LNC- New T6-Series

Operator's Manual

2013/9 Ver：V01.00.000(4408110124)

Leading Numerical Controller

LNC Technology Co., Ltd.

Table of Content

1 CNC Operation 1
1.1 Operation Devices Type 1
1.2 LCD Display Device 2
1.3 Screen and Function Description 8
1.3.1 Display Screen Layout 8
1.4 POS Function 9
1.4.1 DIRMNG (File Management) 27
1.4.2 PROCHK (Program Checking) 38
1.4.3 MDI 40
1.4.4 COMM (Communication of Files（RS232）) 41
1.5 WEAR 43
1.5.1 WEAR 43
1.5.2 GEOM (Geometry OFFSET) 46
1.5.3 W. SHIFT (Work Coordinate Shift) 47
1.5.4 MACRO 48
1.5.5 WORK 50
1.6 LATHE EASY-CAM 51
1.6.1 Function Description 51
1.6.2 Operation Procedure of How to Produce Working Path Program 51
1.6.3 Operation Screen and Function Key Instruction 53
1.7 GRAPH 76
1.7.1 Function Introduction 76
1.7.2 GRAPH 76
1.7.3 SET 77
1.8 DGNOS 79
1.8.1 ALARM 79
1.8.2 IOCSA 83
1.8.3 MLC2 84
1.8.4 SYSTEM 87
1.8.5 Working Parameter Page 89
1.8.6 SYSUPD 90
1.9 SOFTPL 99
1.10 PARAM 101
1.10.1 PARAM 101

1.10.2 USROPT 102
1.10.3 TLIM 104
1.10.4 NET SET 105
1.10.5 【Change password】 111
1.10.6 【Switch user】 112

2 OP Panel Operation 113

2.1 OP 113
2.2 LED SIGNAL 114
2.3 AXIS SELECTION 114
2.4 MODE SELECT 114
2.5 SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH 116
2.6 Additional Function Selection 117
2.7 EMG-STOP 119
2.8 CYCLE START&FEED HOLD 119
2.9 Feedrate/Rapid Rate Adjusting Button 120
2.10 PROGRAM PROTECTION LOCK 121
2.11 POWER ON/OFF 121

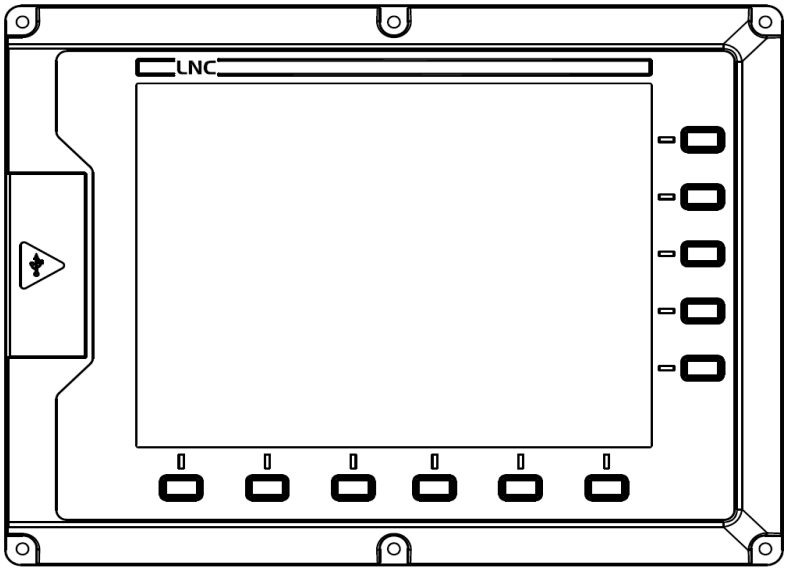
3 RS232 COMMUNICATION SOFTWARE 123

3.1 Transmission line preparation 123
3.2 PC end installation (use LNC ReCON 232 software) 124
3.3 NC end 126
3.4 Communication （RS232）-【 DNC file upload and download functions 】 140
3.4.1 Path Program Upload & save (using ReCON software) 140
3.4.2 Path Program Upload & save (using ReCON software) 143
3.4.3 Path Program Upload & save (using general software) 146
3.4.4 Path Program Upload & save (using general software) 150
3.5 Setting & Attention for USB to RS232 port 155

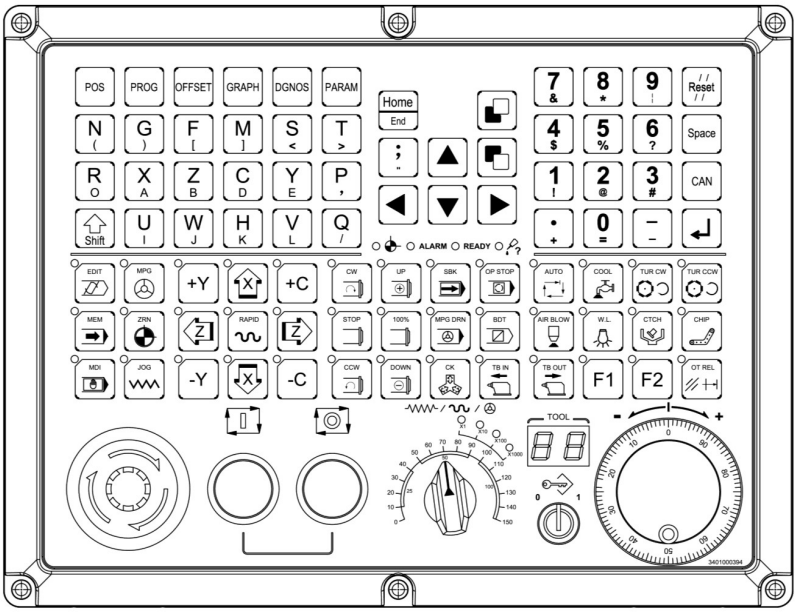
1 CNC Operation

1.1 Operation Devices Type

The control panels can be divided into two units: **LCD MONITOR** and MDI**OP**. The MDIOP data input area is for users to edit programs word by word, or edit setting. And MDIOP operation area is a panel for achieving different working process, such as multiple function keys, keys and pulse generator (hand wheel) and etc. OP will have different designs due to different tool machine designs. But, this system has one standard panel that can be selected by tool machine producers. At below we will introduce LCD display device and MDIOP data input area.MDIOP operation area will be introduced at the Chapter Two.



LCD MONITOR

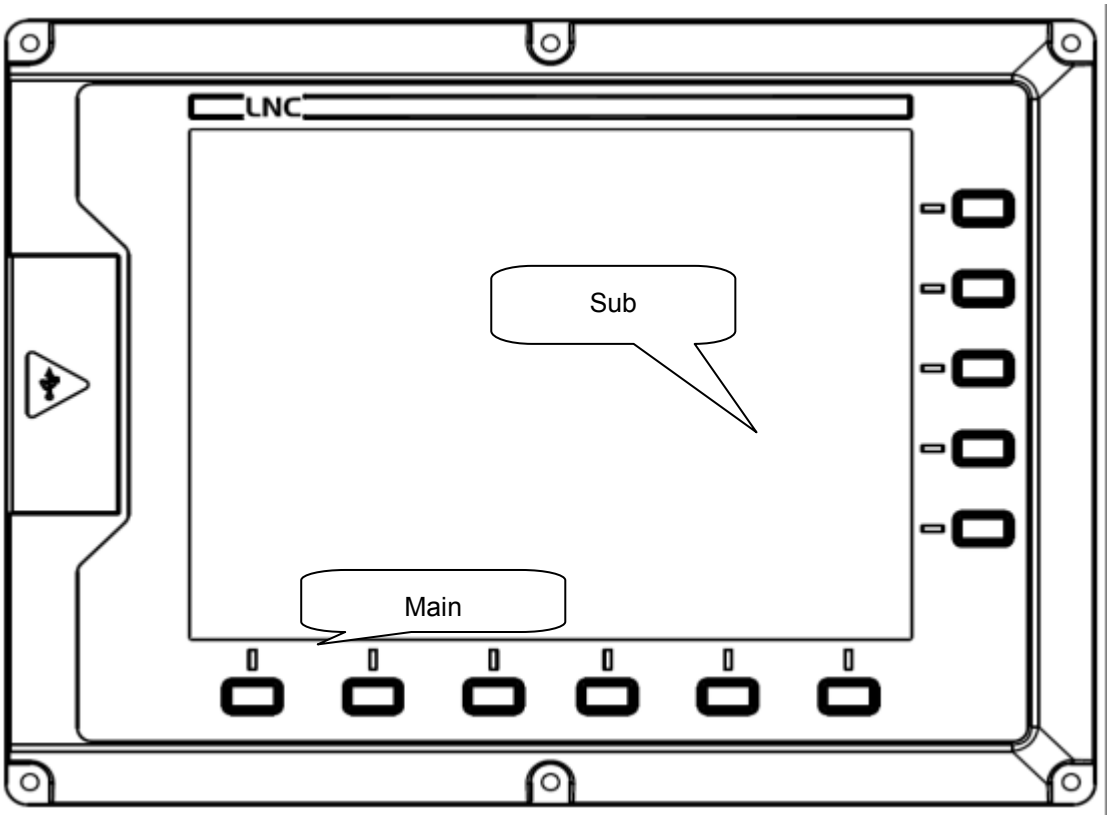


MDIOP VERTICAL

1.2 LCD Display Device

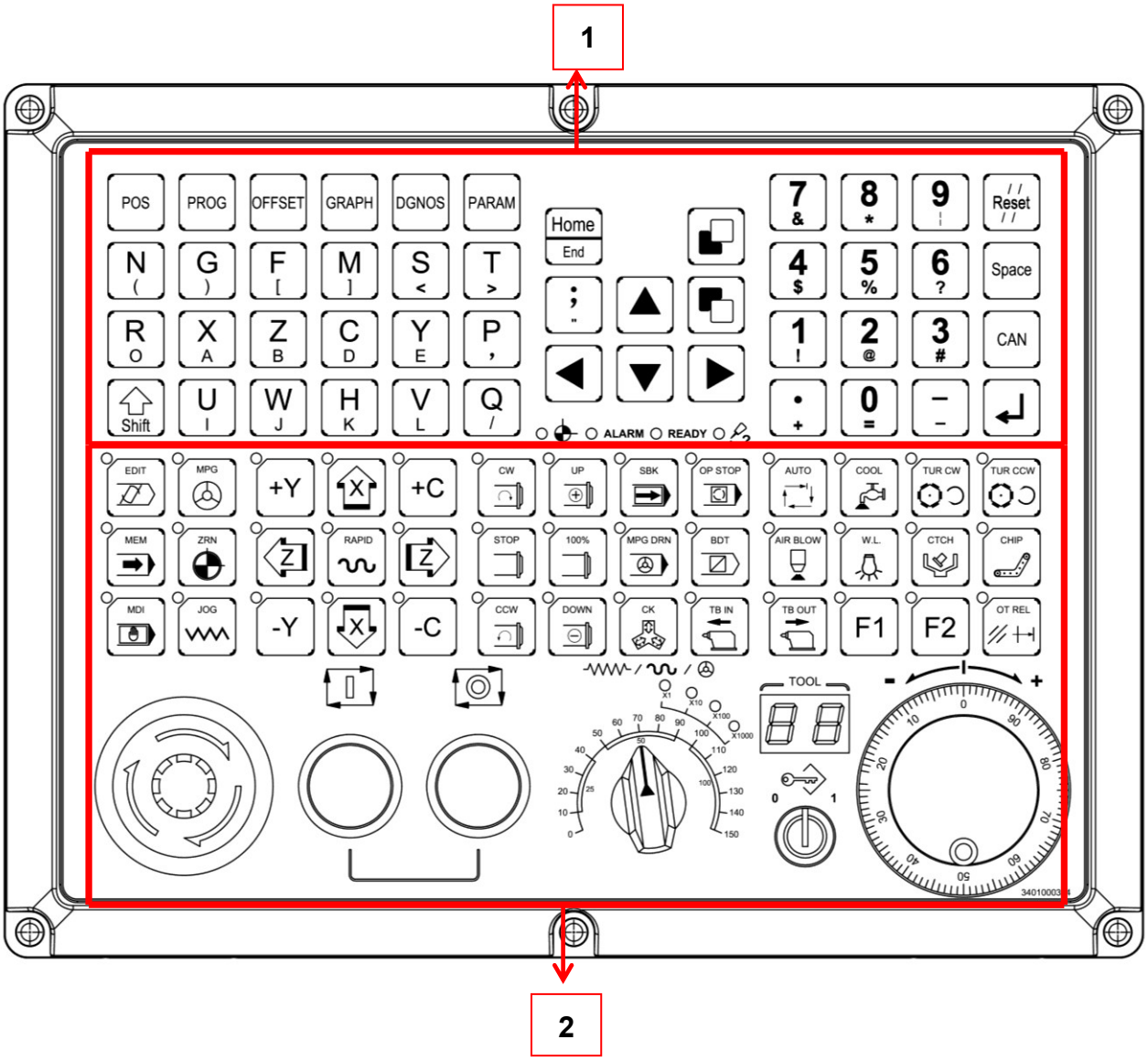
You will three types of function keys at the LCD display device:

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. Pressing the corresponding function button to select the wanted function.
3. **Additional function keys**：
   (1). LCD switches(power ON/OFF)
   (2). PS/2 Keyboard, can connect with keyboard to control.
   (3). PS/2 Mouse, no support now.



MDIOP can divide into two sections.

1. MDI DATA INPUT AREA：For users to input data.
2. OP OPERATION AREA ：For users to control operation.

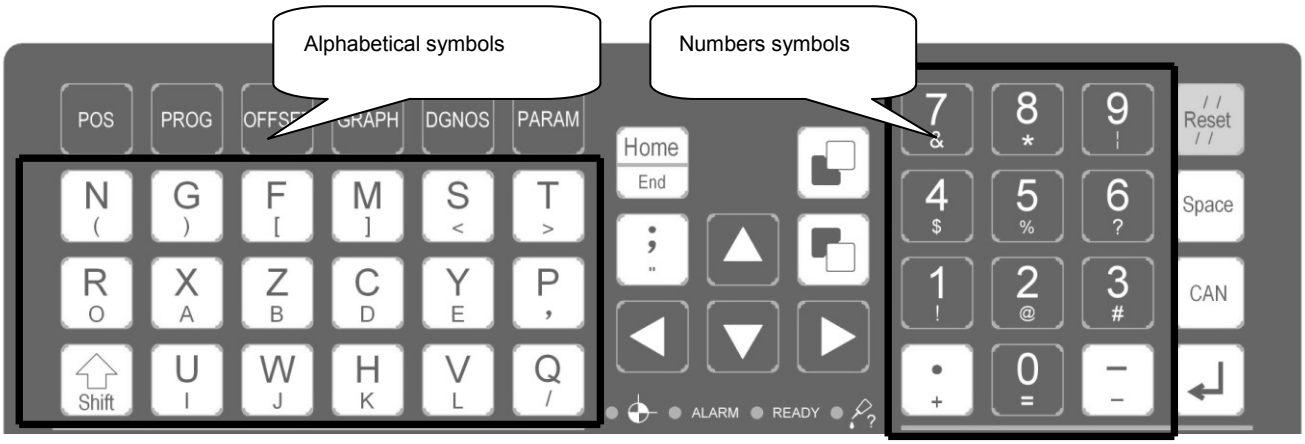


MDI DATA INPUT AREA：

A. CNC Function Keys

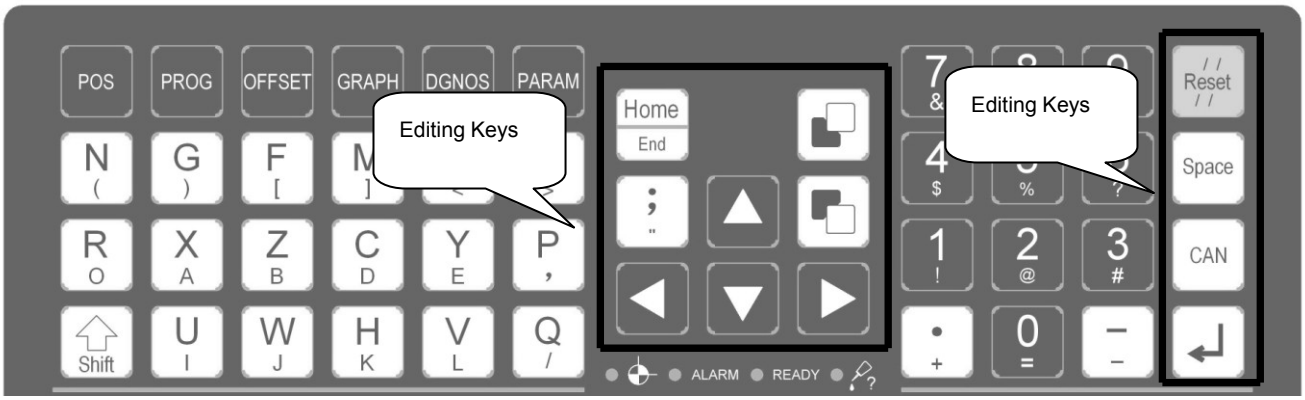
Name	Functions	Description
POS	POS	to display positions
PROG	PROG	to edit and to display program (with 1.Edit Mode2.Auto Mode3.Manual Mode)
OFFSET	OFFSET	to set and to display tool offsest
GRAPH	GRAPH	to draw tool path
DGNOS	DGNOS	to display instance messge at DGNOS page
PARAM	PARAM	to display parameter screen

B. Character & Symbol Keys：

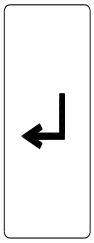
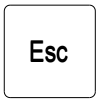


These characters, symbols and numbers are used for program editing and data key-in. There are some symbols that are diminished down at right-bottom of these keys. If want to use these symbols, please press SHIFT and the symbol key at the same time.

C. Editing Keys：



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

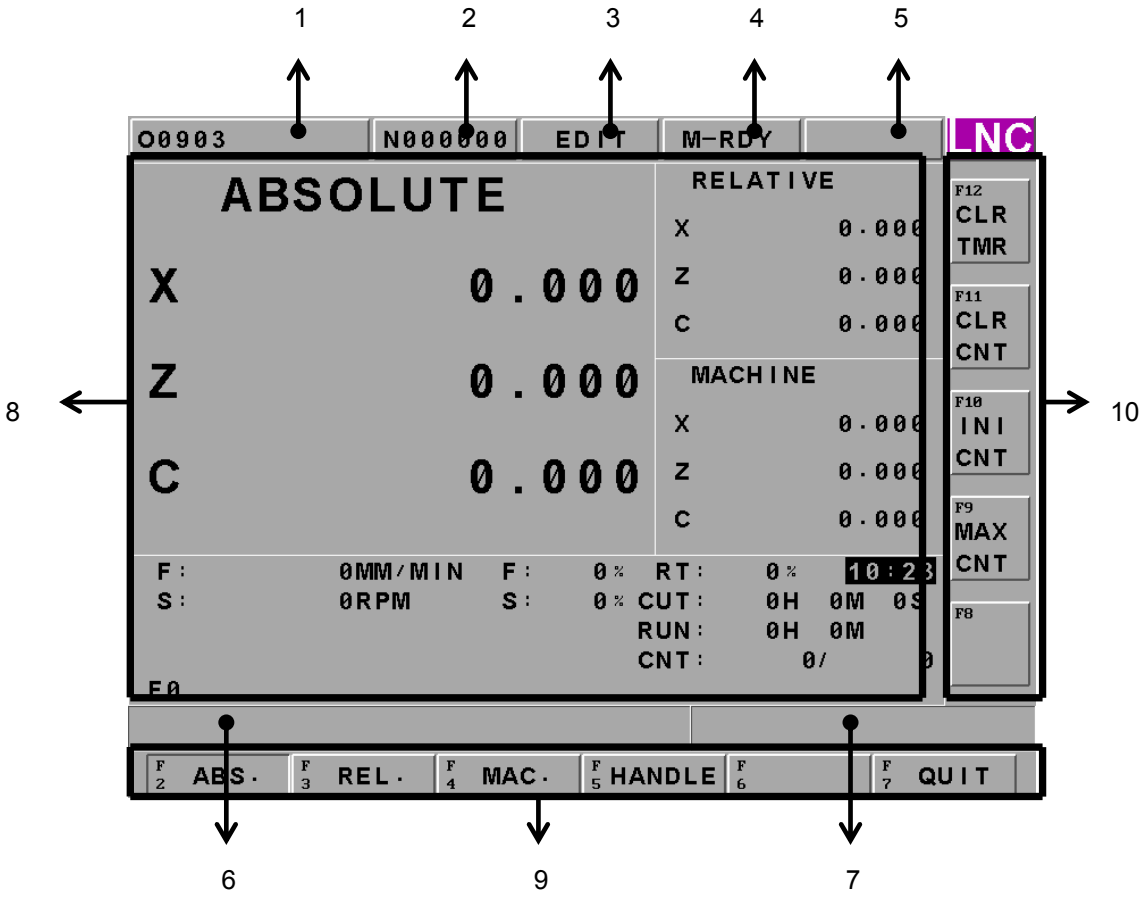
Name	Editing	Description
RESET		1. To reset system to return default. 2. When alarm removing, cancel alarm. 3. When start working, to cancel working. 4. At edit mode, cursor returns to program head.
PAGE↑		To turn to the previous page.
PAGE↓		To turn to the next page.
Enter		1. Input at the data input area, press enter to confirm. The column will be filled with the data. 2. At file management, move cursor to the file location, press enter to open the file. 3. At edit mode, press enter to insert space line.
ESC		Return back Note: this function will be valid under BIOS mode.
↑		Under edit mode, move cursor upper. To move cursor upper.
↓		Under edit mode, move cursor down. To move cursor down.
←		Under edit mode, move cursor left. To move cursor left.

Name	Editing	Description
→		Under edit mode, move cursor right. To move cursor right.
Insert		To switch enter/replace characters modes.
Delete		To delete.
Space		To key-in into empty space.
Shift		To key-in special symbols with use of symbol number keys. Note: you need to press SHIFT and symbol buttons at the same time.
Home		To return cursor to the beginning of the sentence when editing program.
End		To return cursor to the end of the sentence when editing program.
TAB		When editing program, to move cursor right with 8 space.
CAN		To cancel the previous character.

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

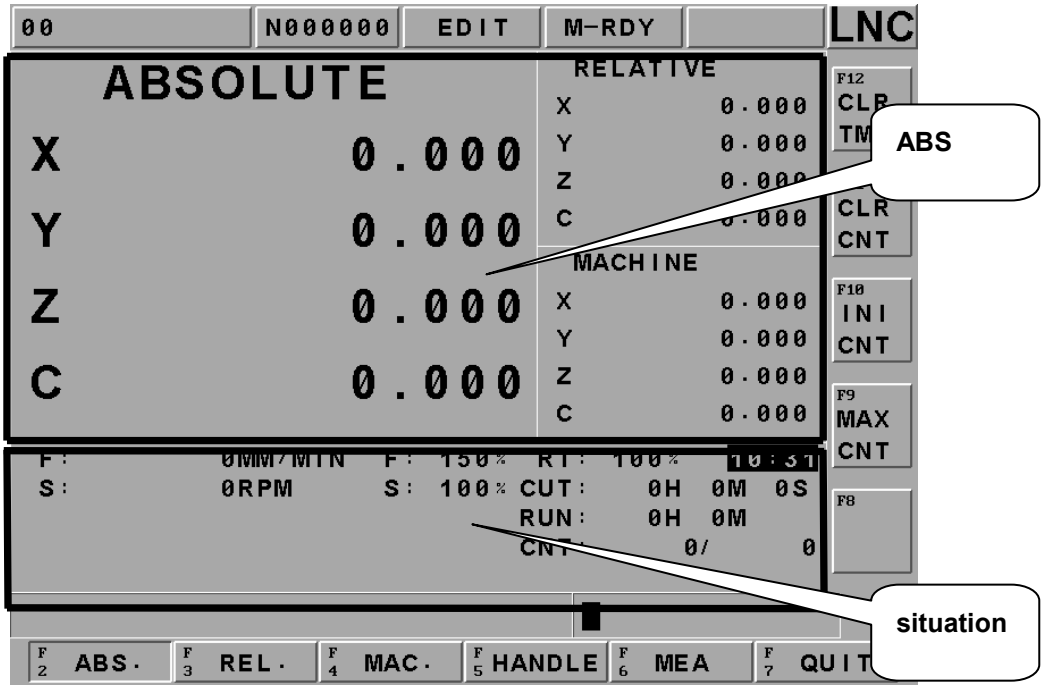
5：Wrong alarm/warning message

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

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

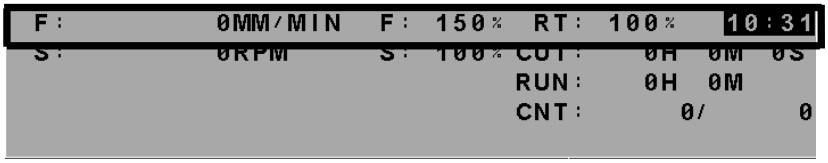
Actual Feed Rate

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%>.

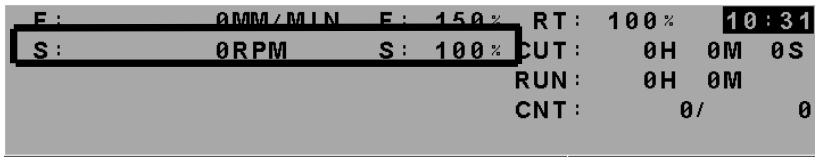
Time：Current time.



Rotation situation

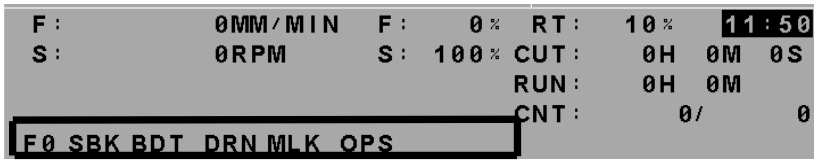
Rotational speed (rpm/min)：Current spindle actual rotation. Unit is rpm/min.

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）

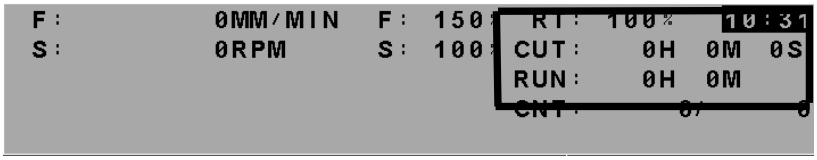


Situation of CUT, RUN, CNT

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.

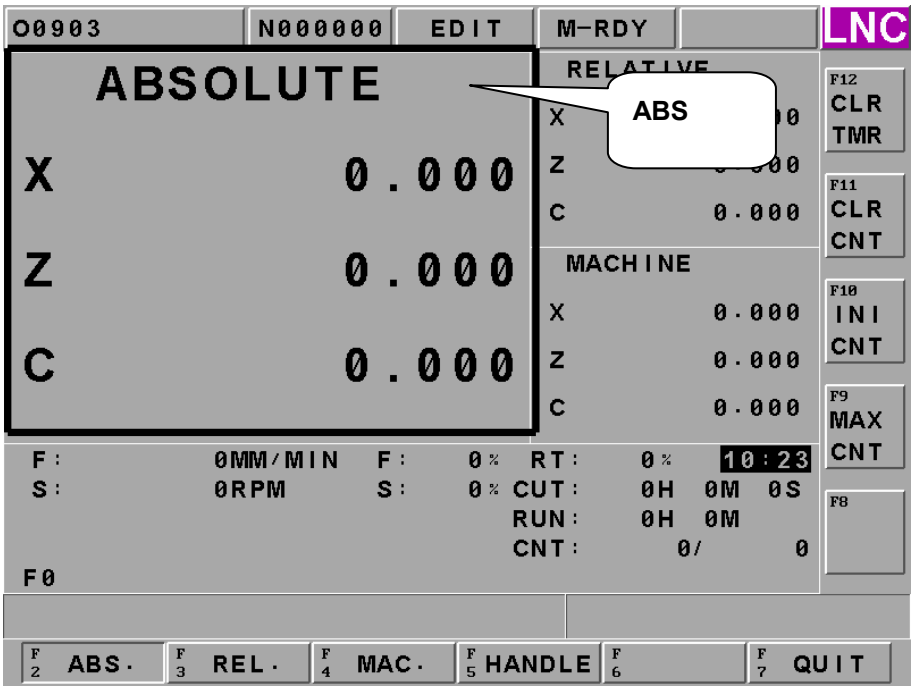
RUN(H"M)：Running time is the total cutting time after every booting, until the system shuts down, the time will be reset to be zero. You can also use【TIME CLEAR】 to make it to be zero.

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】

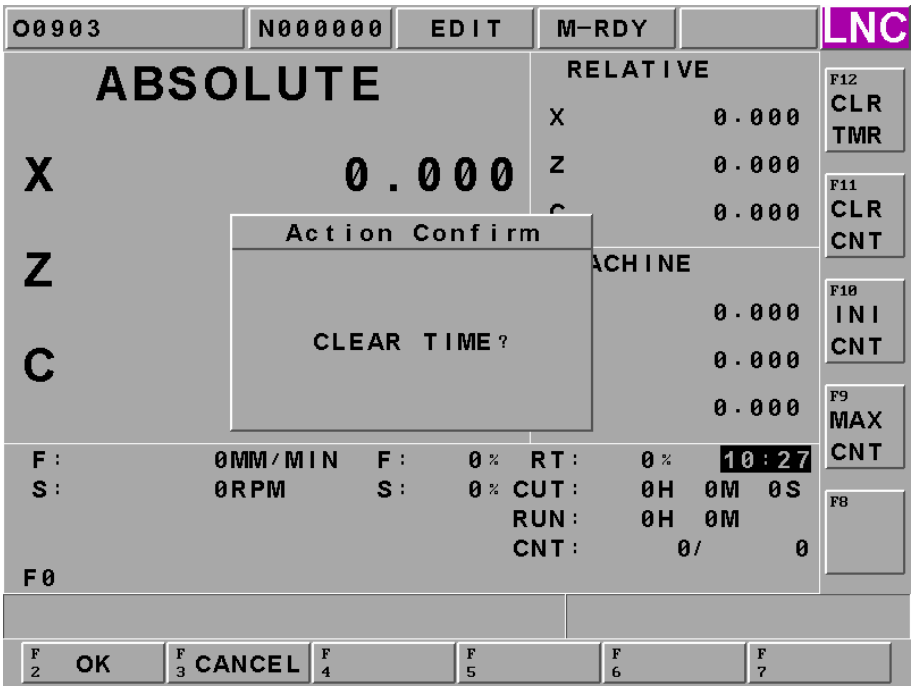
Pressing【**ABS.**】to enter into absolute coordinate screen. Details will be shown in bigger words at the upper left side of the screen. Using the program home (means the workpiece coordinate) to show tool's current location, the absolute coordinate will be changed according to the movement of tool. This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



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

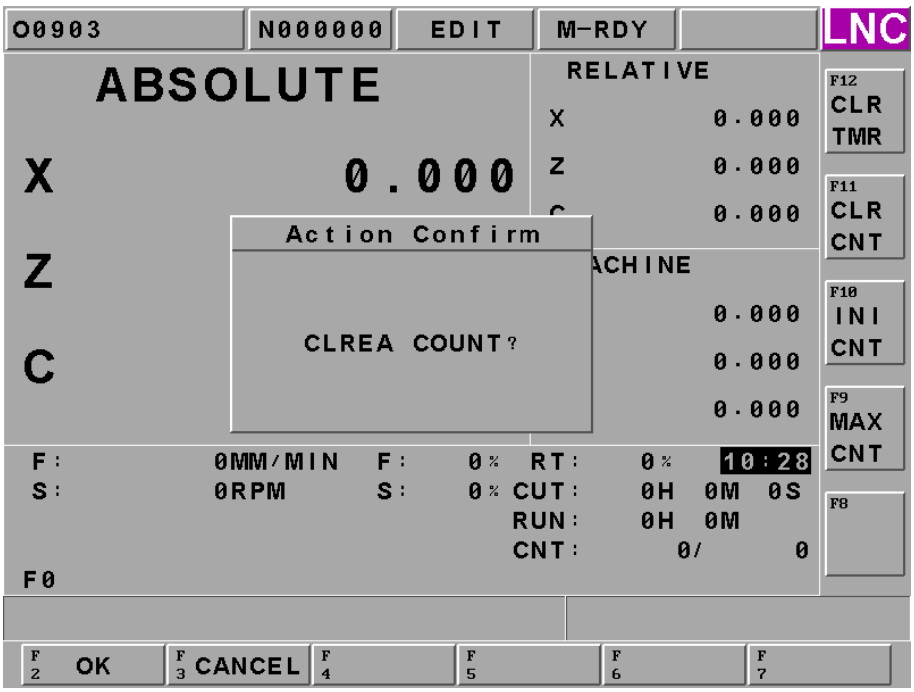
【Time Clear】

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



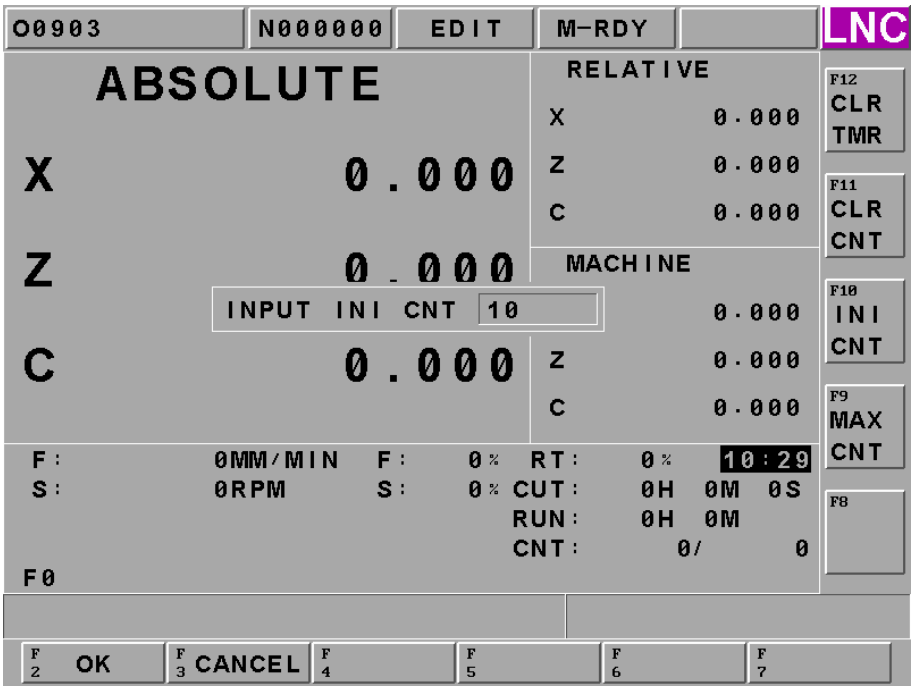
【Workpiece Clear】

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



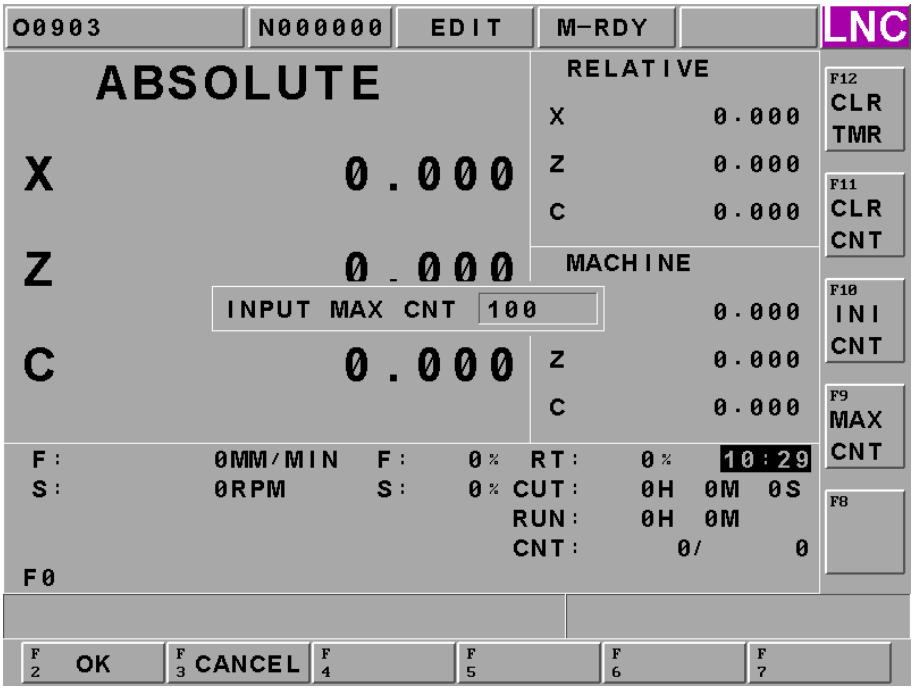
【Initial Workpiece】

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



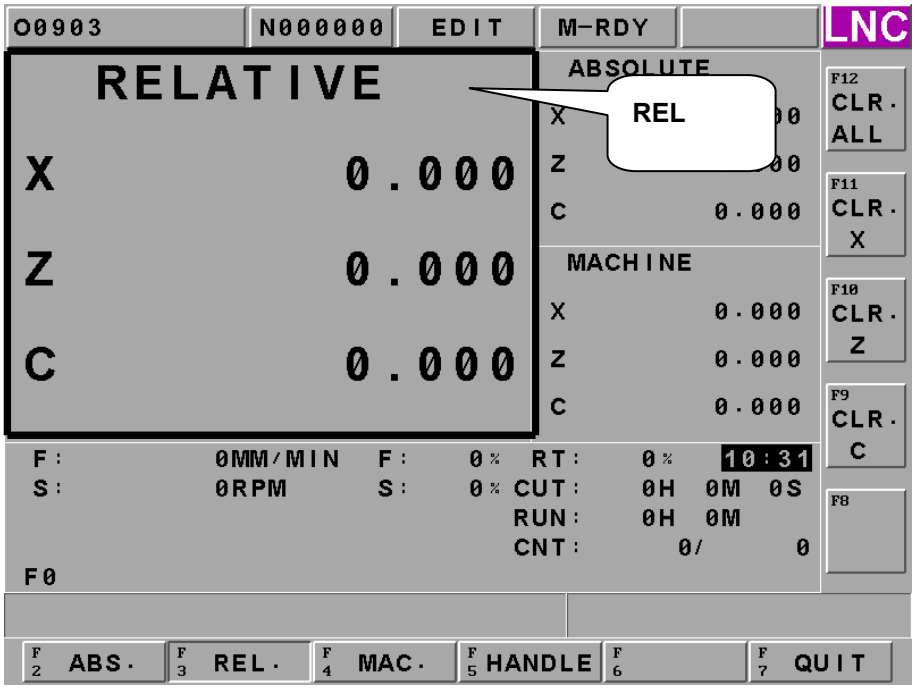
【Max Workpiece】

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：

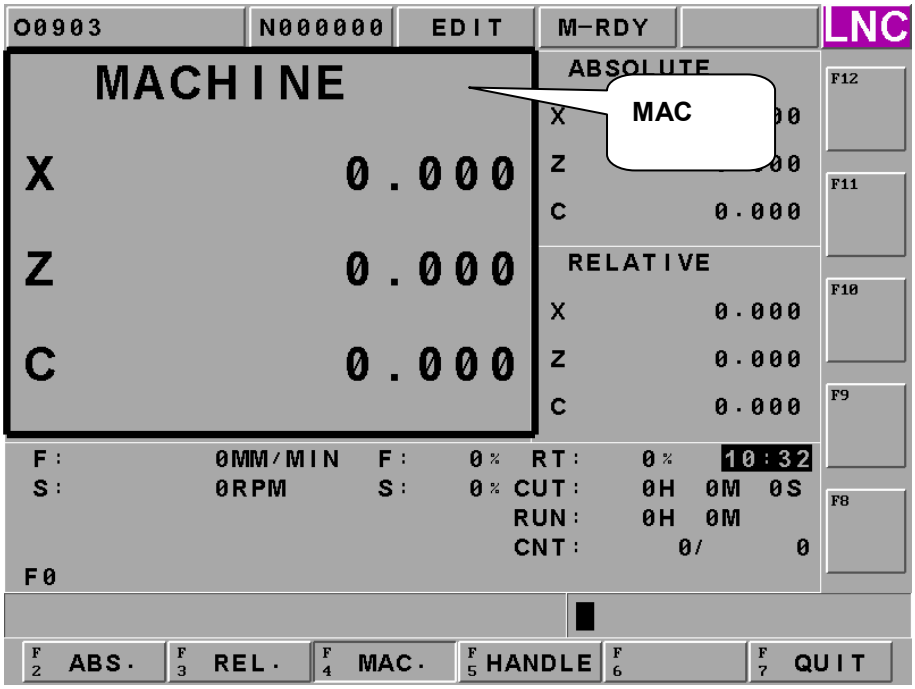
If want to set the 3 axes relative coordinate values to zero separately or at the same time, users only need to press the corresponding sub-function buttons **【CLR. X】**, **【CLR. Y】**, **【CLR. Z】**, **【CLR. ALL】** to execute the corresponding clean.

