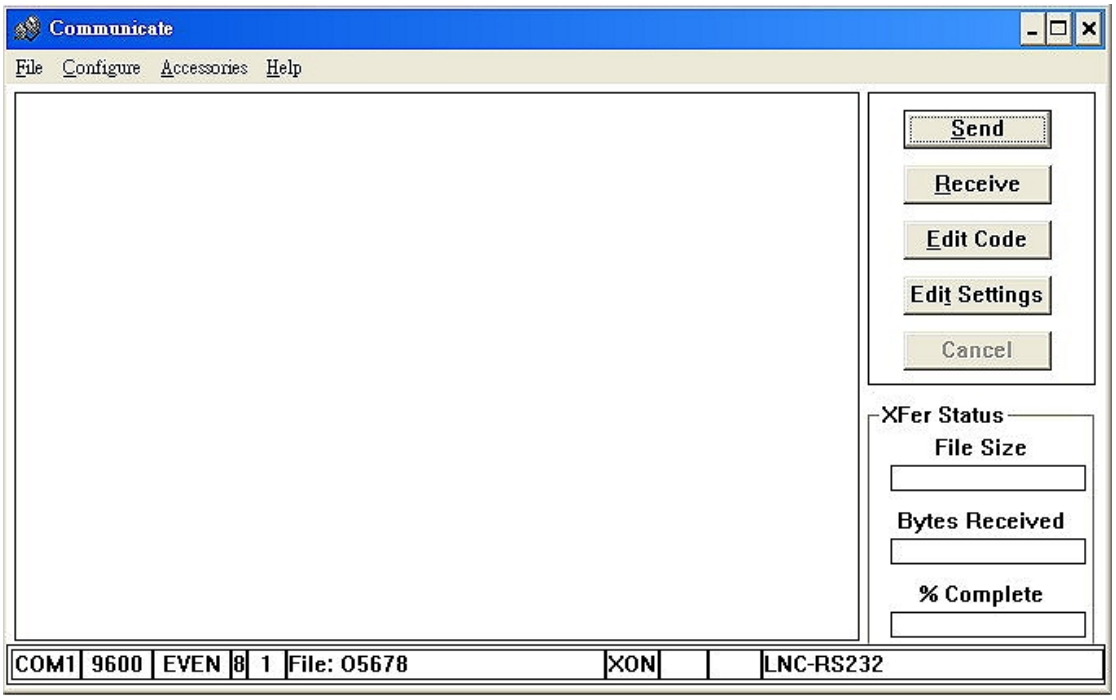
Press file and go to [Edit]> [Open Code File]



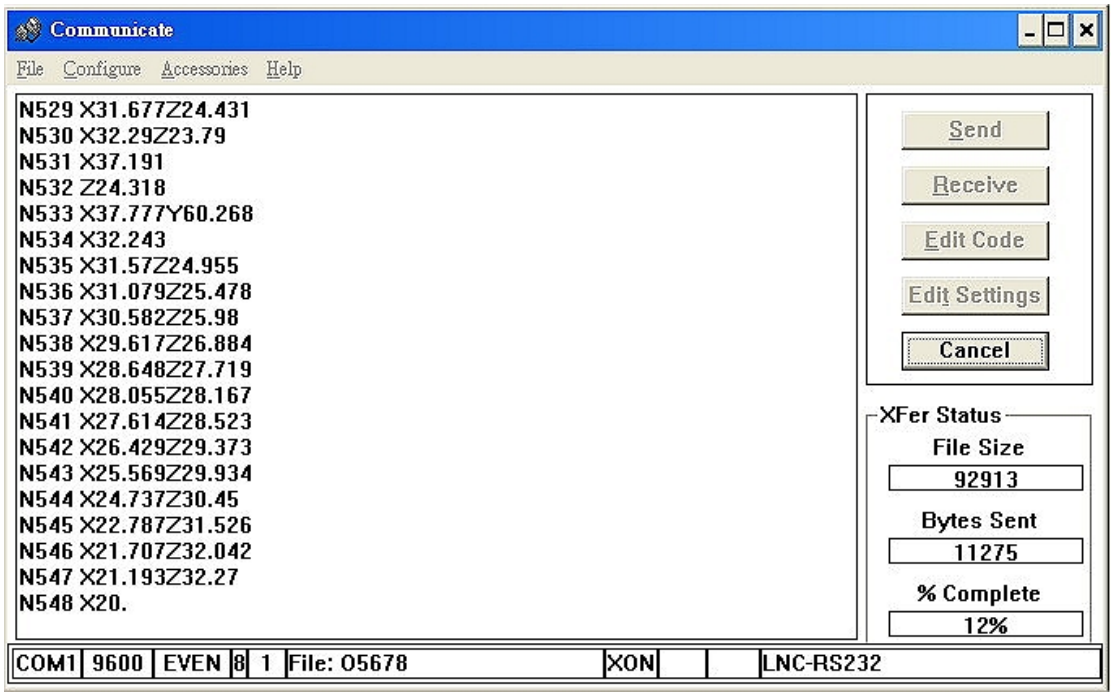

Select the file that you want to do DNC transfer and press OK.



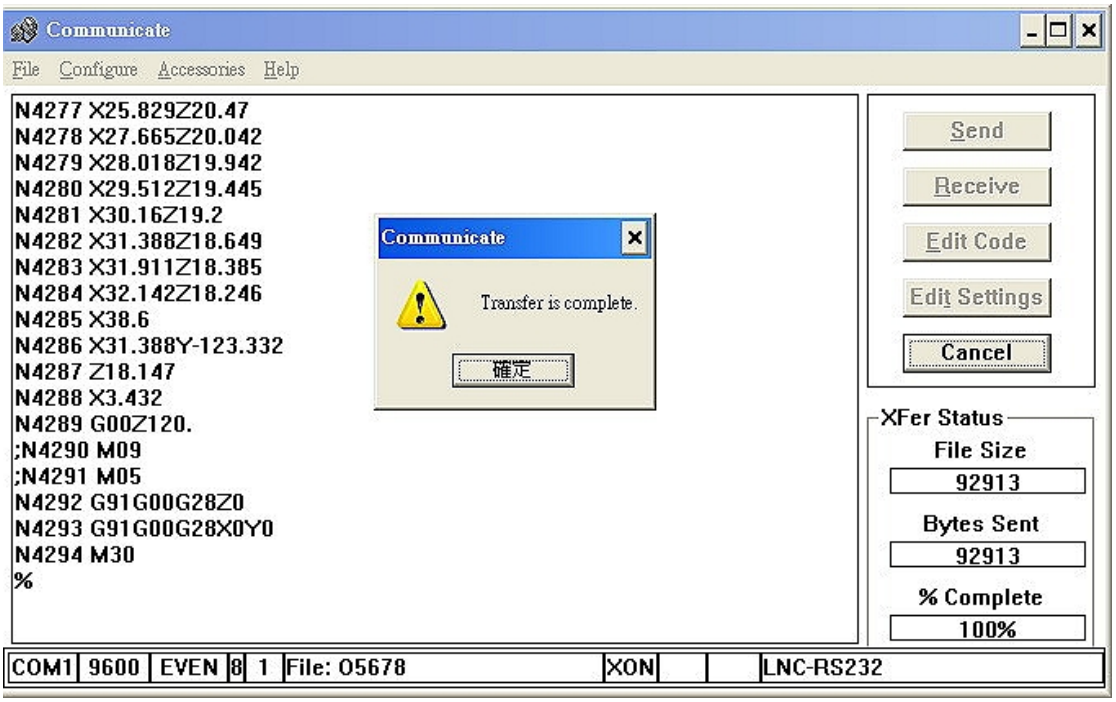
Press [Send]




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.



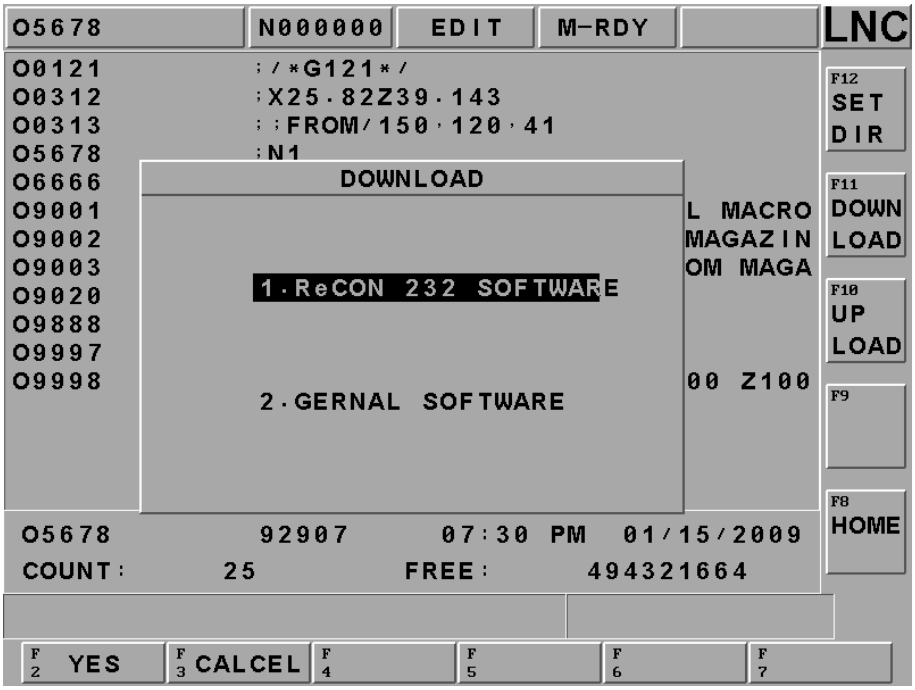
## 3.4 Communication （RS232）-【 DNC file upload and download functions 】

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

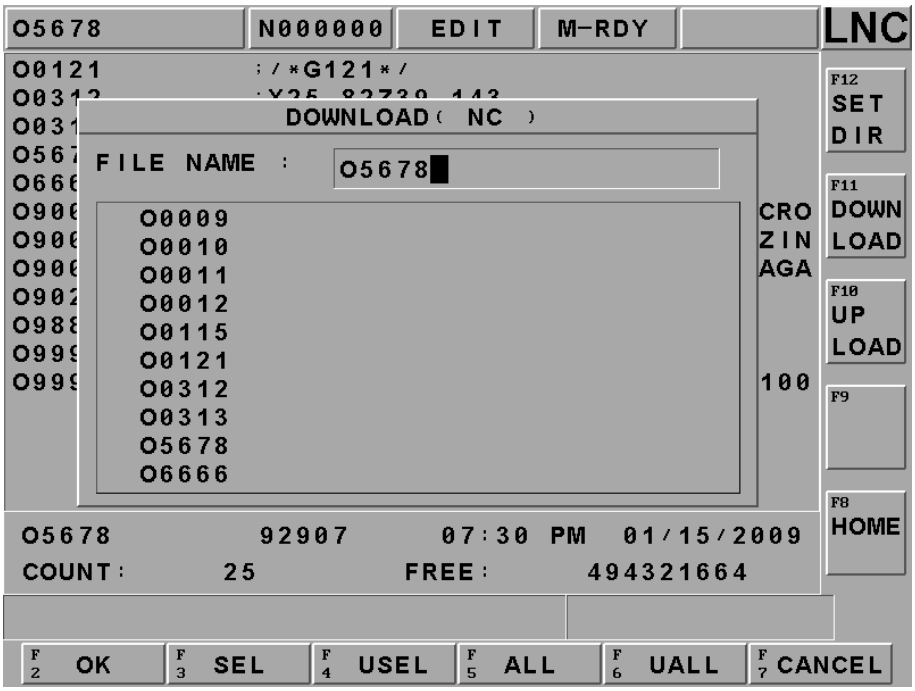
This is a device that could make you transfer the file from PC end to NC end and also save the file at NC end(H.DD/Disk ON Chip/Disk ON Modular), Details as below：