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.

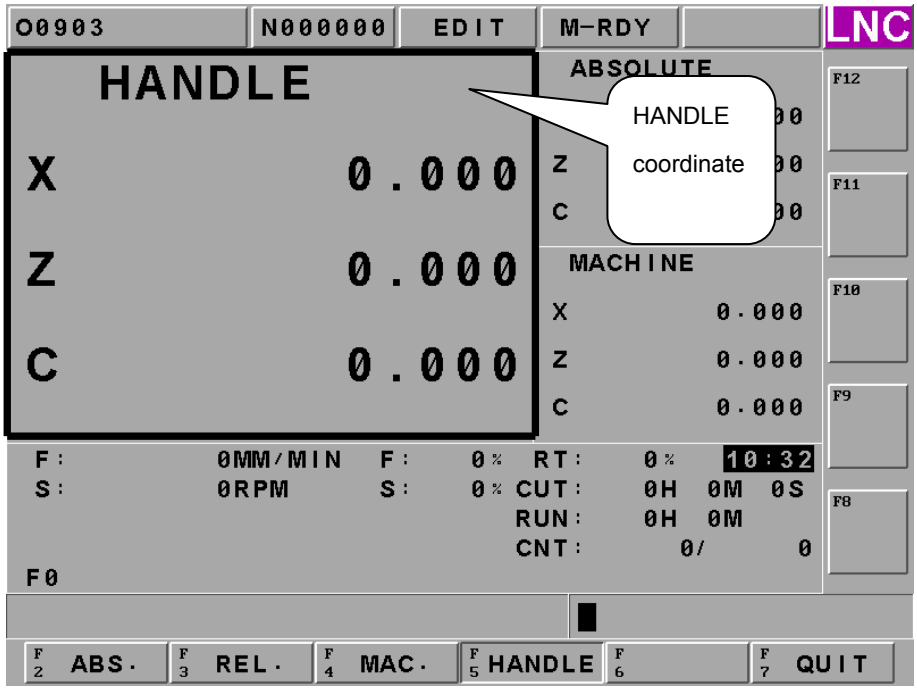


Machine coordinate is the distance of the present position corresponding to the reference point

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.



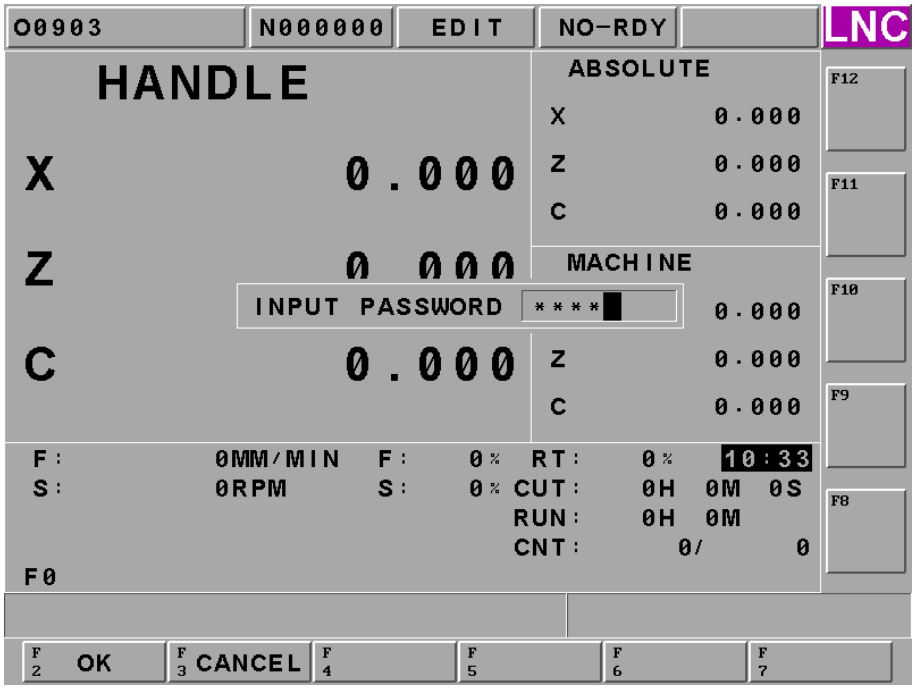
Note： Because the absolute coordinate will not be changed by MPG interruption, therefore the tool path later will have certain offset with original program path. This offset need to be modified by manual return zero to clear. And about the offset value, you can refer to MPG coordinate.

About this detailed function, you can refer to maintenance manual-MPG interruption signal selection chapter.（C BITS 066 ~ 071）and MPG rates（REGISTER 014）

ABS	will not change because of MPG interruption to show program coordinate.
REL	will not change because of MPG interruption.
MAC	will change because of MPG interruption to chow exact position of machine.

【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）

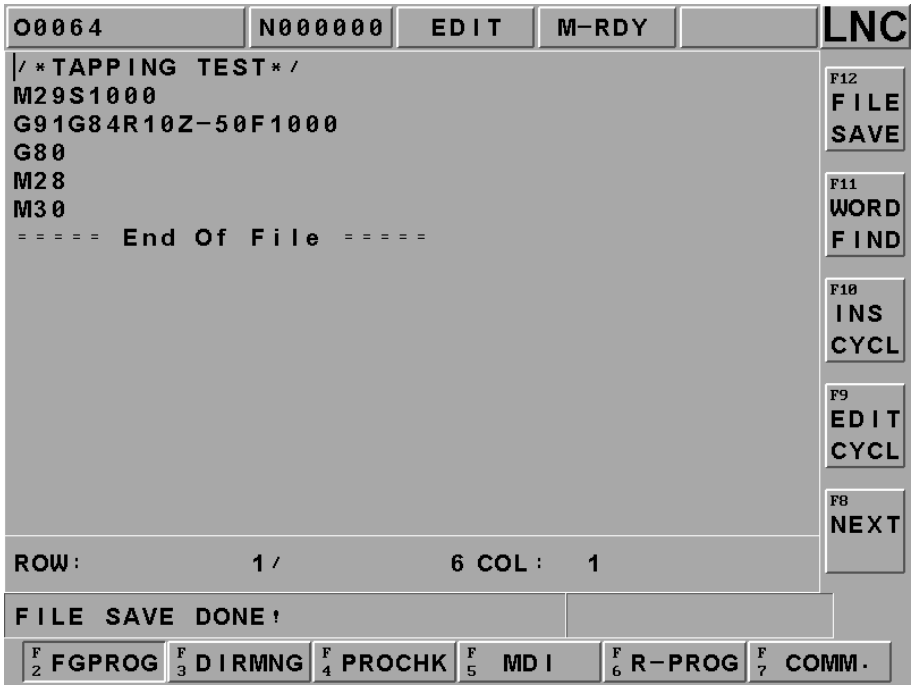
Pressing **<PROG>** to enter into program function group screen. This function group provides part program editing, file managing, checking, rebooting, RS232 transmitting and other related functions.

【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】

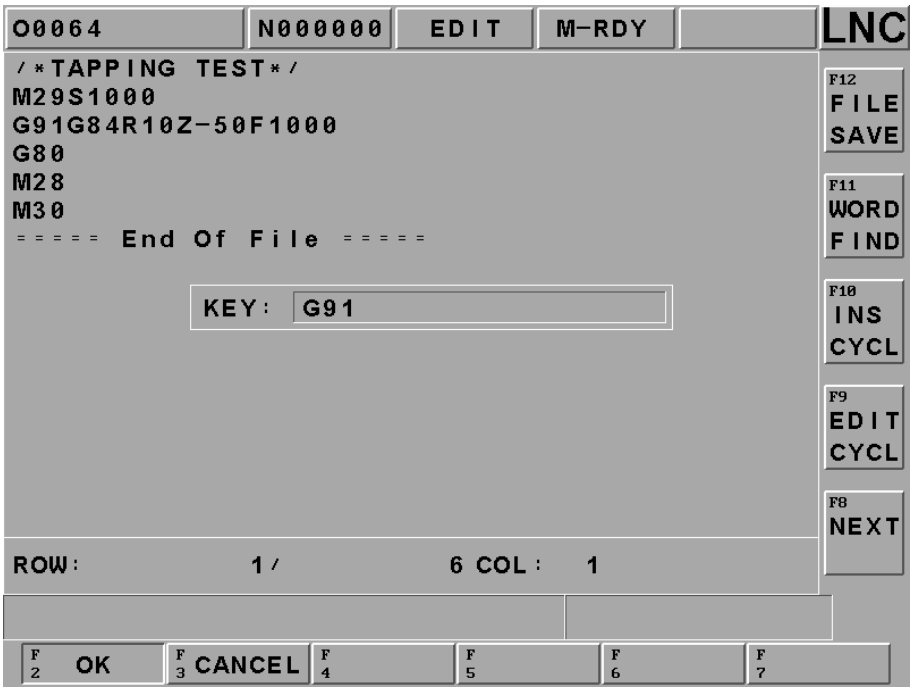
Press right side【FILE SAVE】function key to save current editing program or the file will save automatically when group page changing.



- 【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.

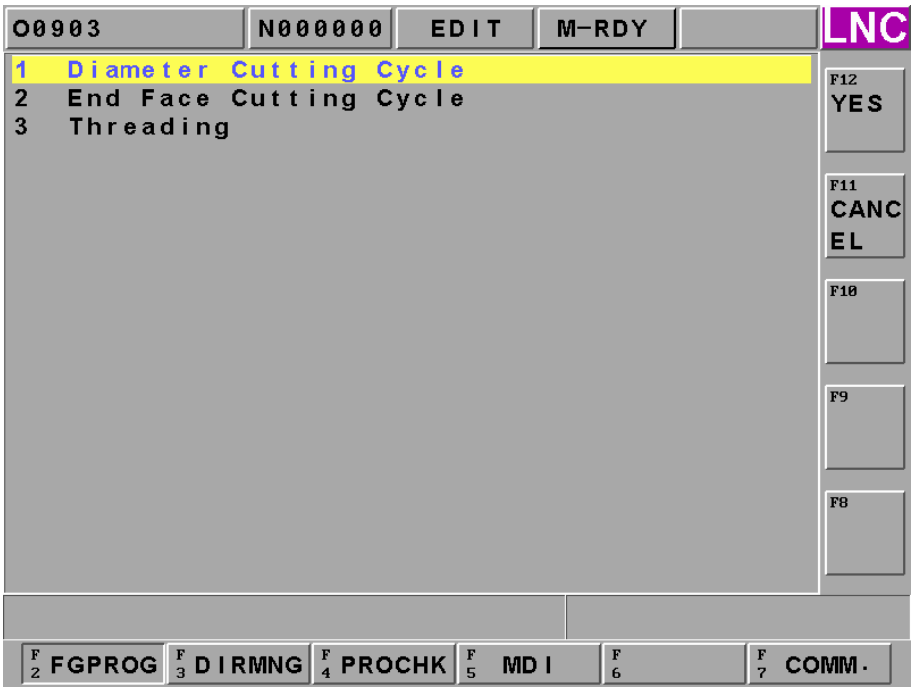
If the system can't find the word, it will show no words finding.



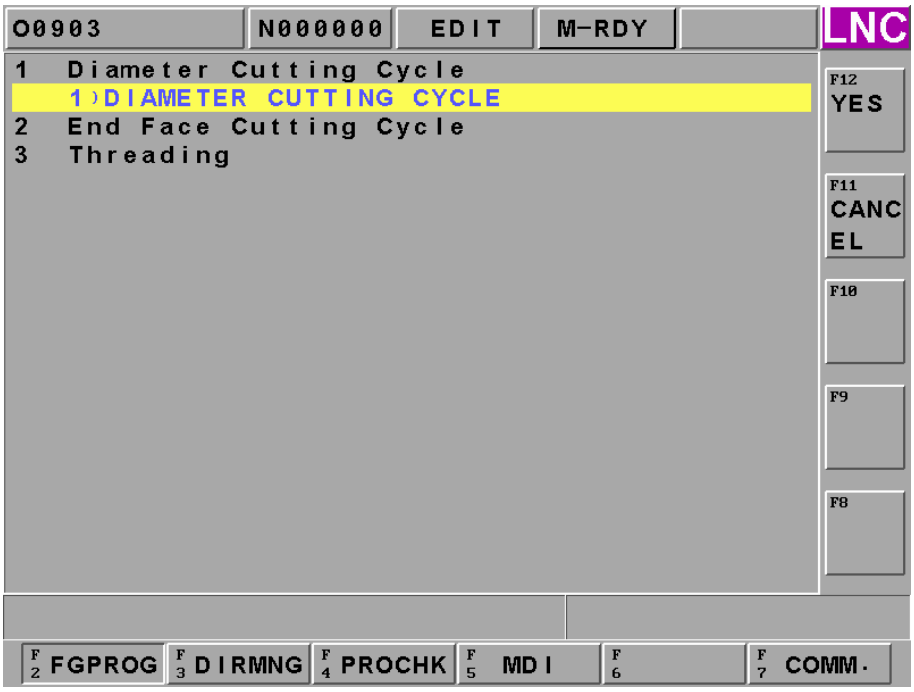
- 【INS CYCL】

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

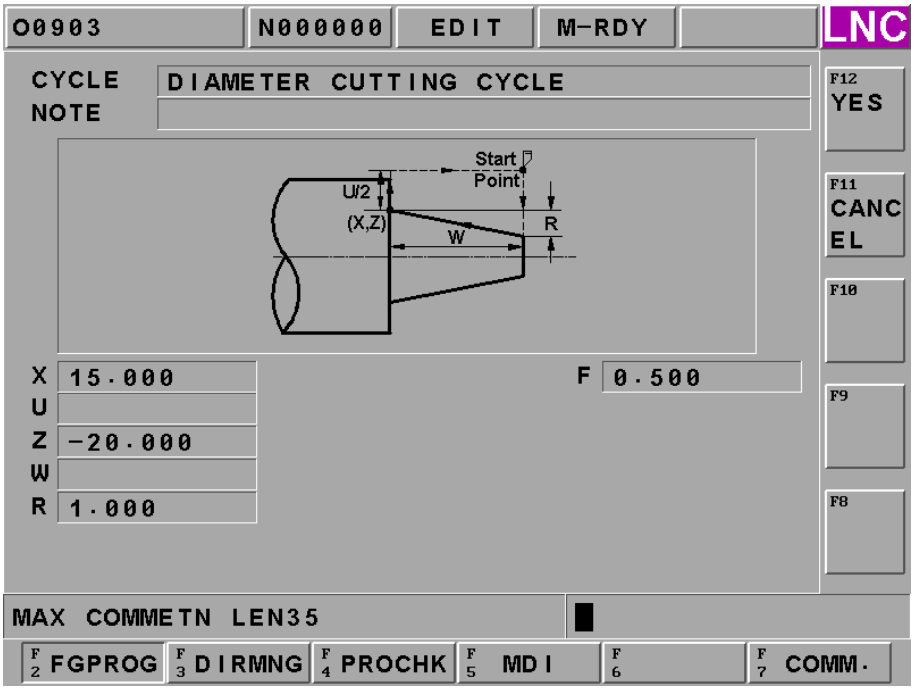
After pressing【**INS CYCL**】, the screen will be as below：



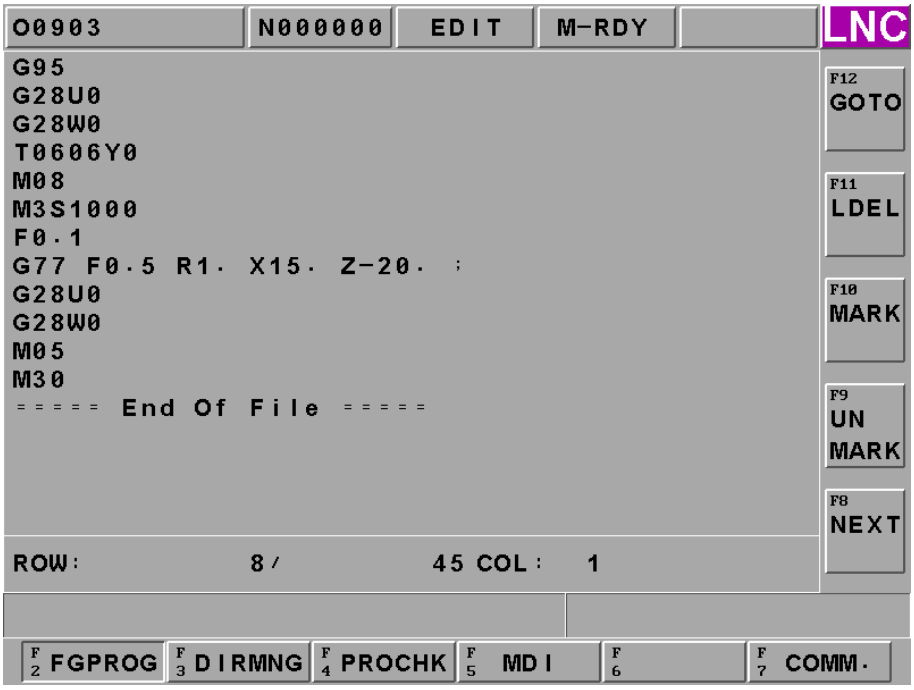
Pressing “Input” after selecting working method：



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】

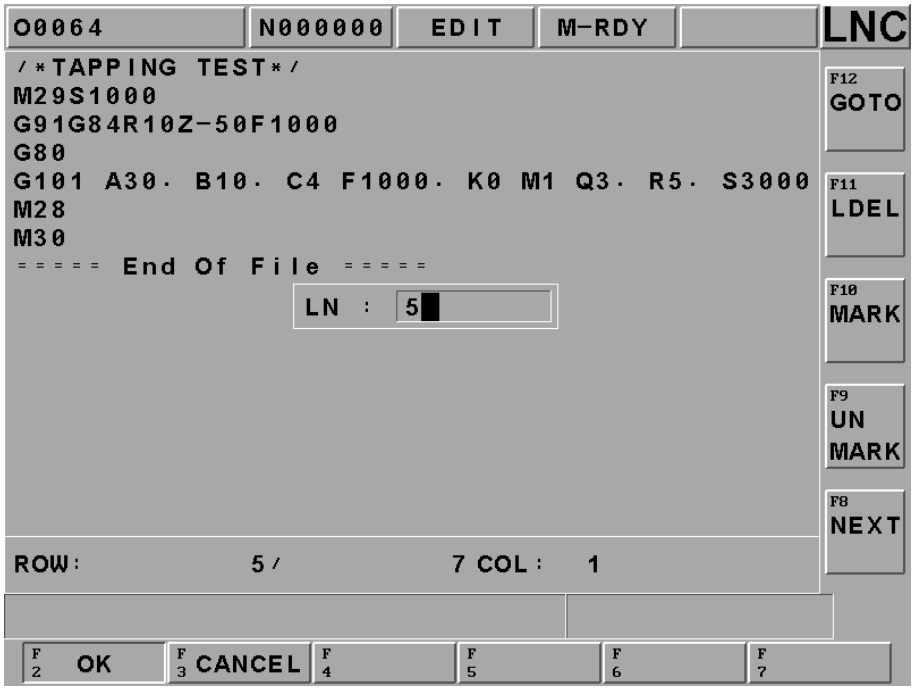


- **【EDIT CYCL】**

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.

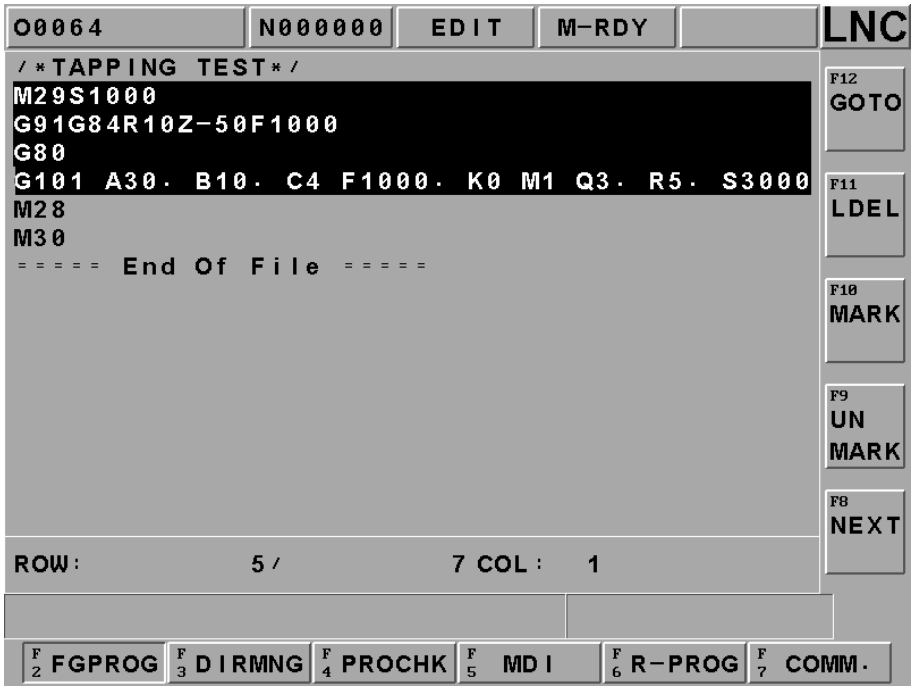
- 【LDEL】

  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.



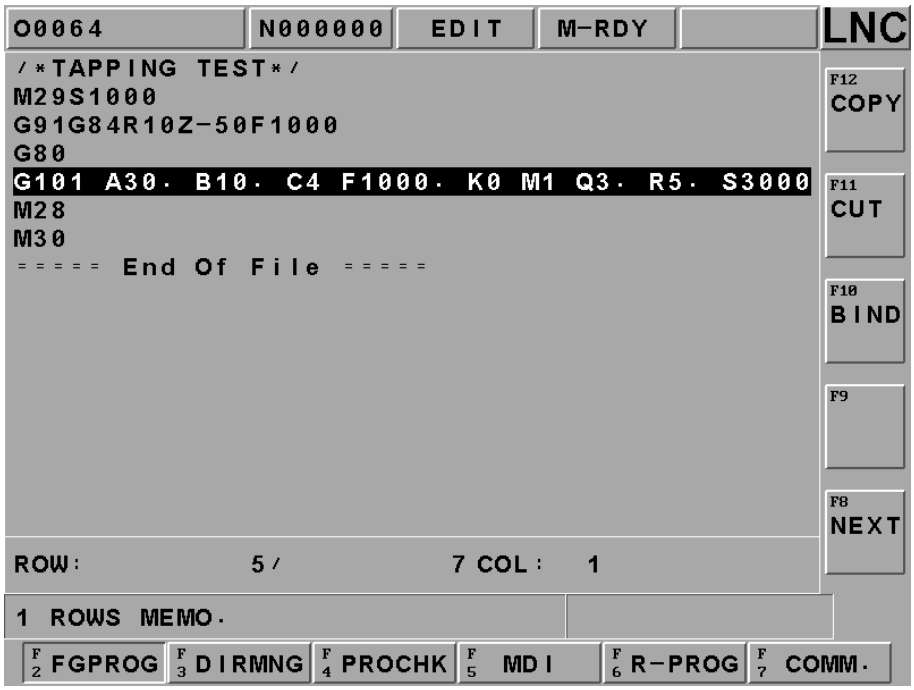
- 【UNMARK】

  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】

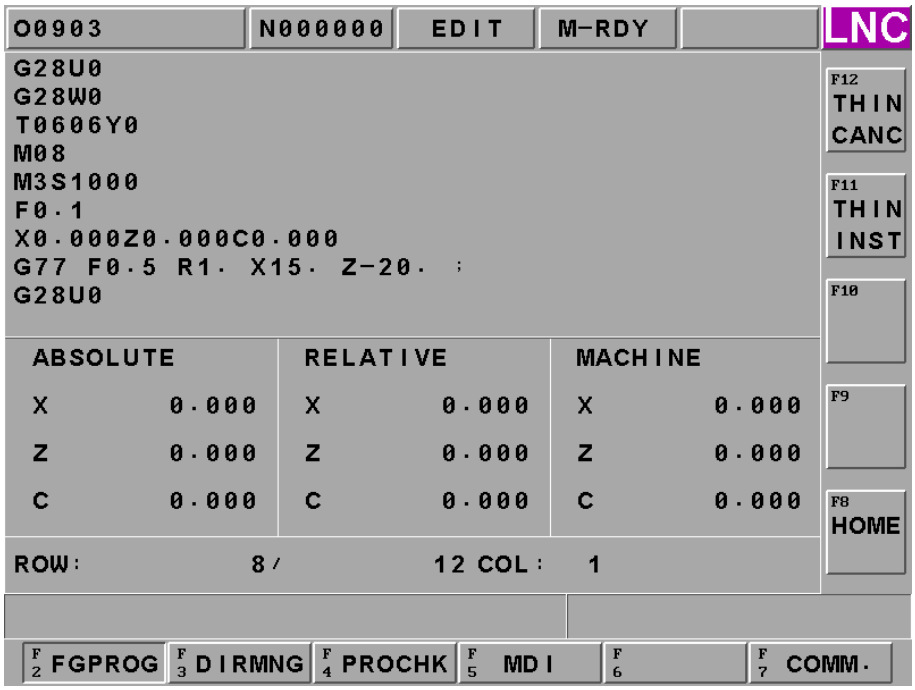
  To cut down the marked program in local range.

- 【BIND】

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

- 【Teach Mode】/【Delete Teach】

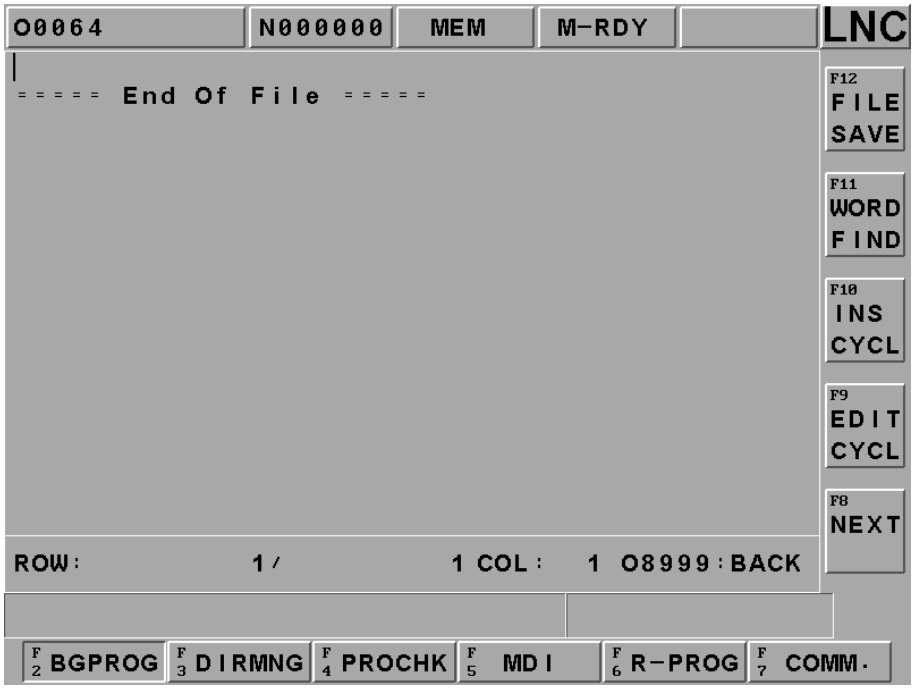
Press this button to show ABS/REL/MAC at the down side for reference. At the right side of sub-function keys will show【INSERT COORDINATE】for users to insert current location's absolute coordinate quickly, detail is as below:



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.



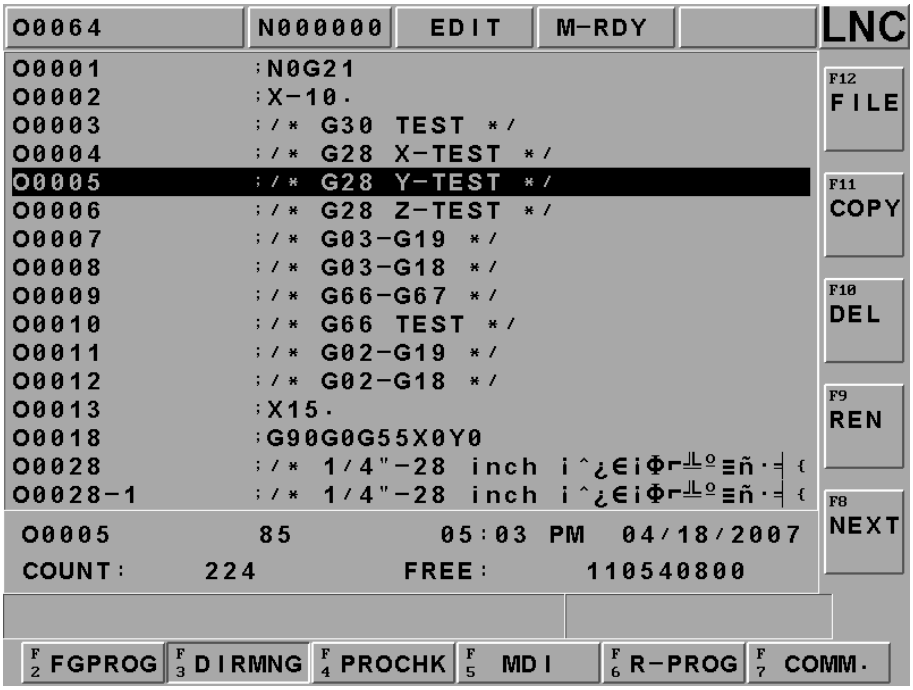
1.4.1 DIRMNG (File Management)

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

- **【FILE】**

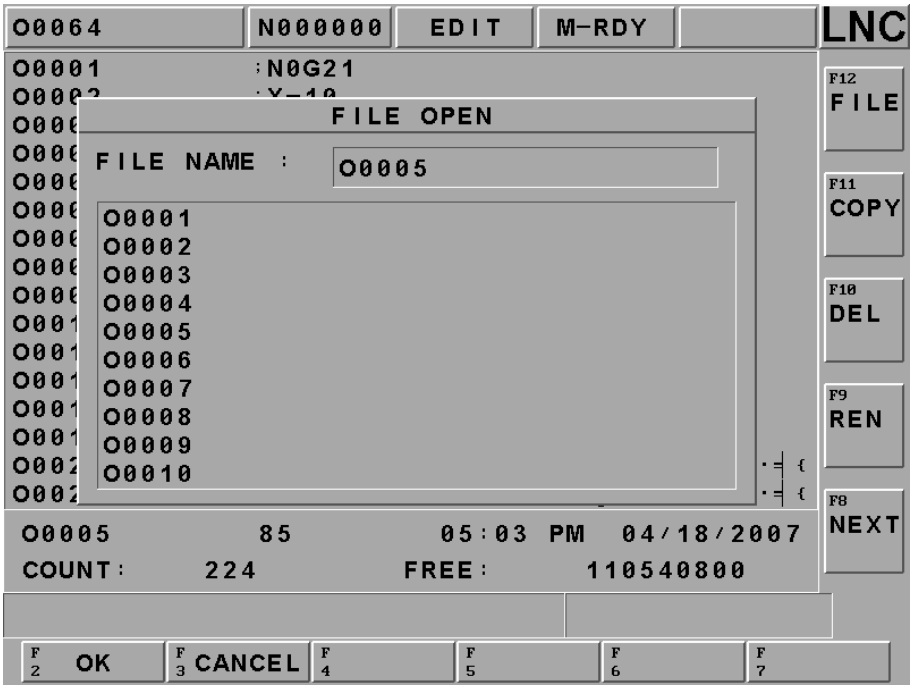
  Solution 1

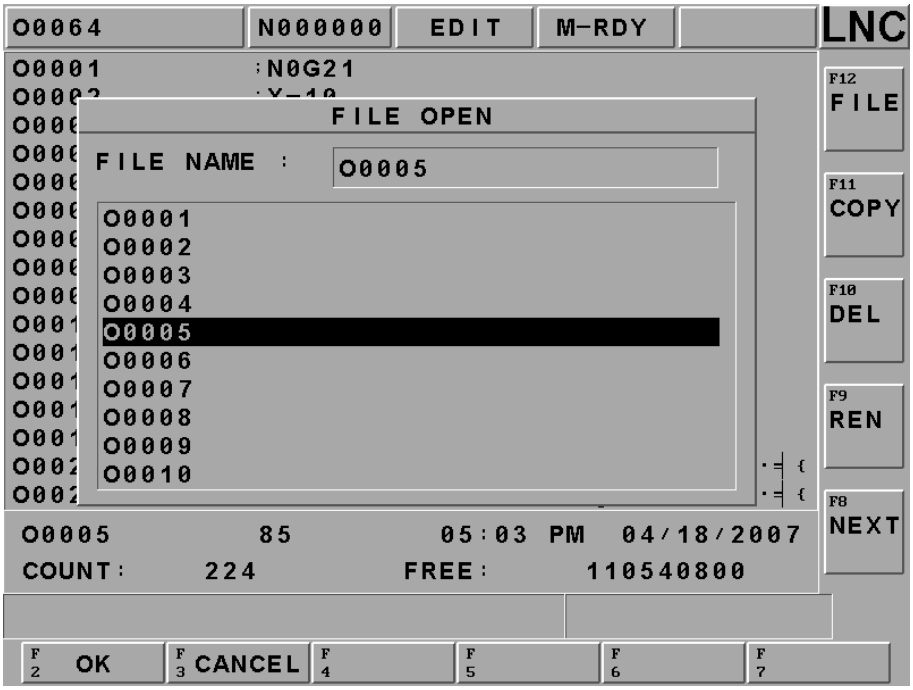
  Able to use direction key to choose the wanted open file after entering into the file management screen. Also, users are able to open a file after pressing <INPUT>.



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.

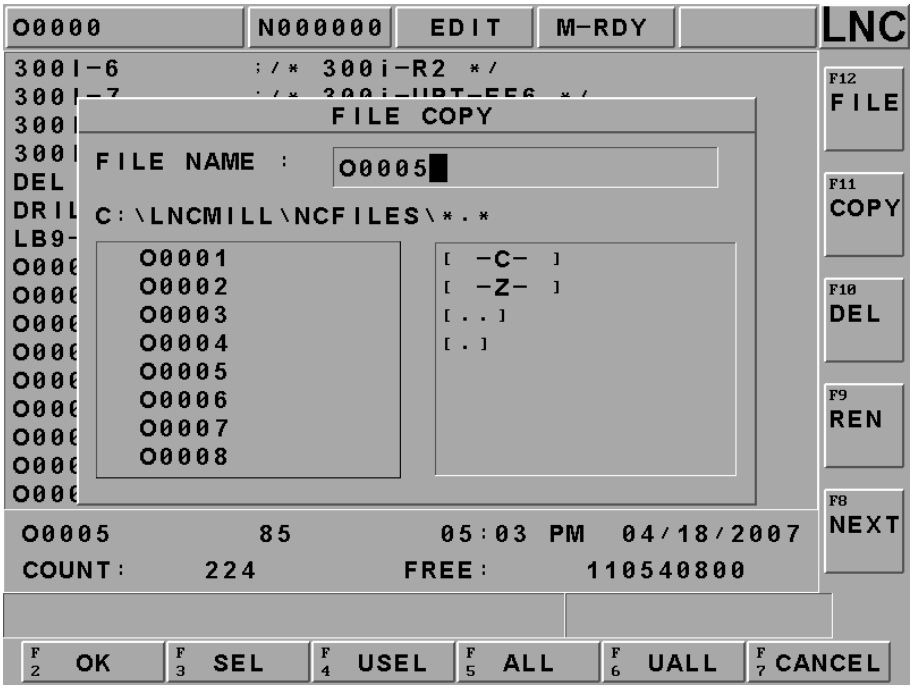




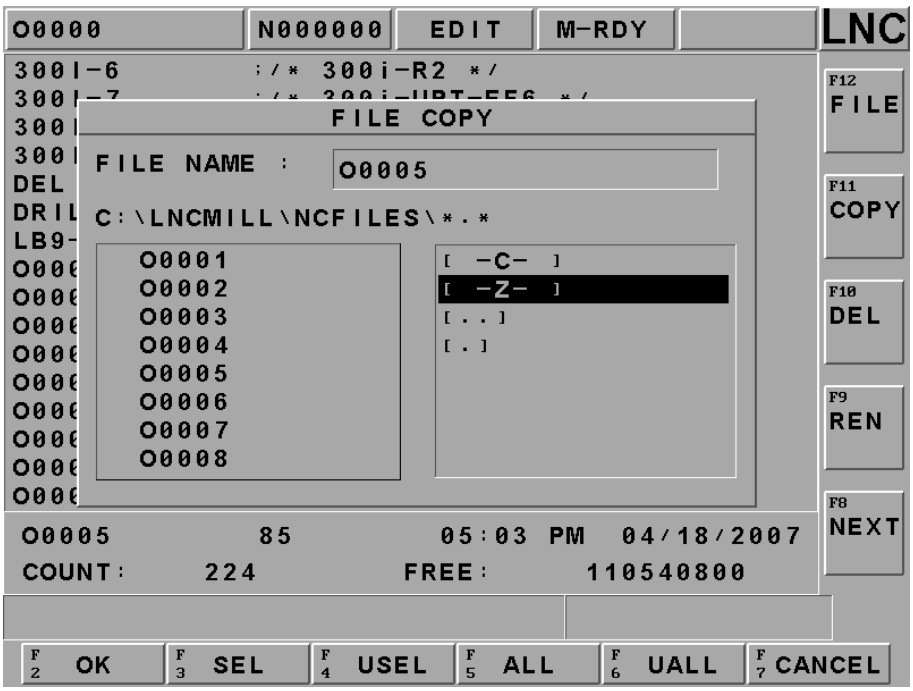
Note：The opened file is a background program under auto mode. Automatically setting the opened file to the present program under editing mode means NOT able to open the file under other modes.

- 【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:

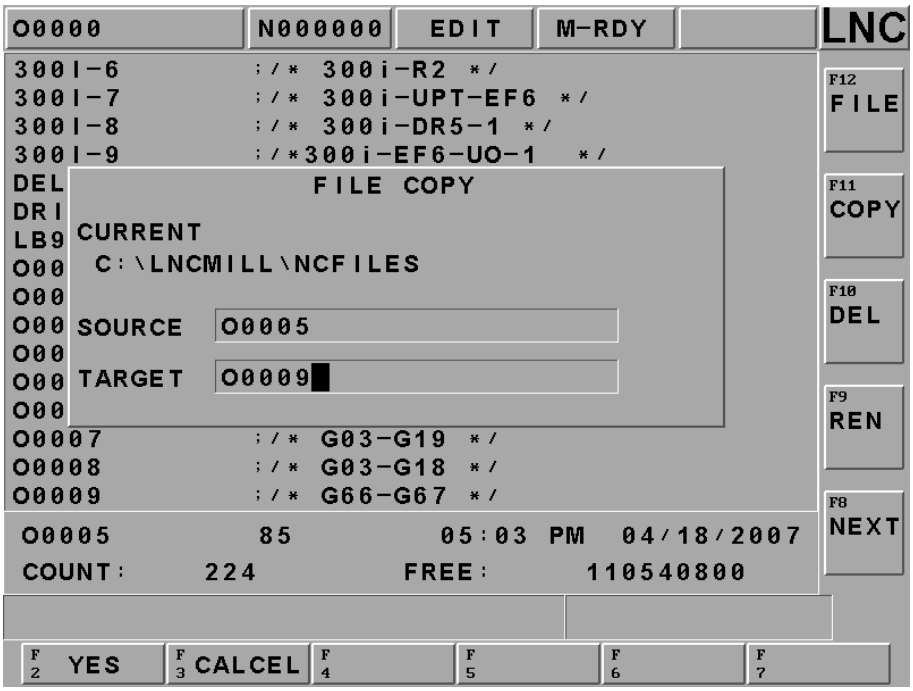


Figure 1.4-1 Screen of Key-In Destination File Name

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

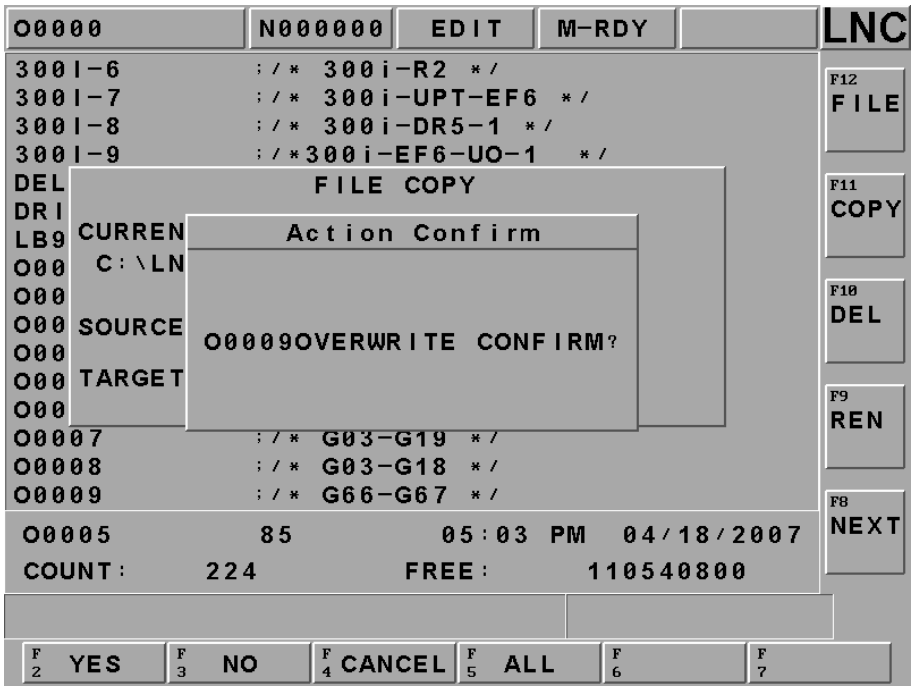


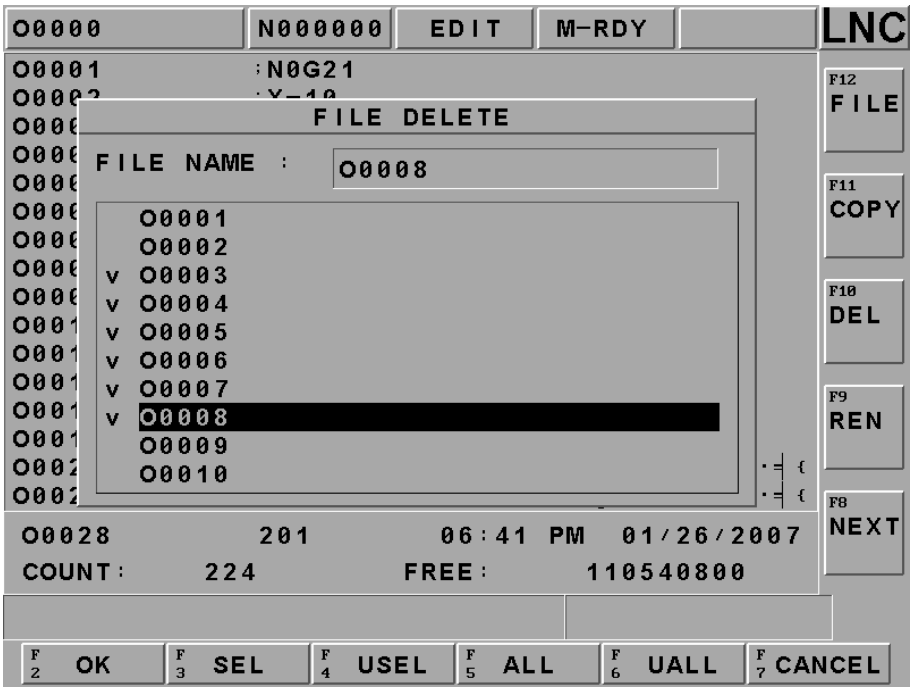
Figure 1.4-2 Overwriting

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.



- 【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.

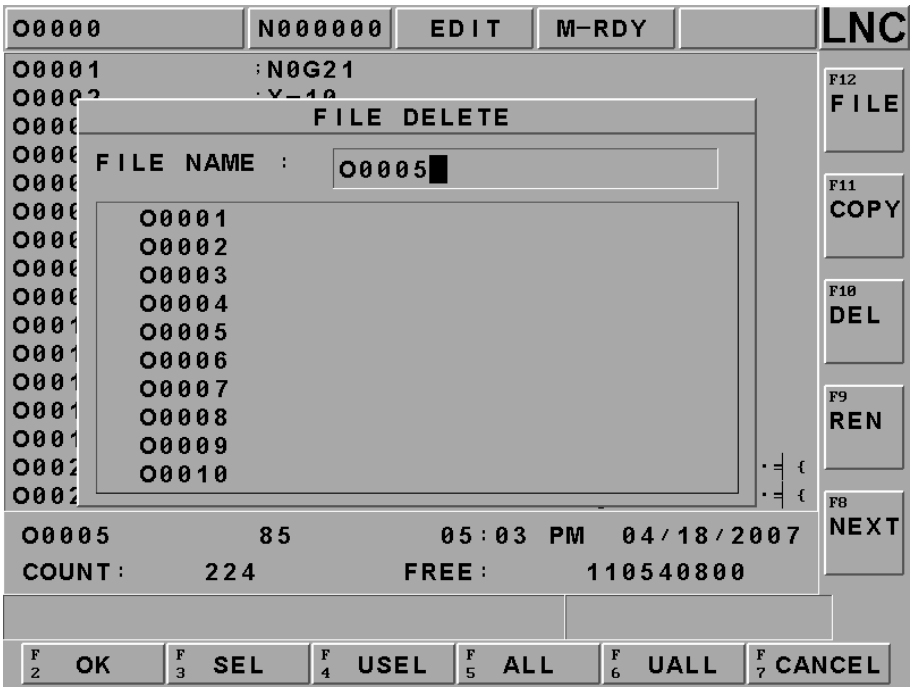
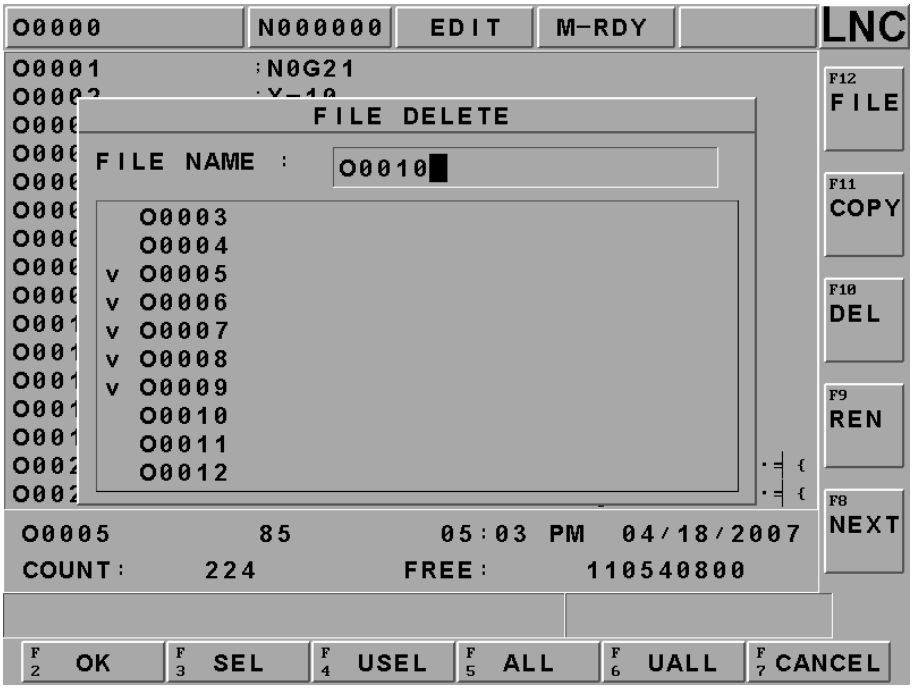


Figure 1.4-3 【DEL】screen

- **【COPY】MULTI-DELETE**

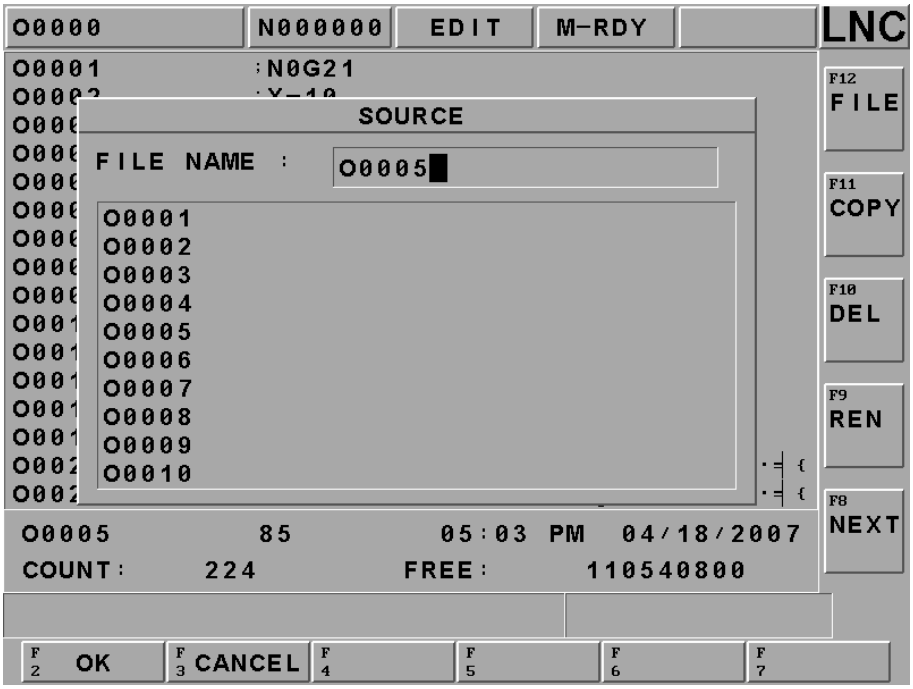
Except single file delete, our controller also offer multi-file delete function. In the dialogue box mode, you can press left or right button to move cursor to switch line to the left side. Press select to choose check box and press confirm to delete. After pressing confirm, the system will pop up hint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

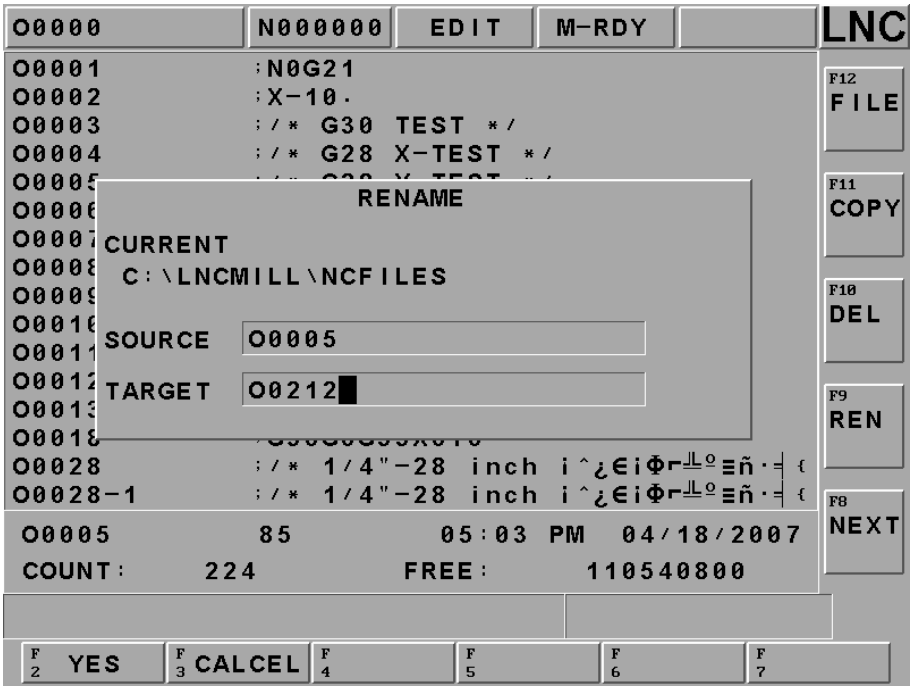


【REN】

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.

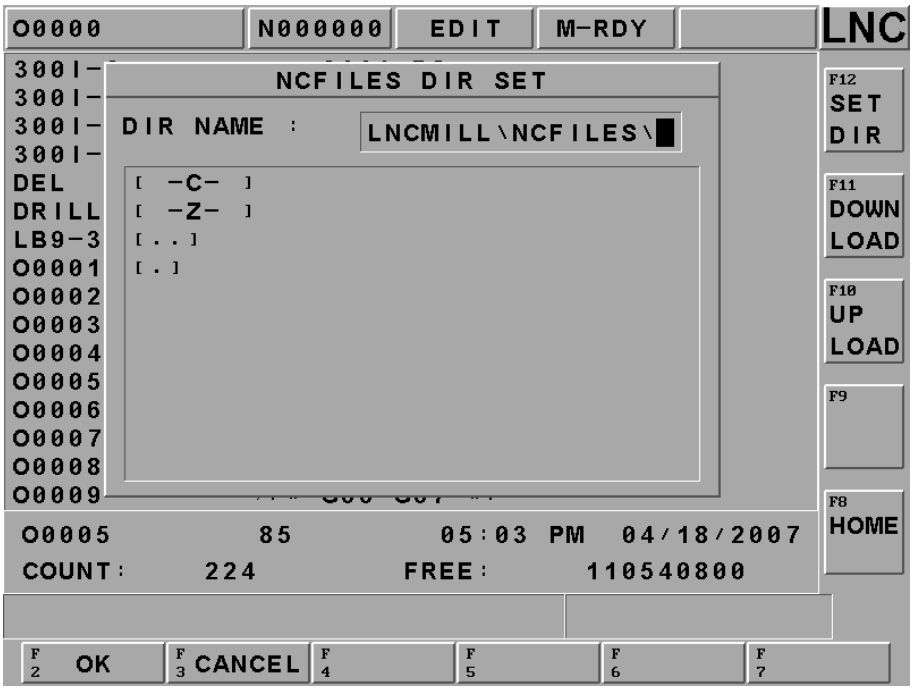
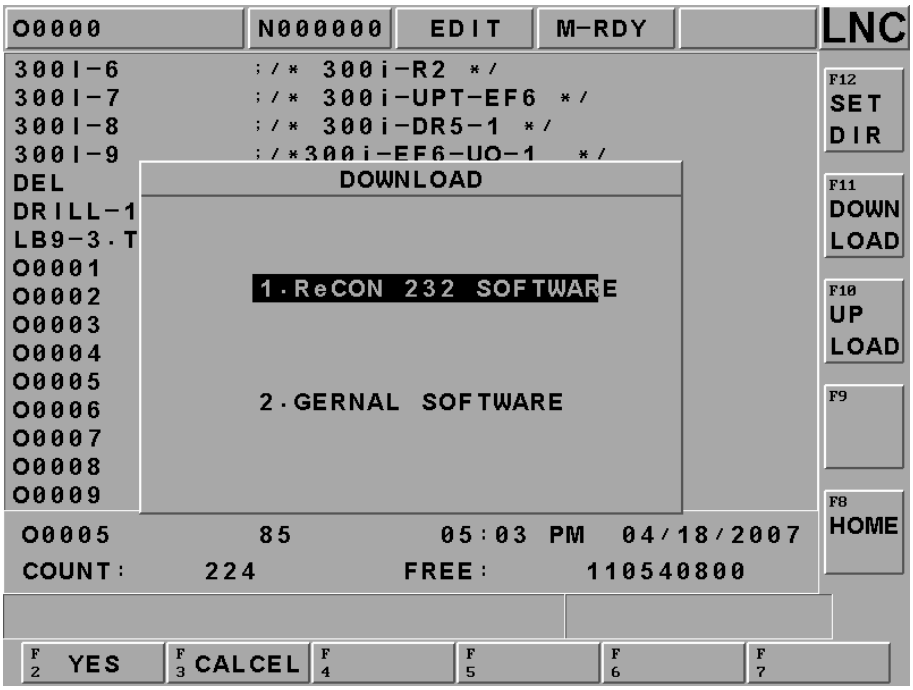


Figure 1.4-4 Screen when Setting Part Program Menu

- 【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.

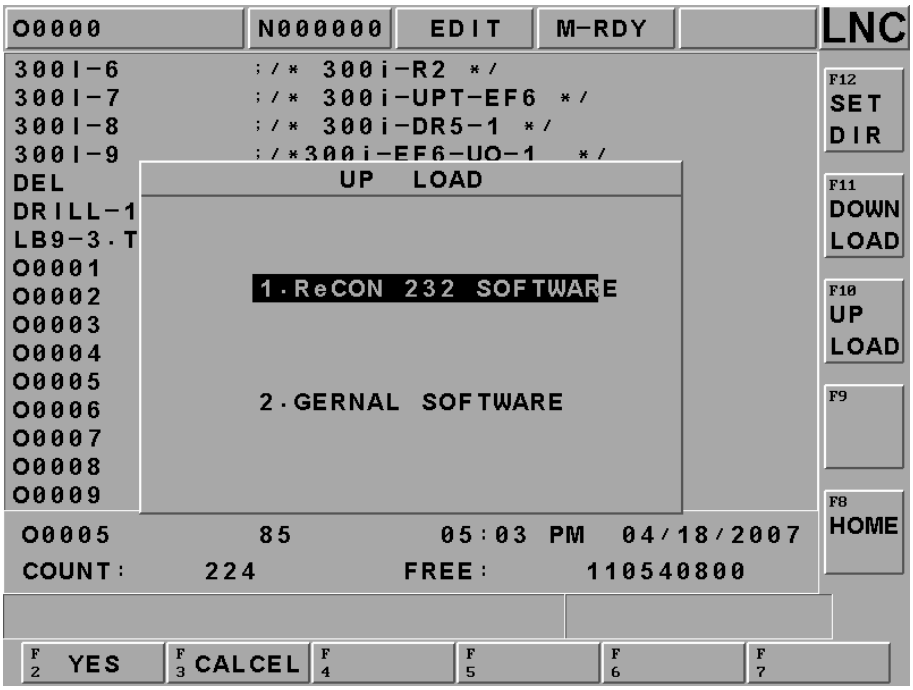
Note：As for detail information, please refer to RS232 communication software chapter.



- 【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.

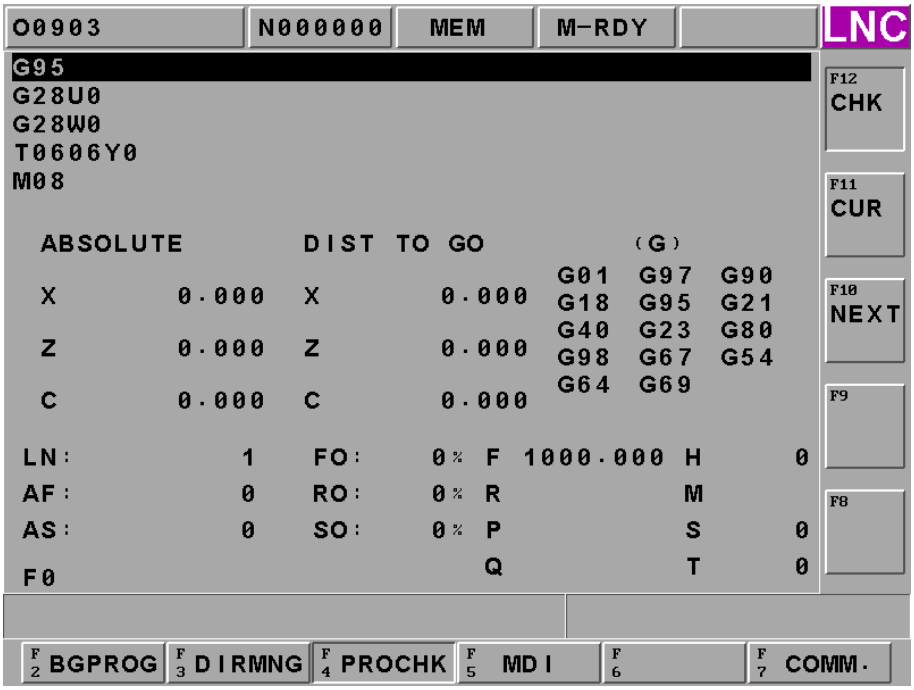


1.4.2 PROCHK (Program Checking)

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.

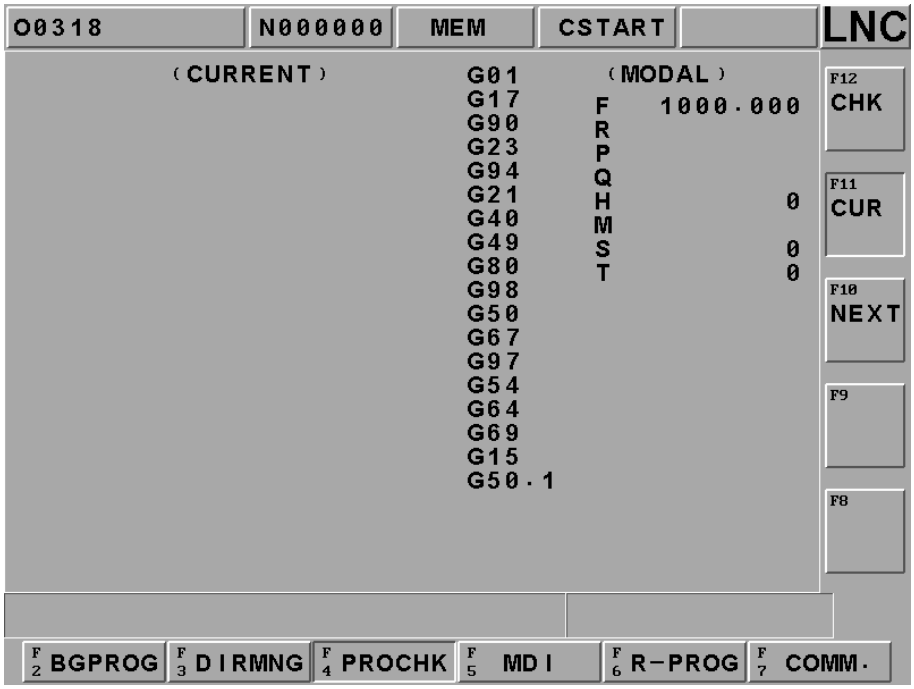
- 【**SINGLE CHECK**】

  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.



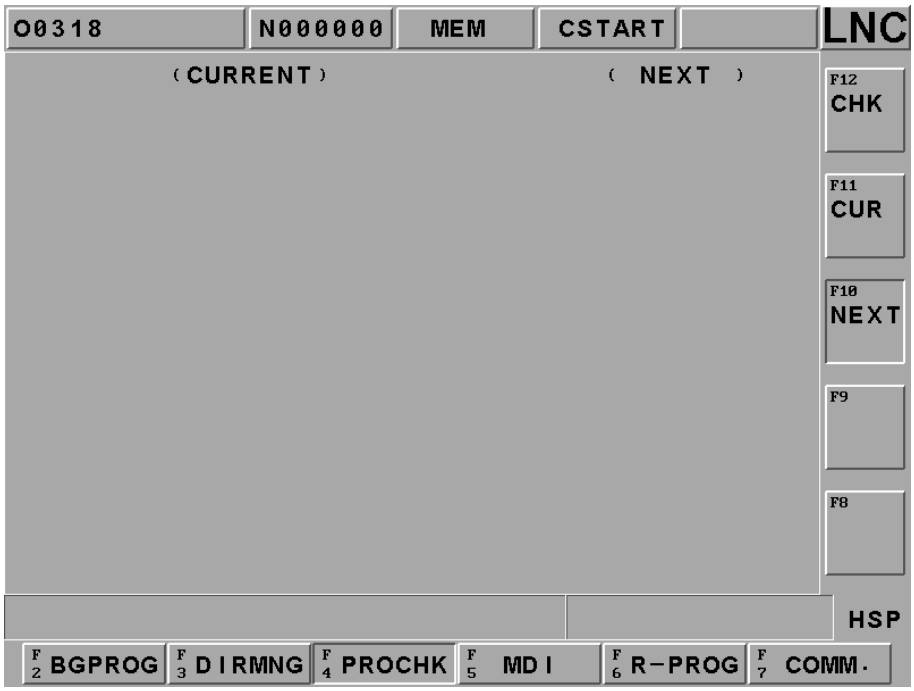
- **【Current Single Block】**

  To display the current single block.



- **【Next Single Block】**

  To show the next single block.

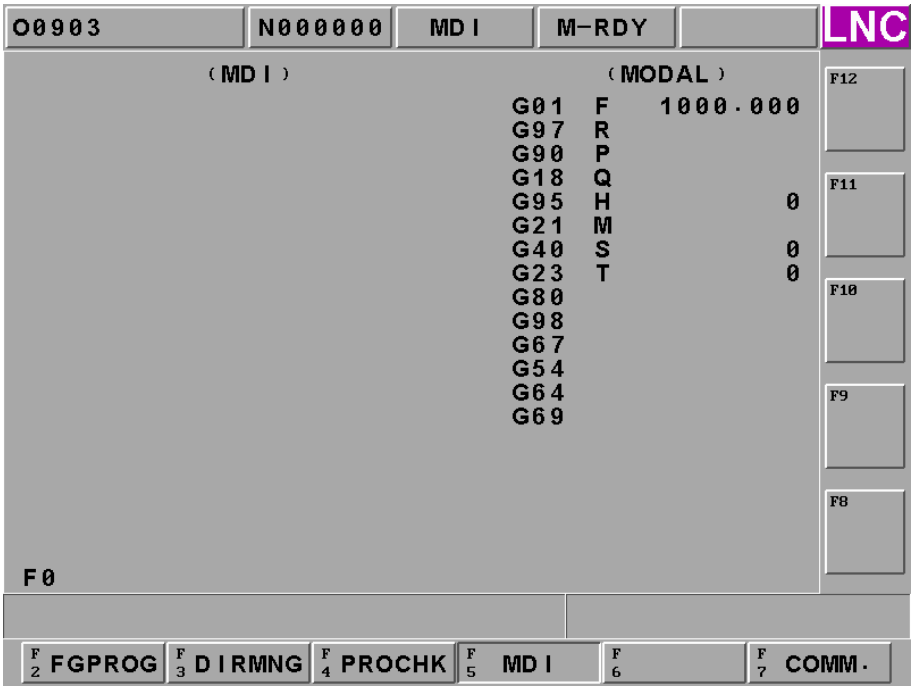


1.4.3 MDI

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.



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.

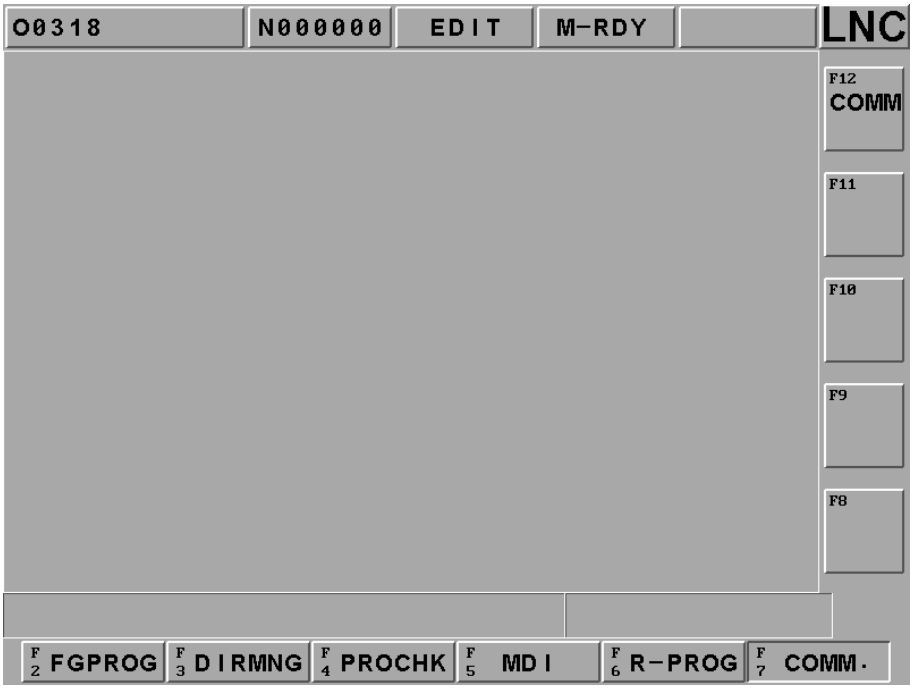


Figure 1.4-5【**COMM**】 screen

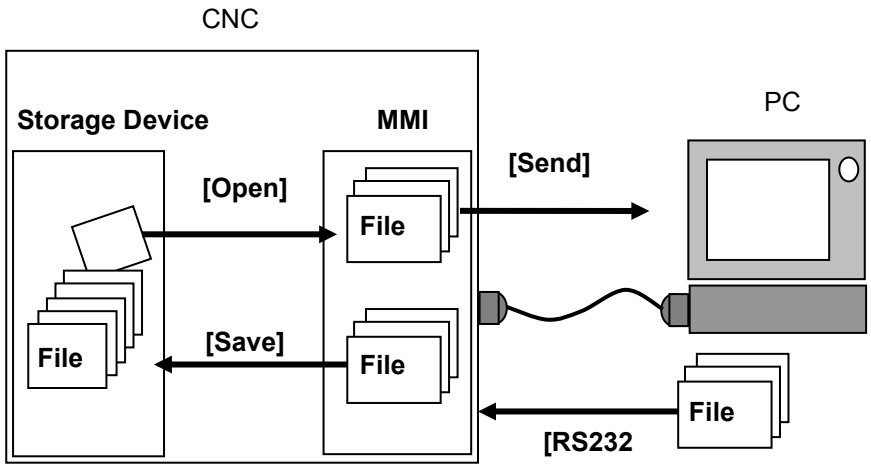


Figure 1.4-6 Files Transmitting/Receiving Figure between Controller and Other PCs

- 【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.

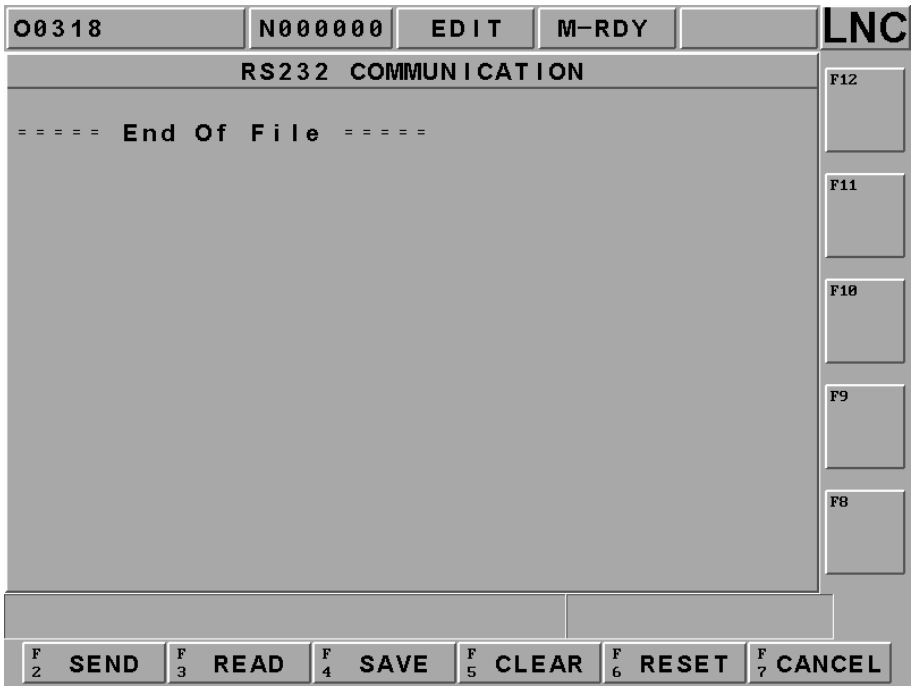


Figure 1.4-7 send main screen

【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 WEAR

Press <WEAR> to go in to WEAR page,【WEAR】、【GEOM】、【W.SHIFT】、【MACRO】、【WORK】。 Users can change this datas under MDI mode and machine preparation complete.

1.5.1 WEAR

Press【**WEAR**】to go in to WEAR page, users can input datas under MDI mode.

1. Wear offers total 30 settings, use <PAGE ↓ > 、<PAGE ↑ > to change pages.
2. Move cursor to the numbers you want to change, input setting value at data input line, press <INPUT> to write in controller.
3. When using a tool offset number in a program, absolute coordinate value will change by the offset value, the value is

   Absolute coordinate value = machine coordinate value – offset – G54~G59 offset –（tool offset +tool wear offset）

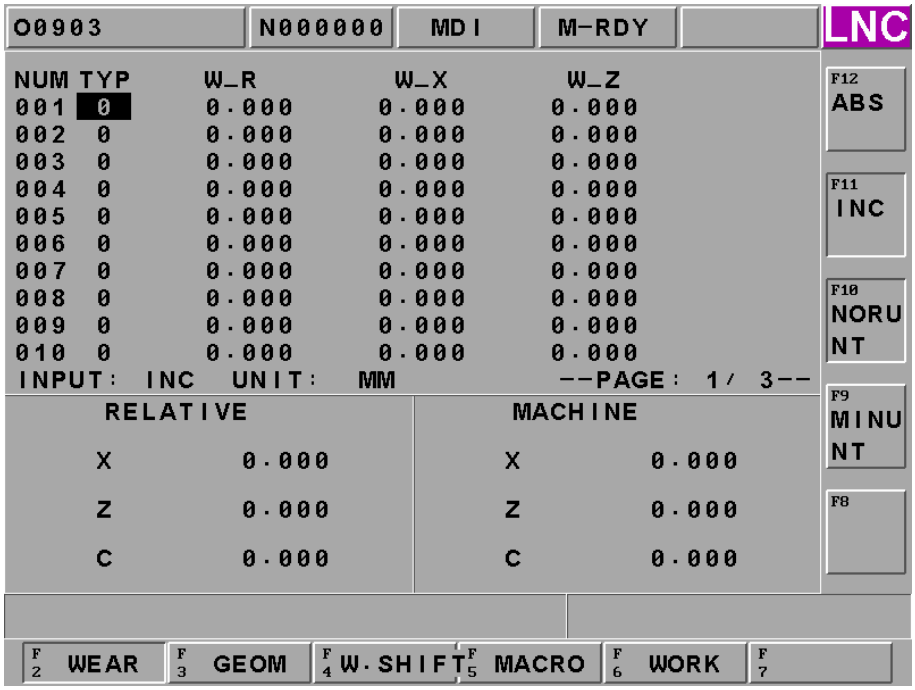


Figure 1.5-1 Tool wear offset page

【**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.

Figure and each column as below：

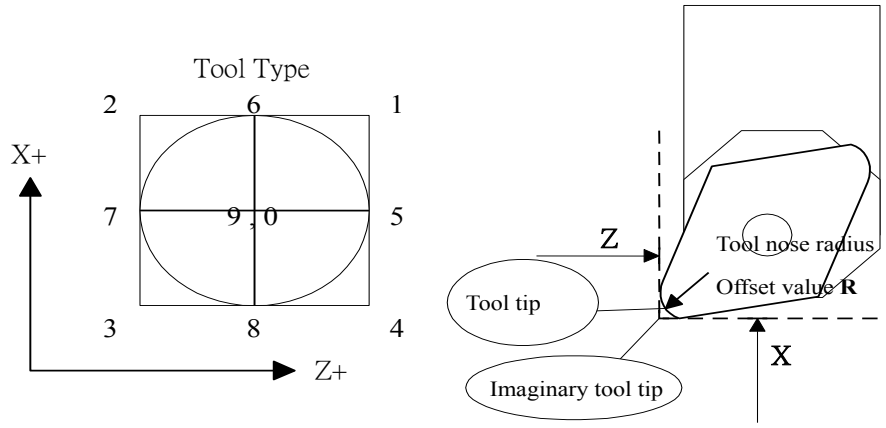


Figure 1.5-2

Type：Tool type

Tool Nose Wear：Diameter Wear Offset

X Axis Wear ：X Axis Wear Offset

Z Axis Wear ：Z Axis Wear Offset

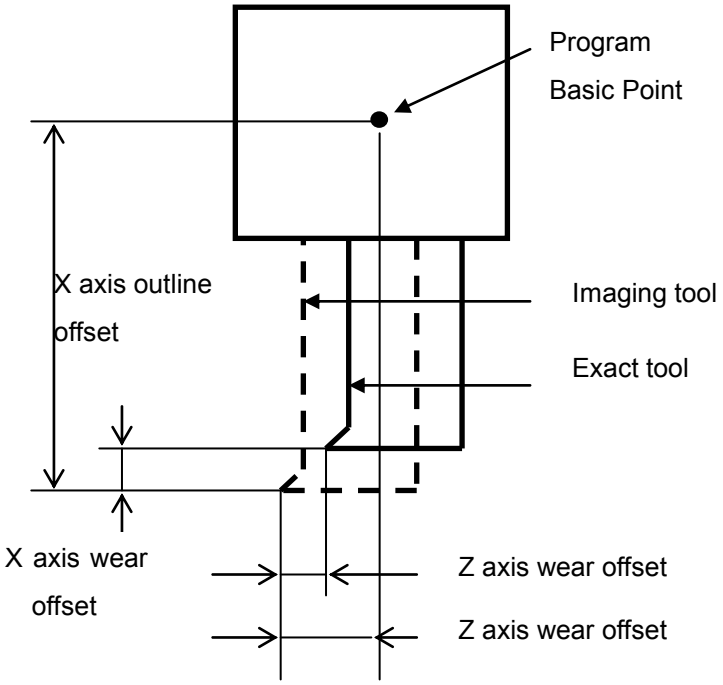


Figure 1.5-3

Tool Wear Offset Password

When users input wear offset into part program, system will pop out dialog window to let you input password, when the password is correct, the data will be valid. If users didn't get used to input password, at this time users can make Pr.0899 to be 0, then next time you don't have to input password again. Password protection situation s are as below：

NC State	Pr. 0206 Wear Offset when traveling	Pr. 0899 Use CE	Password Protection	Noted
Preparation Not Ready	—	—	—	Cursor Lock, can't input
Preparation Complete	—	—	N	Not in the program processing.
Cycle Start	0	—	N	No wear offset during traveling.
Feedhold	1	0	N	Password protection Off
Blocks Pause	1	1	Y	

Tool Offset Password Setting Chart

1.5.2 GEOM (Geometry OFFSET)

Pressing 【**GEOM**】 button to enter into the below screen so users are able to modify under MDI mode.

1. GEOM Outline offset offers 30 types of setting, use <PAGE ↓ > 、<PAGE ↑ > to change setting screen. Move the cursor to the one that you want to change, input data into this line, press <INPUT> to write into controller.
2. Everytime when using a tool offset number in a part program, the coordinate will change according to absolute coordinate. The data will be:
   Offset=Machine Coordinate–WORK SHIFT–G54 ~ G59 Offset–Input data

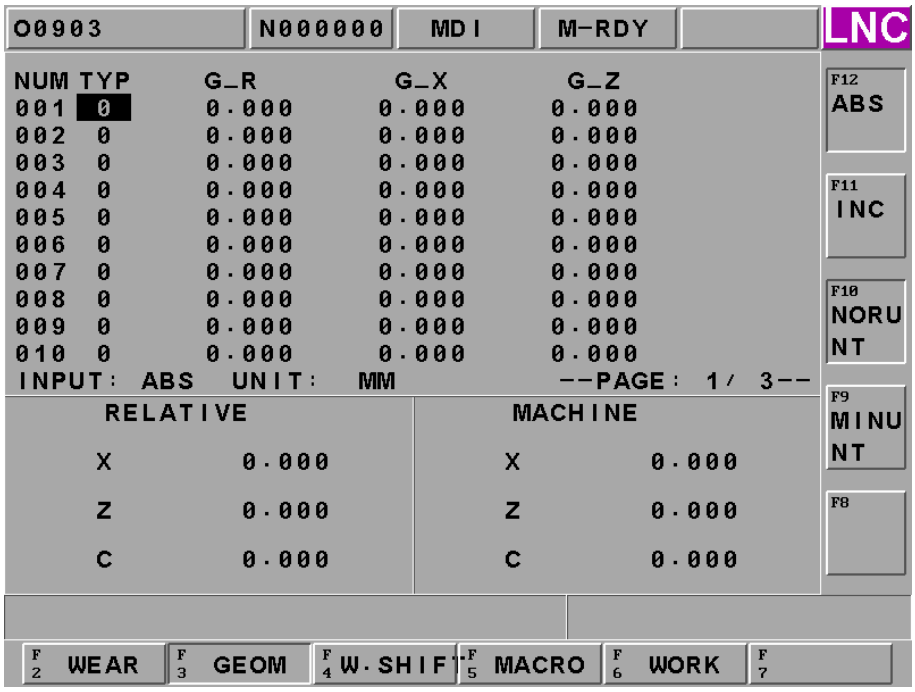


Figure 1.5-4 Tool Outline Offset Screen

【**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.