Operation for NC&PC end

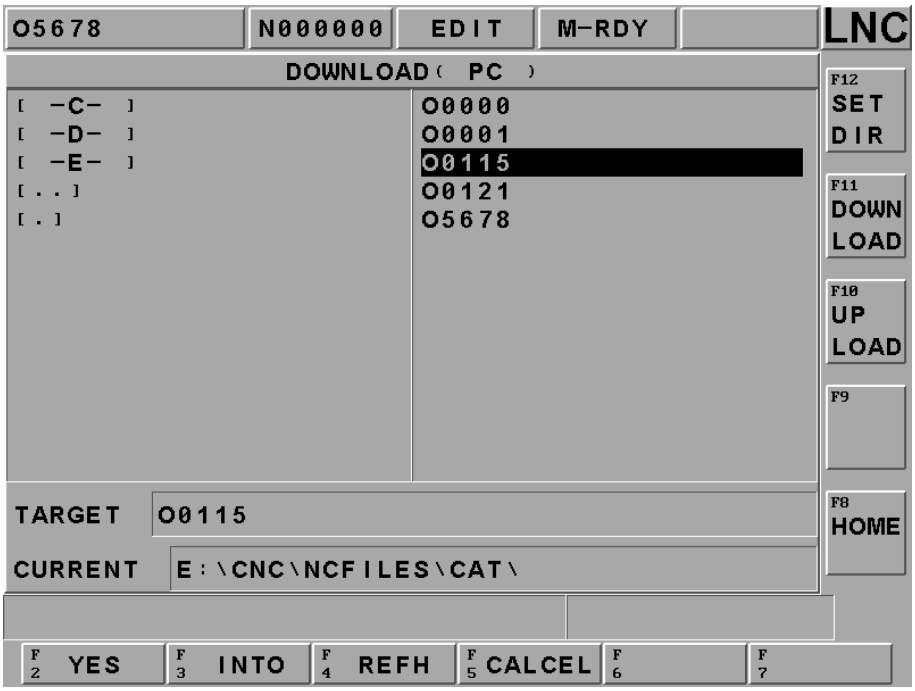
1. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
2. Use PROG group, change to EDIT mode.
3. Go to 【DIRMNG】 and choose the file that you want to transfer to PC end and press [DOWN LOAD]
   Select 1. LNC ReCON 232 software to upload.



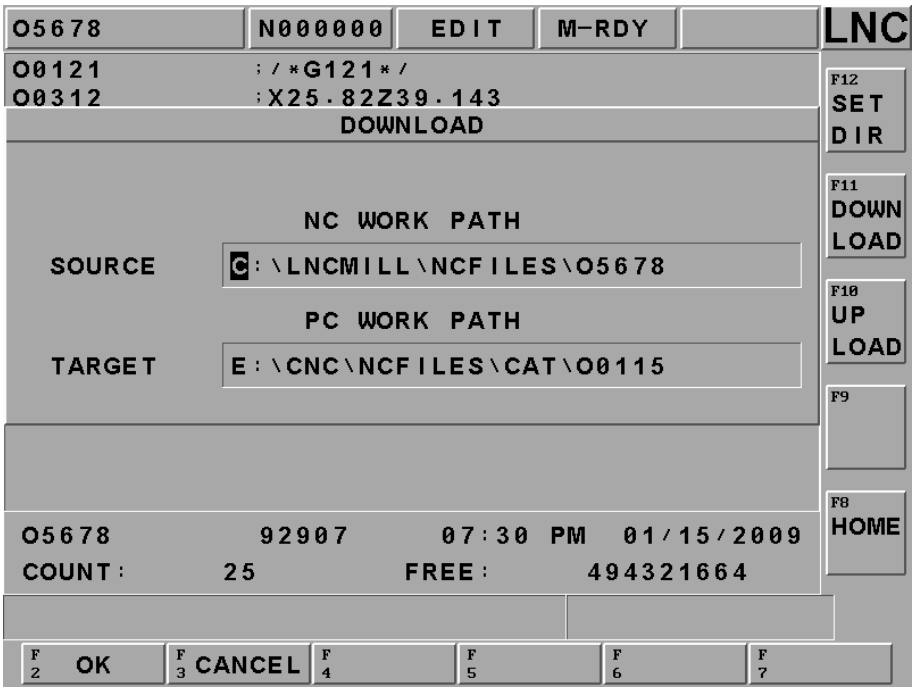
4. Choose the file O5678 you want to download from controller and press OK




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



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

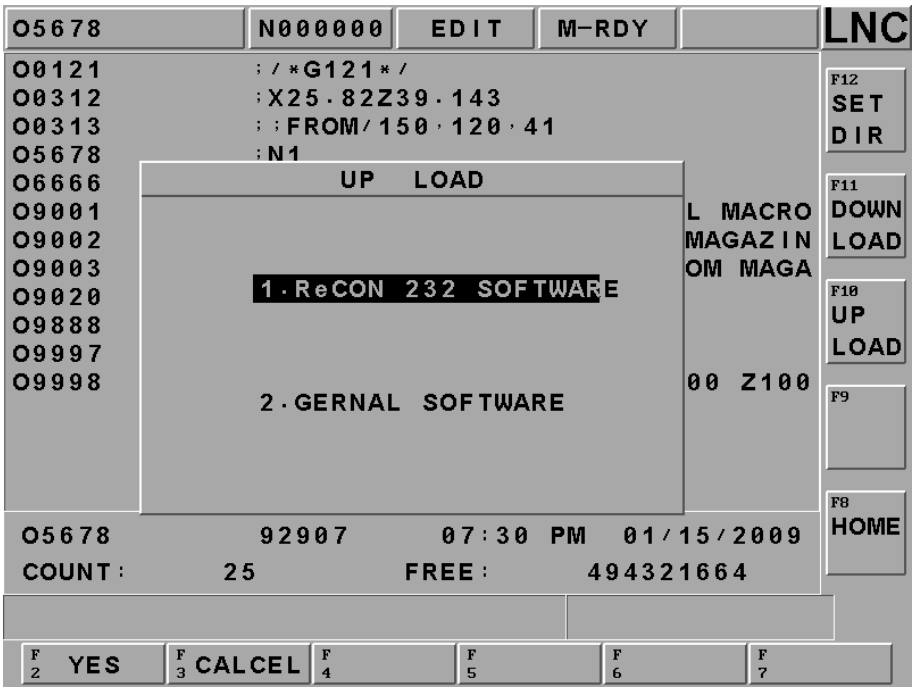


### 3.4.2 Path Program Upload & save (using ReCON software)

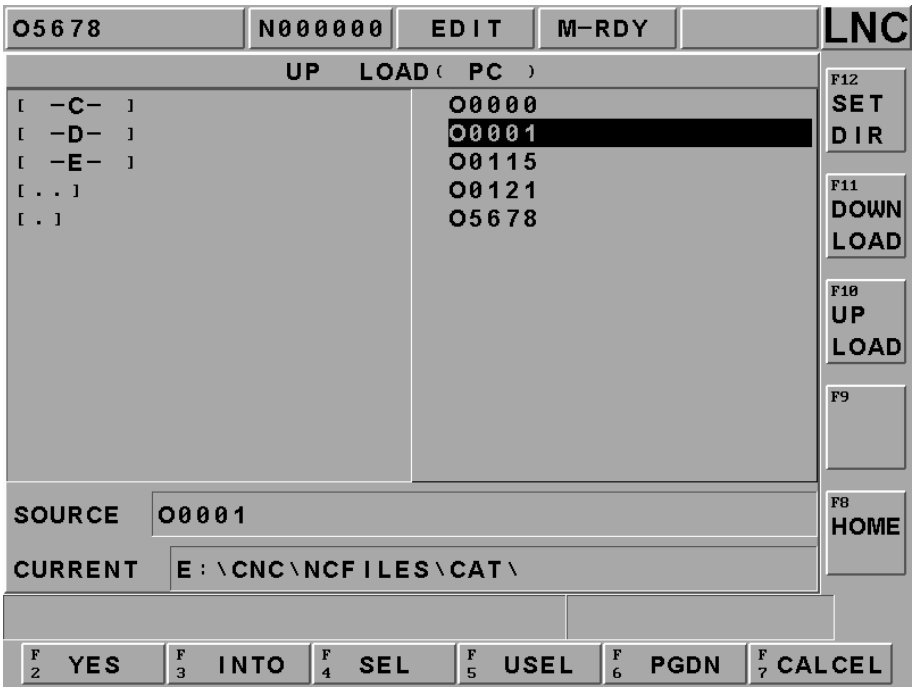
Operation Steps for NC&PC end

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] and choose the file that you want to transfer to PC end and press [UP LOAD].

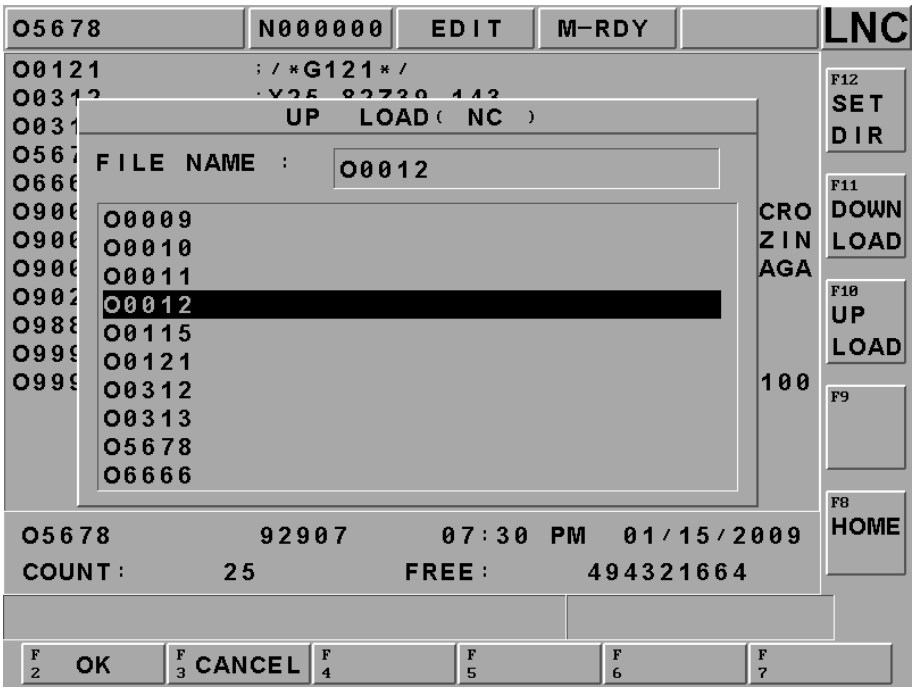
   Select 1. LNC ReCON 232 software to upload.