Each column as below：

Type：Tool type

Tool Nose Diameter：Diameter Outline Offset

X Axis Tool Length：X Axis Outline Offset

Z Axis Tool Length：Z Axis Outline Offset

Users can input change data under complete prepared MDI mode. But for X axis and Z axis tool length, users can use ”MXxx” and ”MZzz” to do offset setting.

1.5.3 W. SHIFT (Work Coordinate Shift)

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

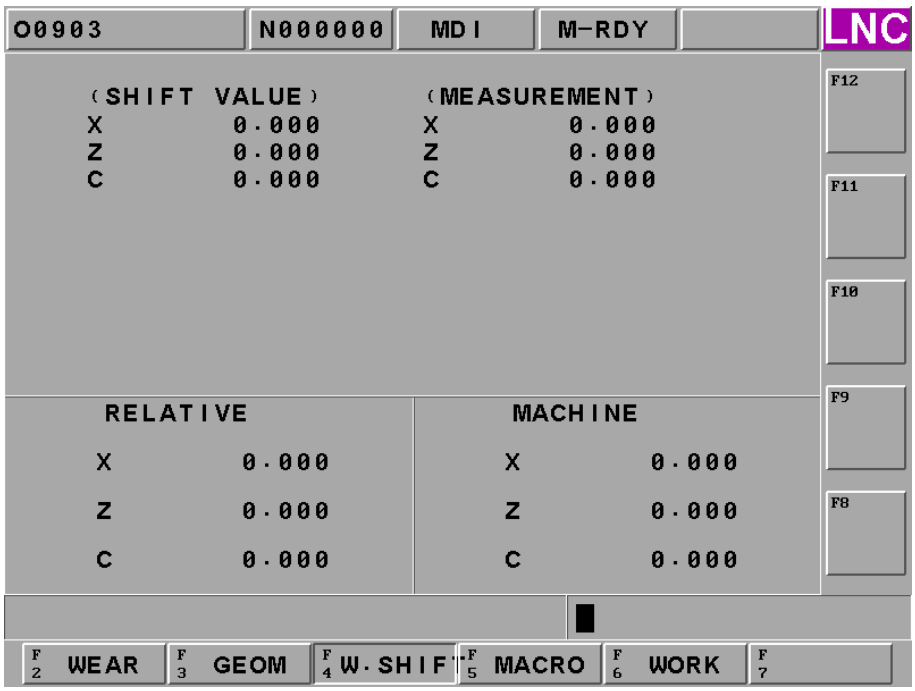


Figure 1.5-5 Setting WORK SHIFT Screen

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

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

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

Shift value = machine coordinate value – entry value - G54~G59 shift – (wear offset value + geometry offset value)

1.5.4 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. Or input @100 at input line and press enter, it will start to search @100 automatically.
2. Move the highlighter to the wanted modify position and then to press <INPUT> key after entering the wanted value.

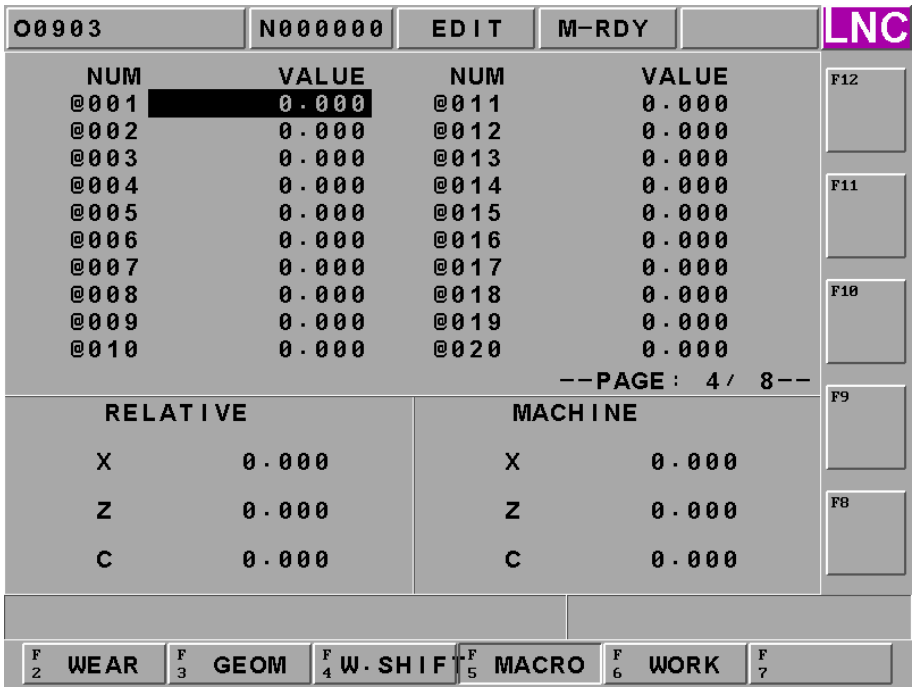


Figure 1.5-6 Local Variables of MACRO Variables

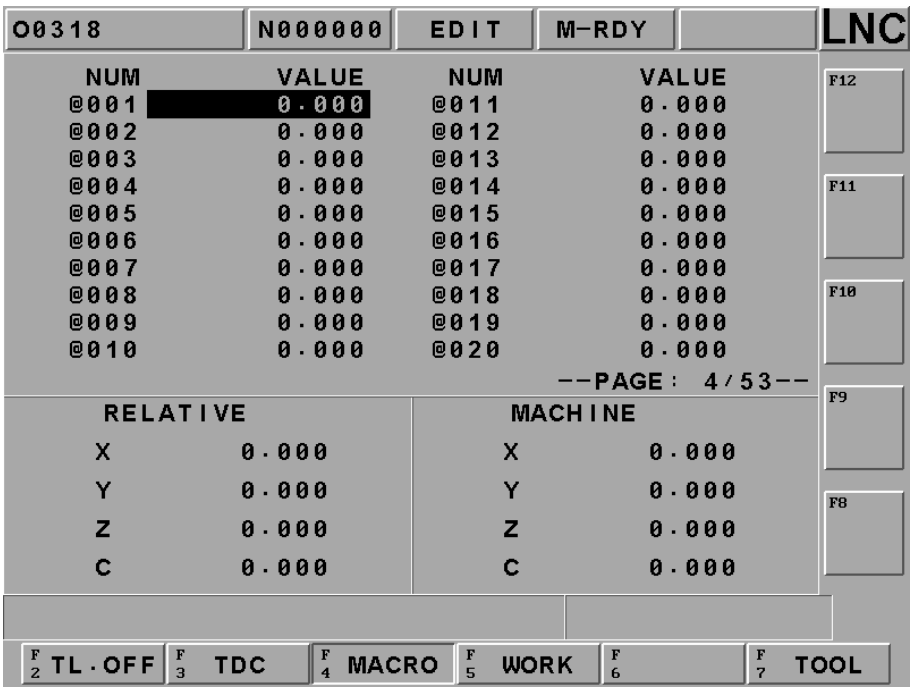
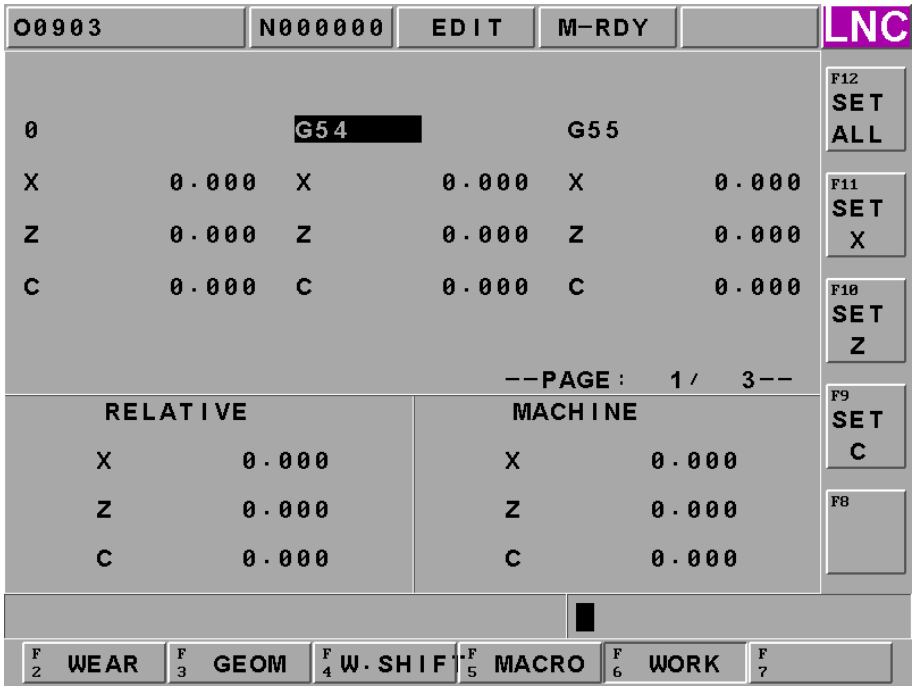


Figure 1.5-7 Global Variables of MACRO Variables

1.5.5 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. Users are able to write 2 axes at the same time, such as Xxxx. Zxxx. But, remember to put decimal point in order to prevent confusion（i.e., X100. Z200.）



Controller offers【Teach in】functions for users to input working coordinates. 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.

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.
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.

1.6 LATHE EASY-CAM

1.6.1 Function Description

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

1.6.2 Operation Procedure of How to Produce Working Path Program

1. **Open a New Project File**
   Pressing 【DIRMNG】button on the main function screen to change the screen to the file management screen. Pressing【New】 button on the project management screen and key-in the project name into the entry column and then pressing【OK】button. The screen will change to the main function screen after opening file successfully.
2. **New Procedure**
   Pressing【PROC ADD】button on the main function screen.
3. **Working Method Selection**
   Pressing 【ENTER】button when yellow cursor occurs at the working method column and a working method selection menu will popup. Using 【UP】 or 【DOWN】 key to select the needed working method and then pressing 【ENTER】 or 【OK】 button. Then, the selected working method will appear on the working method column.
4. **Tool Number Selection and Tool Information Key-In**
   When the yellow cursor occurs at the tool number column, pressing【ENTER】button to change the screen to the tool setting screen. Key-in the needed information according to different working method and then pressing【ENTER】button. After that, pressing 【RTN】button to change the screen to the main function screen. The chosen tool number will appear on the tool number column.
5. **Key-in Head Parameter Info**
   When the yellow cursor occurs at the head parameter column, pressing【ENTER】key to switch the screen to the head parameter screen. After key-in the needed head parameter data, pressing 【ENTER】key to switch the screen back to the main function screen. Then, a blue “DEFINE” will occur on the head parameter column.

6. **Key-in Cutting Parameter Information**

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

7. **Key-in Geometry Define Information**

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

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

1.6.3 Operation Screen and Function Key Instruction

◎ **Main Function, the screen is as below**

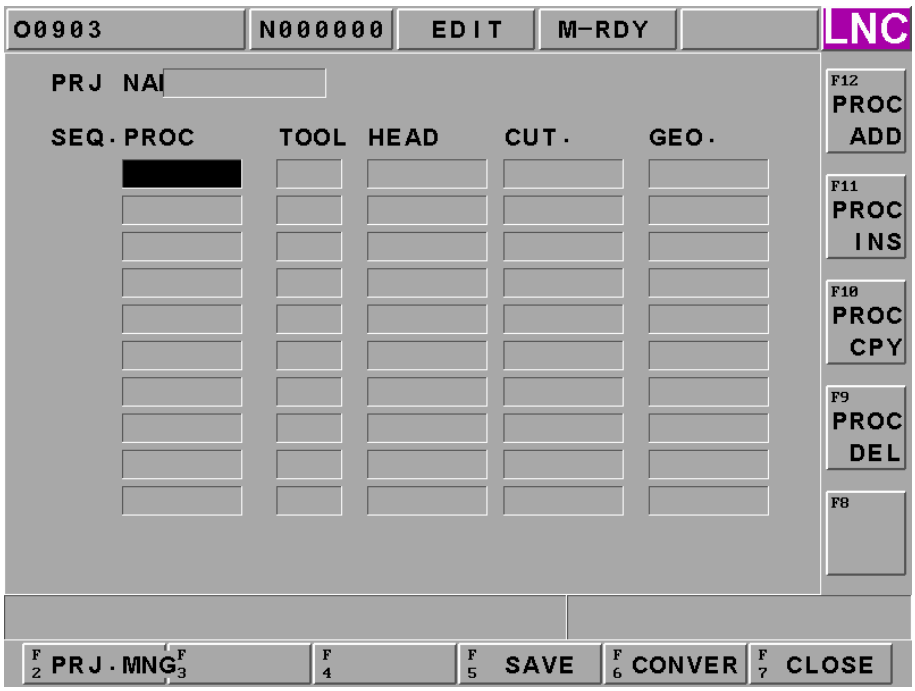


Figure 1.6-1 Main Function Page

【Column Description】

※ Project Name：display the currently opened project name.

※ Procedure：display procedure order number. The system will auto edit this column.

※ Working Method：display the selected working name. A “clear working name” dialog box will be popup when users press【ENTER】key at this column.

※ Tool #：display the selected tool number. The screen will switch to the tool setting screen when users press 【ENTER】 at this column. 4 types of tool number, which are A type for drilling tool, B type for profile lathe tool, C type for grooving tool, D type for screwing tool. Number means the tool number from the tool list on the tool setting page.

※ Head Parameter：red indicates “un-define” and blue indicates “define”. “Un-define” means users select to enter into the head parameter screen that never been in before. On the other hand, “define” means users have been to the head parameter screen. Screen will switch to the head parameter screen when users press【ENTER】key at this column.

※ Cutting Parameter：red indicates “un-define” and blue indicates “define”. “Un-define” means the cutting parameter screen that users haven’t been in before. On the other hand, “define” means users have been to the head parameter screen. Screen will switch to the cutting parameter screen when users press【ENTER】key at this column.

※ Geometry Define：red indicates “un-define” and blue indicates “define”. “Un-define” means the geometry define screen that users haven’t been in before. On the other hand, “define” means users have been to the geometry define screen. Screen will switch to the geometry screen when users press 【ENTER】 key at this column. Screen will switch to different geometry setting screens according to different working methods.

【Function Keys at Screen Bottom】

※ **PRJ. MNG**：switching screen to the project management windows.

※ **SAVE**：saving the currently opened project.

※ **CONVER**：switching the currently opened project data to the work path program.

※ **CLOSE**：closing the currently opened project.

【Function Key at Right of Screen】

※ **PROC ADD**：adding a new program to the next row of the row where the yellow cursor occurs.

※ **PROC INS**：insert a new program to the previous row of the row where the yellow cursor occurs.

※ **PROC CPY**：copy a row of program where the yellow cursor occurs to the next row of the copied row.

※ **PROC DEL**：delete all procedures where yellow c sor occurs.

◎ **Screen of Project Management is as below**

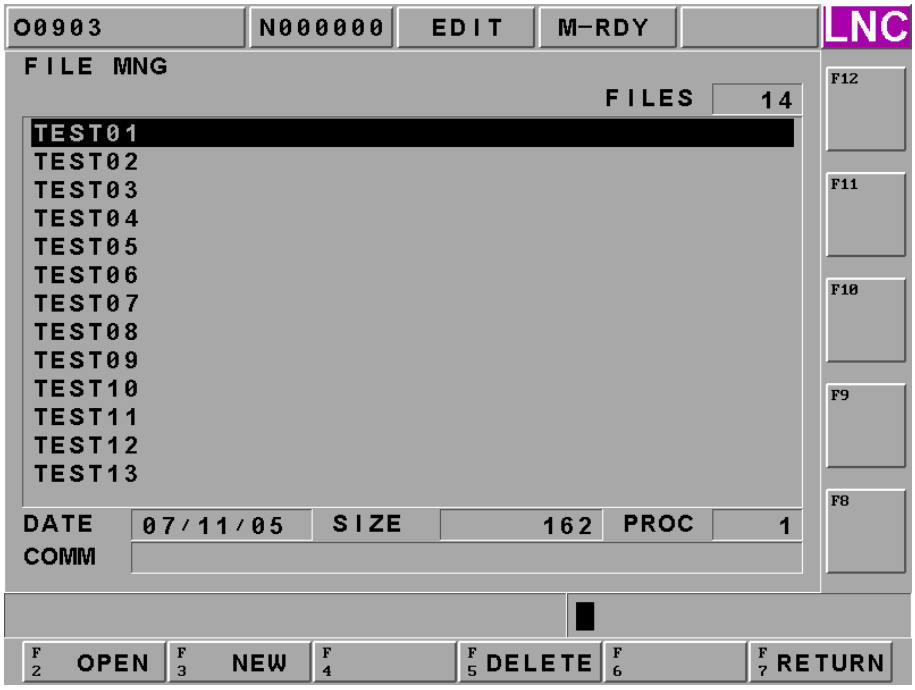


Figure 1.6-2 Project Management Page

【Column Description】

※ Project #：display the current project number.

※ Display Dialog Box: display all project list

※ Project Date：display the project date where the yellow cursor occurs.

※ File Size：display the file size where the yellow cursor occurs.

※ Procedure #：display the total procedure # of a project where the yellow cursor occurs.

※ File Description：display the project description where the yellow cursor occurs. Pressing 【LEFT】or 【RIGHT】key to move the yellow cursor to the column. Then, key-in data into that column and press 【ENTER】key to write “NOTE” for the project.

【Function Keys at Screen Bottom】

※ OPEN：downloading a project where the yellow cursor occurs and switching to the main function page.

※ NEW：building a new project name and the screen will switch to the main function screen after confirm the key-in project name.

※ DELETE：deleting a project where yellow cursor occurs.

※ RETURN：switching screen to the main function screen.

【Function Keys at Right of Screen】

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

※ Cancel：pressing 【OPEN】 or 【DEL】 key at the screen bottom first. Then, pressing【DEL】 key after pressing 【OPEN】 key at screen bottom to cancel opening new file. Pressing 【CANCEL】 key after pressing 【DEL】 key at screen bottom to cancel deleting file.

◎ **Screen of Working Method is as below:**

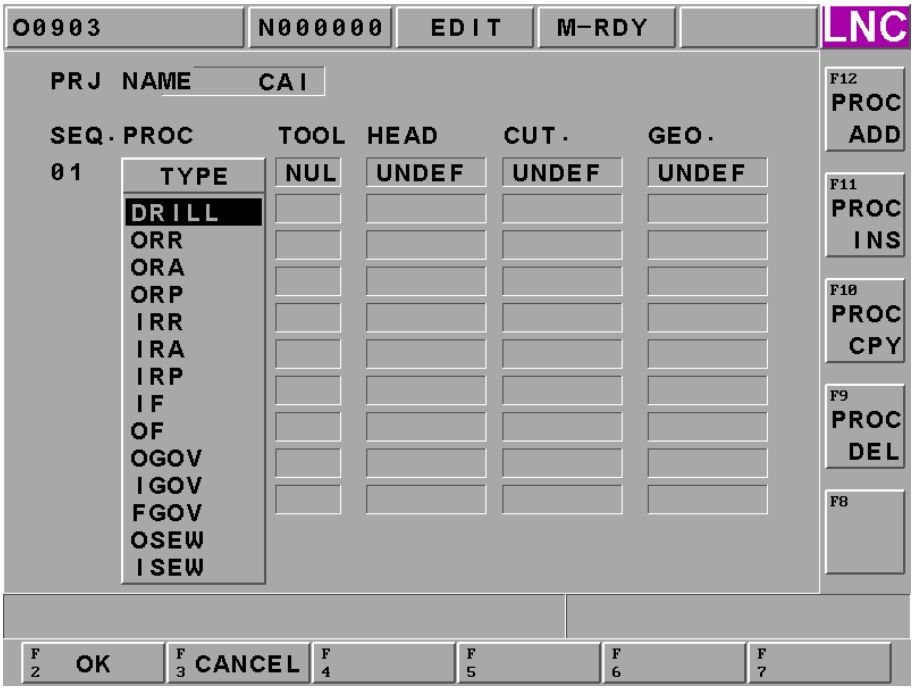


Figure 1.6-3 Working Method Page

【Column Description】

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

⊙ **Drilling**：drilling one internal hole on the material

⊙ **ORR**：outer rough diameter. Cutting most of material part that is made by straight line and arch. Tool moving direction is according to the perpendicular Z-axis direction

⊙ **ORA**：outer stock removal in turning. Clearing the most material part that is made by line and arch. Tool moving direction is according to the perpendicular X-axis direction.

⊙ **ORP**：outer pattern repeating. Clearing the most material of the pattern that is made by straight line and arch. Tool moving direction is according to the curve outline.

⊙ **IRR**：internal rough diameter. Cutting most of material part that is made by straight line and arch. Tool moving direction is according to the perpendicular Z-axis direction.

⊙ **IRA**：internal stock removal in turning. Clearing the most material part that is made by line and arch. Tool moving direction is according to the perpendicular X-axis direction.

⊙ **IRP**：internal pattern repeating. Clearing the most material of the pattern that is made by straight line and arch. Tool moving direction is according to the curve outline.

⊙ **IF**：internal finishing cycle. Clearing the rest of the material of the outline that is combined by straight line and arch.

⊙ **OF**：outer finishing cycle. Clearing the rest of the material of the outline part that is combined by straight line and arch.

⊙ **OGOV**：grooving on outer surfaces,. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.

⊙ **IGOV**：grooving on internal surface. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.

⊙ **FGOV**：grooving on end faces. Using grooving tool to cut a concave shape on the working piece. Outer surface, inter surface or end-faces is according to the grooving position.

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

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

【Function Key at Screen Bottom】

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

※ Cancel：cancel selecting working method action and switch screen to the main function screen.

【Function Key at Right of Screen】

None.

◎ **Screen of Tool Setting is as below screen**

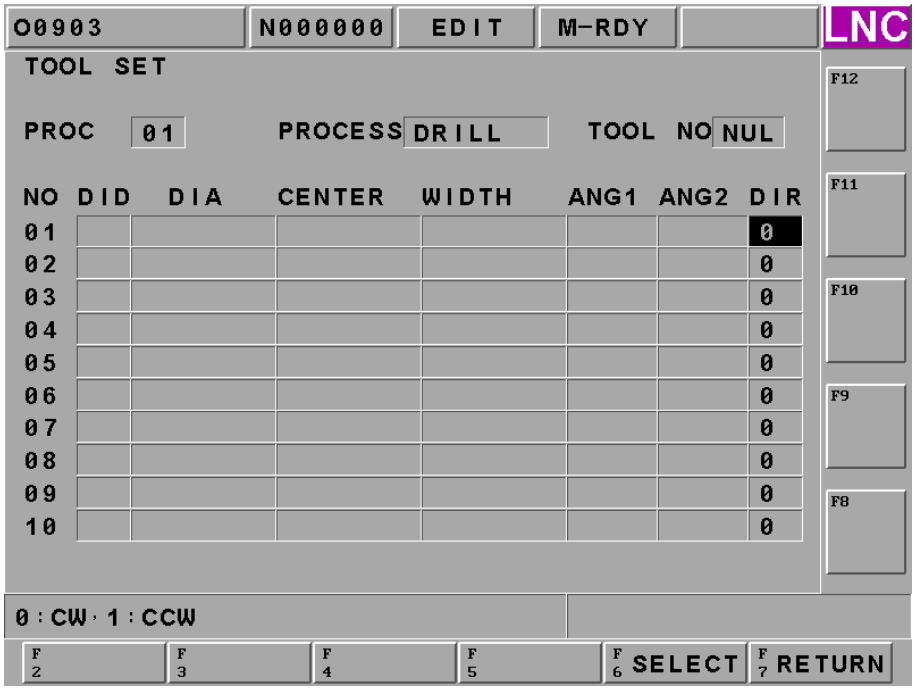


Figure 1.6-4 Tool Setting Page

【Column Description】

※ Procedure：display the present editing procedure order number.

※ Working Method：display the working method name of the present editing procedure.

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

※ Tool Clear Dialog Box at Screen Bottom: able to display and edit 10 sets of data. Users need to key-in different columns according to different working methods. The description of each column is as below:

⊙ Tool NO.：tool parameter number, The same type, able to do CAM calculation according to difference parameter. Selecting the NO. 1 tool parameter when one tool is at the first procedure. But, for the 2nd procedure, users can select the 2nd tool parameter for CAM calculation.

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

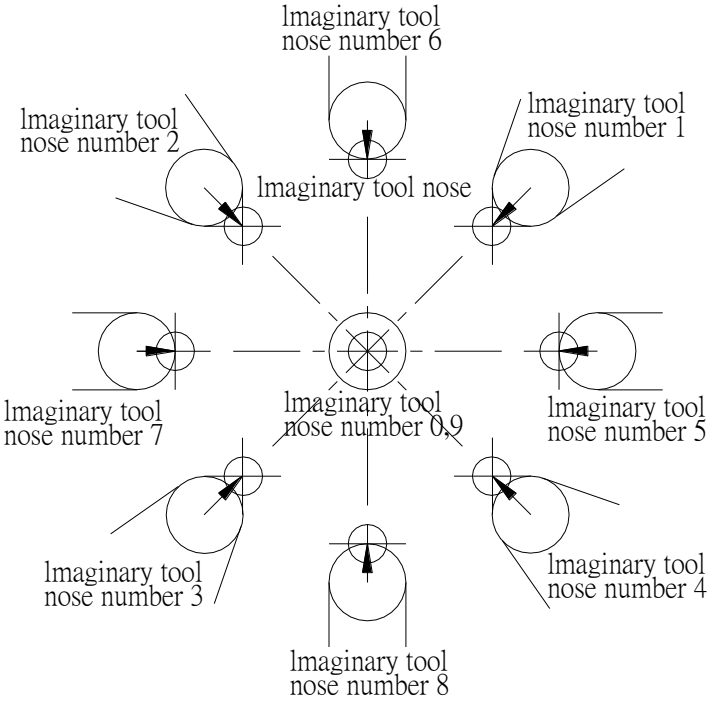


Figure 1.6-5　Corresponding Position Figure of Imagery Tool Tip Point and Tool Noise Center Point

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

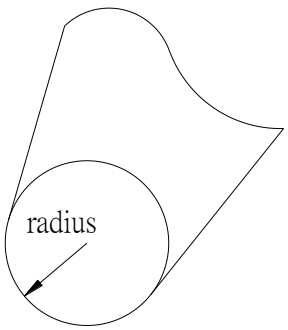


Figure 1.6-6　Scale-In Figure of Tool Tip

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

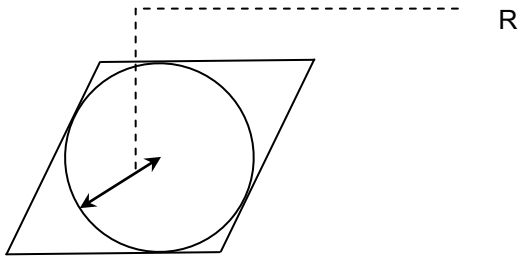


Figure 1.6-7　Internal Tool Radius

⊙ Tool Width：Tool width

⊙ Feed Angle：Feel angle. To set the needed setting column, external pattern, axis, wheel and fine-lathe to 90 and to set the internal pattern, axis, wheel and fine-lathe to 270.

⊙ Cutting Angle：the max. effective cutting angle. To set the needed setting column, external pattern, axis, wheel and fine-lathe within the range of 0～90 when doing profile cutting and to set the internal pattern, axis, wheel and fine-lathe within the range of 270～360.

⊙ Rotary Direction：Spindle rotary direction, clockwise (CW) or counter clockwise (CCW)

【Function Keys at Bottom at the Screen】

※ Selection: select the needed tool parameter data, tool number at the upper screen indicates the selected tool number.

※ Return: switch to the main function screen.

◎ **Screen of Head Parameter is as below**：

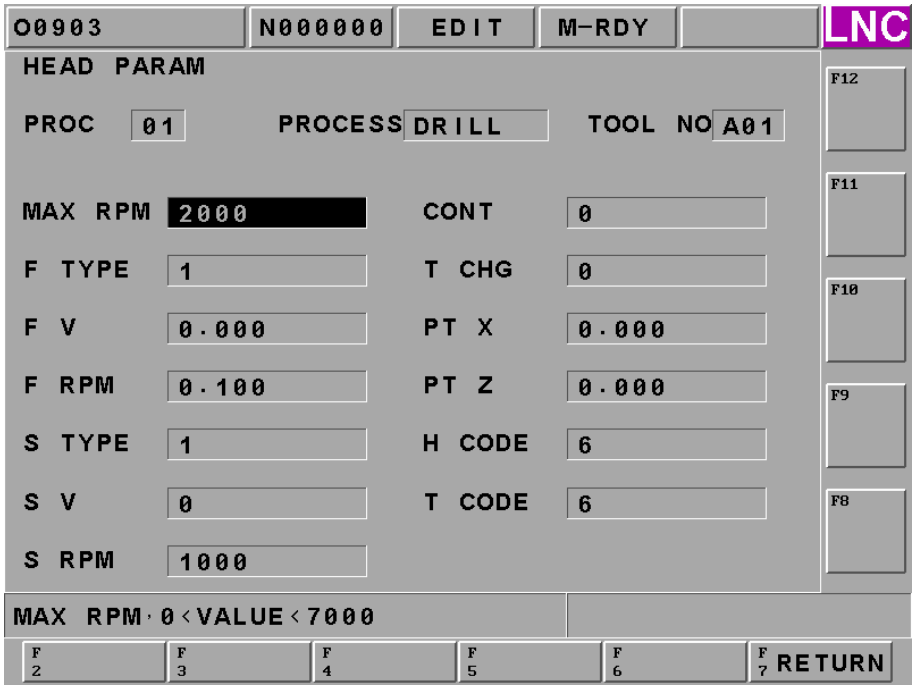


Figure 1.6-8 Header Parameter Page

【Column Description】

※ Procedure：display the currently editing procedure number.

※ Working Method：display the working method name of the currently editing procedure.

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

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

⊙ **Max. RPM**：spindle max. rotational speed.

⊙ **F TYPE**：0 means using feed/min, 1 means using feed/rotate.

⊙ **F V**：when the feed unit is selected as 0, users must key-in any value that is larger than 0 into this column.

⊙ **F RPM**：when the feed unit is selected as 1, users must key-in any value that is larger than 0 into this column.

⊙ **S TYPE**：0 means using fixed cutting speed, 1 means using fixed rotating speed.

⊙ **S V:** when fixed cutting/rotate is selected as 0, users must key-in any value large than 0 into this column.

⊙ **S RPM**：when fixed cutting/rotate is selected as 1, users must key-in any value larger than 0 into this column.

⊙ **CONT**：whether or not to use cutting liquid. 0 means no need to use it, 1 means use it.

⊙ **T CHG**：0 means not need to use tool-exchanging point, 1 means use the user default tool-exchanging point.

⊙ **PT X**：user default of tool-exchanging X coordinate.

⊙ **PT Z**：user default of tool-exchanging Z coordinate.

⊙ **H Code:** tool OFFSET number

⊙ **T Code:** actual tool number on turret.

【Function Keys at Screen Bottom】

※Return: switch to the main function screen.

【Function Keys at Right of the Screen】

None.

◎ **Cutting Parameter is as the below screen**

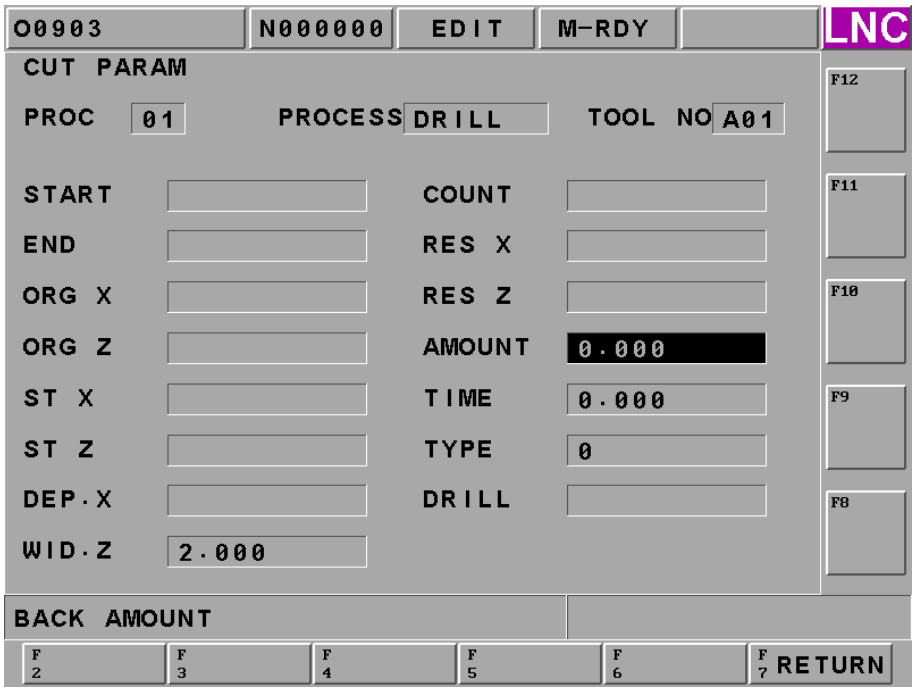


Figure 1.6-9 Cutting Parameter Page

【Column Description】

※ Procedure：display the currently editing procedure order number.

※ Working Method：display the working name of the currently editing procedure.

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

※ Cutting Parameter Column：the needed key-in column will be difference due to different working methods.

⊙ **START**：0.

⊙ **END**：all linear and arch numbers of profile.

⊙ **ORG X**：0.

⊙ **ORG Z**：0.

⊙ **ST X**：material at the upper right point is the diameter coordinate.

⊙ **ST Z**：material at the upper right point is the Z coordinate

⊙ **DEP. X**：lathe cutting depth

⊙ **WID. Z**：lathe cutting width.

⊙ **COUNT:** lathe-cutting repetitive times.

⊙ **RES X**：lathe-cutting preservation amount.

⊙ **RES Z**：0

⊙ **AMOUNT:** lathe-cutting escape amount.

⊙ **TIME:** drilling dwell time.

⊙ **TYPE:** the lathe-cutting mode selection.

⊙ **DRILL:** pre-drilling of internal lathe-cutting.

【Function keys at Screen Bottom】

※ Return: switch to the main function screen.

【Function Keys at Right of Screen】

None.

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

◇ **Screen of Geometry Drilling is as below:**

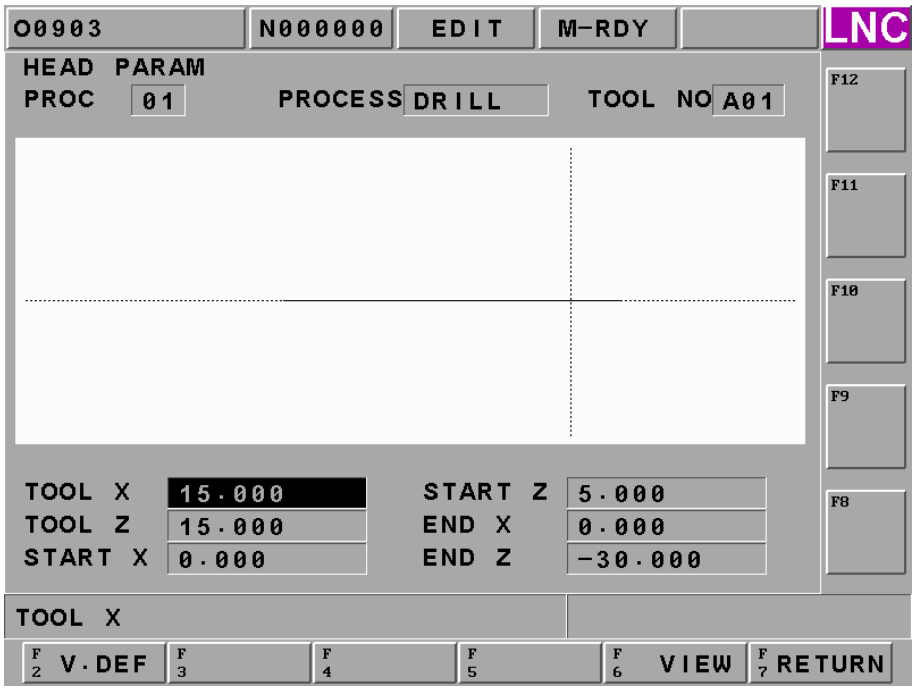


Figure 1.6-10 Geometry Drilling Page

【Column Description】

※ Procedure：display the currently editing procedure order number.

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

※ Geometry Drilling Column：

⊙ **TOOL X**：tool point diameter coordinate

⊙ **TOOL Z**：tool point Z coordinate

⊙ **START X**：starting drilling hole point diameter coordinate, 0.

⊙ **START Z**：start drilling hole point Z coordinate.

⊙ **END X**：final drilling hole point diameter coordinate, 0.

⊙ **END Z**：final drilling hole point Z coordinate

【Function Keys at Screen Bottom】

※ Windows Scaling：scale-in or return to original size of figure shape of display box at center of screen.

※ Preview：preview the Figure shape of the key-in data

※ Return：switching screen to the main function screen.

【Function Keys at Right of Screen】

※ SCALE：pressing the【SCALE】key at bottom of the screen for the “SCALE” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use 【UP】, 【DOWN】, 【LEFT】 and 【RIGHT】key to move the dialog box. Moving the dialog box to the wanted scale-in position and pressing the 【ENTER】key to complete the action.

※ Fittest Size: pressing 【WINDOW】at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.

※ Return：pressing the【SCALE】key at bottom of the screen to switch to the geometry drilling screen.

◇ **Screen of Geometry Profile is as below**

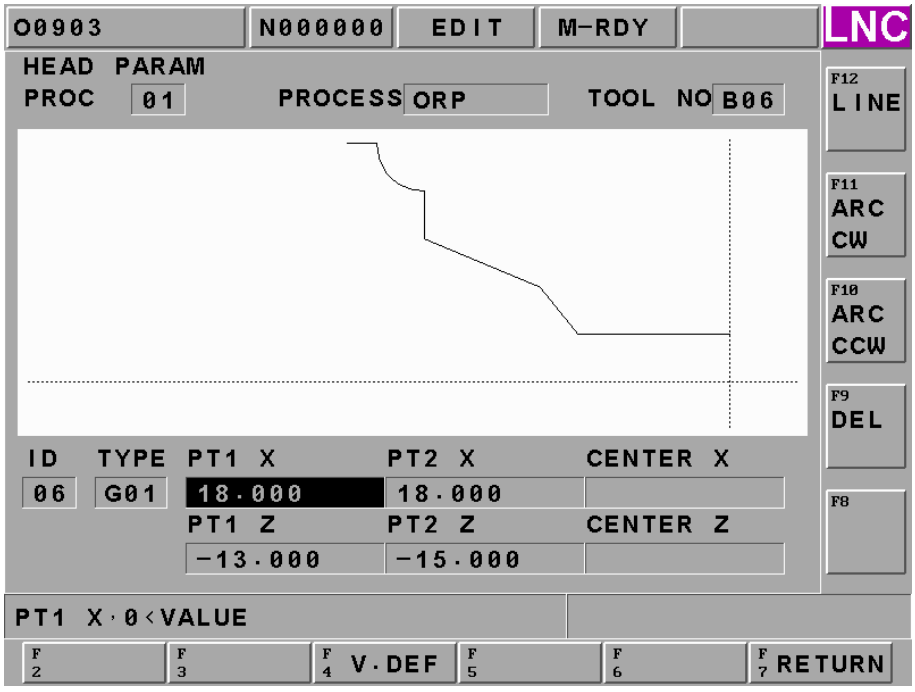


Figure 1.6-11 Geometry Profile Page

【Column Description】

※ Procedure：display the currently editing procedure order number

※ Working Method：display working name of the currently editing program

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

※ Order #：display profile linear order number, the system will auto-edit number.

※ Type：display profile linear type, G01 is linear, G02 is clockwise, and G03 is counter-clockwise.

※ Geometry Profile Column: key-in the final-point coordinate of linear; cycle needs to key-in multiple center-point coordinate.

⊙ **PT1 X：**starting point diameter coordinate.

⊙ **PT1 Z：**starting point Z coordinate.

⊙ **PT2 X：**final point diameter coordinate.

⊙ **PT2 Z：**final point Z coordinate.

⊙ **CENTER X：**circle center diameter coordinate.

⊙ **CENTER Z：**circle center Z coordinate.

【Function Keys at Screen Bottom】

※ Windows scale-in：scale-in or return to original size of the Figure shape of the dialog box at screen center.

※ Return: switch to the main function screen.

※ Confirm：pressing【LINE】or【CW】or【CCW】key at right of the screen to display key-in data in Figure form and to display the confirm dialog box at the center of the screen.

※ Cancel：pressing【LINE】or【CW】or【CCW】key at right of the screen to cancel key-in linear motion.

【Function Keys at Right of Screen】

※ Line：adding a new line. Users can key-in data after pressing this key. After pressing this key, users can key-in data. Then, pressing【OK】key at screen bottom to finish adding new line.

※ CW：drawing a arch by clockwise direction. Users are able to key-in data after pressing this key. Pressing【OK】key at screen bottom to complete this new adding action.

※ CCW：drawing a arch by counter clock-wise direction. Users are able to key-in data after pressing this key. Pressing【OK】key at screen bottom to complete this new adding action.

※ Delete：delete the last line of the present profile.

※ Zoom In：pressing the【SCALE】key at bottom of the screen for the “SCALE” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use 【UP】, 【DOWN】, 【LEFT】 and 【RIGHT】key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the 【ENTER】key to complete the action.

※ Fittest Size：pressing【WINDOW】at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.

※ Return：when pressing【SCALE】 key at screen bottom to switch to the geometry profile screen.

◇ **Screen of Geometry Grooving is as below**

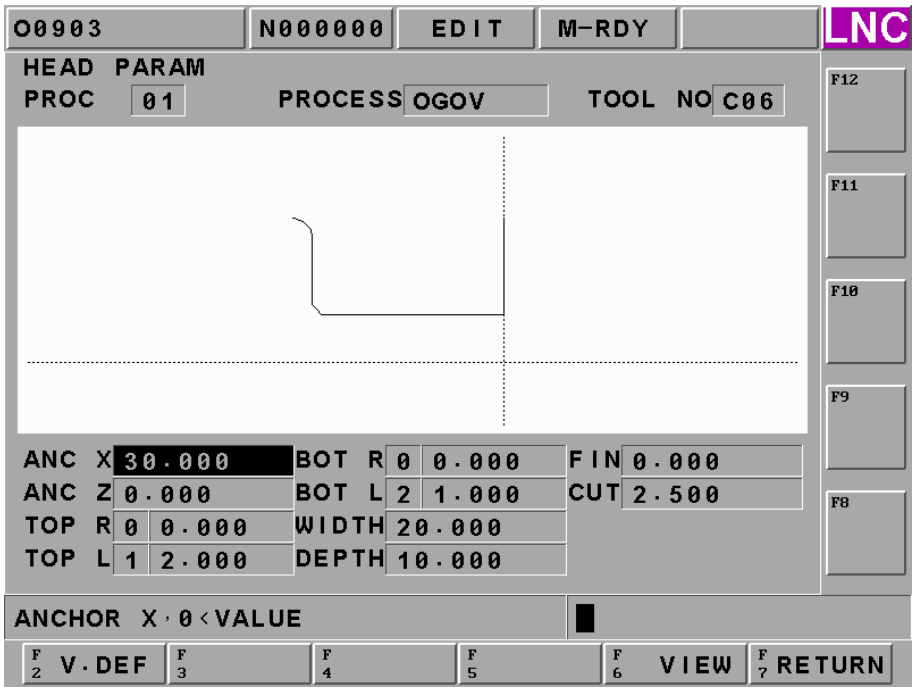


Figure 1.6-12 Geometry Grooving Page

【Column Description】

※ Procedure：display the currently procedure code number

※ Working Method：display the working name of the currently editing program.

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

※ Geometry Grooving Column：

⊙ **ANC X**：Reference diameter coordinate.

⊙ **ANC Z**：Reference Z coordinate.

⊙ **TOP R**：upper right chamfer angle condition and amount, condition（0 = right angle, 1 = rounding angle, 2 = chamfer）but, the amount cannot be smaller than the grooving-tool's tool noise radius.

⊙ **TOP L**：upper left chamfer angle condition and amount, condition（0 = right angle, 1= rounding angle, 2 = chamfer）but, the amount cannot be smaller than the grooving-tool's tool noise radius.

⊙ **BOT R**：lower right chamfer angle condition and amount, condition（0= right angle, 1= rounding angle, 2= chamfer）but, the amount cannot be smaller than the grooving-tool's tool noise radius.

⊙ **BOT L**：lower left chamfer angle condition and amount, condition（0= right angle, 1= rounding angle, 2= chamfer）but, the amount cannot be smaller than the grooving-tool's tool noise radius.

⊙ **WIDTH**：Grooving width.

⊙ **DEPTH**：Grooving depth.

⊙ **FIN**：preservation amount when cutting.

⊙ **CUT**：everytime's cutting width.

【Function Keys at Screen Bottom】

※ Windows Scale：scale-in or return to original size of figure shape of display box at center of screen.

※ Preview: preview the Figure shape of the key-in data

※ Return: switch back to the main function screen.

【Function Keys at Right of Screen】

※ Scale：pressing the 【SCALE】 key at bottom of the screen for the "Zoom In" key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use 【UP】, 【DOWN】, 【LEFT】 and 【RIGHT】key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the 【ENTER】key to complete the action.

※ Fittest Size：pressing 【WINDOW】at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.

※ Return：pressing【SCALE】key at screen bottom to switch the screen to the geometry grooving screen.

◇ **Screen of Geometry Thread is as below**

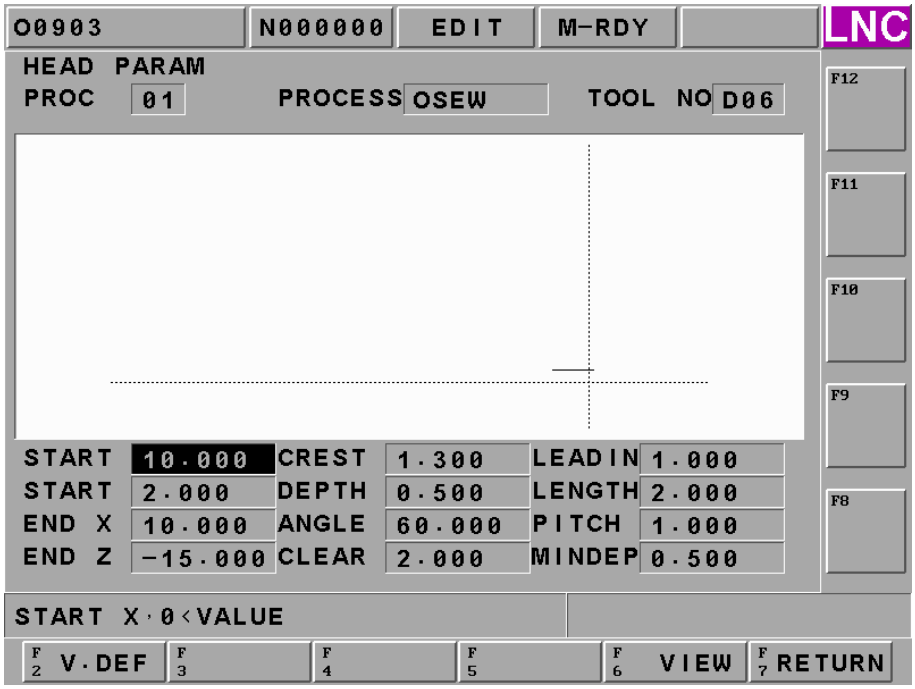


Figure 1.6-13 Geometry Thread Page

【Column Description】

※ Procedure: display the currently procedure code number.

※ Working Method: display the working name of the currently working programs.

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

※Cutting Parameter Column：

⊙ **START**：start tapping diameter coordinate

⊙ **START Z**：start tapping Z coordinate

⊙ **END**：start tapping diameter coordinate

⊙ **END Z**：end tapping Z coordinate

⊙ **CREST**：thread height

⊙ **DEPTH**：1st cutting depth

⊙ **ANGLE**：tapping angle

⊙ **CLEAR**：safety tool-hanging height

⊙ **LEADING**：leading cutting amount

⊙ **LENGTH**：post-chamfer angle length

⊙ **PITCH**：tapping distance

⊙ **MINDEP**：smallest lathe-cutting depth

【Function Keys at Screen Bottom】

※ Windows Scale：scale-in or return to original size of figure shape of display box at center of screen.

※ Preview：preview the Figure shape of the key-in data

※ Return: switch to the main function screen.

【Function Keys at Right of Screen】

※ Scale：pressing the 【SCALE】 key at bottom of the screen for the “Zoom In” key to occur. Scale-in the selected Figure. A figure dialog box will occur when pressing this key. Users are able to use 【UP】, 【DOWN】, 【LEFT】 and 【RIGHT】key to move the dialog box. Moving the dialog box to the wanted zoom-in position and pressing the【ENTER】key to complete the action.

※ Fittest Size：pressing 【WINDOW】at screen bottom for this function to be occur. This will return to the original size of the Figure shape at inside of the dialog box at screen center.

※ Return：pressing【SCALE】key at screen bottom to switch the screen to the geometry thread screen.

◆ **NOTE**

※ Each parameter content will be different due to different working methods.

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

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

※ Pattern cutting line must be from right to left and cannot be intersect with each other. The max. line sections are 10.

※ Thread lathe-cutting is not able to do facing-thread and variable-lead thread.

※ Dwell time of tapping cutting parameter of drilling lathe-cutting cannot be zero.

◆ Appendix A（Error Code Description）

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

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

◎ Table 1-2 Tool Error Code（T）

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

◎ Table 1-3 Exchange File Error Code（E）

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

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

Error Code	Message
182	Use drilling lathe-cutting tool error

1.7 GRAPH

1.7.1 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.

1.7.2 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.

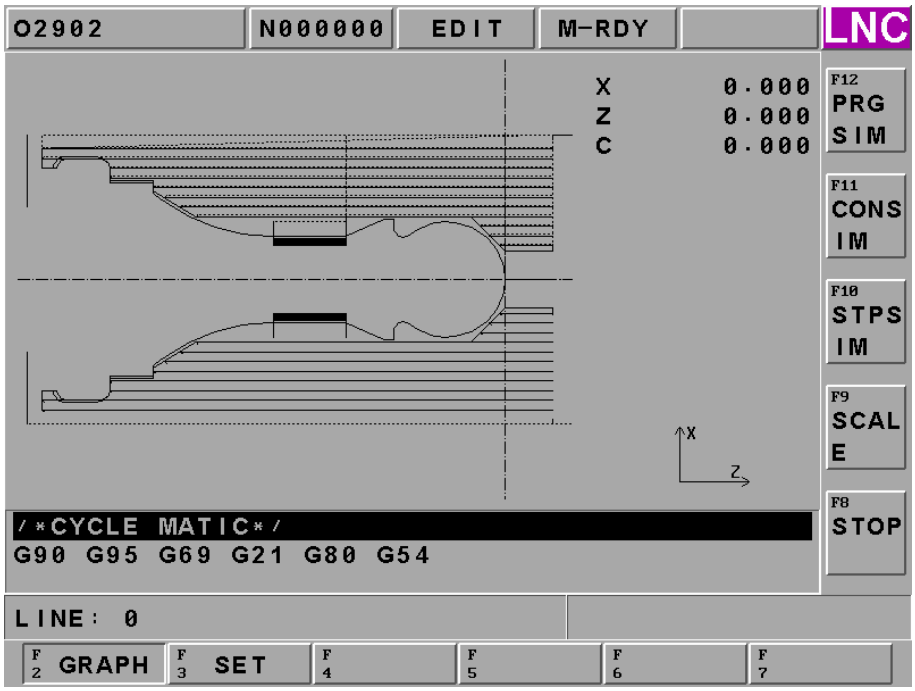


Figure 1.7-1 GRAPH page

【**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.

1.7.3 SET

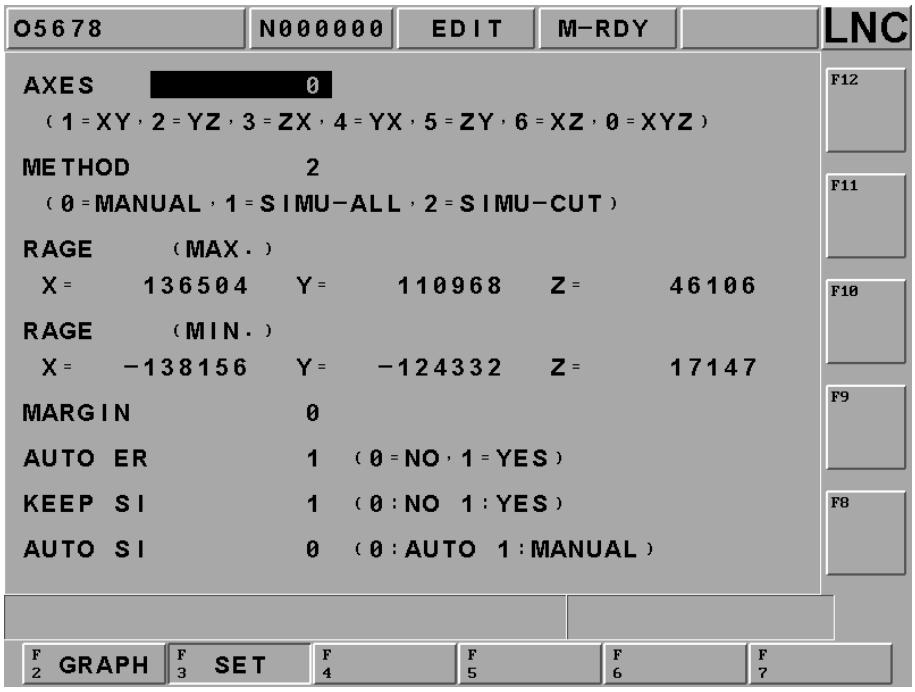
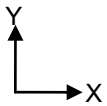
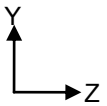
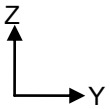
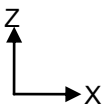
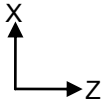
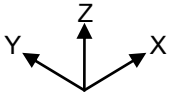
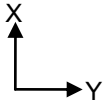


Figure 1.7-2 Main Screen of the Windows Definition

■ **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.

1.8 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.

1.8.1 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.

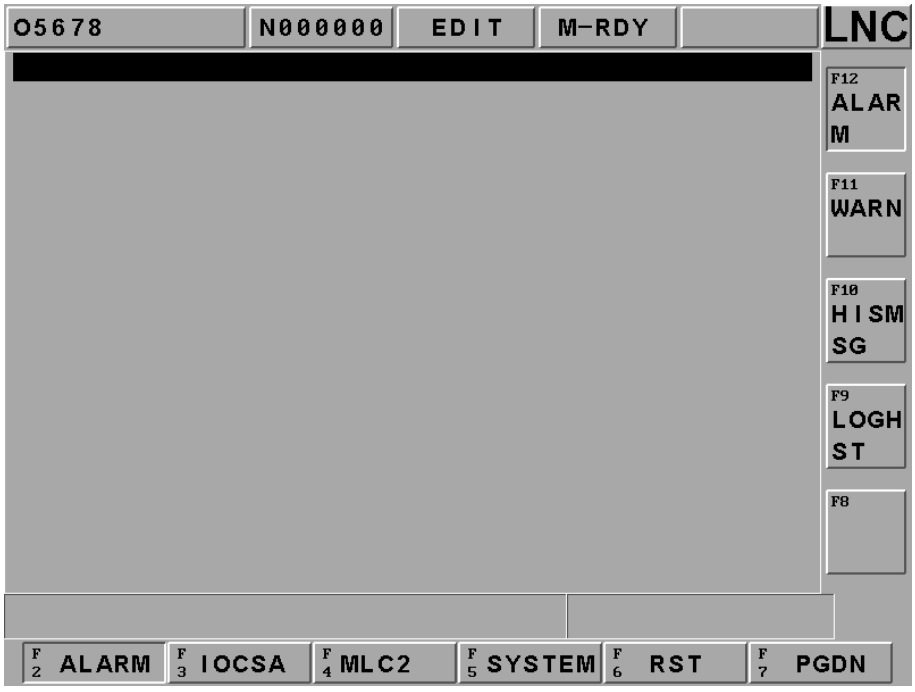
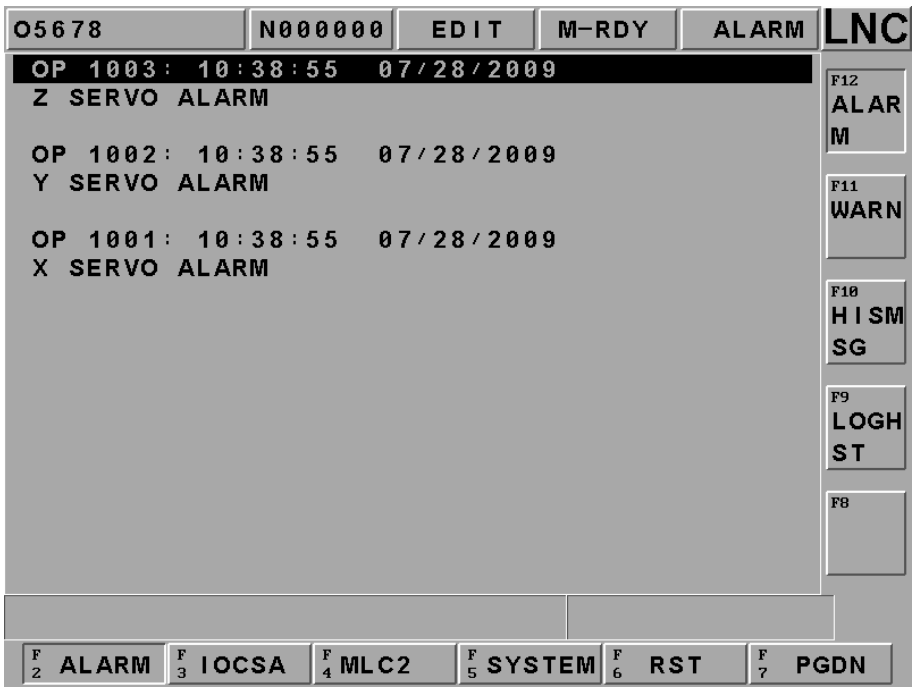
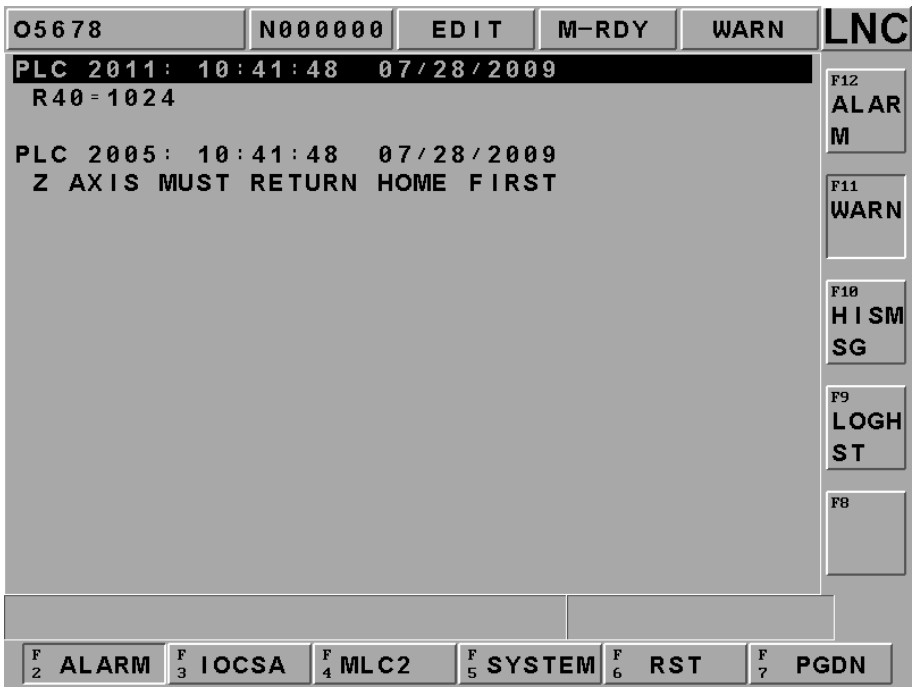


Figure 1.8-1 ALARM screen

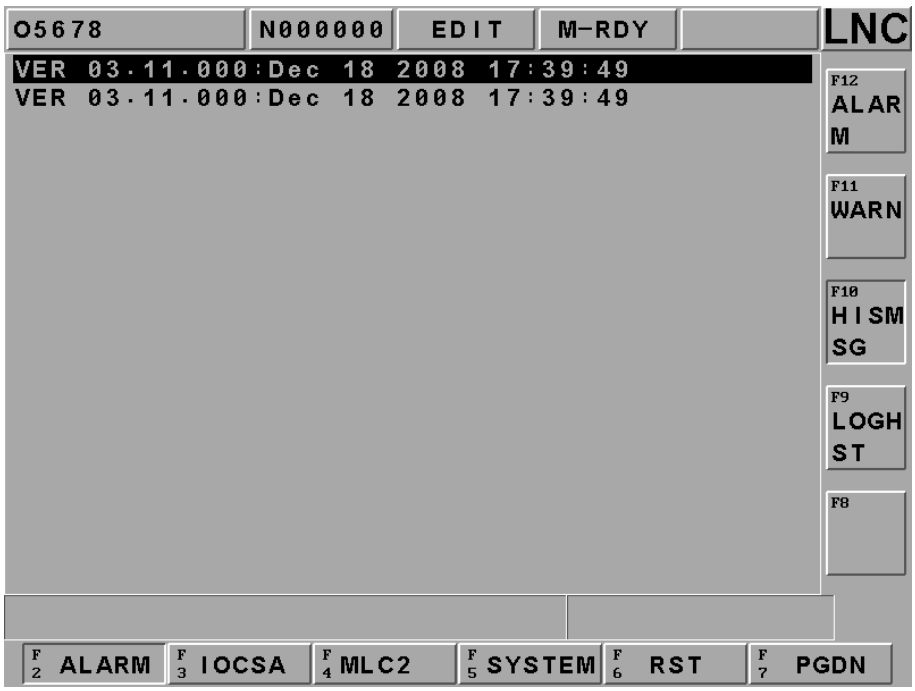
【**ALARM**】Alarm message will occur when there is any problem in system running. This will make the system stop running and also will display the alarm message on the screen. When the problem is solved, must press **<RESET>** in order to release the situation.



【**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.



【**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.

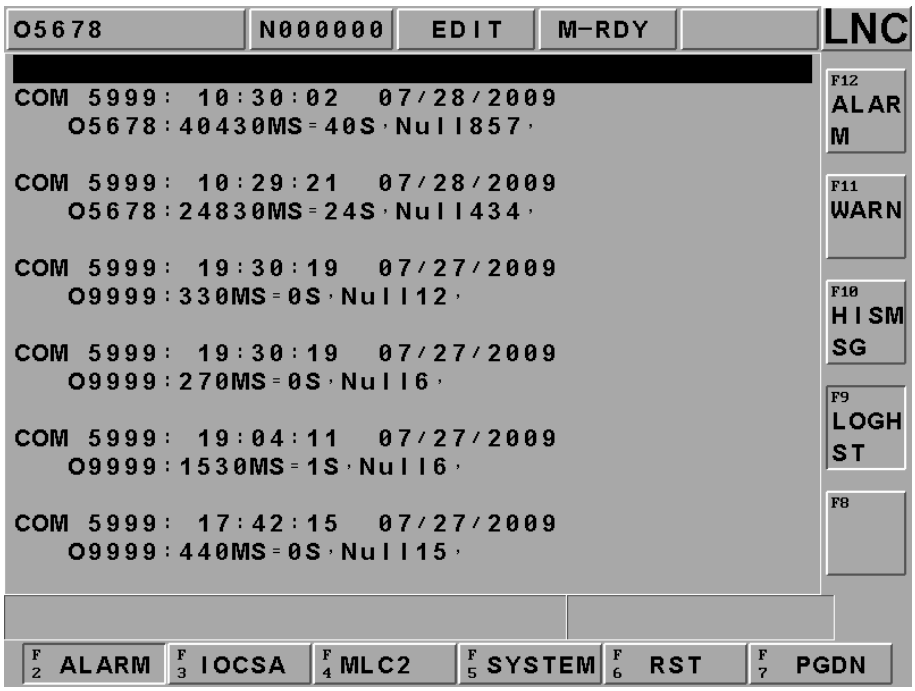


Figure 1.8-2 ALARM Message Screen

1.8.2 IOCSA

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

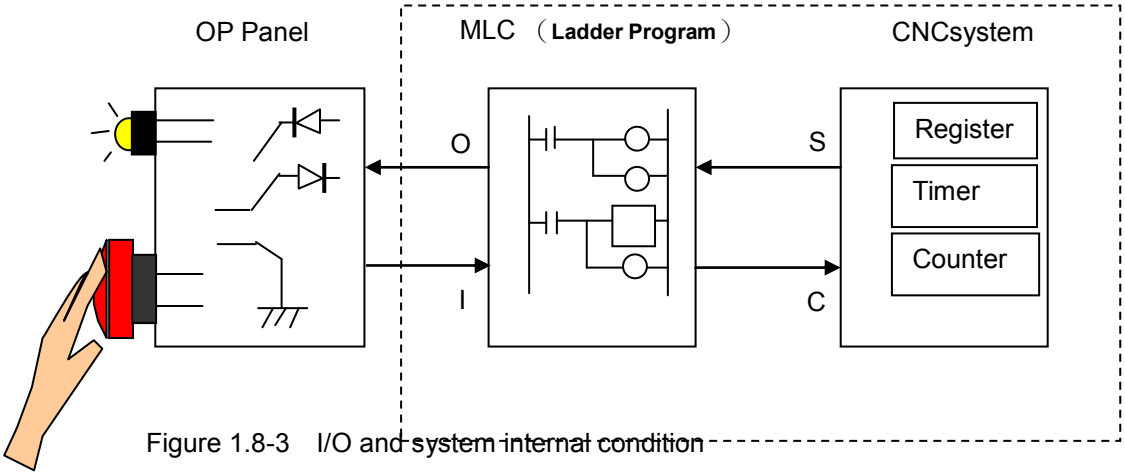


Figure 1.8-3 I/O and system internal condition

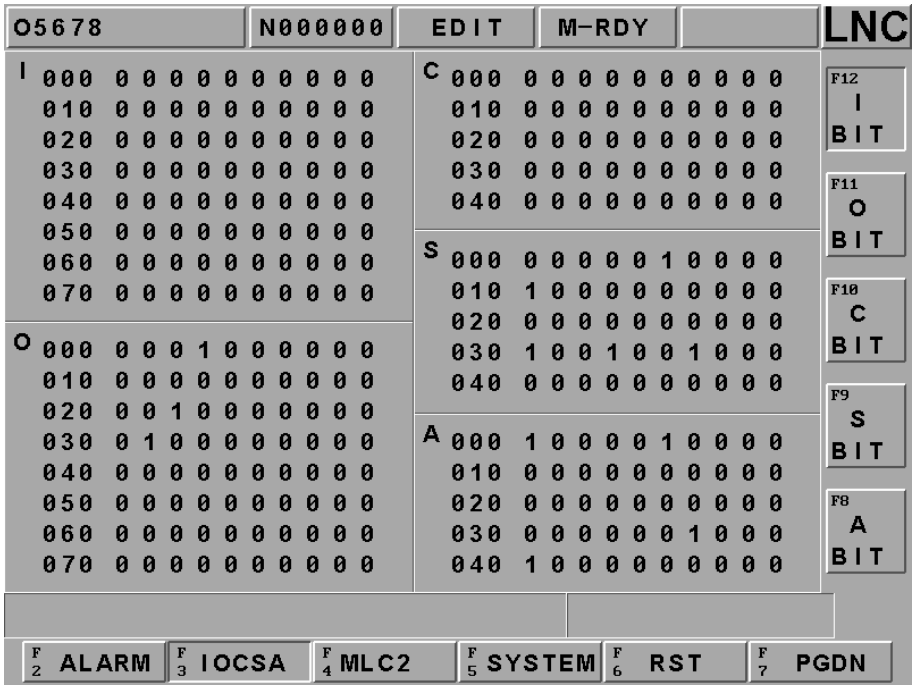


Figure 1.8-4 IOCSA Main Screen

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.

1.8.3 MLC2

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

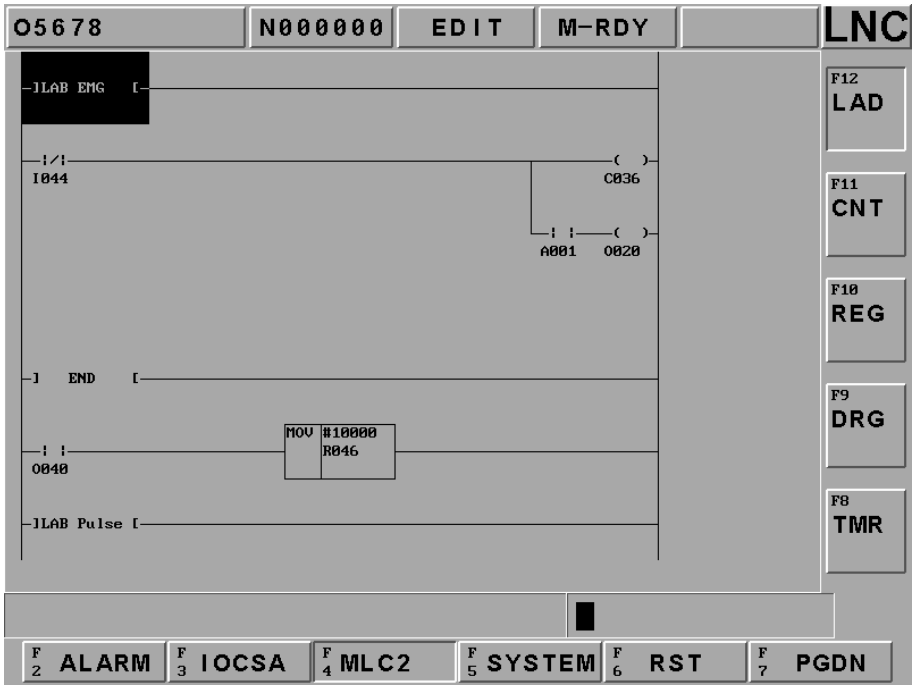
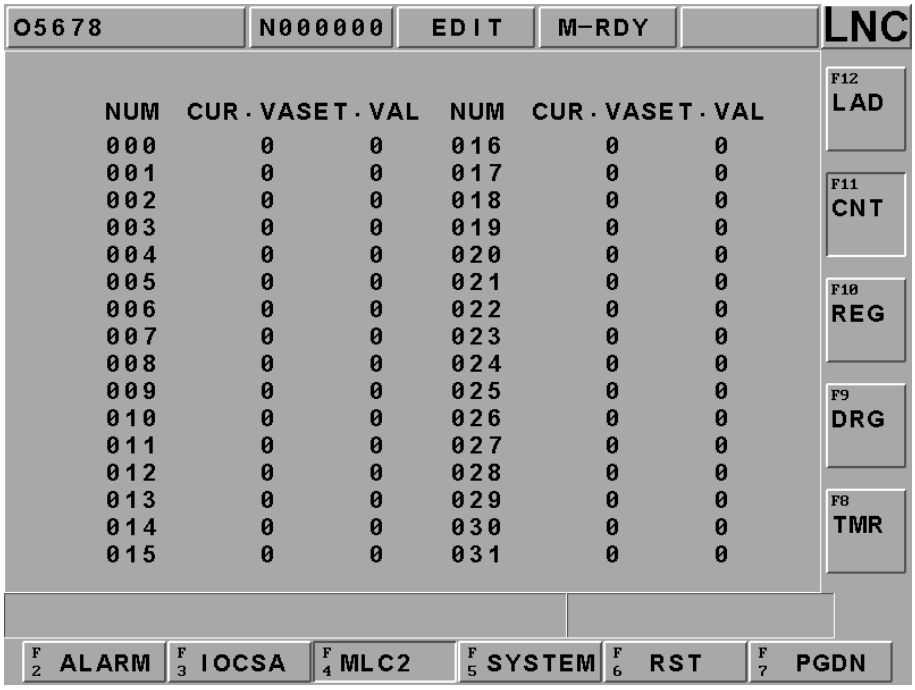


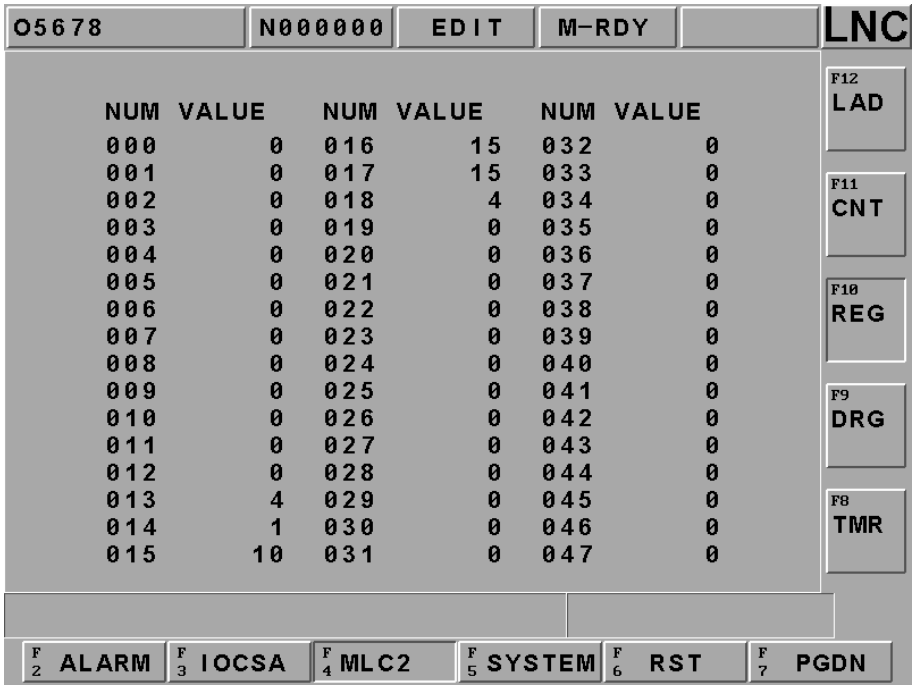
Figure 1.8-5 LADDER Figure

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.

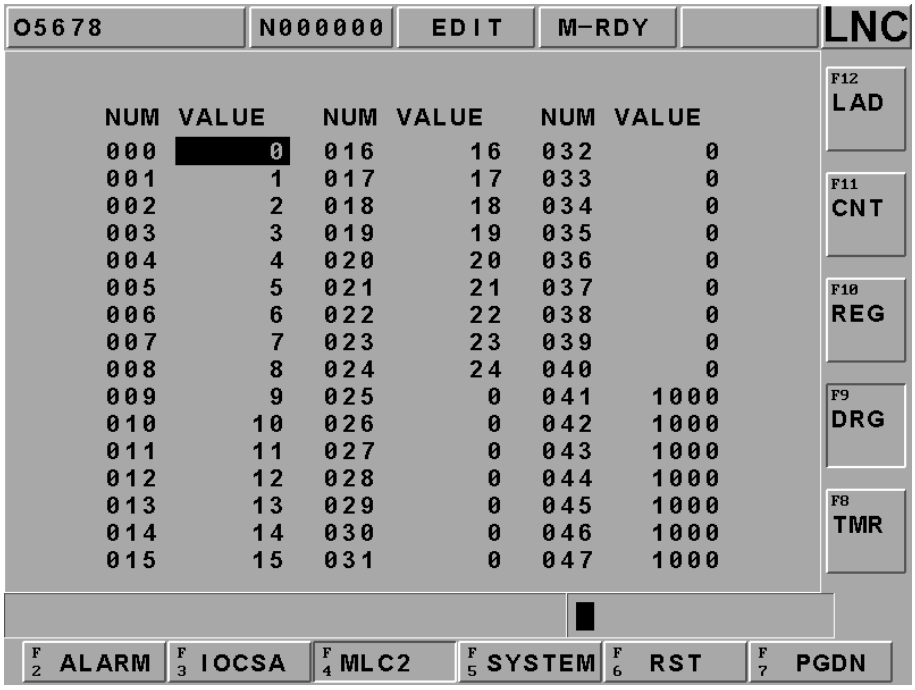
Press【CNT】to check data as below.



Press【REG】to check data as below.

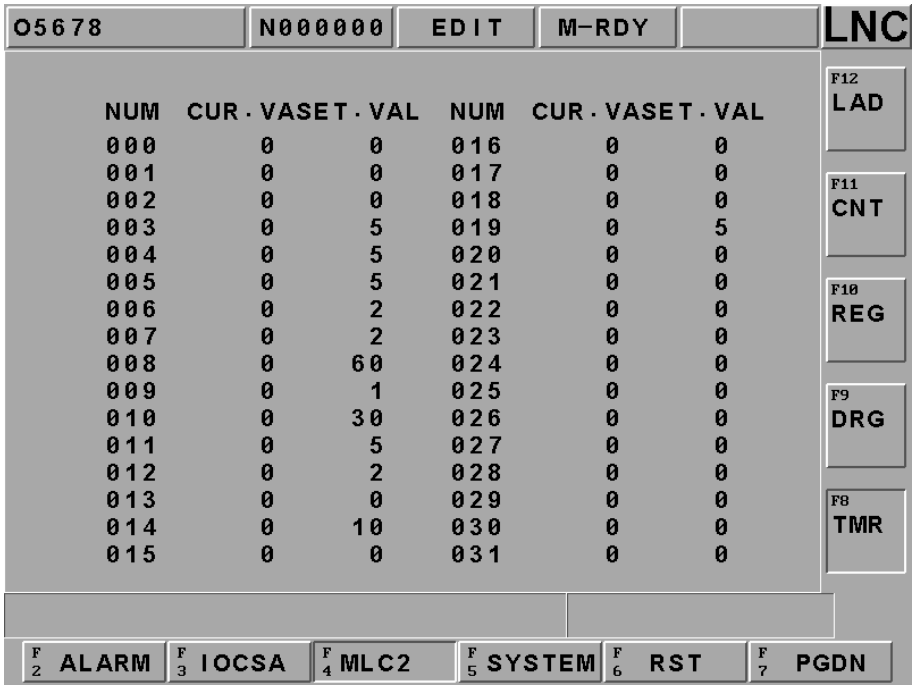


Press【CNT】、【REG】、【DRG】、【TMR】to see data as below.



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	

1.8.4 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.)

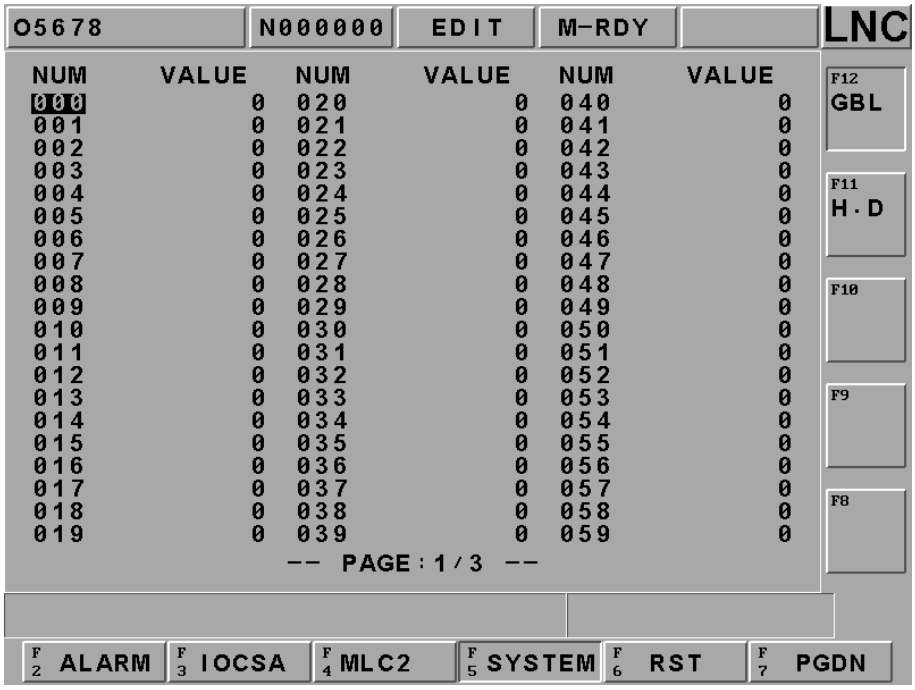


Figure 1.8-6 System Data\System

【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.