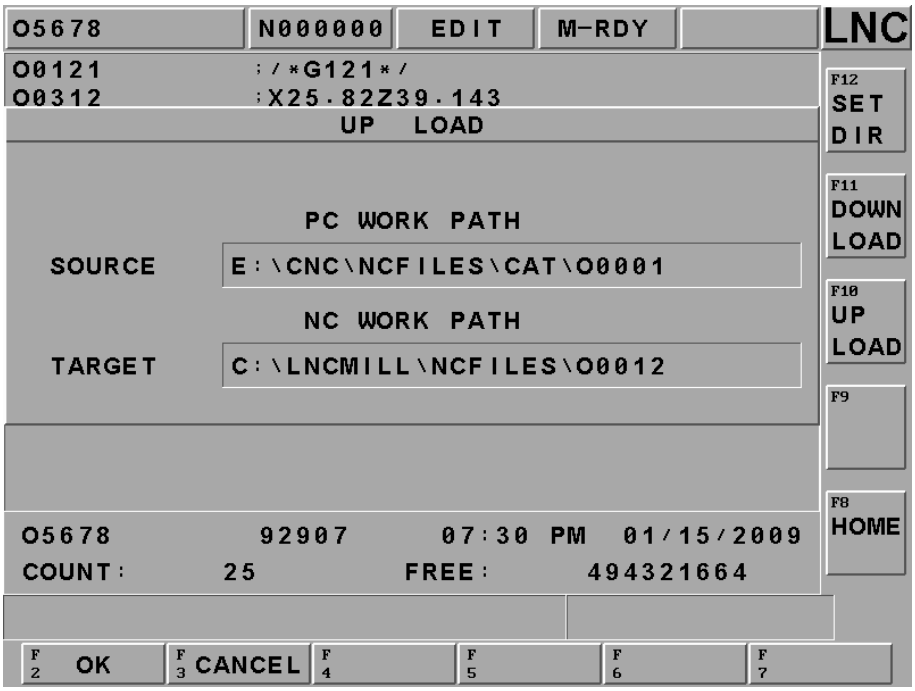
4. Select save path and upload name at PC end.



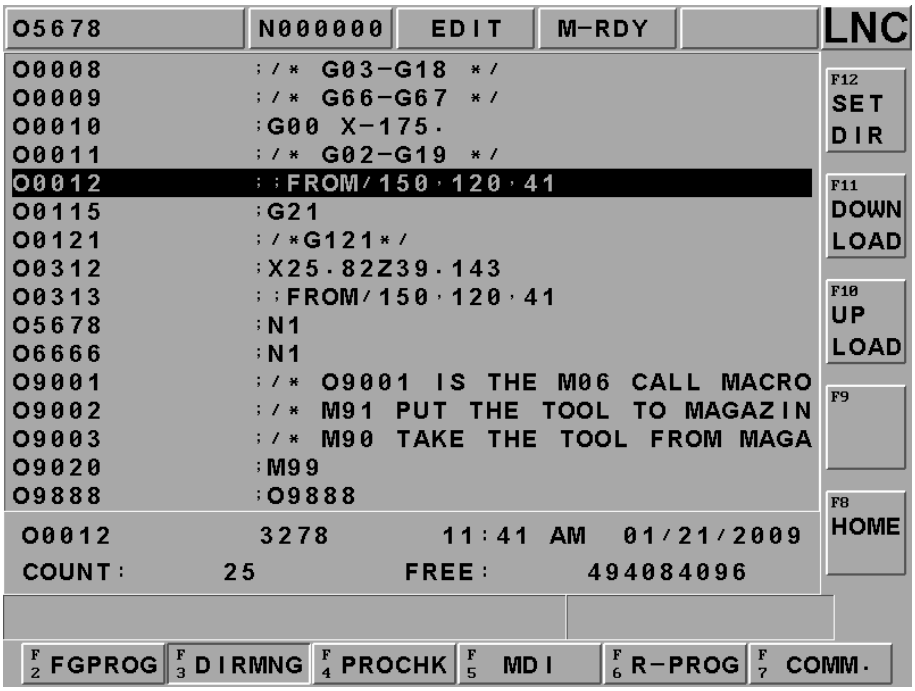

5. At NC end, you could input new name or save at the same file.



6. Confirm and press [OK]



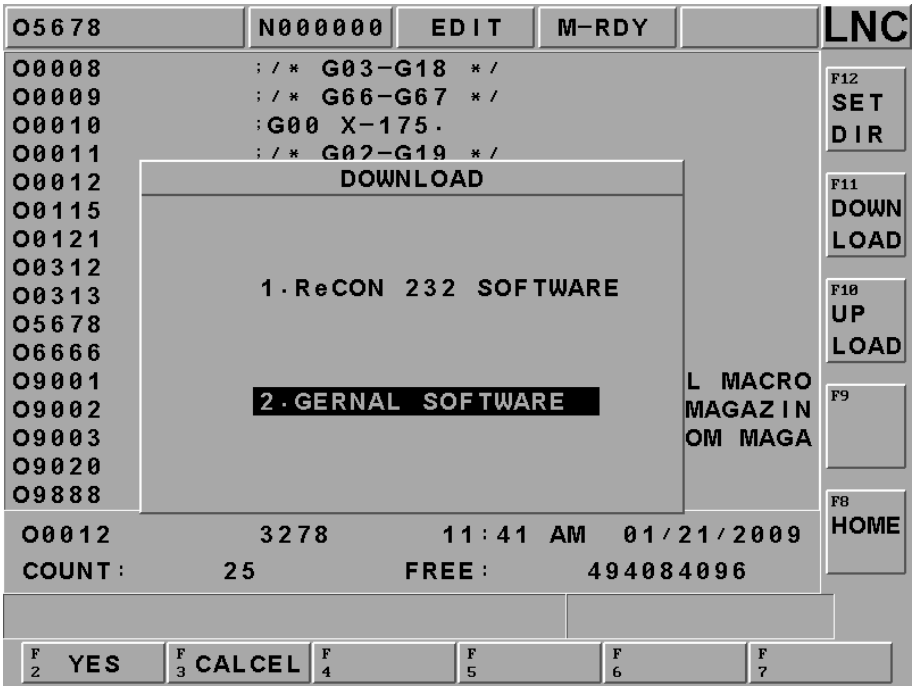

7. Upload file successful.



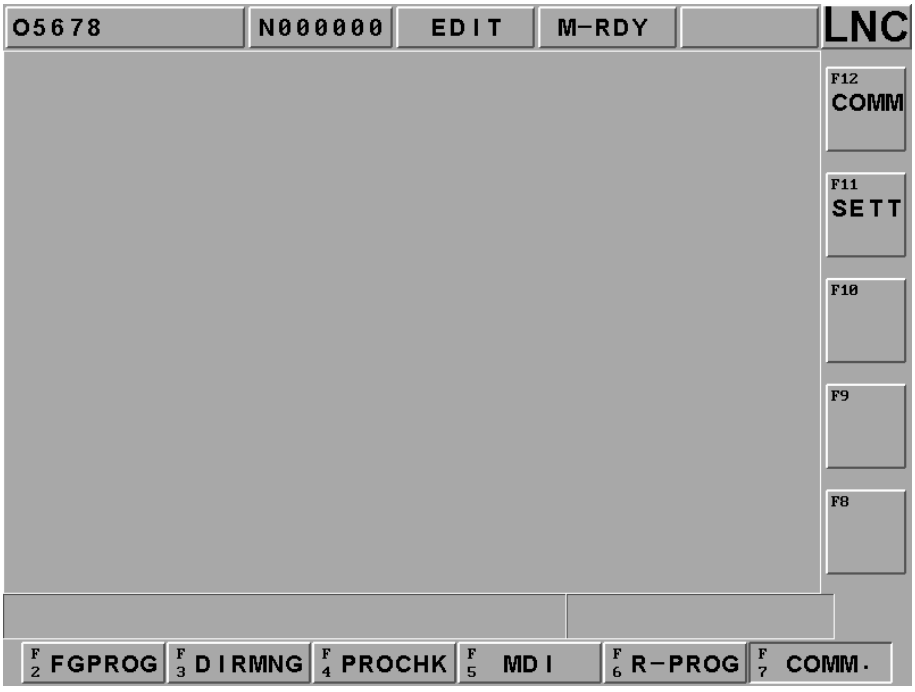
3.4.3 Path Program Upload & save (using general software)

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】and choose the file that you want to transfer to PC end and press [DOWN LOAD].

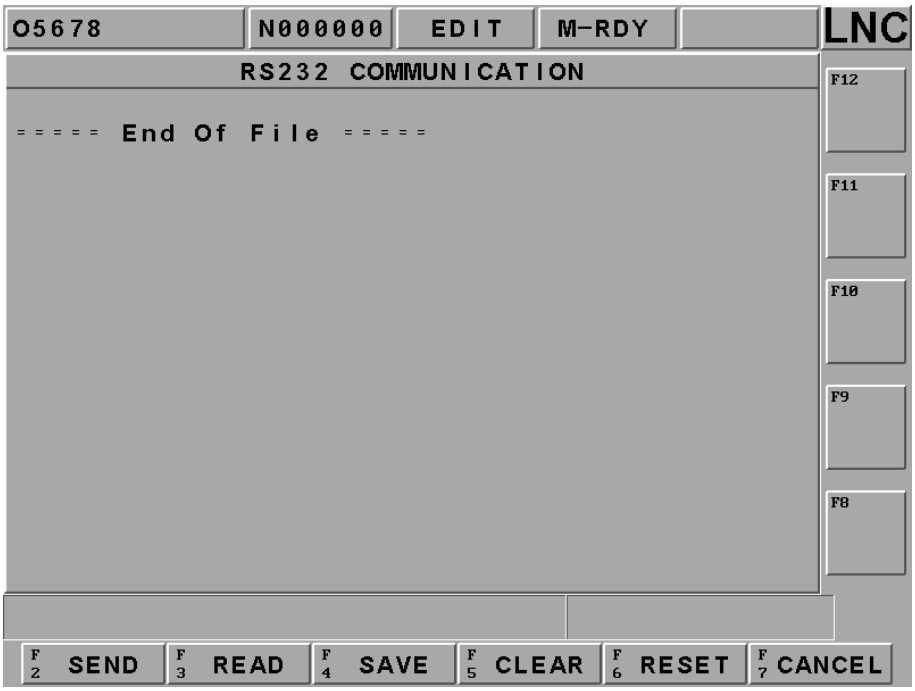
   Select 1. General software to upload.



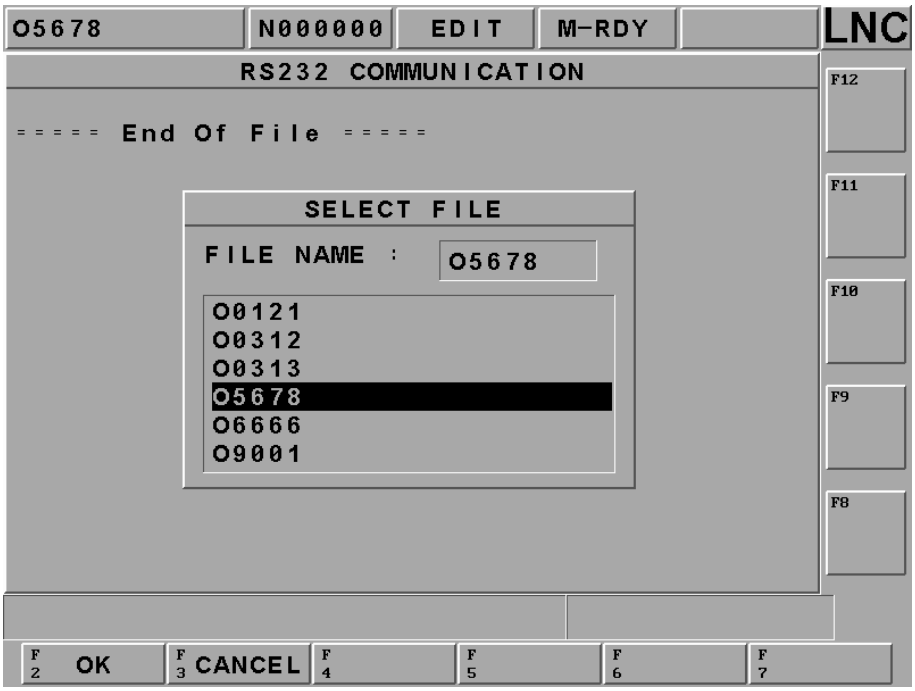

4. Go to Communication page and press [COMM] at the down right side.



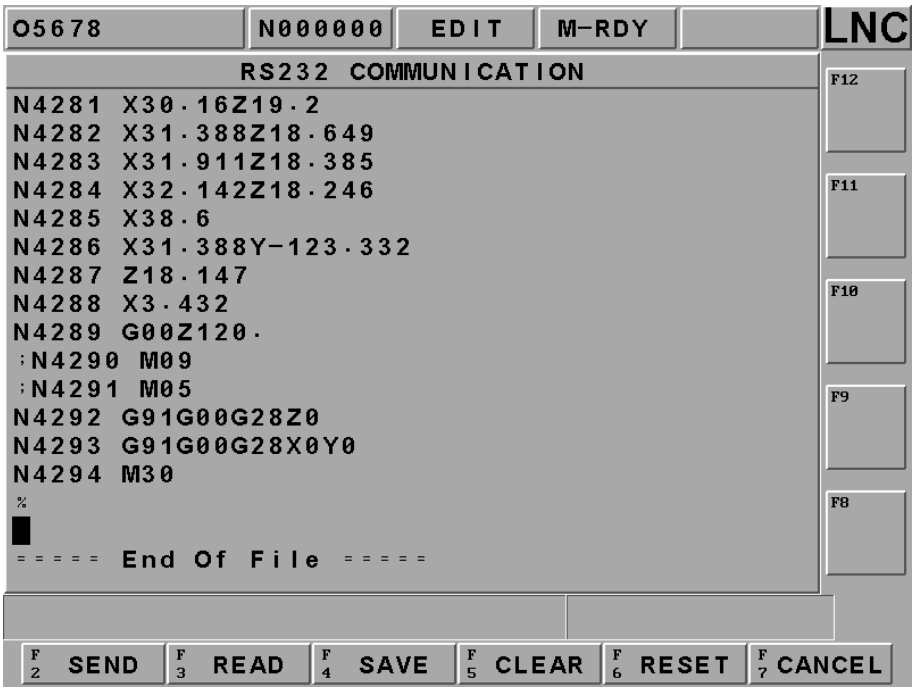
5. Go to RS232 COMMUNICATION file download page.



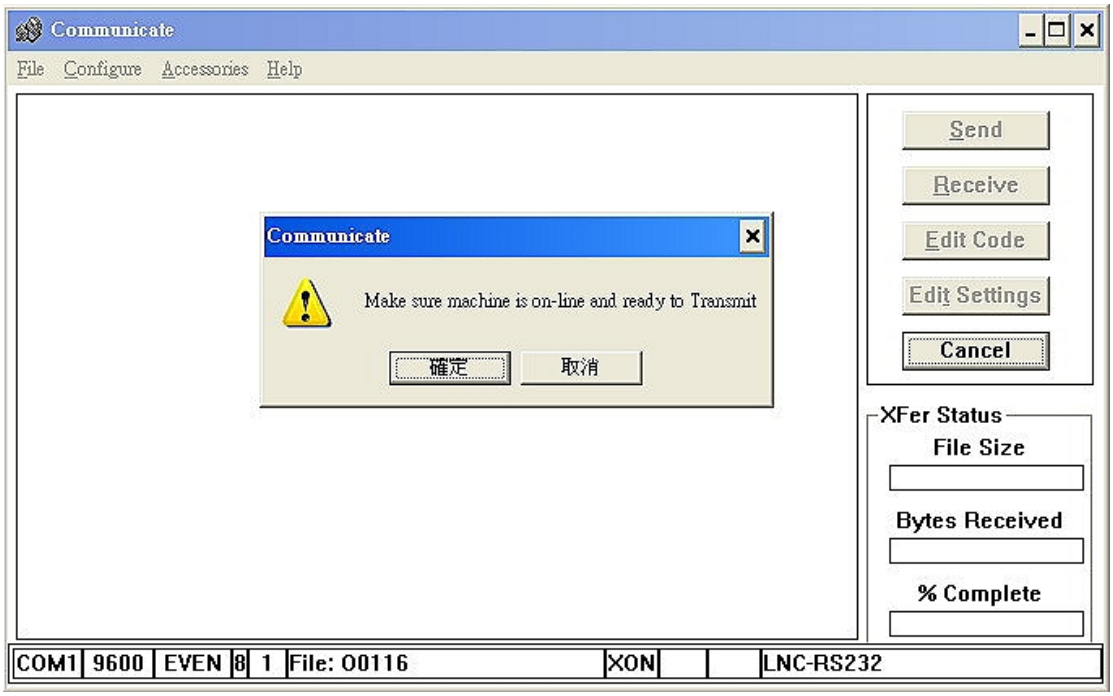

6. Press [SEND], start to transfer files from NC end to PC end, select name and press [OK]



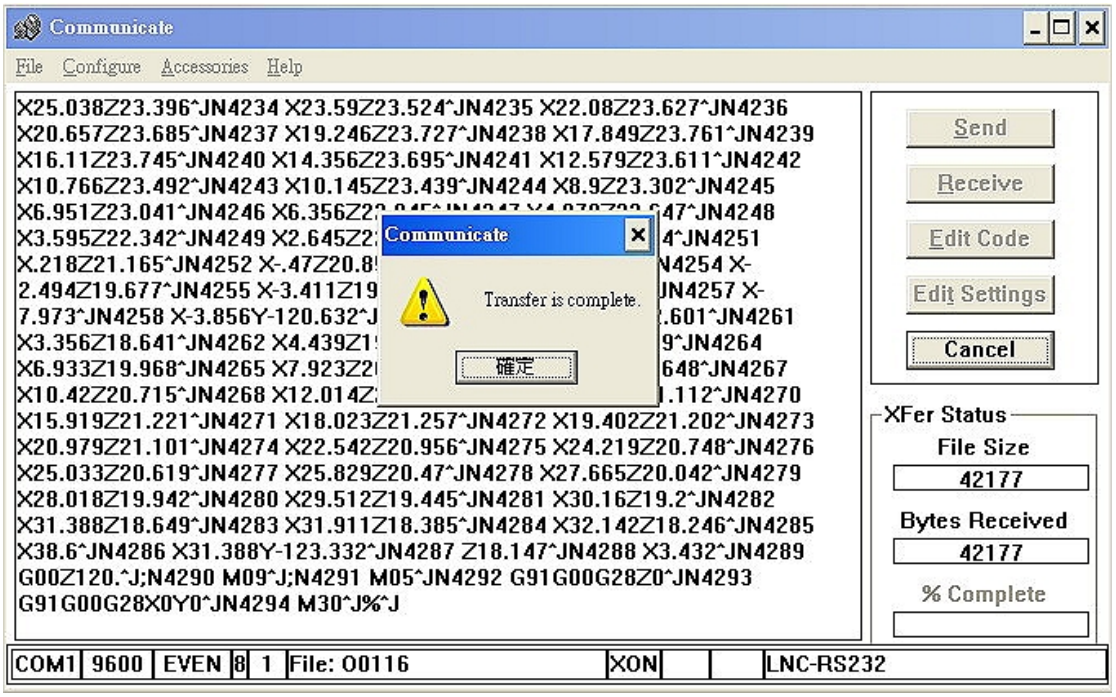
7. NC end's file will start to transfer to PC end.