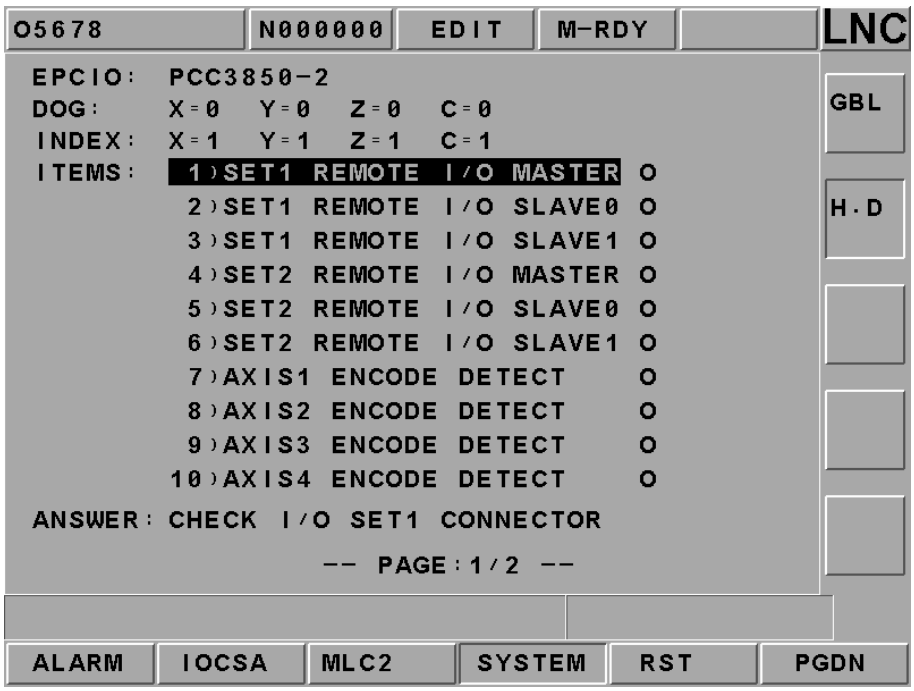


Figure 1.8-7 SYSTEM\H.D

1.8.5 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.

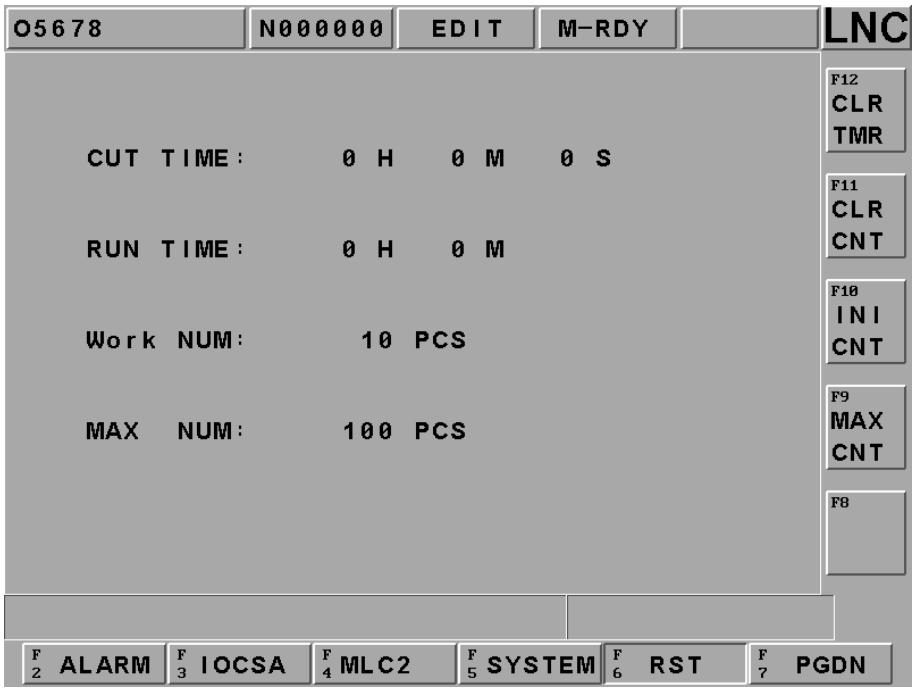


Figure 1.8-8 Working Parameter Page

1.8.6 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：

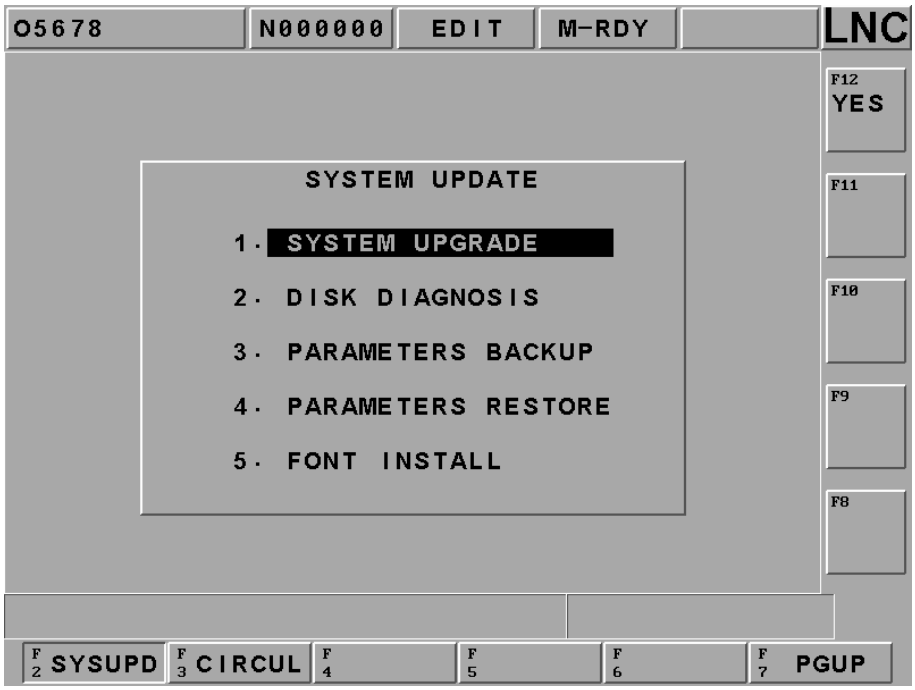


Figure 1.8-9 System Upgrade Screen

- 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.

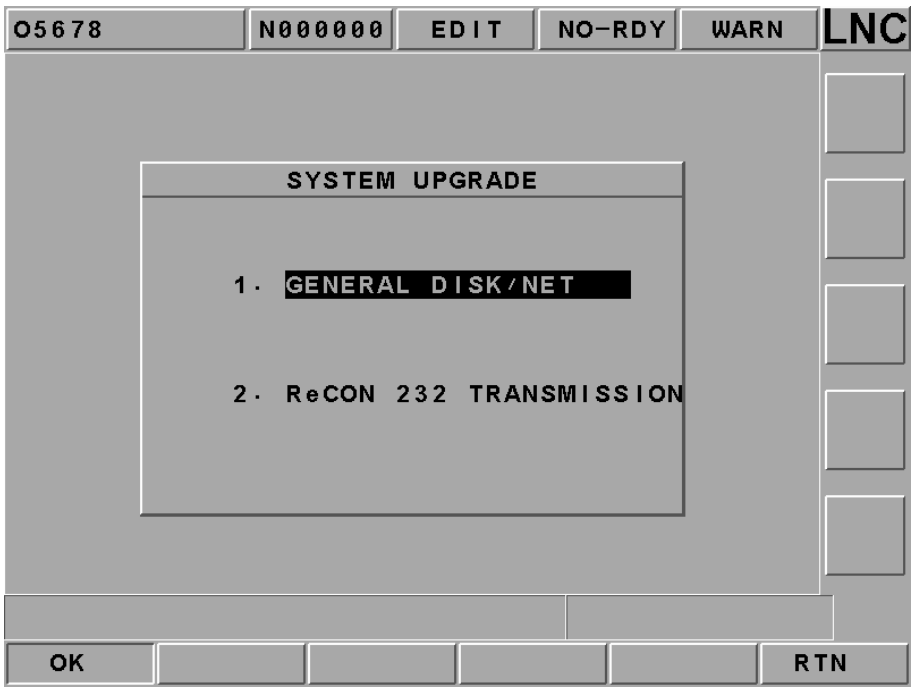
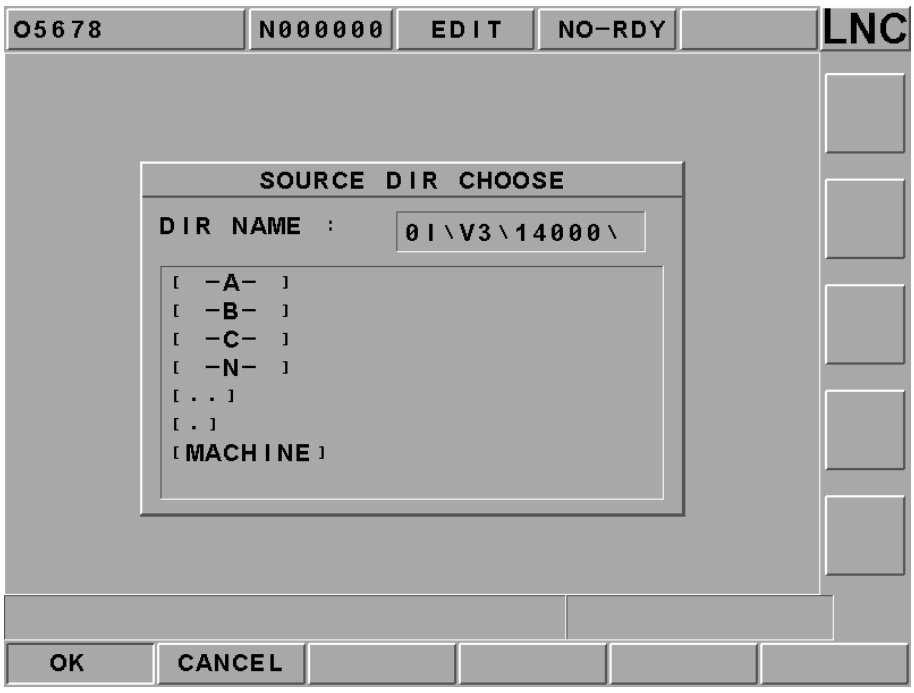


Figure 1.8-10 SYSTEM UPGRADE

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-T600 SERIES
=====================================
Current Version:
   T600_VER_03.12.000

Installing Version:
   T600_VER_03.12.000

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

Please wait, file preparing.

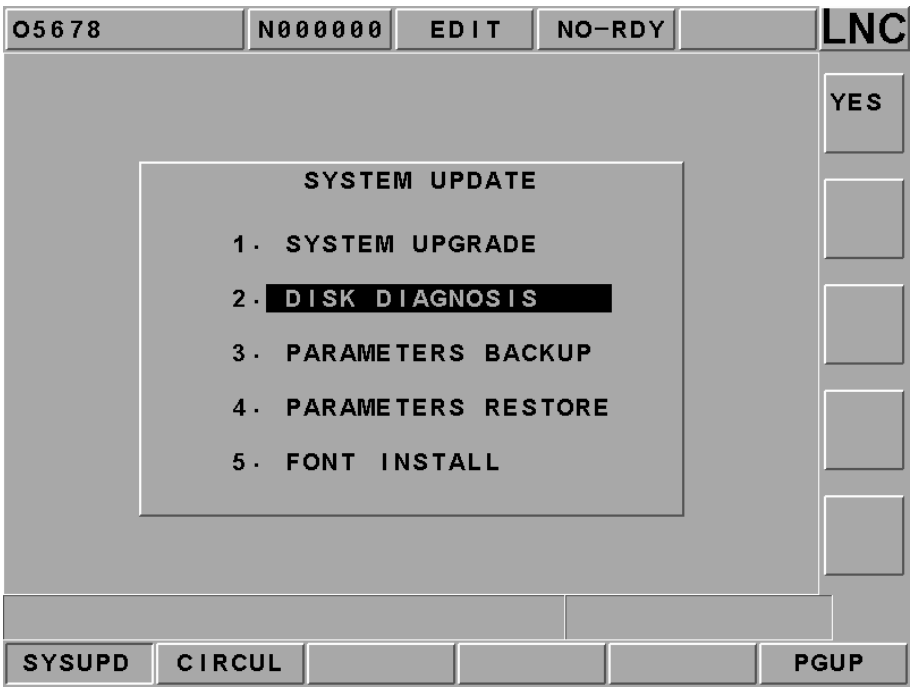
```
Installing...
Please wait,file preparing ...
```

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：

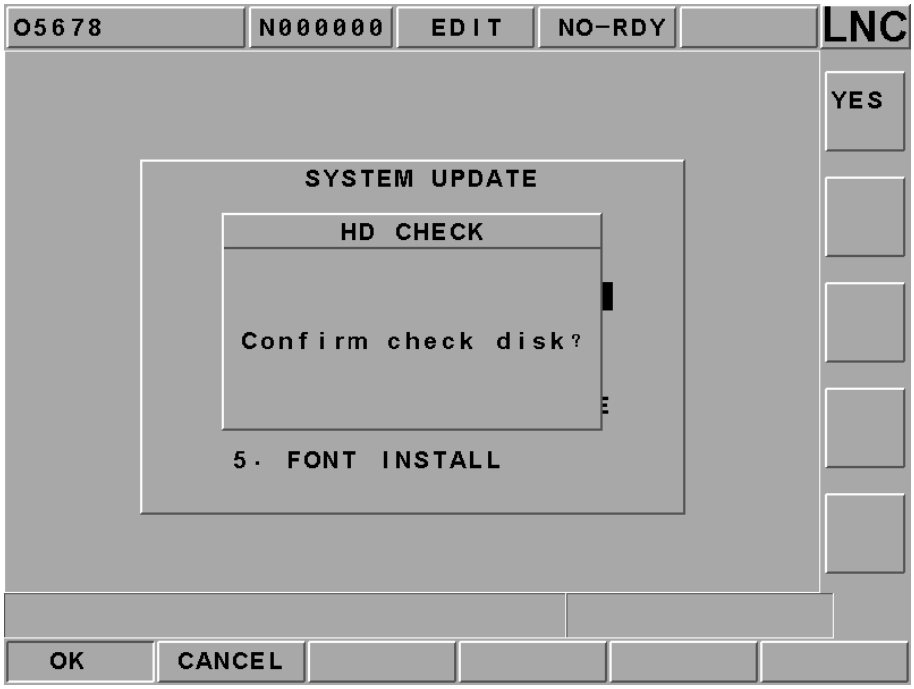


Figure 1.8-11　Confirm Whether or Not to Do the Hardware Checking

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 4 types of working items for users to choose:

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

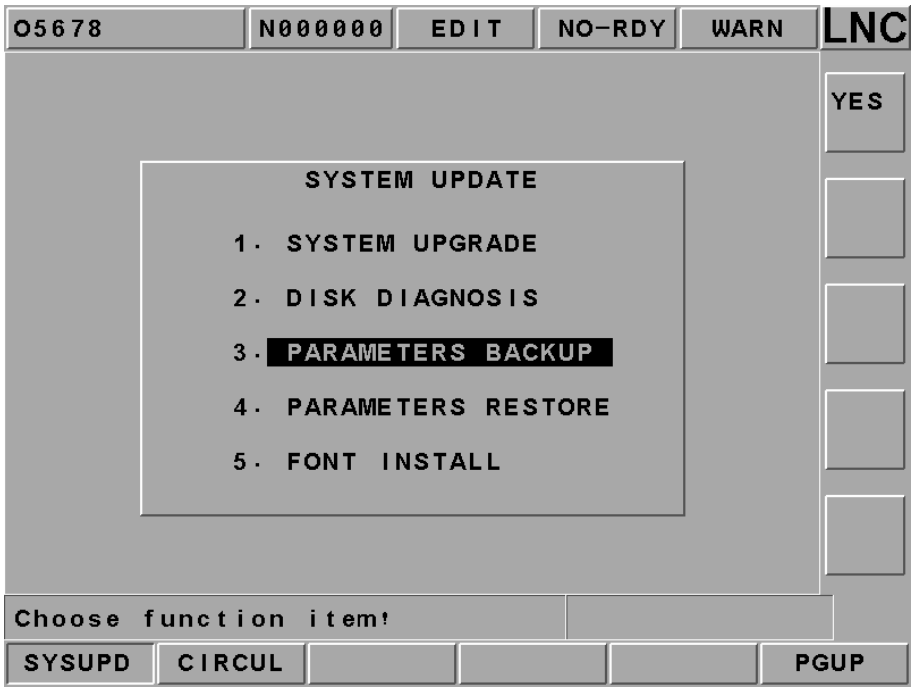
```
1.PCscan
  Virus checking
2.DISK doctor
  disk diagnostic and errer-fix
3.DEFRAG
  Disk access performance enhancement

0.Quit

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

Figure 1.8-12　Hardware Checking Function Selection

- 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.

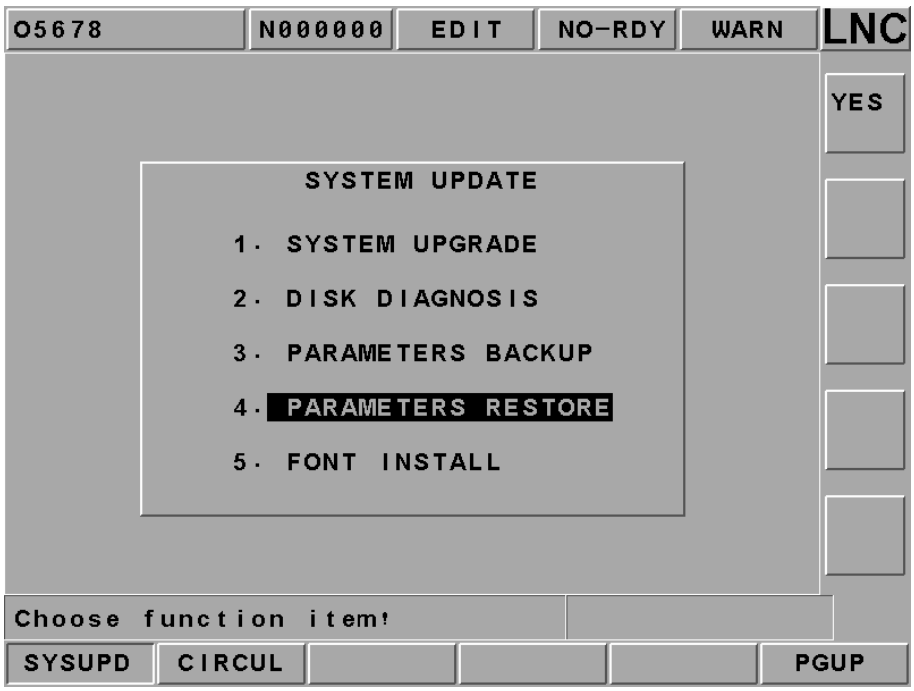
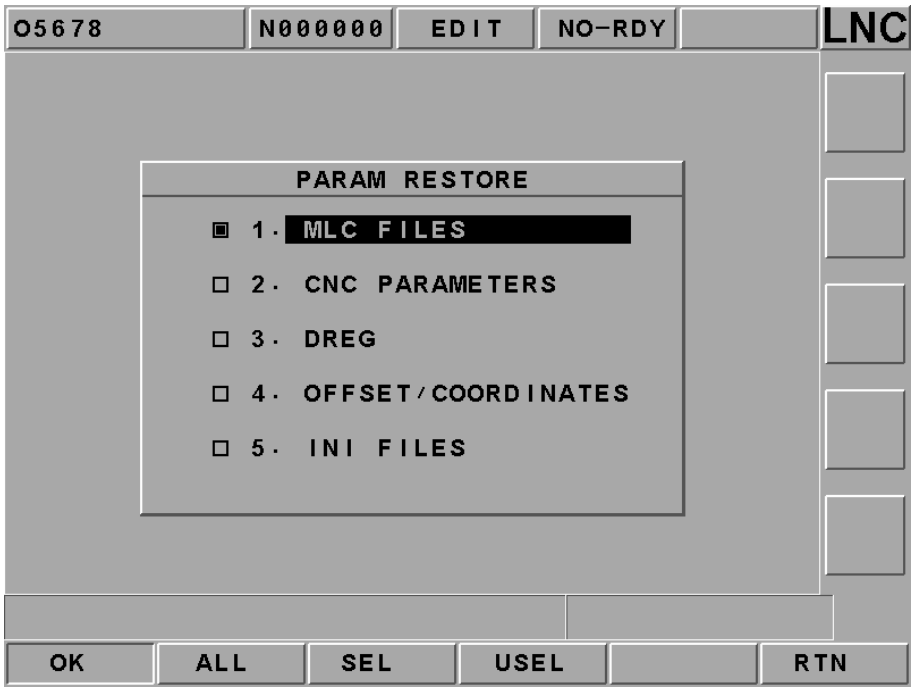


Figure 1.8-13 PARAM BACKUP Selection Screen

- PARAM LEAD-IN：



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

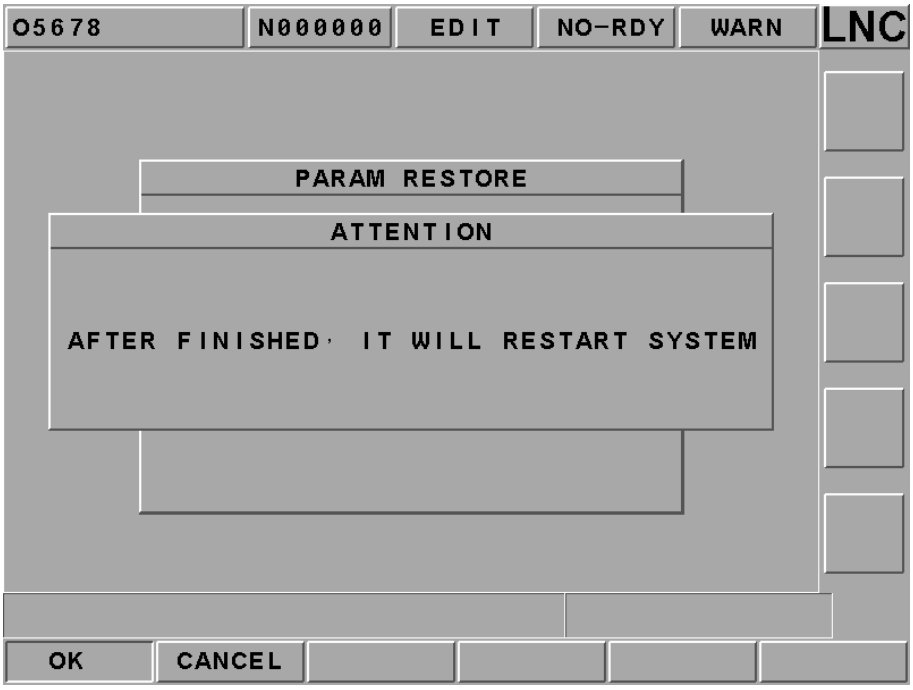


Figure 1.8-14　PARAM LEAD-IN SCREEN

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.

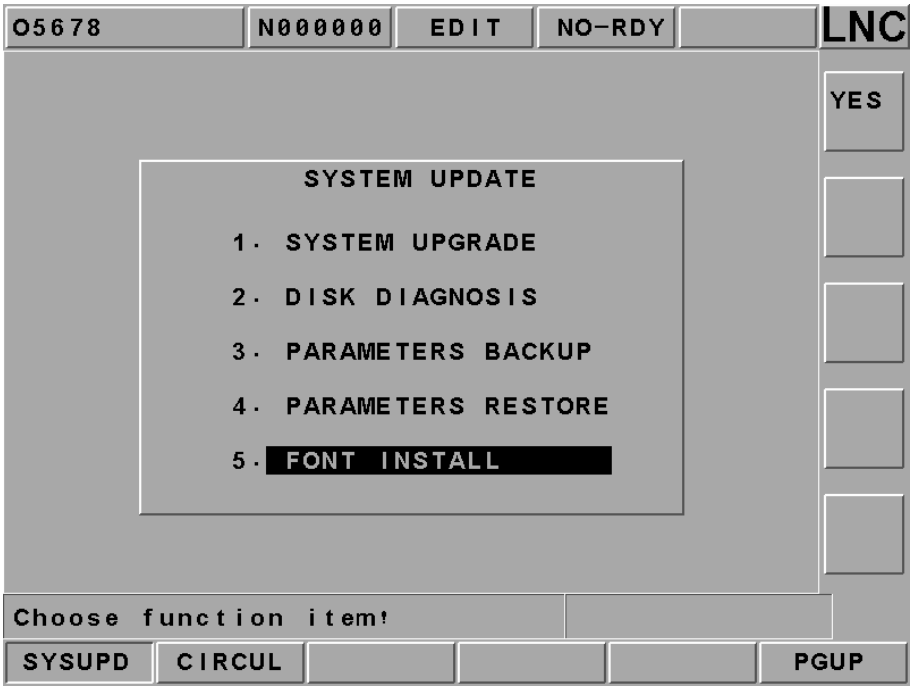
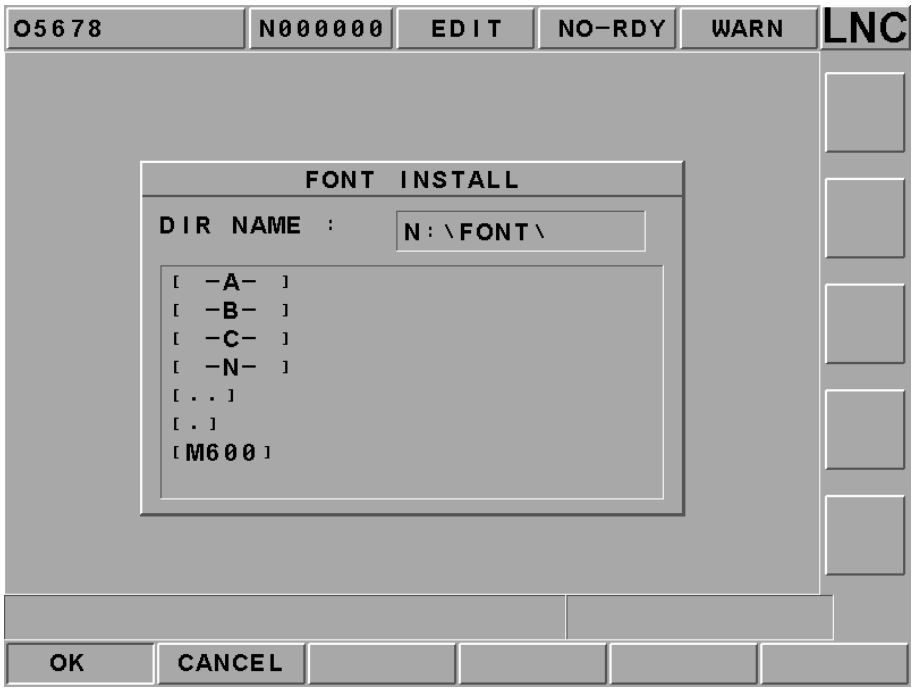


Figure 1.8-15　Reminding Users to Reboot after Key-in Parameter

- **FONT：**



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

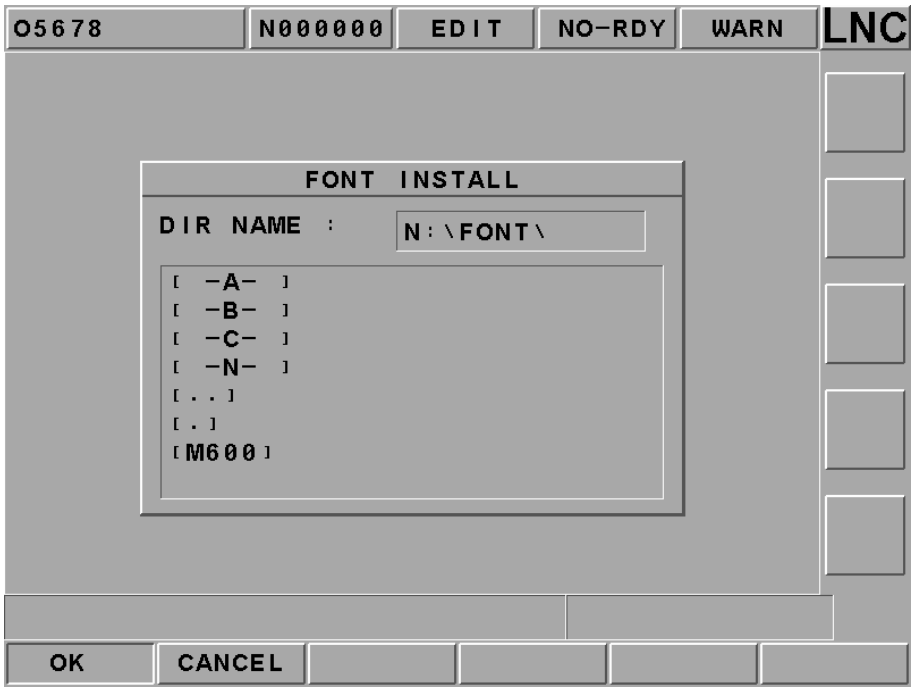


Figure 1.8-16　Text Font Installation Source Page

1.9 SOFTPL

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

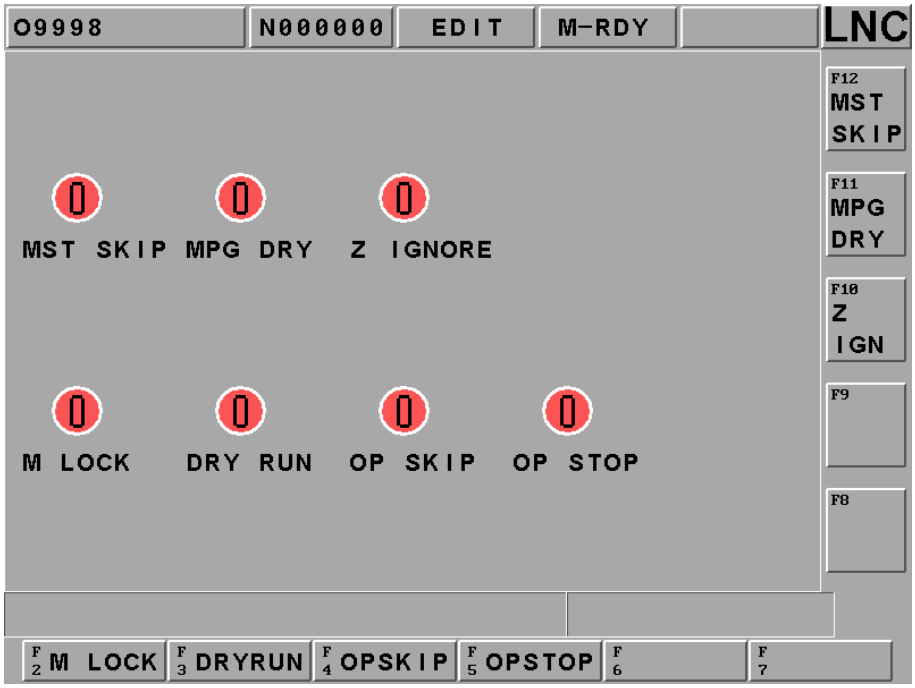


Figure 1.9-1 Soft Panel Function Screen

Using these corresponding functions switches separately. The function picture changes from red to green on the screen, which means the function is enabled. Turning it back again to disable the function

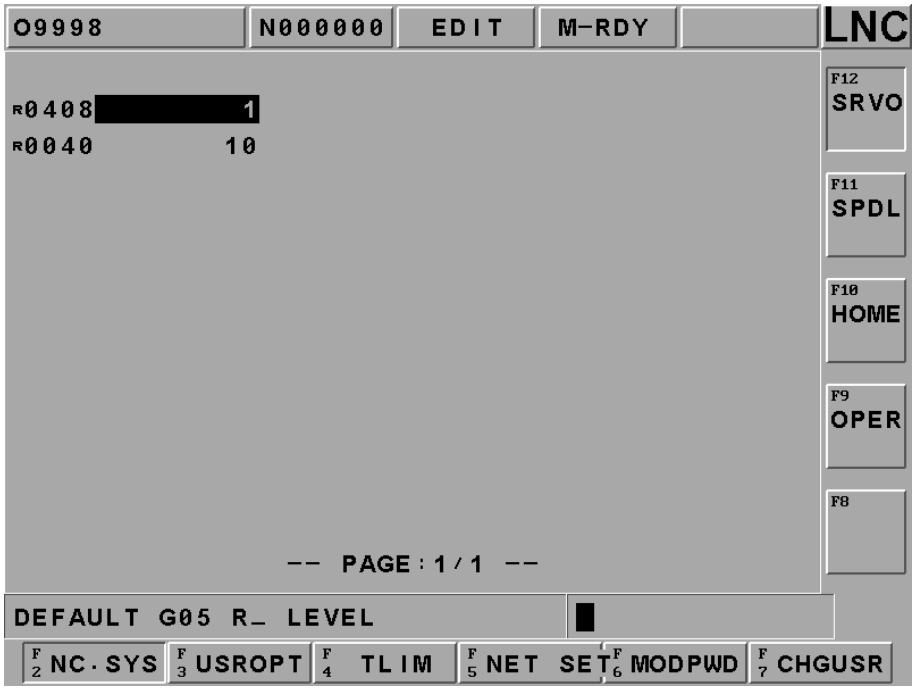
The Below Table Describes Each Soft Panel Function Selection Switches：

Function Switches	ON	OFF
M LOCK	The controller will continue executing program while the program is executing. But, the moving command of the servo axis will NOT be outputted. So actually, the servo axis is completed stop (no movement).	The moving command of the servo axis will be outputted with the program while the program is executing.
DRY RUN	It will follow the parameter's G01 velocity, not the program's velocity(the velocity of Pr,1004 setting)	The designated speed of the servo feed program.
OP SKIP	Program begins with "/" symbol will be skipped and not be executed.	Single block that begins with "/" symbol will be executed normally.
OP STOP	The program will stop when the program executes up to M01. If want to continue executing, users need to press "CYCLE START".	Controller will ignore M01 and execute the next single block directly.
MST Ignore	Ignore M, S, T codes commanding.	Consider M, S, T codes commanding.
MPG DRY RUN	System starts to process, hand wheel can control the program processing, and the program coordinate and servo axis to change together. The faster MPG runs, the faster program runs. But MPG will not faster then program feetrate. When MPG stops, the program will also stop. MPG CCW will not control program.	Hand wheel can not modify the program coordinate and let the servo axis moves according to it.
Z Ignore	Ignore any command from Z Axis	Process any command from Z Axis.

1.10 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】**

1.10.1 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.

1.10.2 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.

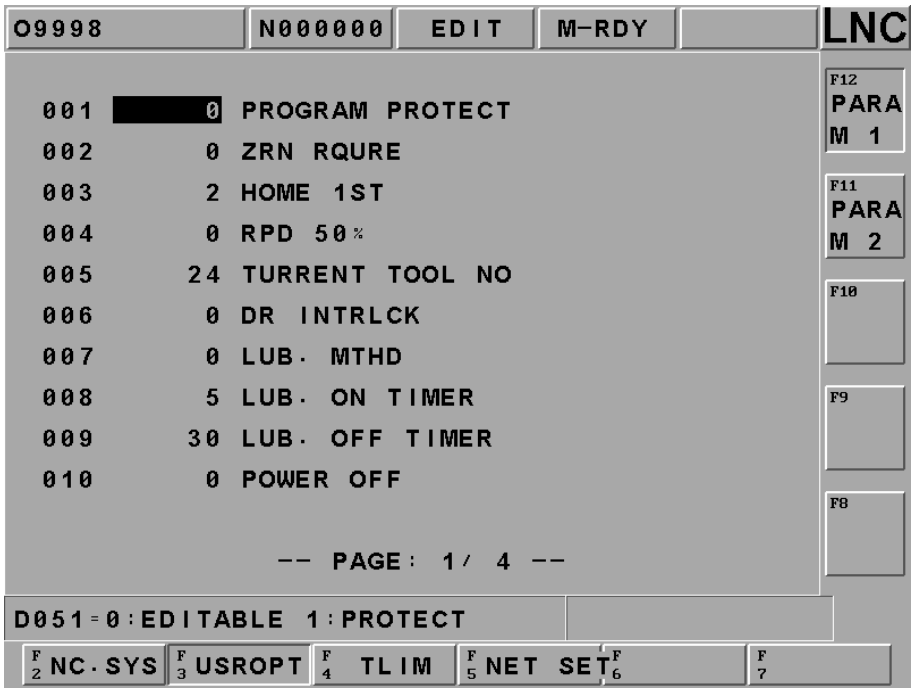
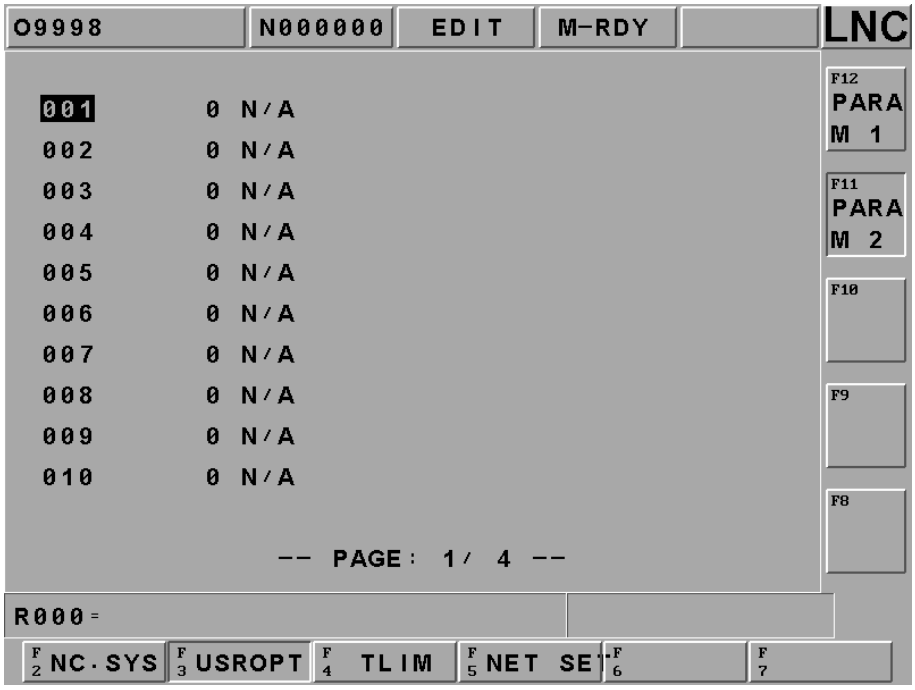


Figure 1.10-1 USROPT

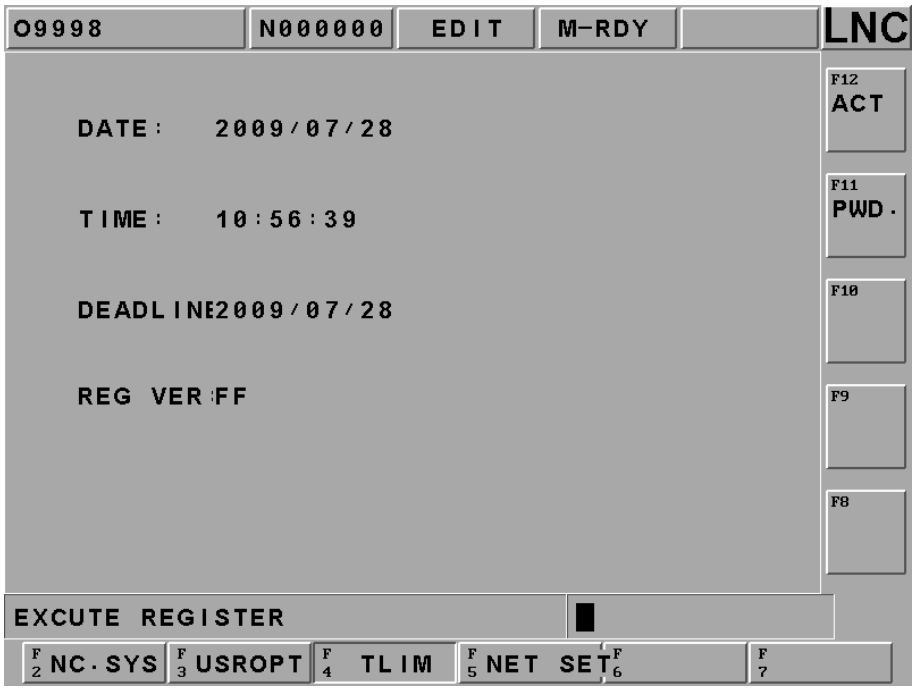
Press【PARAM 2】page, there are 40 sets of parameters. The way to change the parameter is the same to system parameter, please refer to last chapter.



1.10.3 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.】

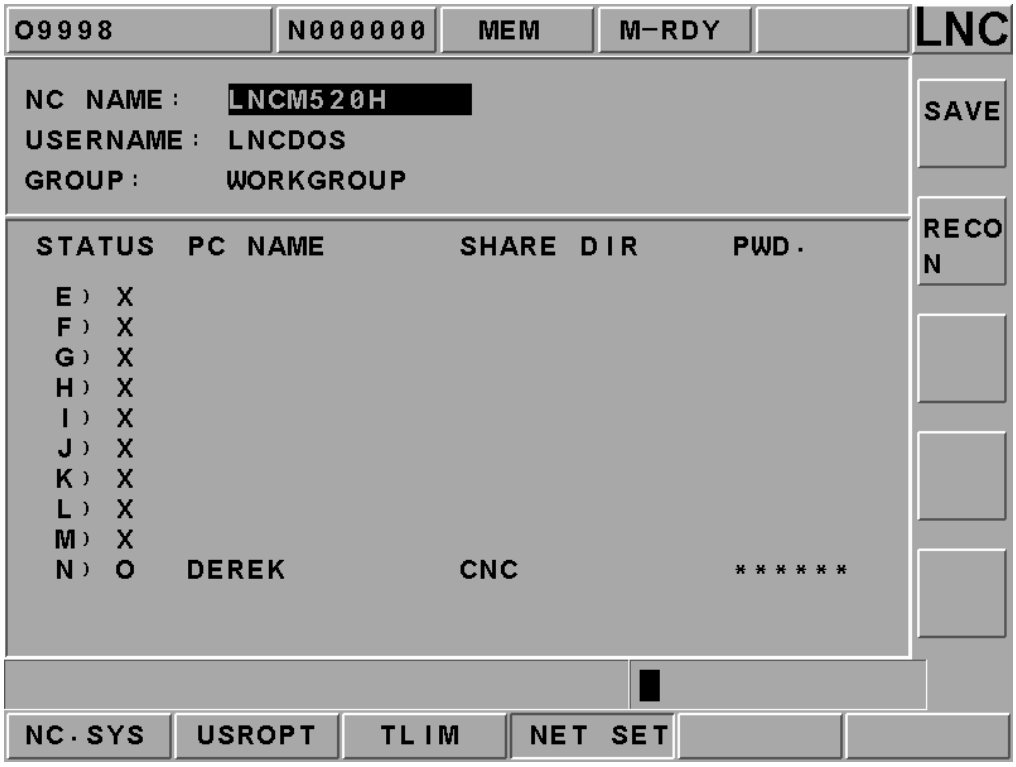
1.10.4 NET SET

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.

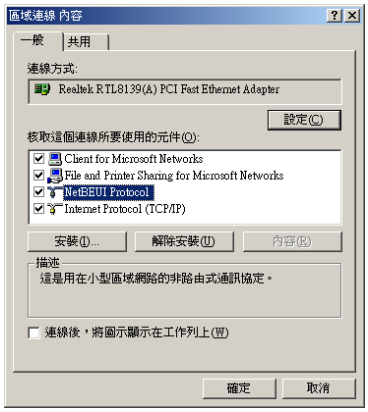
Share File：The share files on the PC

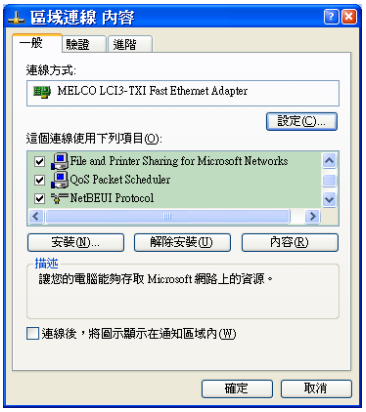
Password：This is the password for controller to log into PC. Also means PC end LNCDOS user's password.

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.

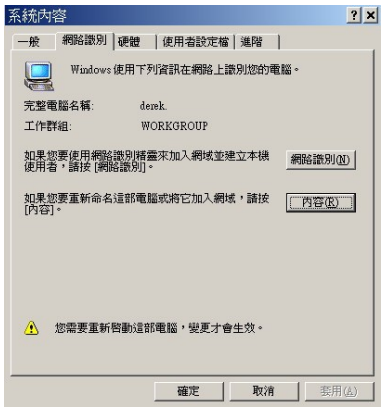
Note：Nwlink NetBIOS, this protocol can't be drove, otherwise the connection will fail.

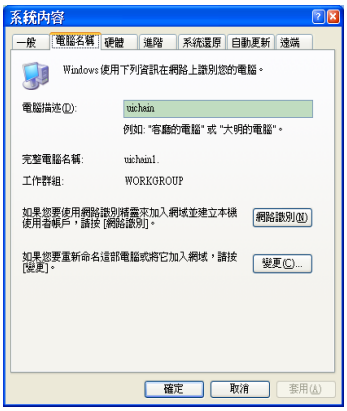




WINDOWS-98 **WINDOWS-2000** **WINDOWS-XP**

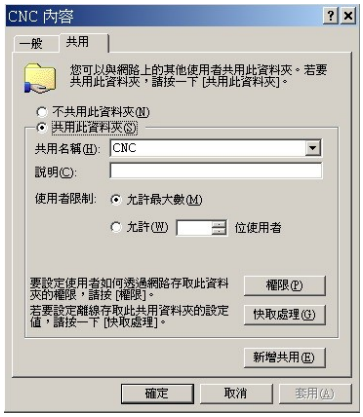
B.Confirm [PC name] and [Work Group] is correct setting.（Controller side's PC name and work group name should be the same, and vice versa）





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

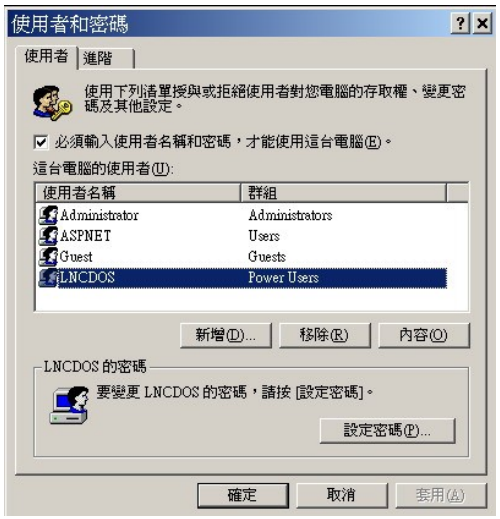
C.To share PC end's file, and make sure the share name is the same with controller side.

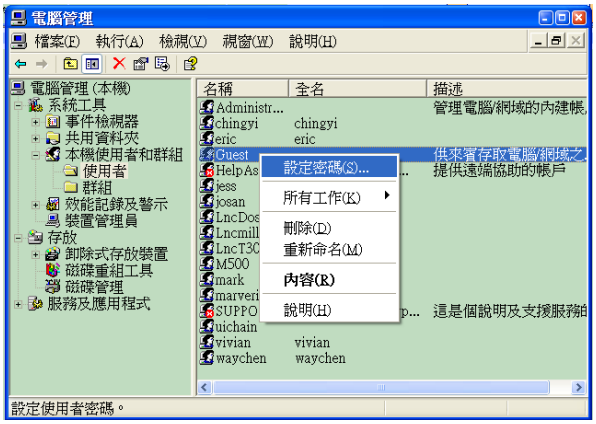


WINDOWS-98 **WINDOWS-2000** **WINDOWS-XP**

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

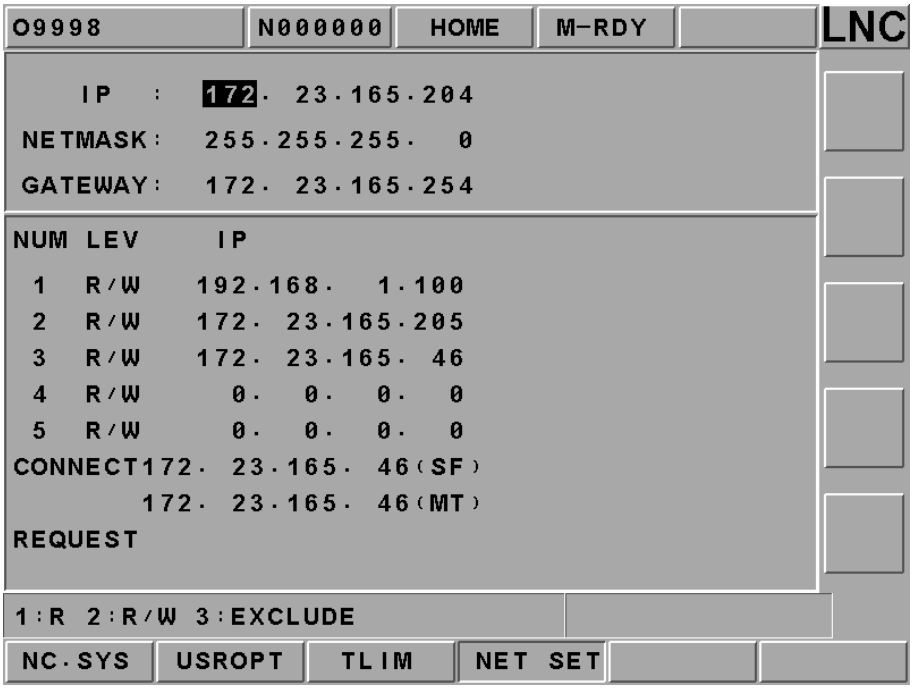
（Window98：If you have files to share, no need to add new users.）





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.

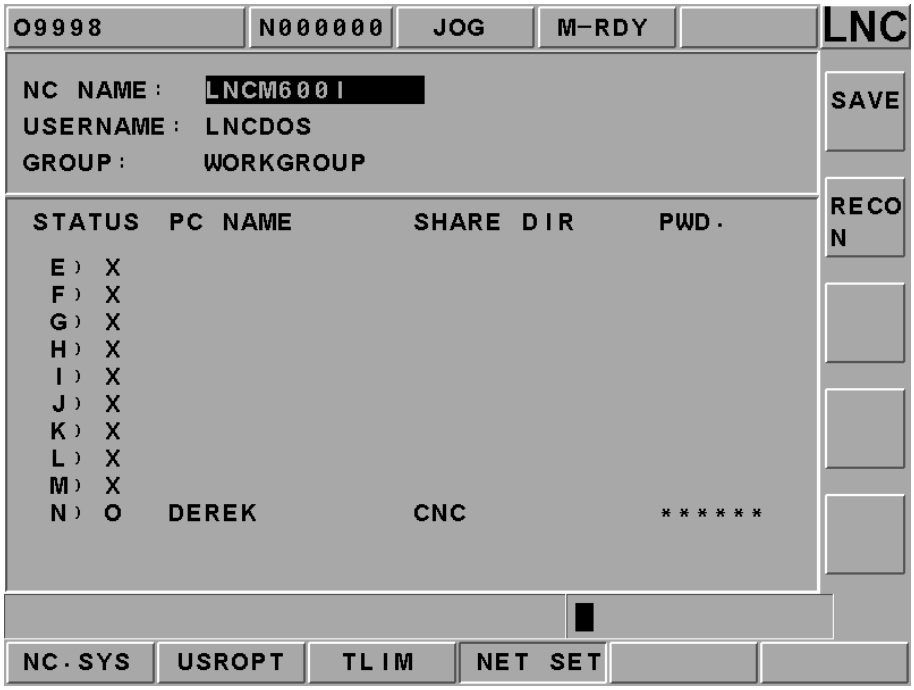
3. GATEWAY：Please set this gateway the same with PC end.

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

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

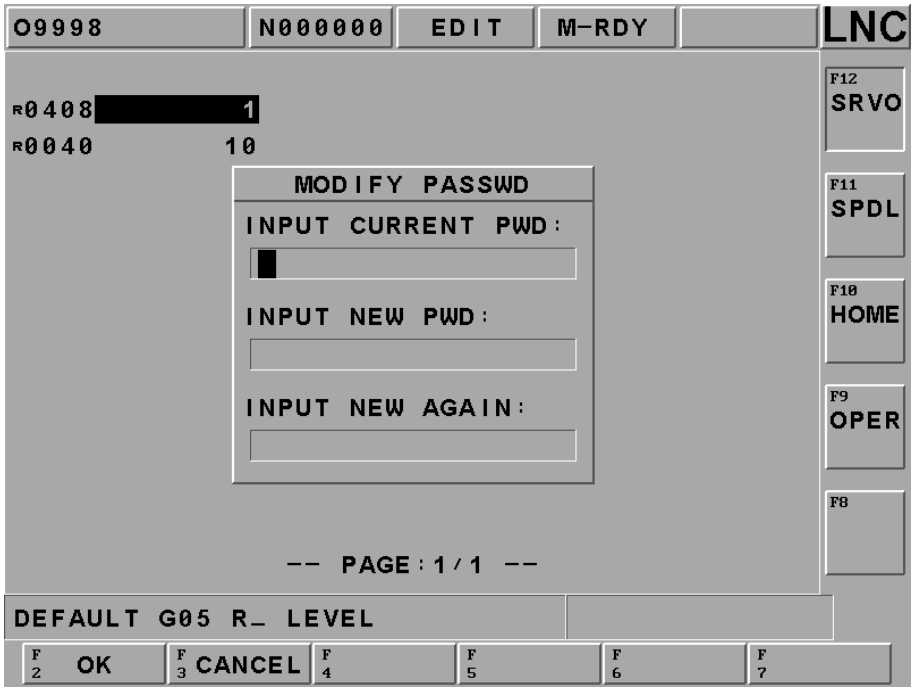
If you want to cancel the Ip address, you can move cursor to this are and press INPUT to make the data to be 0. Next time when you connect to PC, it will show a dialogue again to request you fill up the IP address.

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.

1.10.5 【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.

1.10.6 【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.

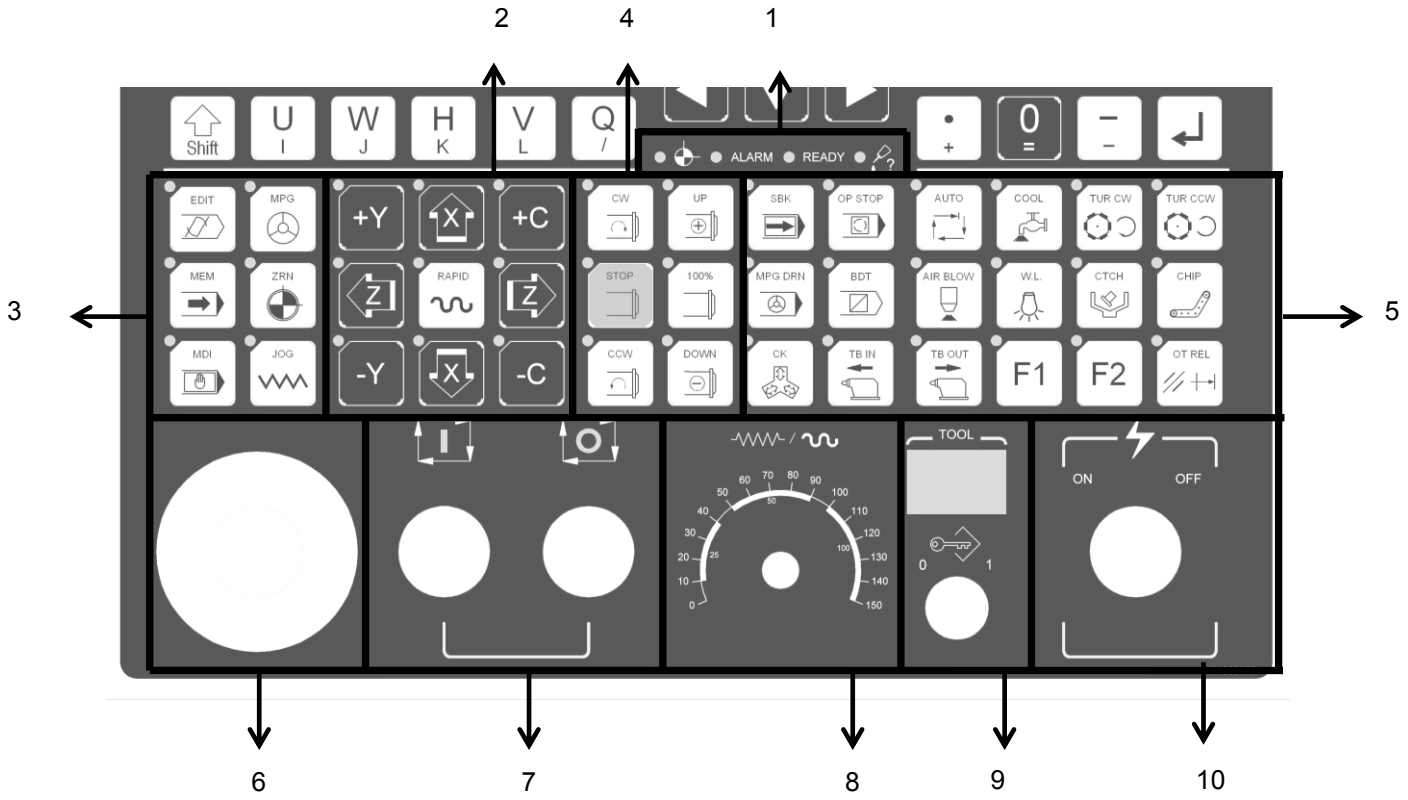
2 OP Panel Operation

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 11 sections according to different functions：

(1) LED SIGNAL
(2) AXIS SELECTION
(3) MODE SELECT
(4) SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH
(5) AUXILIARY
(6) EMERGENCY STOP
(7) CYCLE START & FEED HOLD
(8) FEEDRATE ADJUSTMENT
(9) PROGRAM PROTECTION KEY
(10) POWER ON/OFF



2.2 LED SIGNAL

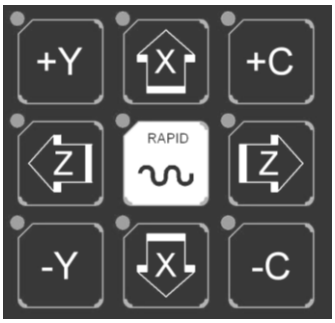
1：X/Y/Z/C AXIS ZRN

2：CONTROLLER ALARM

3：POWER ON, LED ON

4：LUBRICATION INSUFFICIENT, LED ON

2.3 AXIS SELECTION



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

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

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

2.4 MODE SELECT

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

(1). (**EDIT**)

Users can edit new program or modify old one.

(2). (**MEM**)

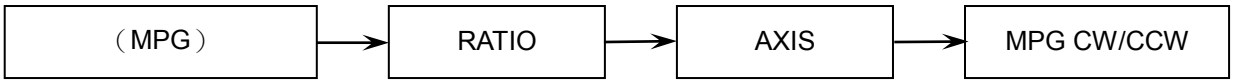
Users can do auto program executing.

(3). (**MDI**)

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

(4). (**MPG**)

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



(5). (**ZRN**)

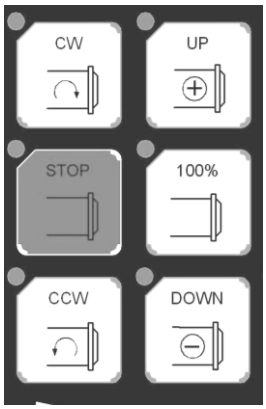
Operating home return of each axis. When changing to this mode, pressing home return direction button of each axis (the same button as JOG). This axis will start home return procedure according to the setting speed of the parameter until reaching DOG. Also, servo axis starts searching for the home position and it will stop when reaching home point. At this time, this axis +ive direction light will be ON. Also, when users switches to HOME, this light will be ON in order to remind users to define machine to complete HOME RETURN motion. Please do the Home Return procedure before doing other part program every time reboot the machine in order to ensure each coordinate's accuracy.

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

(6). （**JOG**）

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

2.5 SPINDLE ROTATION& SPINDLE SPEED ADJUSTMENT SWITCH



Spindle Operation Buttons

Under manual mode,（here means JOG、RAPID、MPG）, spindle's rotation can be controlled by this three buttons.

CW ：Spindle CW

STOP ：Spindle Stop.

CCW ：Spindle CCW

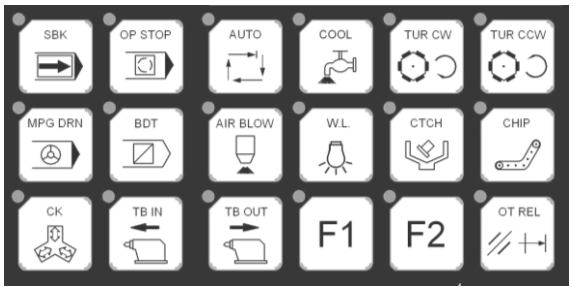
Under this manual mode, no matter the spindle is CW or CCW, the rotation command will be between 0% ~ 120%, one more thing you need to be careful is when changing the rotation direction, you need to press STOP first, otherwise this change will be not effective.