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



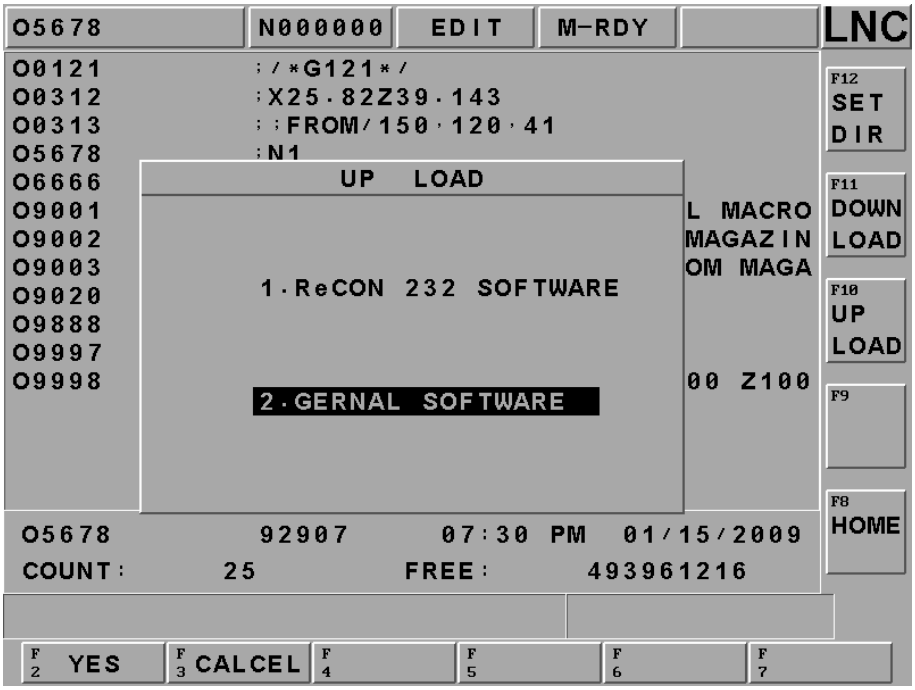
9. After transfer is completed, press [OK]



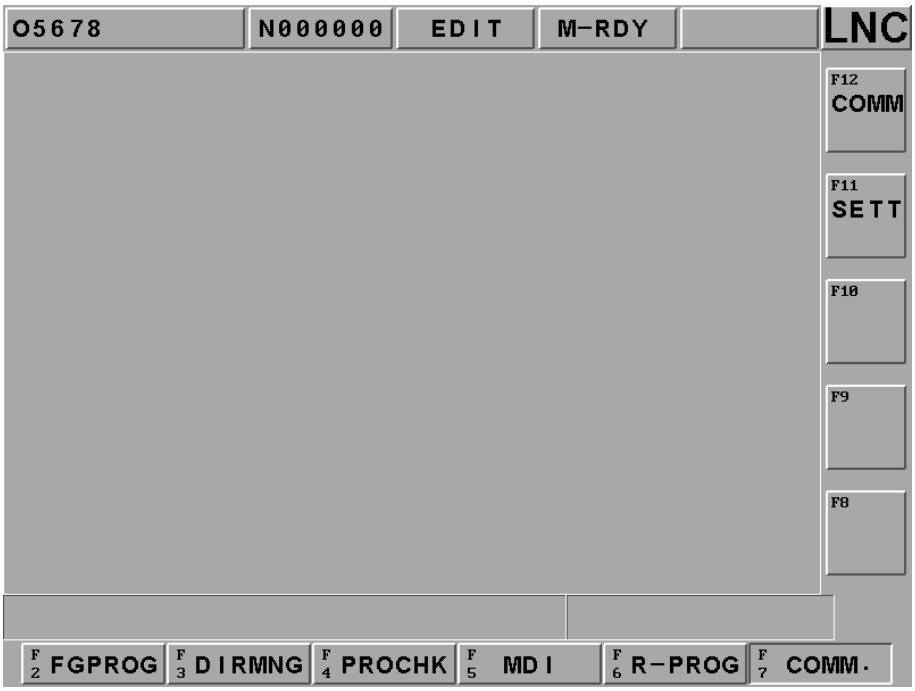
3.4.4 Path Program Upload & save (using general software)

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】 and choose the file that you want to transfer to PC end and press [UP LOAD].

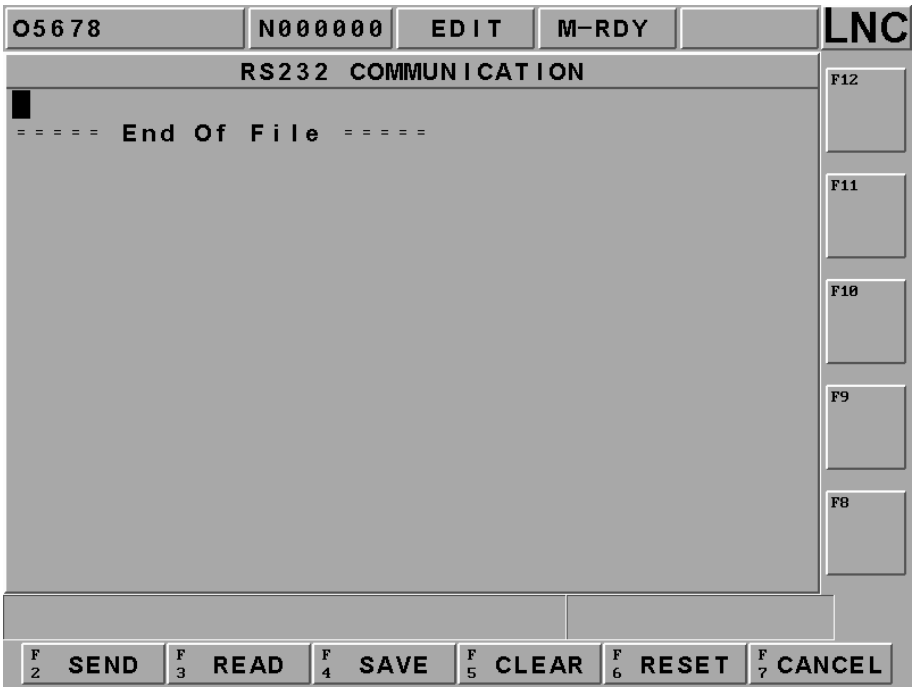
   Select 1. General software to upload.



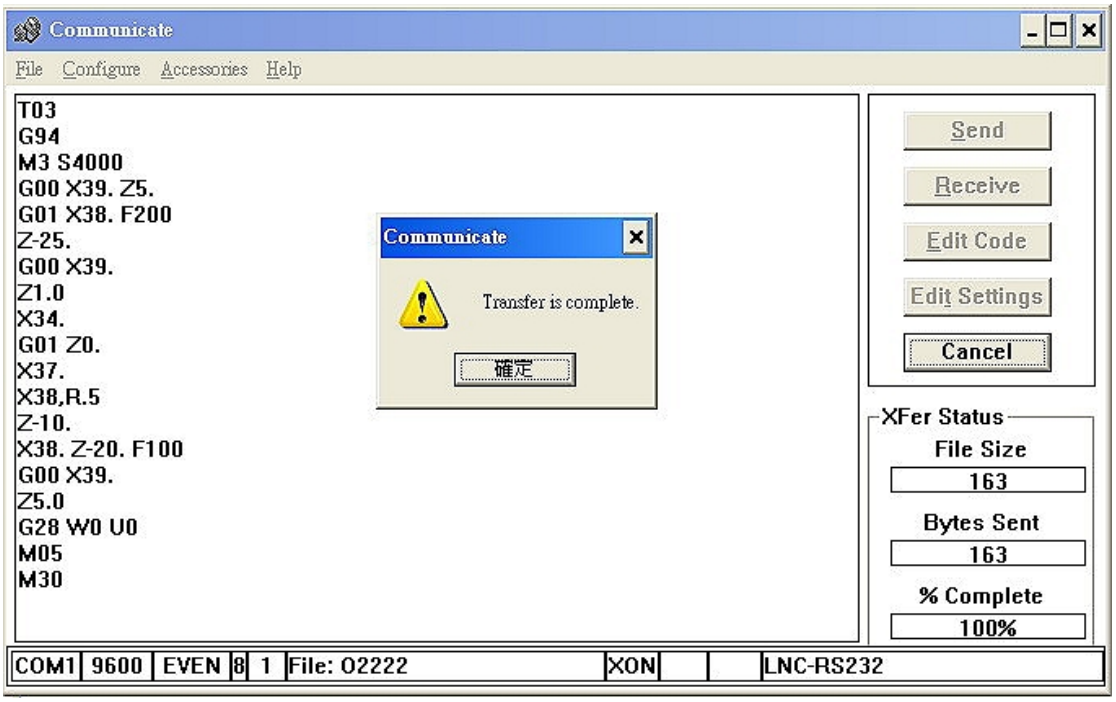
4. Go to communication page and press [COMM] at the down right side.



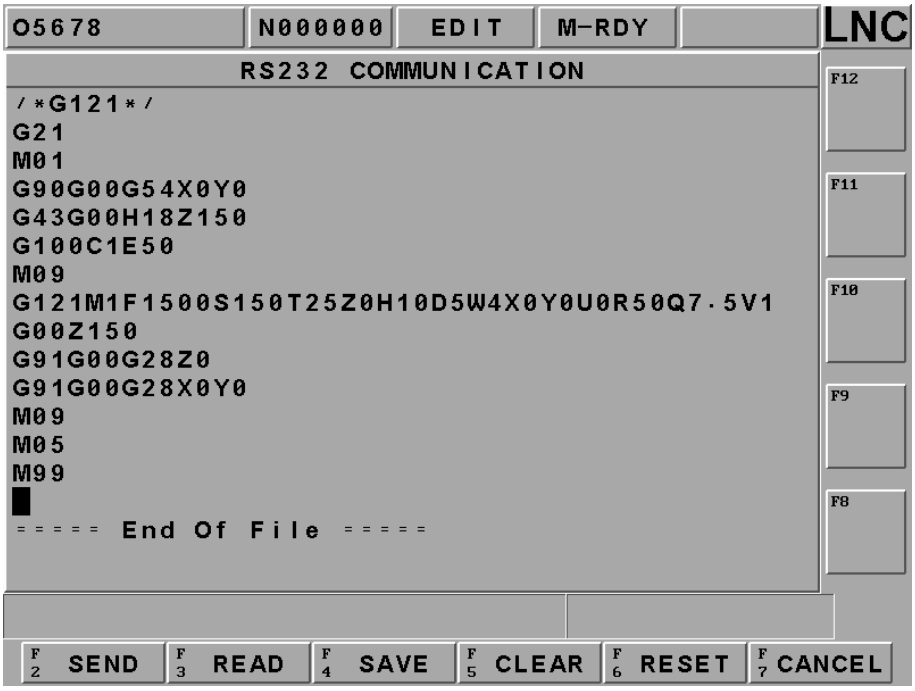

5. Go to RS232 communication.



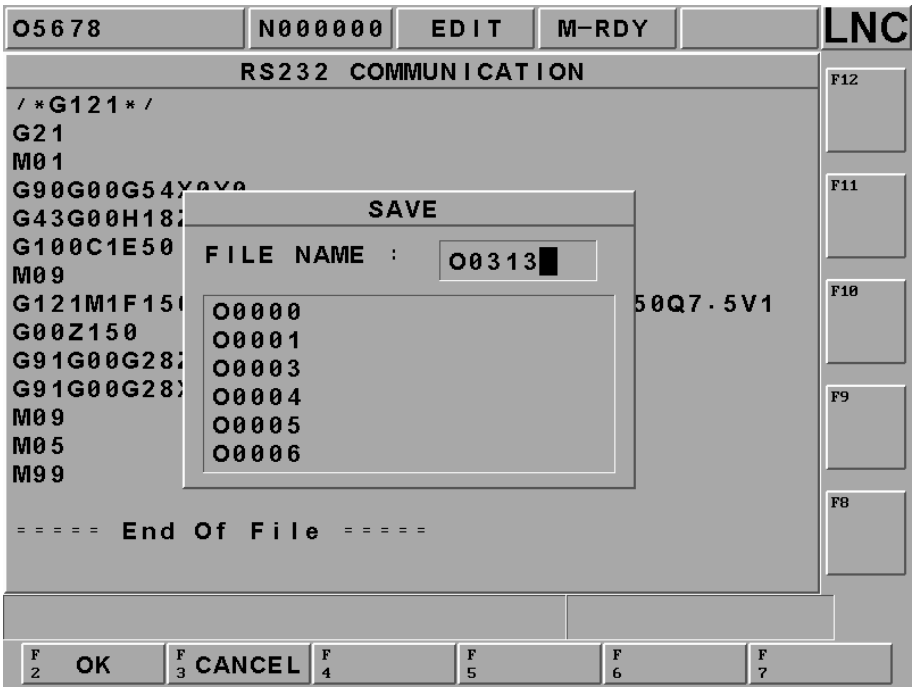
6. After communicate setting at PC end, open an old file for transferring, press [Send], confirm to upload file, press [OK], upload complete .



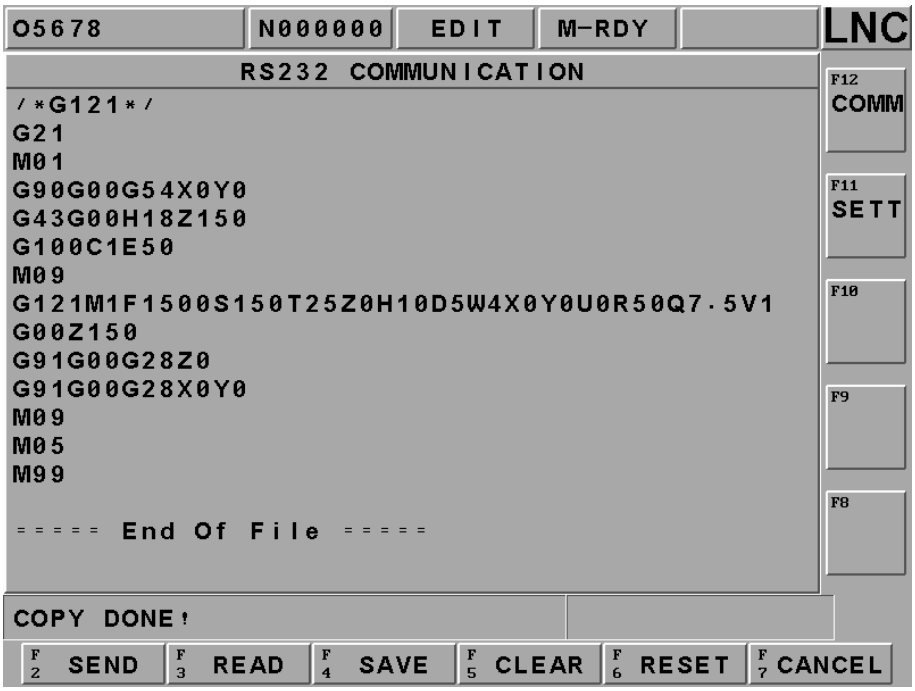

7. PC end will start to receive the files from NC end automatically.



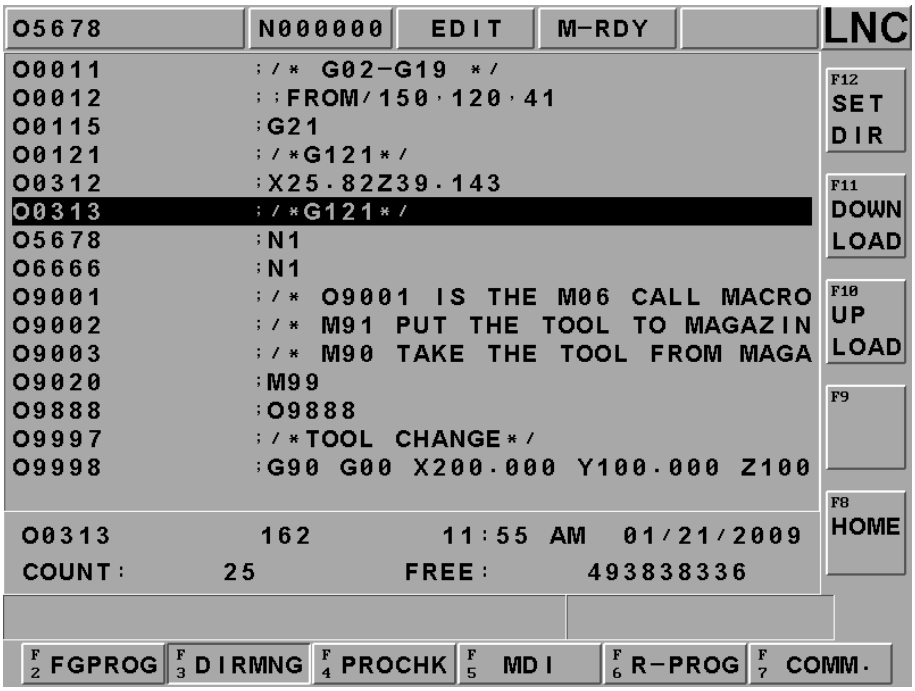
8. After receiving, press SAVE with a new name or SAVE at the same file.



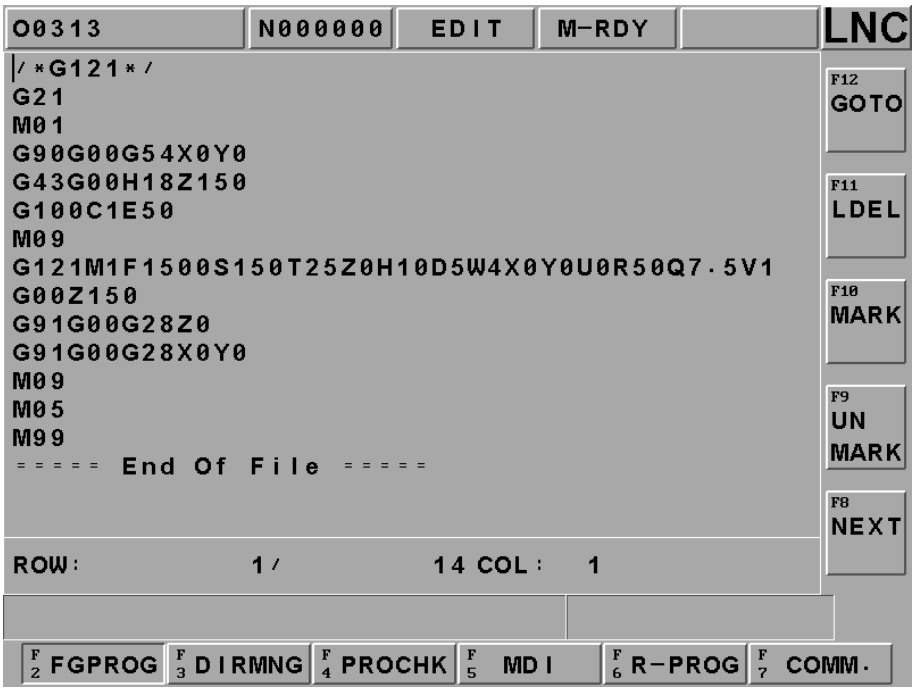

9. Press OK, Copy complete.



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




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

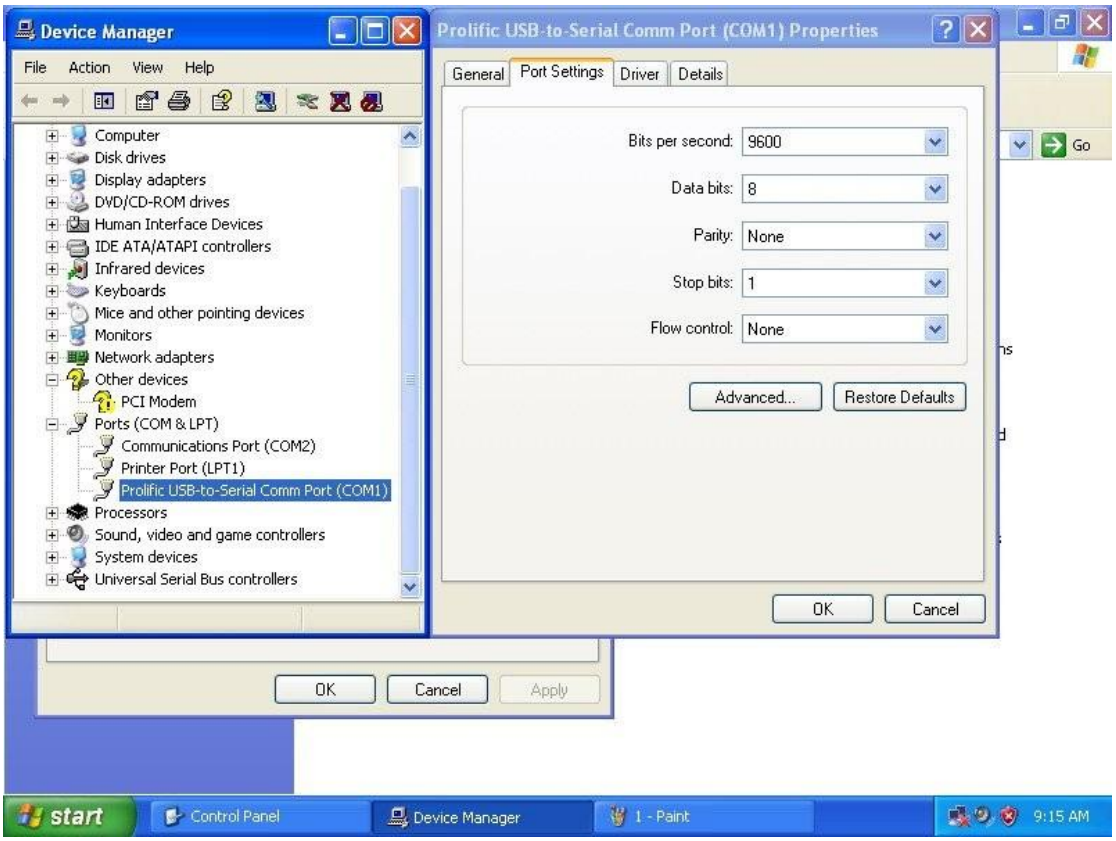


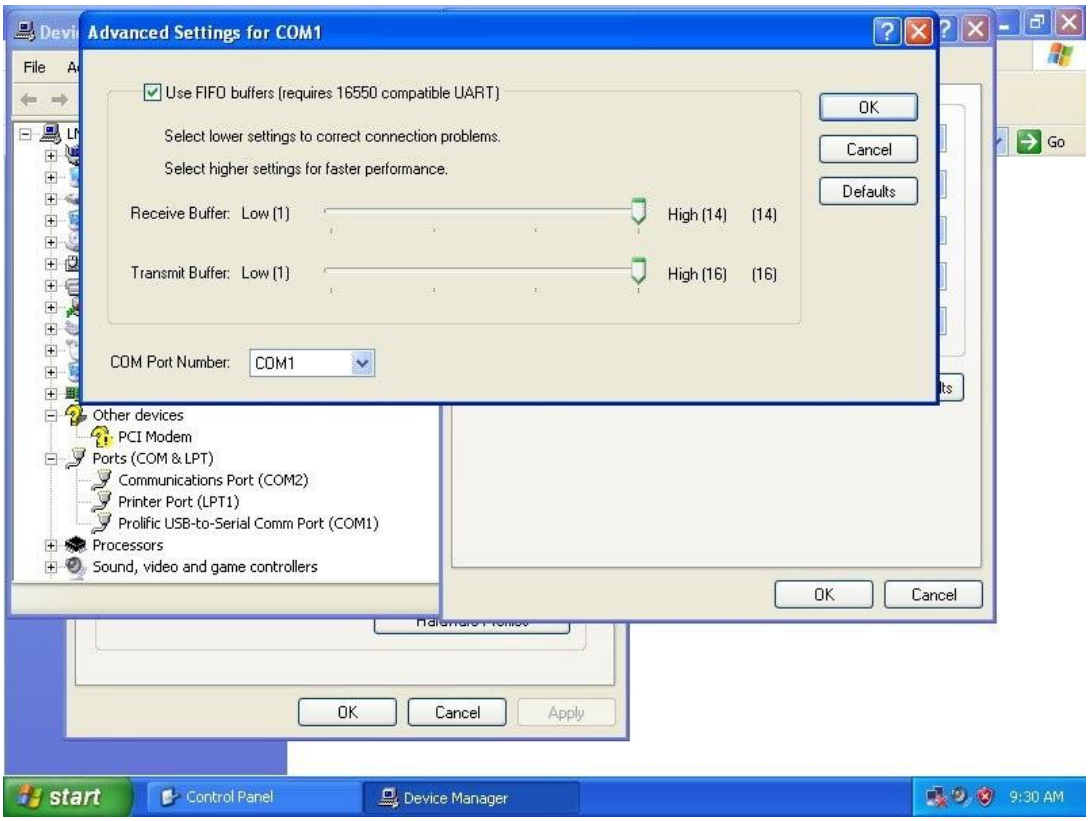
## 3.5 Setting & Attention for USB to RS232 port

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.




Advanced Settings for COM1
Use FIFO buffers (requires 16550 compatible UART)
Select lower settings to correct connection problems.
Select higher settings for faster performance.
Receive Buffer: Low (1) High (14) (14)
Transmit Buffer: Low (1) High (16) (16)
COM Port Number: COM1
OK
Cancel
Defaults
Other devices
PCI Modem
Ports (COM & LPT)
Communications Port (COM2)
Printer Port (LPT1)
Prolific USB-to-Serial Comm Port (COM1)
Processors
Sound, video and game controllers
Apply
start
Control Panel
Device Manager
9:30 AM

# 4 SYSTEM ALARM （Alarm）and WARNING（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 (INTERPREATATION) ALARM. Definitions of the above alarm message are as following：

## 4.1 OP OPERATION ALARM

**OP 1001：X SERVO ALARM**

**OP 1002：Y SERVO ALARM**

**OP 1003：Z SERVO ALARM**

**OP 1004：4$^{TH}$ SERVO ALARM**

**OP 1100：5$^{TH}$ SERVO ALARM**

**OP 1101：6$^{TH}$ 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：DETACH/ATTACH X-AXIS AS MOVING**

**OP 1014：DETACH/ATTACH Y-AXIS AS MOVING**

**OP 1015：DETACH/ATTACH Z-AXIS AS MOVING**

**OP 1016：DETACH/ATTACH 4TH-AXIS AS MOVING**

**OP 1024：DETACH/ATTACH 5TH-AXIS AS MOVING**

**OP 1025：DETACH/ATTACH 6TH-AXIS AS MOVING**

(1) Move is not allowed when switching.

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

(1) 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 （-）**

(1) Press rest to reboot system

**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 （-）**

(1) Check each axis hardware travel limit.

## 4.2 INT INTERPRETATION ALARM

**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**

**INT 3054：（H）TOTAL DEPTH IS 0**

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

**INT 3056：（W）EACH CUT WIDTH IS 0**

**INT 3060：HOLES TOO DENSITY**

**INT 3061：HOLES COUNT MUST>=2**

**INT 3062：R MUST > Z**

**INT 3070：WRONG DATA：R=0**

**INT 3071：WRONG DATA：2R<=PHI**

**INT 3072：WRONG DATA：V>PHI**

**INT 3073：WRONG DATA：Q=0**

**INT 3074：WRONG DATA：V>=Q**

**INT 3075：WRONG DATA：（PHI+2Q）>=2R**

**INT 3076：WRONG DATA：I（J）=0**

**INT 3077：WRONG DATA：I（J）-2R<=0**

**INT 3078：WRONG DATA：2Q+PHI>=I（J）**

**INT 3079：WRONG DATA：2V+PHI>I（J）**

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

**INT 3081：WRONG DATA：2（R-V）<PHI**

**INT 3082：WRONG DATA：2（R-V）<=PHI**

**INT 3083：WRONG DATA：2V+PHI>=I（J）**

**INT 3084：WRONG DATA：0.414*PHI+I<=2C**

**INT 3085：WRONG DATA：0.414*PHI+J<=2C**

**INT 3090：T COMMAND ERROR**

**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 3110：FETCH ERROR**

**INT 3111：LACK OF FILENAME（P ADDRESS IS NOT ENTERED）**

(1) Lock 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 file name.

(3) Press RESET to clear wrong warning message.

**INT 3113：FILE NOT EXIST**

(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**

(1) There is no M02 or M30 error in part program.

(2) Modify error in part program.

**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 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 wrong warning message.

**INT 3124：MACRO UNDERFLOW（G67）**

**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 warning message.

**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 3129：SINGLE BLOCK ONLY 1 SET OF SYSTEM M CODE(Ex. M00, M01, M02, M30, M98)**

(1) Check if this single block using system M code or customized M code at the same time.

**INT 3130：COORDINATE ERROR**

**INT 3131：UNKNOWN PLANE**

**INT 3132：USE G02/G03, 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 3133：CALL MACRO ACCEPT ONLY 1 M CODE**

(1) Check if this single block call Macro whether to use multiple sets of M code.

**INT 3134：IN SINGLE BLOCK, M CODE (USER) CAN NOT BE OVER 3**

(1) Check if this single block use more than 4.

**INT 3140：SEND TABLE1 ERROR**

**INT 3141：NO FREE VARIABLES**

**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 3151：IP MAINTAIN ERROR**

**INT 3152：CAN CYCLE EXECUTE G27，G28，G29，G30**

(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 3154：ILLEGAL IN CC**

(1) Executing illegal motion in CANNED CYCLE.

(2) Please cancel any illegal motion in CANNED CYCLE before executing.

(3) Press RESET to clear wrong warning message.

**INT 3156：ILLEGAL G31 IN COMPENSATION**

**INT 3157：G10 P CODE OUT OF RANGE**

**INT 3158：G10 L/E CODE OUT OF RANGE**

**INT 3160：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 3161：DNC RS232 LOSS DATA PACKGE**

**INT 3162：DNC RS232 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 RANGE IS 1-10**

**INT 3164：PATH FOLLOW AND FOLLOW AXIS HAS CONFLICT IN MOVEMENT**

**INT 3165：G54,P IS OVER RANGE**

**INT 3166：G10 COMMAND HAS ILLEGAL**

**INT 3167：G05 P/L COMMAND NEED TO SHOW TOGETHER**

**INT 3173：IN DO, N VALUE SHOULD OVER 0 OR SMALLER THEN 4**

**INT 3174：WHILE DO RANGE OVERLAP**

**INT 3175：LOOP FROM OUTSIDE TO GO IN WHILE DO**

**INT 3176：DO-END HAS NO 1:1**

**INT 3200：GOTO LINE IS THE SAME WITH CURRENT ONE**

**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：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：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 3205：COMP VECTOR LENGTH 0**

(1) DETERMINE values is 0.

(2) Check part program.

(3) System error, please contact supplier.

**INT 3206：COMP INTERFERENCE**

(1) Over cutting interference occurs.

(2) Decrease compensation radius or modify part program pathway.

(3) Press RESET to clear wrong message error.

**INT 3301：THIS AXIS IS ONE OF THE SIMULTANEOUS AXIS, CAN NOT GIVE MOVE COMMAND TO IT**

**INT 3303：CHANING AXIS COMMAND CAN NOT BE REPEAT WHEN CHANGING**

**INT 3999：ALARM NUMBER IS OVER**

## 4.3 MOT MOTION RELATED ALARM

**MOT 4001：X AXIS ERROR COUNTER OVERFLOW**

**MOT 4002：Y AXIS ERROR COUNTER OVERFLOW**

**MOT 4003：Z AXIS ERROR COUNTER OVERFLOW**

**MOT 4004：4TH AXIS ERROR COUNTER OVERFLOW**

**MOT 4202：5TH AXIS ERROR COUNTER OVERFLOW**

**MOT 4203：6TH AXIS ERROR COUNTER OVERFLOW**

(1) Motion board servo 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、1142~1145.

(2) Clicking RESET to set new parameter.

(3) After changing the parameter, please reboot.

**MOT 4006：X AXIS SERVO LAG OVERFLOW MORE THAN PR.0002**

**MOT 4007：Y AXIS SERVO LAG OVERFLOW MORE THAN PR.0003**

**MOT 4008：Z AXIS SERVO LAG OVERFLOW MORE THAN PR.0004**

**MOT 4009：4TH AXIS SERVO LAG OVERFLOW MORE THAN PR.0005**

**MOT 4204：5TH AXIS SERVO LAG OVERFLOW MORE THAN PR.0250**

**MOT 4205：6TH AXIS SERVO LAG OVERFLOW MORE THAN PR.0251**

(1) Servo axis servo lag over parameter 0002 ~ 0005、0250、0251setting value.

(2) Check whether the setting speed is too fast or the parameter 0002 ~ 0005、0250、0251 setting value is too small.

(3) Clicking RESET to continue operating.

(4) If reset parameter, must reboot.

**MOT 4012：SINGLE BLOCK COMMAND ERROR**

**MOT 4014：X AXIS ON HOME DOG**

**MOT 4015：Y AXIS ON HOME DOG**

**MOT 4016：Z AXIS ON HOME DOG**

**MOT 4017：4TH AXIS ON HOME DOG**

**MOT 4206：5TH AXIS ON HOME DOG**

**MOT 4207：6TH AXIS ON HOME DOG**

(1) Servo axis is on HOME DOG

(2) Press RESET, use JOG to move out from HOME DOG to go ZRN.

**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 2nd soft limit is smaller than the smallest value.

(2) Checking parameters 1034~1041、1146~1149.

(3) Reset parameter and then reboot.

**MOT 4025：G10 P RANGE ERROR**

(1) P value is over range.

(2) Please check the part program.

**MOT 4026：X AXIS ENCODER WIRING ERROR**

**MOT 4087：Y AXIS ENCODER WIRING ERROR**

**MOT 4096：Z AXIS ENCODER WIRING ERROR**

**MOT 4097：4TH AXIS ENCODER WIRING ERROR**

**MOT 4098：5TH AXIS ENCODER WIRING ERROR**

**MOT 4099：6TH AXIS ENCODER WIRING ERROR**

(1) ENCODER brake or 5V power error.

(2) Check wiring or connector.

**MOT 4027：HOME DOG LENGTH TOO SHORT**

(1) Return HOME, the DOG is too short or speed too fast.

(2) Enlarge DOG or slow return HOME speed.

**MOT 4035：CMR SETTING WRONG**

(1) DMR setting error.

(2) Check Pr.0068 ~ 0070，0072，0100 ~ 0107，0270 ~ 0275，1112 ~ 1115，1119~1120

(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 0053 ~ 0057、0258~0259 range.

(3) Re-setting parameter and re-booting.

**MOT 4039：HOME LOW SPEED ERROR**

(1) 2nd home speed setting error.

(2) Checking whether or not the setting value is over setting parameter 1108~1111、1140~1141 range.

(3) Re-setting parameter and re-booting.

**MOT 4040：X CMP NO. ERROR**

**MOT 4041：Y CMP NO. ERROR**

**MOT 4042：Z CMP NO. ERROR**

**MOT 4043：4TH CMP NO. ERROR**

**MOT 4136：5TH CMP NO. ERROR**

**MOT 4137：6TH CMP NO. ERROR**

(1) Pitch compensation section setting error.

(2) Checking Parameter# 0112~0115、0280~0281range.

(3) Re-setting parameter and re-booting.

**MOT 4044：CMP INTERVAL ERROR**

(1) Pitch compensation section setting error.

(2) Checking Parameter # 1018~1021range.

(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) Check motor card interpolation setting or check home sensor.

**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) Check I/O board.

(2) Check all connectors on I/O board.

**MOT 4048：SPINDLE ORITENTATION 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 ORITENTATION 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 0021 that makes spindle motor has missing step in the Pulse Mode.

**MOT 4050：SPINDLE ORITENTATION 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 # 0184).

(3) Checking whether or not the spindle orientation sensor is broken.

**MOT 4051：NO SPINDLE SPEED DEFINE**

Run tuning or orientation, if SP still can't not find sensor after a circle, system will send alarm.

(1) SP speed is not defined or blank.

(2) Key in S_ before tapping command.

**MOT 4052：LAG IS OVER WHEN TAPPING**

(1) Check if the rigid tapping tuning is done.

(2) Check if Pr.1078 too small.

**MOT 4053：SP SPEED WILL OVER WHEN TAPPING**

(1) Check if SP speed is over for this gear to receive.

(2) Check if Pr.1060 too big, recommend value is 100.

**MOT 4054：RAGID TAPPING AXIS SPEED WILL BE OVER**

While tapping, Z axis speed will be over than Pr. 1004 setting.

(1) Check if rigid tapping return ACC Pr.1060 is too big. If ACC is on when returning,（P1060 is bigger then 100）, then cutting axis will be ACC, recommend value 100.

**MOT 4055：SP SERVO LAG OVER PR.1075**

(1) Check Ladder.

(2) Check if SP rotate when tapping, if not, check SP drives.

(3) Check SP Encoder wiring.

(4) Check if Pr.1075 too small.

**MOT 4057：SPINDLE SPEED CAN'T ARRIVE**

**MOT 4058：OVER SOFTLIMIT**

**MOT 4059：SP ERROR COUNTER OVER**

**MOT 4062：X AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

**MOT 4065：Y AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

**MOT 4068：Z AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

**MOT 4071：4TH AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

**MOT 4084：5TH AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

**MOT 4081：6TH AXIS ABSOLUTE ENCODER VALUE OUT OF TOLERANCE**

After servo axis re-finds home, NC will ask servo absolute encoder to do returning zero. After completing, 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 this time, NC will follow 4th 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 4063：X AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

**MOT 4066：Y AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

**MOT 4069：Z AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

**MOT 4072：4TH AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

**MOT 4079：5TH AXIS ABSOLUTE ENCODER TRANSMISSION OVER TIME**

**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 4064：X AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

**MOT 4067：Y AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

**MOT 4070：Z AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

**MOT 4073：4th AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

**MOT 4080：5th AXIS ABSOLUTE ENCODER CHECK SMM ERROR**

**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 4090：X AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 416**

**MOT 4091：Y AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 417**

**MOT 4092：Z AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 418**

**MOT 4093：4TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 419**

**MOT 4094：5TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 420**

**MOT 4095：6TH AXIS SERVO LAG TOO BIG, CHECK WIRING OR PR. 421**

**MOT 4100：COM REPEAT CHECK PR.1810 AND 1811**

**MOT 4104：X AXIS PMC/INT OUTPUT ERROR**

**MOT 4105：Y AXIS PMC/INT OUTPUT ERROR**

**MOT 4106：Z AXIS PMC/INT OUTPUT ERROR**

**MOT 4107：4TH AXIS PMC/INT OUTPUT ERROR**

**MOT 4108：5TH AXIS PMC/INT OUTPUT ERROR**

**MOT 4109：6TH AXIS PMC/INT OUTPUT ERROR**

**MOT 4121：X AXIS COMMANDED UNDER DETACHED**

**MOT 4122：Y AXIS COMMANDED UNDER DETACHED**

**MOT 4123：Z AXIS COMMANDED UNDER DETACHED**

**MOT 4124：4th AXIS COMMANDED UNDER DETACHED**

**MOT 4127：5th AXIS COMMANDED UNDER DETACHED**

**MOT 4128：6th AXIS COMMANDED UNDER DETACHED**

**MOT 4126：RESTART PROCEDURE CAN NOT GIVE MOVE COMMAND**

**MOT 4129：LIMIT AND INDEX HAS CERTAIN SAFE DISTANCE, USE GRID TO PROTECT**

**MOT 4130：X AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4131：Y AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4132：Z AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4133：4TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4134：5TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4135：6TH AXIS MULTI-HOME LINEAR SCALE HOME PITCH ERROR**

**MOT 4138：FOLLOW SP NAME ERROR**

**MOT 4139：ANALOG VOLTAGE CHECKING CARD DO NOT EXIST**

**MOT 4140：ANALOG VOLTAGE CHECKING MAX AND MIN SETTING ERROR**

**MOT 4141：EXECUTE RETURN HOME WHEN AT PREPARE COMPLETE**

**MOT 4142：STOP INT M CODE, M CODE REPEAT, CHECK PR.211~220**

**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**

**MOT 4149：X AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.845**

**MOT 4150：Y AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.846**

**MOT 4151：Z AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.847**

**MOT 4152：4TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.66**

**MOT 4153：5TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.260**

**MOT 4154：6TH AXIS ABS ENCODER CAN NOT USE ROTATION AXIS, CHECK PR.261**

**MOT 4155：PMC G00 ACC/DEC(PR.398)ERROR**

**MOT 4156：PMC G01 ACC/DEC (PR.399) ERROR**

**MOT 4157：PMC G00 POSITION CHECK RANGE(PR.1831~1836)ERROR**

**MOT 4158：PMC G01 POSITION CHECK RANGE (PR.1837~1842) ERROR**

**MOT 4159：1 SOFTWARE PROHIBIT ERROR**

(1) Check Pr.1900 ~ 1911.

**MOT 4160：2 SOFTWARE PROHIBIT ERROR**

(1) Check Pr.1912 ~ 1923.

**MOT 4161：I/OCARD COMMUNICATION ERROR (RIO2)**

**MOT 4162：OUT OF X AXIS HOME DOG, CHECK PR.1890 OR DOG SIGNAL**

**MOT 4163：OUT OF Y AXIS HOME DOG, CHECK PR.1891 OR DOG SIGNAL**

**MOT 4164：OUT OF Z AXIS HOME DOG, CHECK PR.1892 OR DOG SIGNAL**

**MOT 4165：OUT OF C AXIS HOME DOG, CHECK PR.1893 OR DOG SIGNAL**

**MOT 4166：OUT OF 5TH AXIS HOME DOG, CHECK PR.1894 OR DOG SIGNAL**

**MOT 4167：OUT OF 6TH AXIS HOME DOG, CHECK PR.1895 OR DOG SIGNAL**

**MOT 4168：TWO SETS OF INNER M CODER CAN NOT EXIT TOGETHER**

**MOT 4169：X AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1954 TOO SMALL**

**MOT 4170：Y AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1955 TOO SMALL**

**MOT 4171：Z AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1956 TOO SMALL**

**MOT 4172：4TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1957 TOO SMALL**

**MOT 4173：5TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1958 TOO SMALL**

**MOT 4174：6TH AXIS 2ND FEEDBACK EXAM WRONG, CHECK IF MACHINE INTERFERENCE OR PR.1959 TOO SMALL**

(1) Check if feedback signal was interfered.

(2) Check if have machine interference

(3) Reset corresponding 2nd encoder feedback exam tolerance Pr.1954~1959 value, and press reset to continue.

(4) Enlarge Pr.1954~1959, 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.1960 TOO SMALL**

**MOT 4176：Y AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1961 TOO SMALL**

**MOT 4177：Z AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1962 TOO SMALL**

**MOT 4178：4TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1963 TOO SMALL**

**MOT 4179：5TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1964 TOO SMALL**

**MOT 4180：6TH AXIS AND MAIN AXIS LAG, CHECK CHECK IF MACHINE INTERFERENCE OR PR.1965 TOO SMALL**

(1) Reboot system.

(2) Make Pr.1960~1965 bigger, press reset

(3) Check if have machine interference.

**MOT 4208：SERVO AXIS NO. REPEAT**

**MOT 4209：2ND FEEDBACK EXAM IMPUT PORT REPEAT, CHECK PR.1924~1929**

(1) Check if Pr.1924~1929 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 4950：SYSTEM ALARM**

**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（-）**

**MOT 9019：5TH AXIS OVER SOFTLIMIT（+）**

**MOT 9016：5TH AXIS OVER SOFTLIMIT（-）**

**MOT 9017：6TH AXIS OVER SOFTLIMIT（+）**

**MOT 9018：6TH AXIS OVER SOFTLIMIT（-）**

(1) Check Pr,1006 ~ 1013、1142~1145 and see if need to be adjusted, or servo axes move backward and out of travel limit.

(2) Press RESET and return HOME.

**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 （-）**

(1) Check G22 each axis travel limit and see if need to be adjusted, or modify program command location.

**MOT 9015：OVER CUTTING FEED START SIGNAL WAITTING**