Spindle Speed Adjustment Buttons

When spindle at auto or manual mode, use M3（or M4）to make spindle start to rotate, spindle's exact rotation speed can be adjusted by UP＋ and DOWN－ buttons. Rate can be from 0% to 120%. For example, after giving a command as M3 S1000, and adjust button is at 120%, the exact rotation will be 1200PRM.And if you set it to be 10%, press 100％ button, and the exact rotation speed will become 1000PRM.

2.6 ADDITIONAL FUNCTION SELECTION

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



1. **SBK**

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

2. **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

3. **AUTO**

   System in AUTO mode

4. **COOL**

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

5. **TRU CW**

   This is manual turret CW button, under manual mode,（here means JOG、RAPID、MPG）, press this button（CW LED ON）, turret will rotate CW, until you release this button, turret 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)

6. **TRU CCW**

   This is manual turret CCW button, movement is the same with TRU CW

7. **MPG DRY RUN**

   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

8. BDT

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

9. AIR BLOW

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

10. W.L.

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

11. CTCH

Workpiece conveyor button, press this button to control conveyor.

12. CHIP

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

13. CK

Spindle chuck button, press this button to control chuck.

14. TB IN

Tailstock ejector button, press this button to control tailstock ejector IN.

15. TB OUT

Tailstock ejector button, press this button to control tailstock ejector IN.

16. F1，F2

F1：This buttons are for machine makers to define.

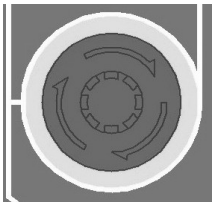
F2：This buttons are for machine makers to define.

17. 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 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

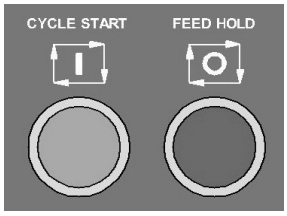
2.7 EMG-STOP



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.

2.8 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, press “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.

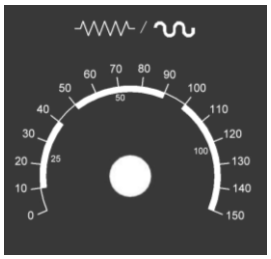
b. **Manual executing in MDI mode**

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

2.9 Feedrate/Rapid Rate Adjusting Button

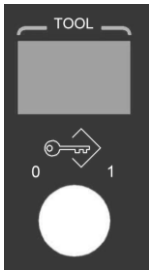


This is the feedrate/rapid rate adjusting button

FEEDRATE：

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

2.10 PROGRAM PROTECTION LOCK & TOOL COUNTER DISPLAY



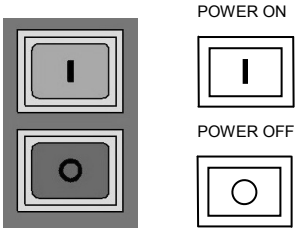
Users can use this program protection lock to lock program edit mode, after locking users can only start program processing via file management, but users can not modify this program.

0：PROGRAM PROTECTION OFF

1：PROGRAM PROTECTION ON

Tool Counter Display: To display tool number while there is tool calling or switching.

2.11 POWER ON/OFF



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

3 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

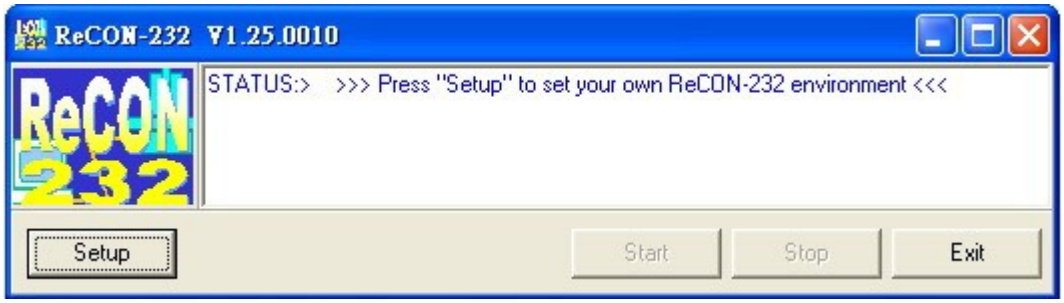
NC(DB9PIN Jack)	PC end(DB25PIN Jack)
pin2(RD)	pin3(TD)
pin3(TD)	pin2(RD)
pin5(SG)	pin7(SG)

3.2 PC end Installation (Use LNC ReCON 232 Software)

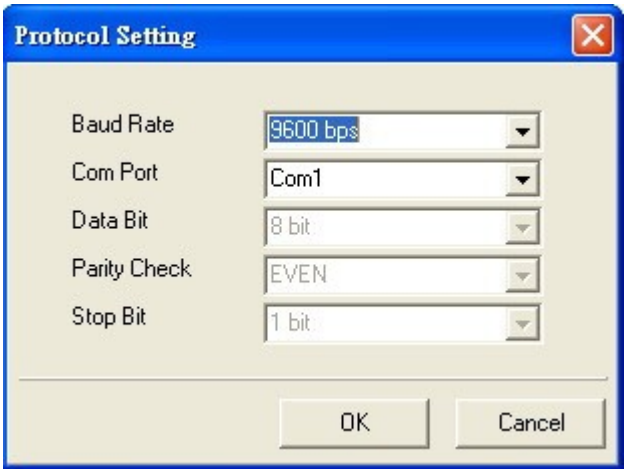
1. Clicks CD> Software\ ReCON-232 V1.25.0010, or double click setup.exe icon, the driver will start to install automatically.
2. According to Wizard's direction to follow next step or click the items to continue.
3. After installation, the desktop will have an icon of ReCON – 232, also at the start> 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) 0:2400 1:4800 2:96□8 3:19200 4:38400
P1813	Transmission file bit(0:7 1:8 bit)	1	
P1814	Transmission stop bit (0:1 1:2)	0	
P1815	Transmission parity bit (0:N 1:E 2:O)	1	
P1816	Transmission mode (0: terminating machine 1:host)	0	
P1817	Transmission code (0ASCII 1EIA 2ISO)	0	

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

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

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

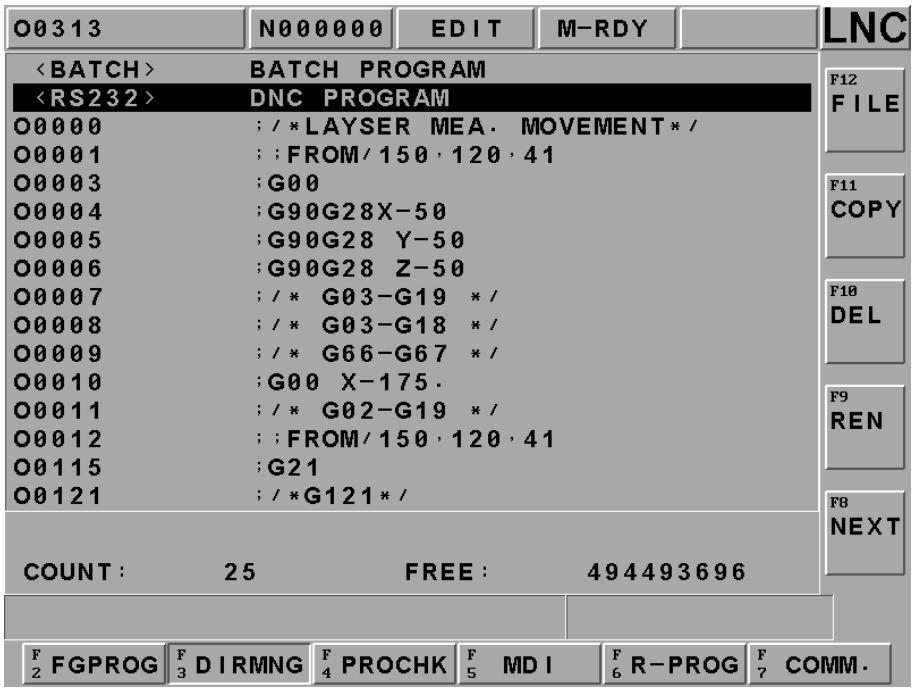
However, DNC's old processing while sending function was still remained (this is called external DNC functions); it is also a very useful tool.

Steps are as below:

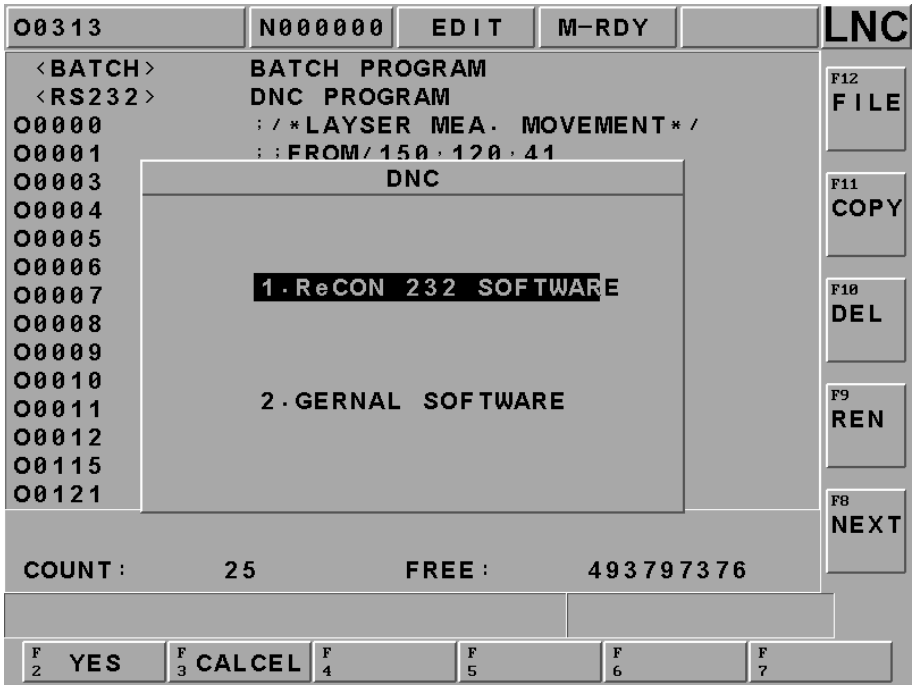
1.Edit RS232 setting at controller and PC end, and make sure they are coordinate.

2.Use PROG group, change to EDIT mode.

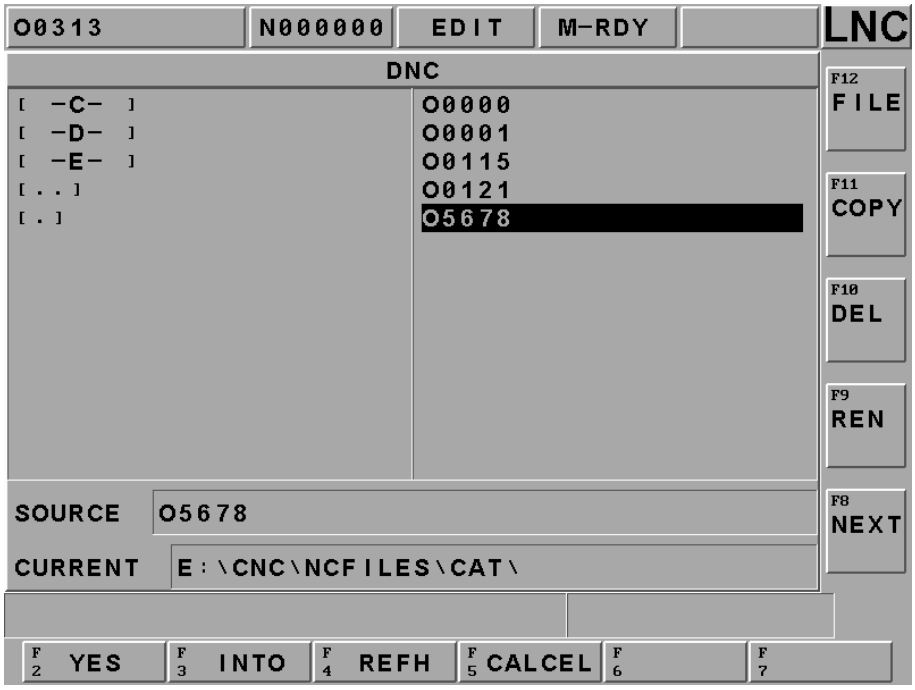
3.Go to 【DIRMNG】, select 《RS232》DNC PROGRAM and press OK.



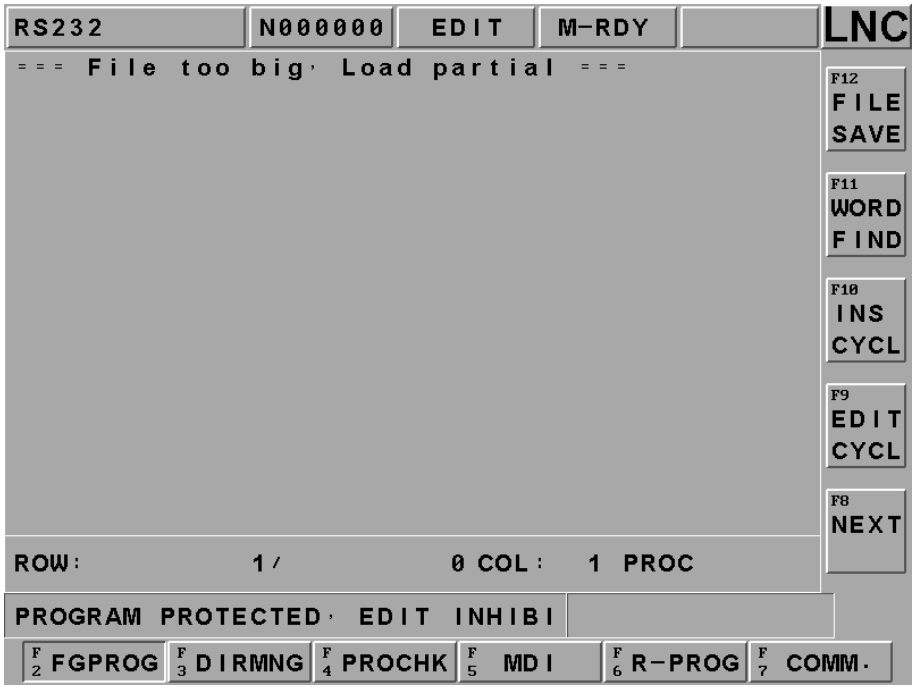
4. Select 1. LNC ReCON 232 software to upload.



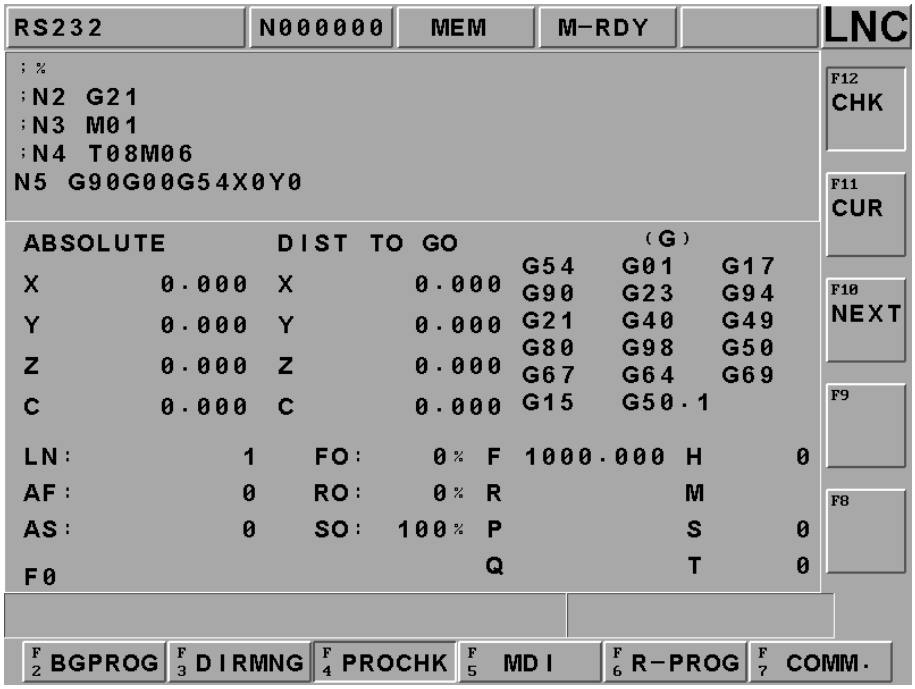
5.Go to PC end disk and select DNC file, press【YES】



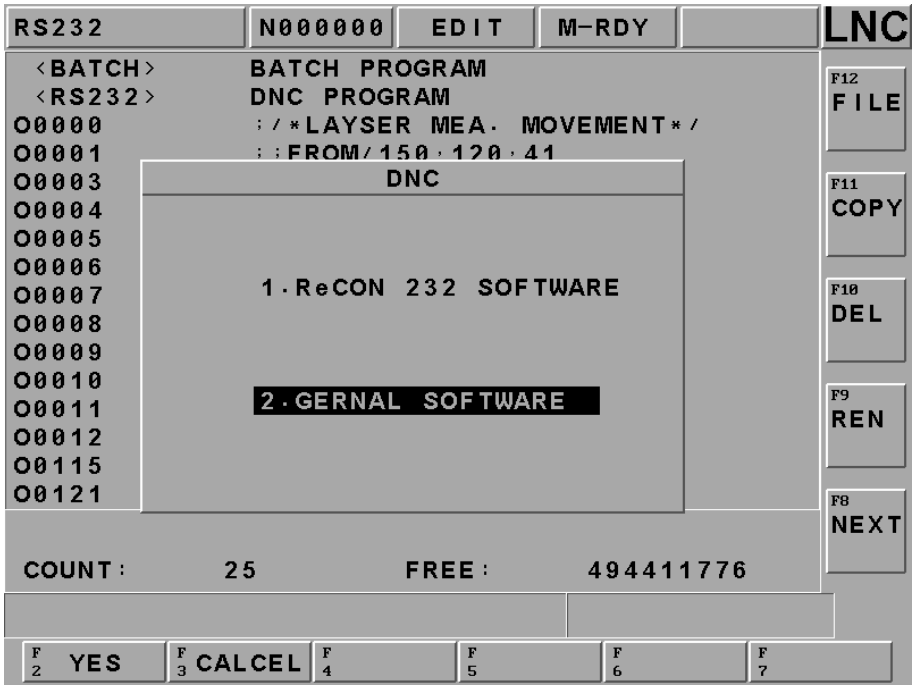
6.Now the file will turn to be RS232 and change mode to MEM.



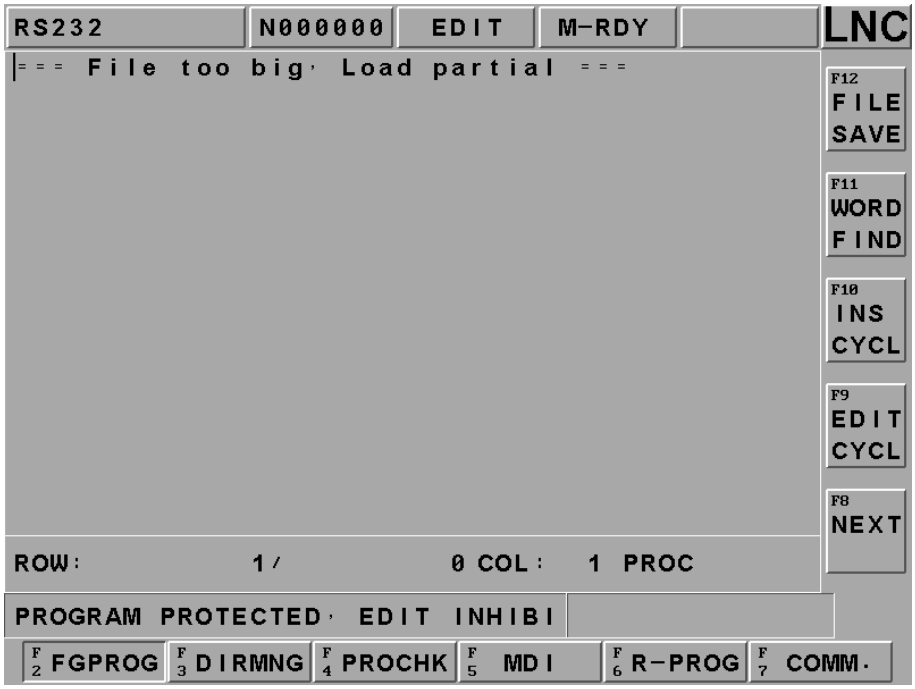
7.Press【CYCLE START】, start to do DNC.



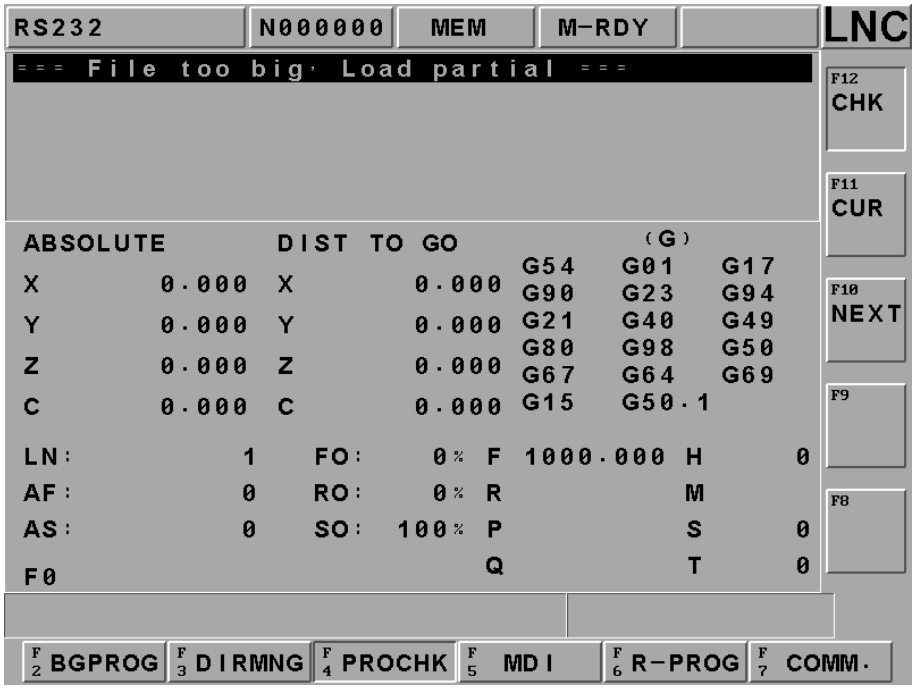
8.Select 2. General software to upload. (Use SM8 sample)



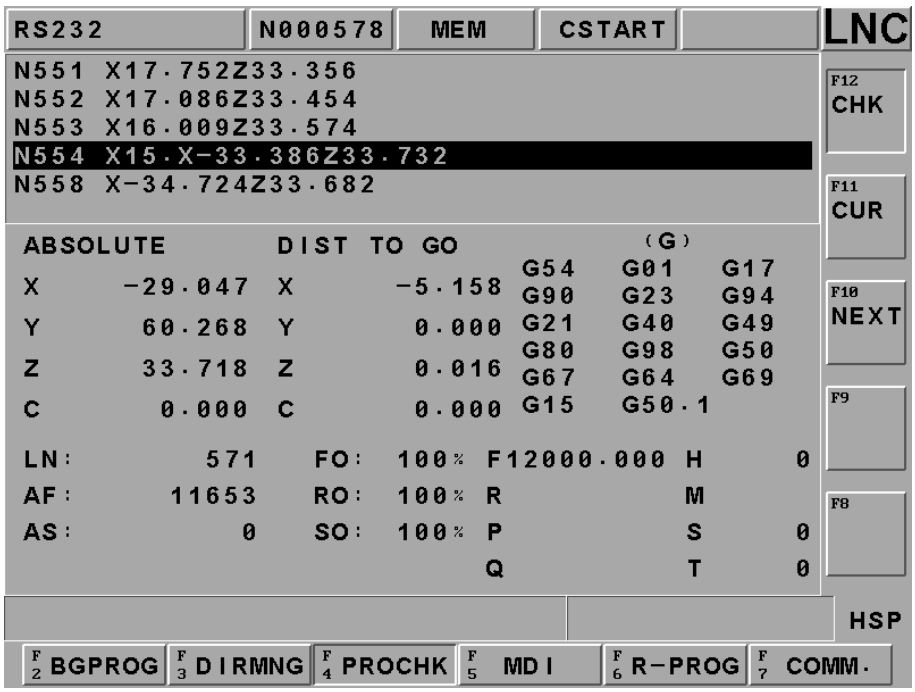
9.Now the file will turn to be RS232 and change mode to MEM.



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

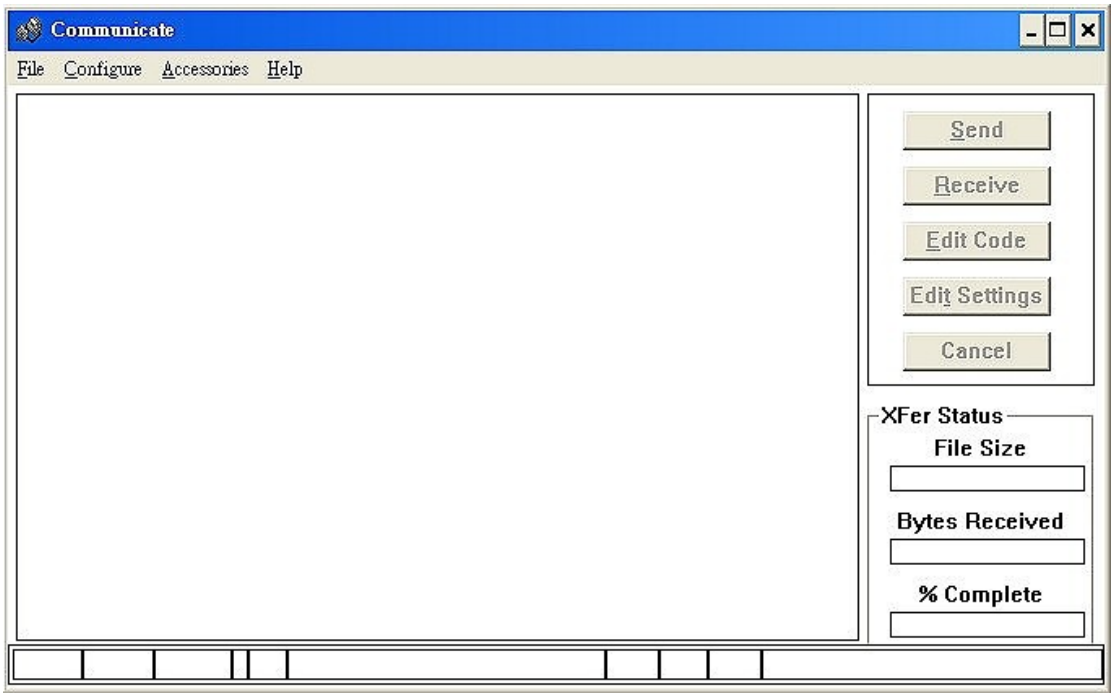


11.Before press Send button at PC end, please make sure NC end is at stand by>Cycle-Start mode. In other words, receiver must be prepared, so the sender can send out files, or after sending, press Cycle-Start to do DNC.



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.

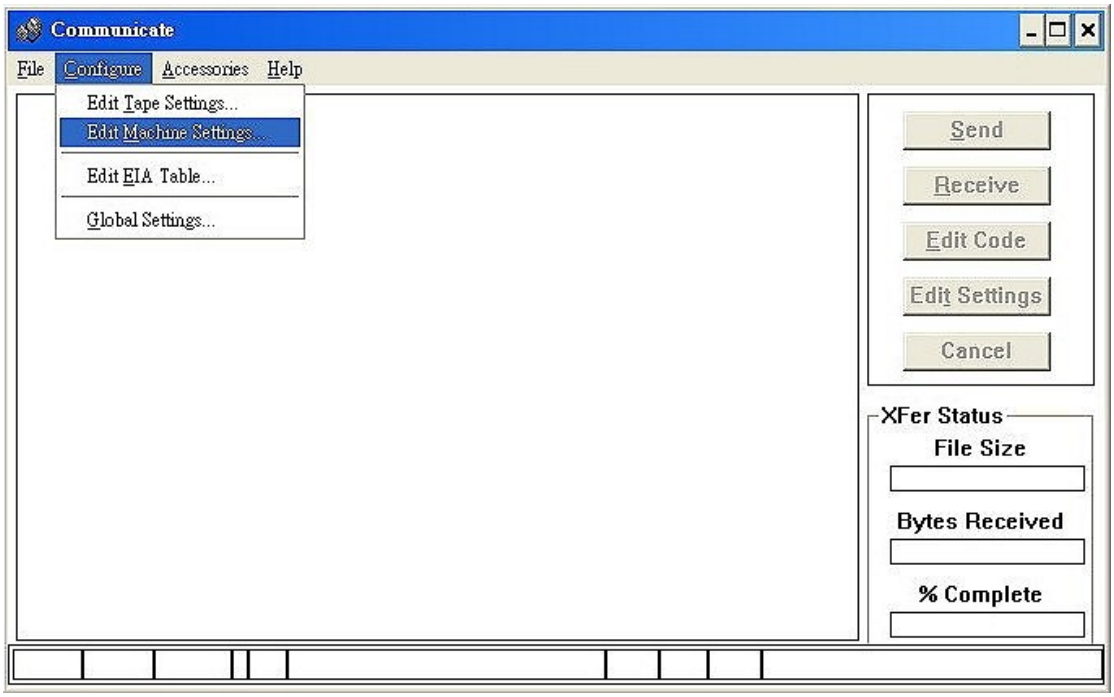
Select the file you want to transfer.

Press Send button, NC end will start to receive.

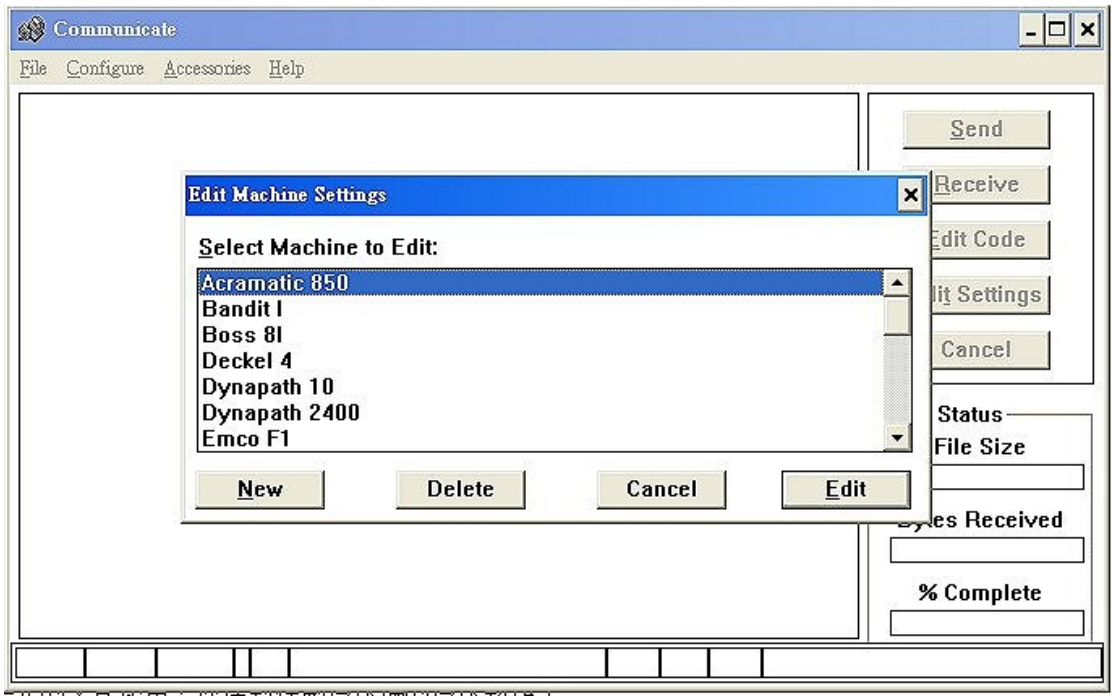
(For safety, we recommend that you make single block enable and minimize override at NC end)

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

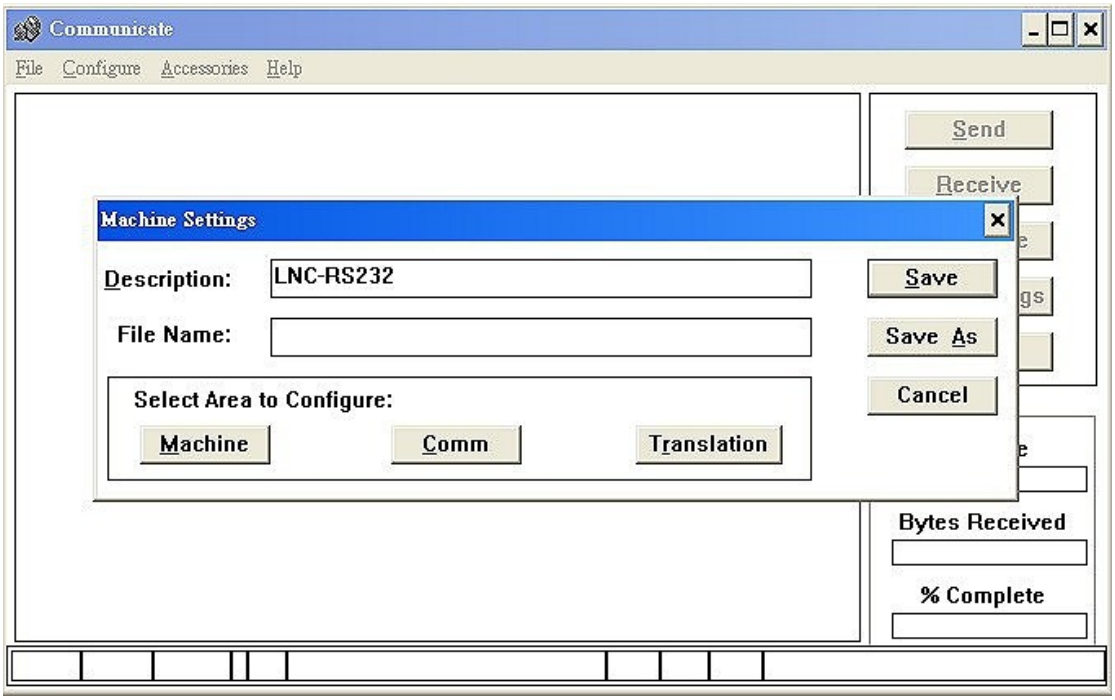
Below is the window of PC end's communication setting page.Go to [Configure]> [Edit Machine Settings] choose a correct setting to connect with LNC.


Communicate
File
Configure
Accessories
Help
Edit Tape Settings...
Edit Machine Settings...
Edit EIA Table...
Global Settings...
Send
Receive
Edit Code
Edit Settings
Cancel
XFer Status
File Size
Bytes Received
% Complete

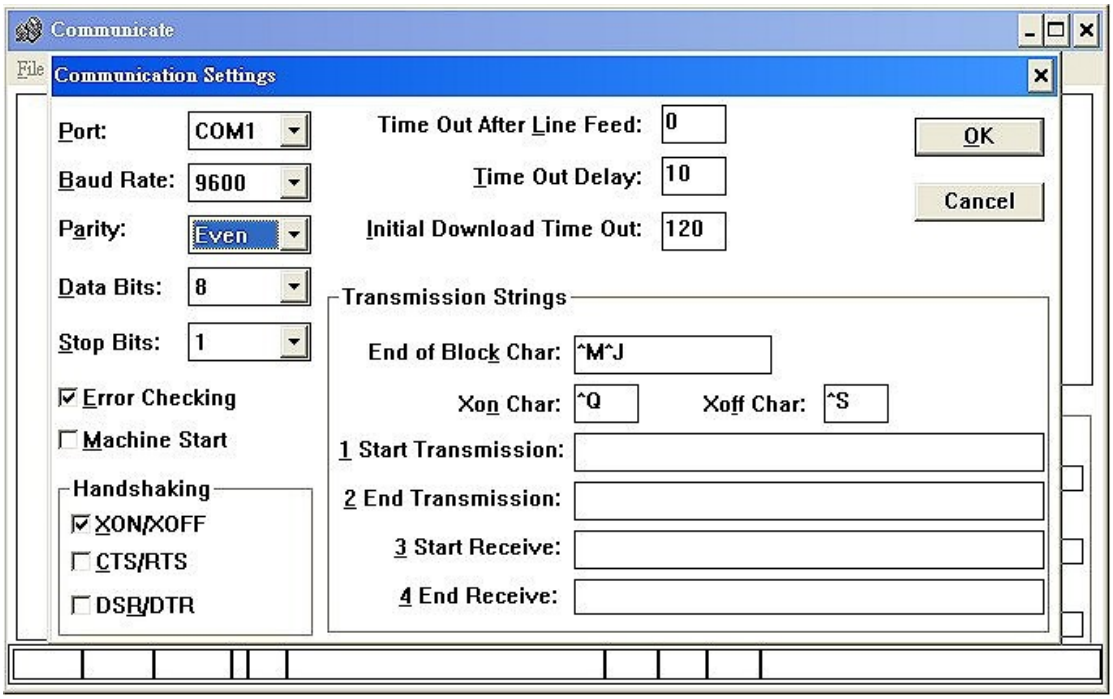
Press [NEW]


Communicate
File
Configure
Accessories
Help
Send
Receive
Edit Code
Cancel
Edit Machine Settings
Select Machine to Edit:
Acramatic 850
Bandit I
Boss 8I
Deckel 4
Dynapath 10
Dynapath 2400
Emco F1
New
Delete
Cancel
Edit
Status
File Size
% Complete

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



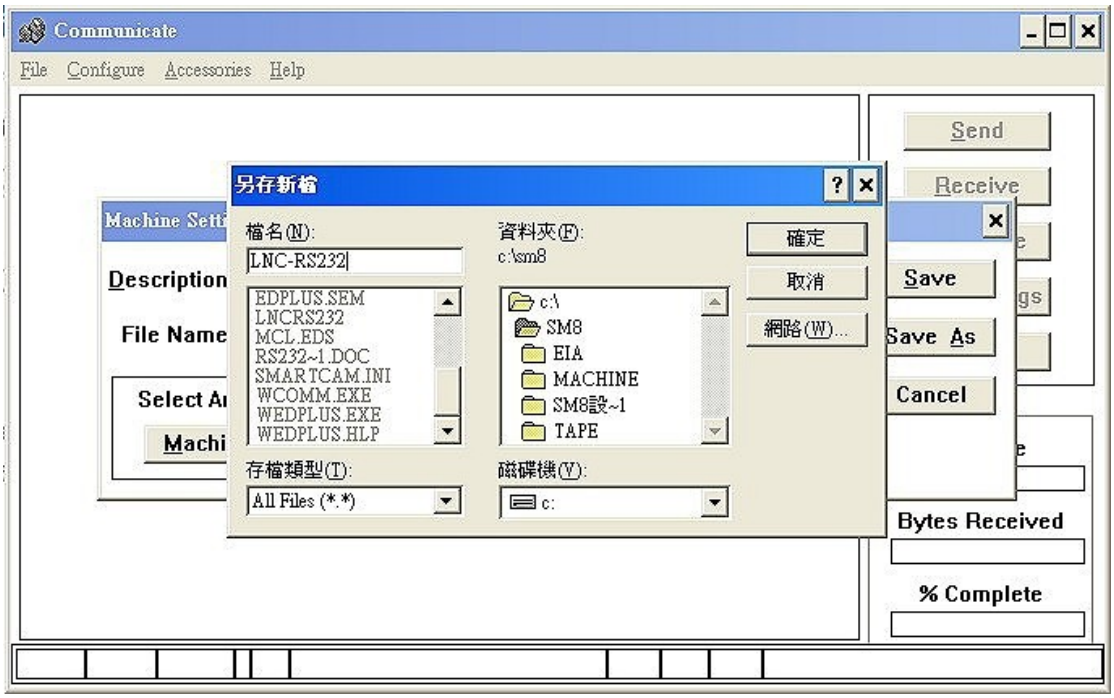
Go to [Comm]>Communication setting, other tables can be the same.



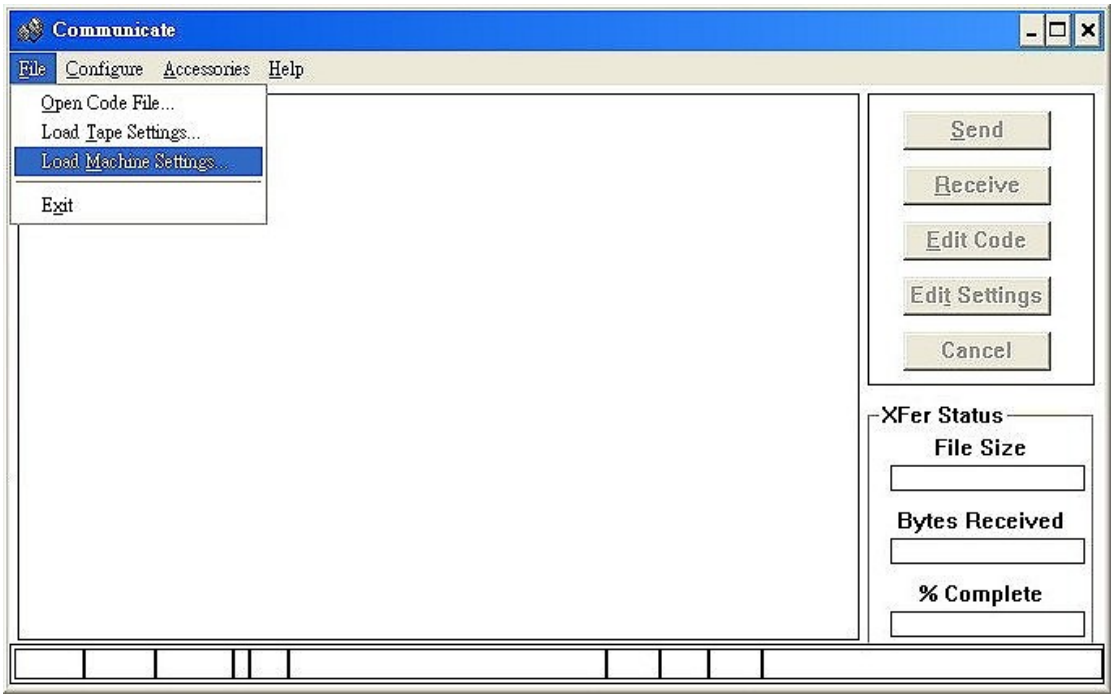
Please click XON/XOFF at HandShaking line; this is a transmission control method which conducts bits by software. If receiver wants to stop transmission from sender, receiver will send a Xon Char (ASCII No.19 bit： ^Q) to sender and stop. But if receiver wants to recover transmission, he will send a Xoff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

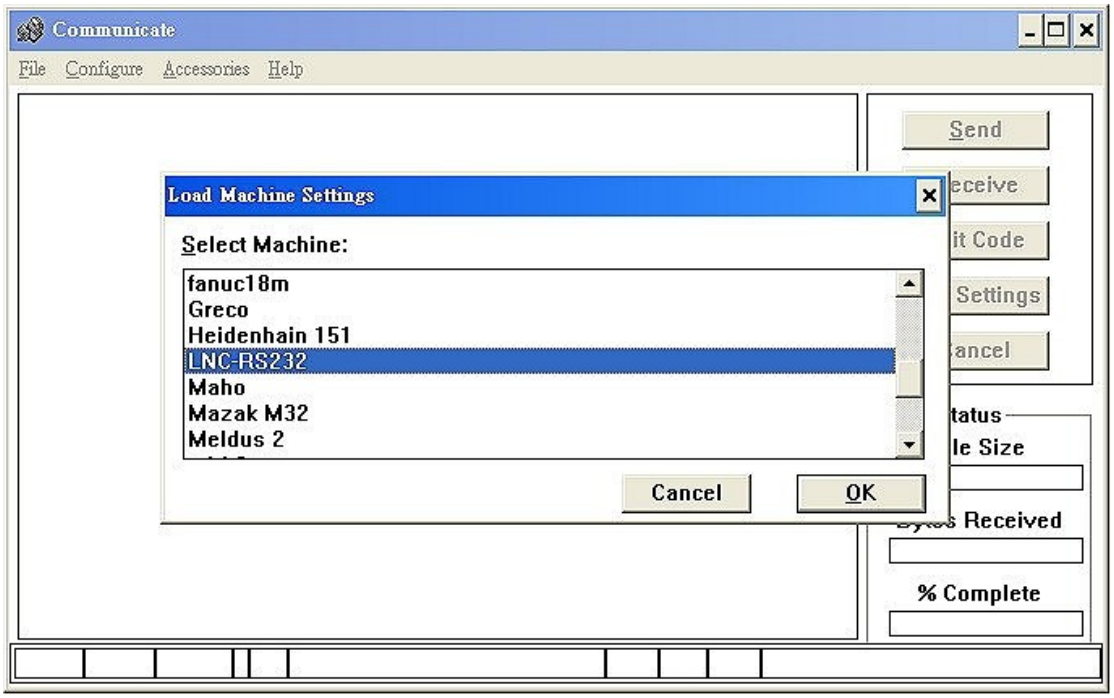
You only need to edit machine setting at the first time, after that all you need to do is to select the same setting.



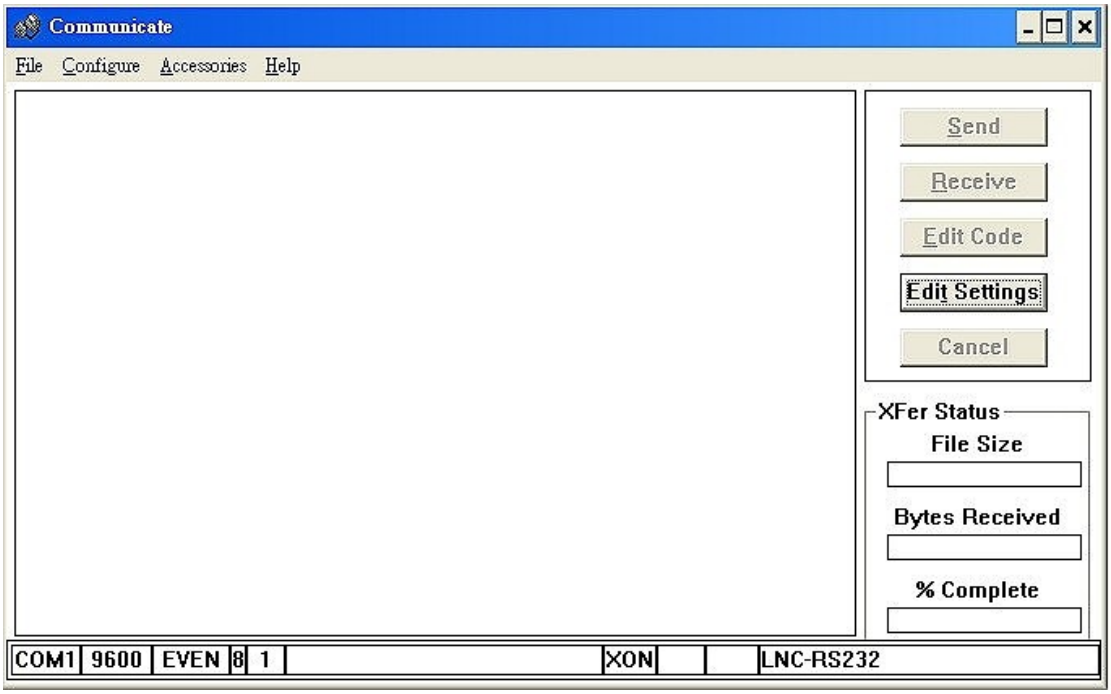
Go to [File]> [Edit Machine Settings]



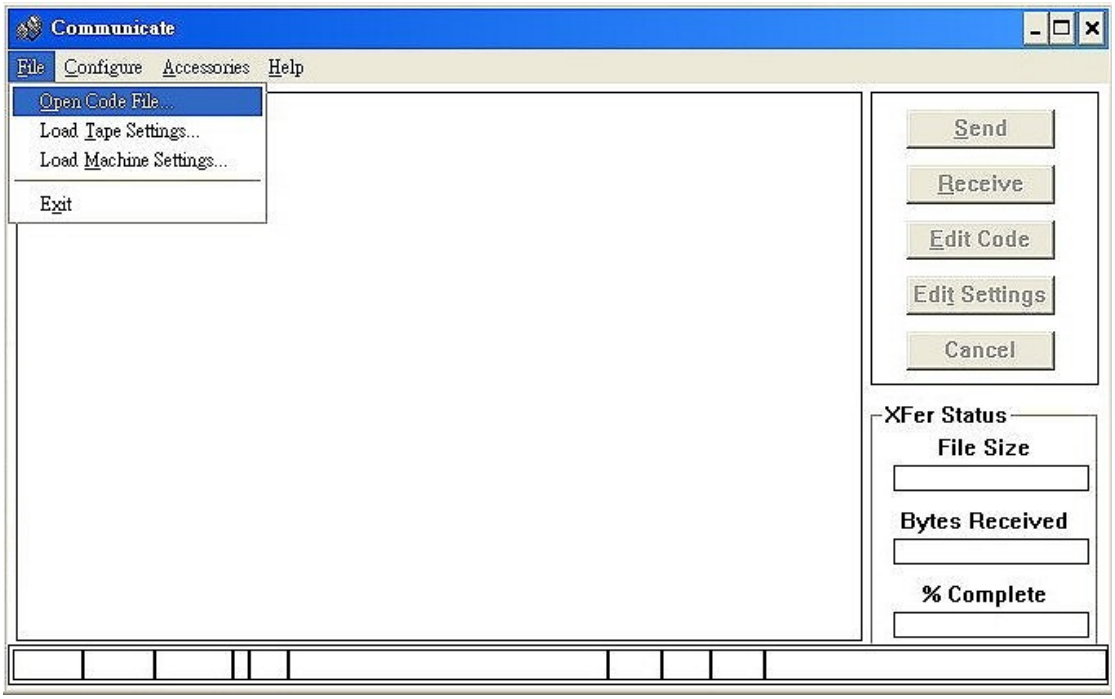
Select the setting of [LNC-RS232] that you just made.



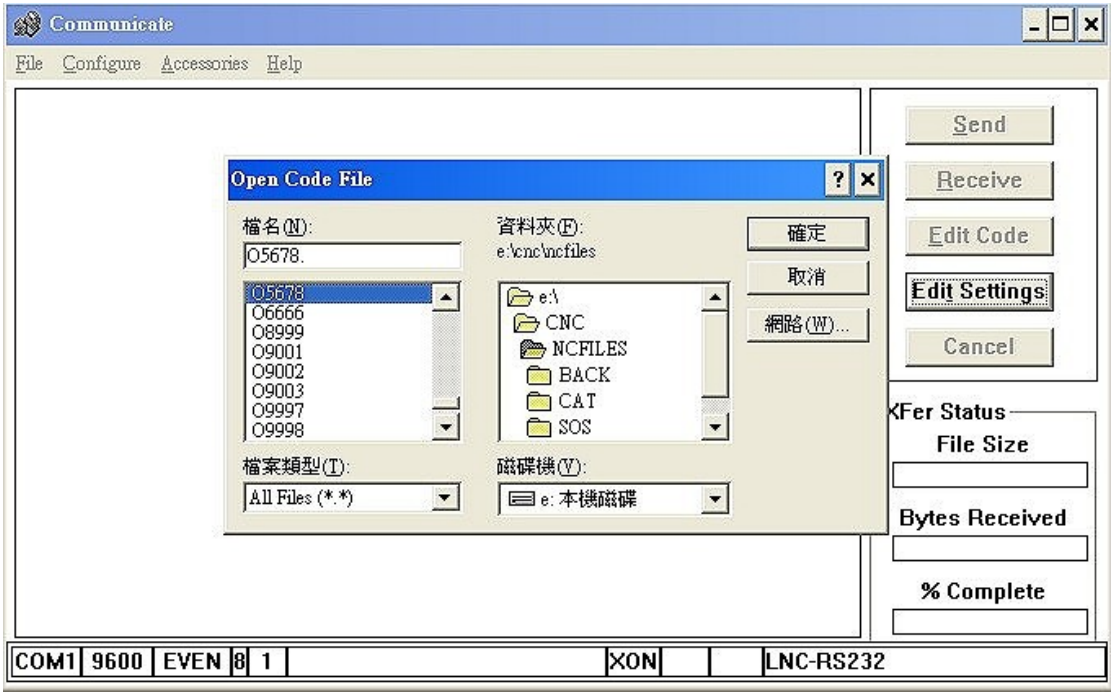
At this time, [Edit Settings] button will pop out, you will see COM1，96□8，EVEN，8，1 at down side.



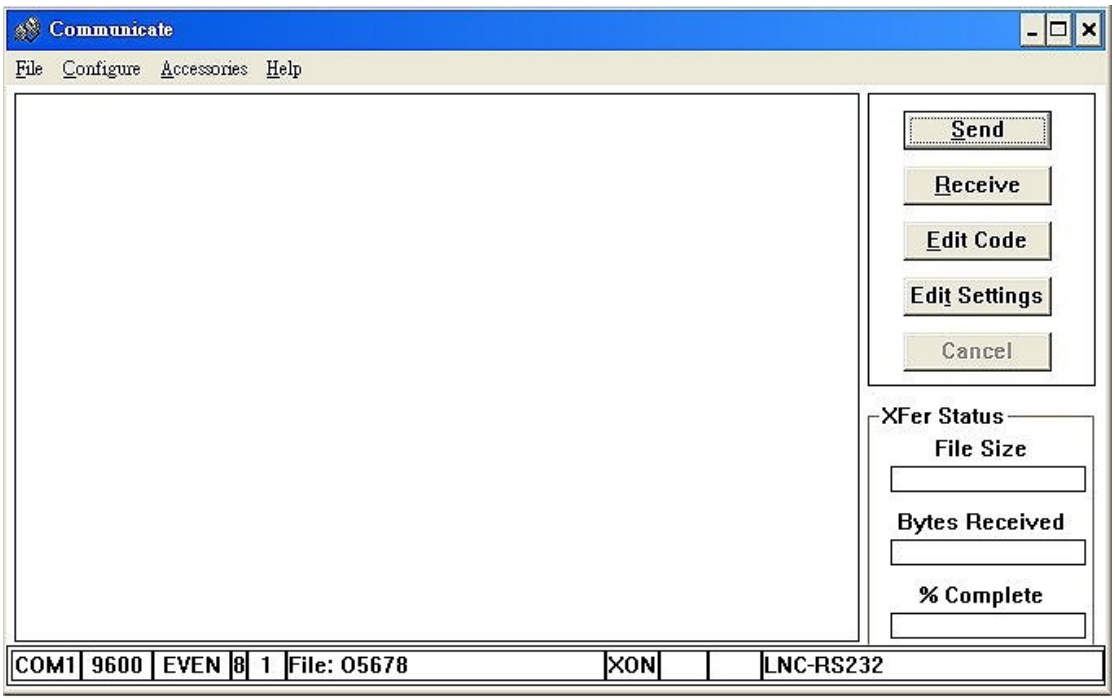
Press file and go to [Edit]> [Open Code File]



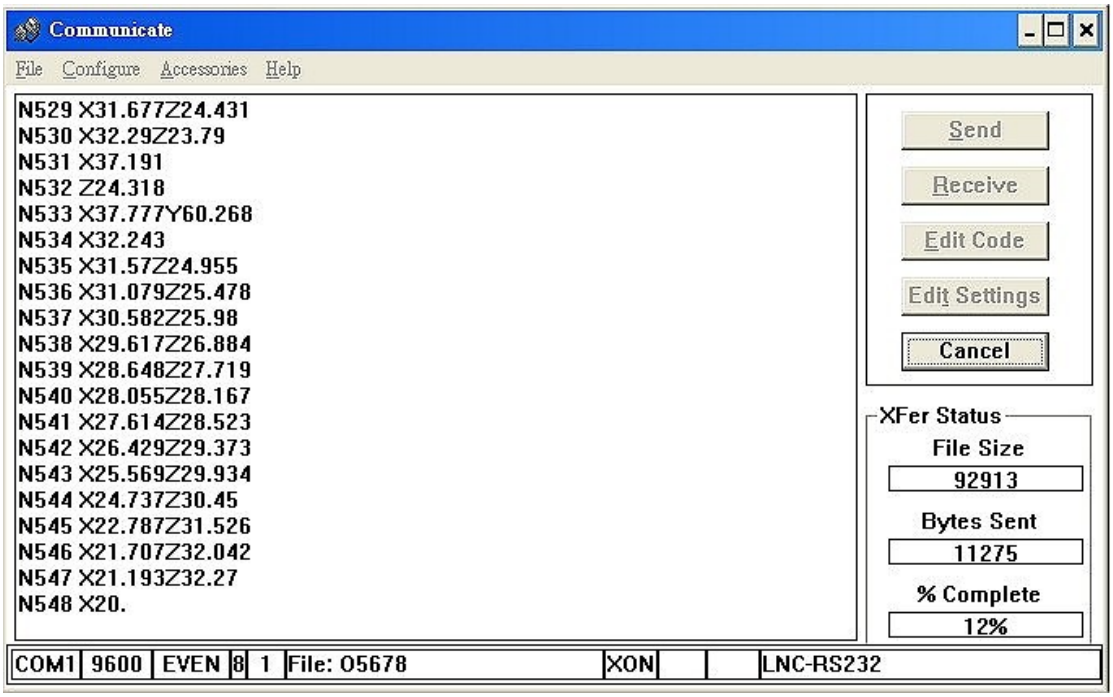
Select the file that you want to do DNC transfer and press OK.



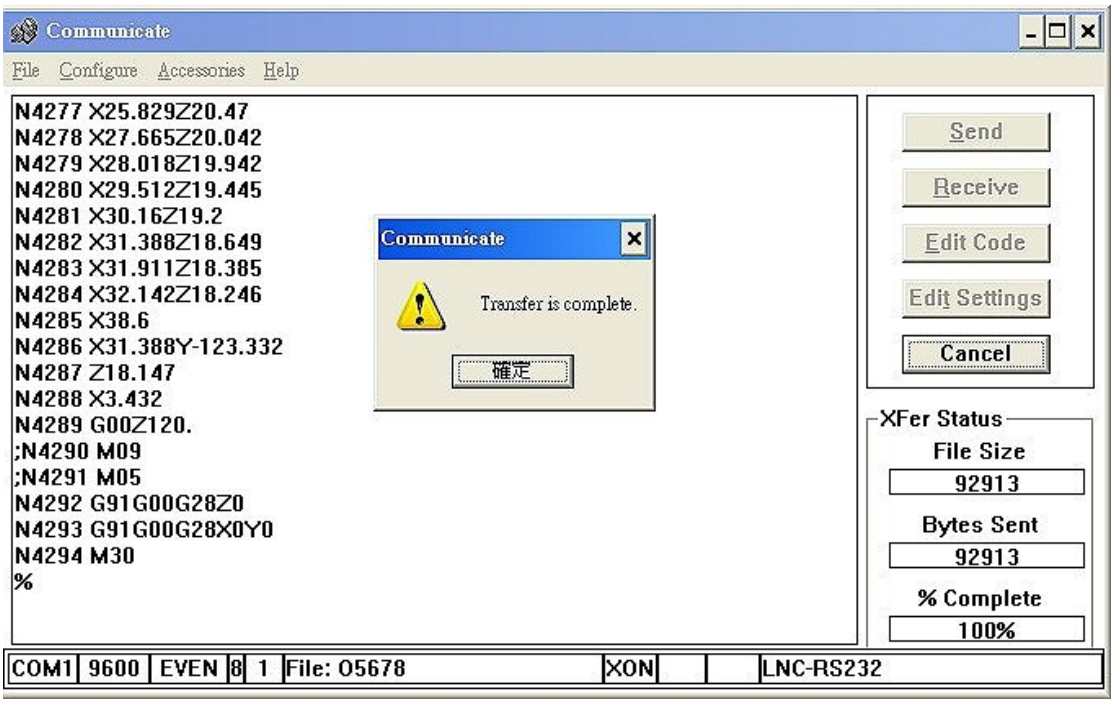
Press [Send]



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.



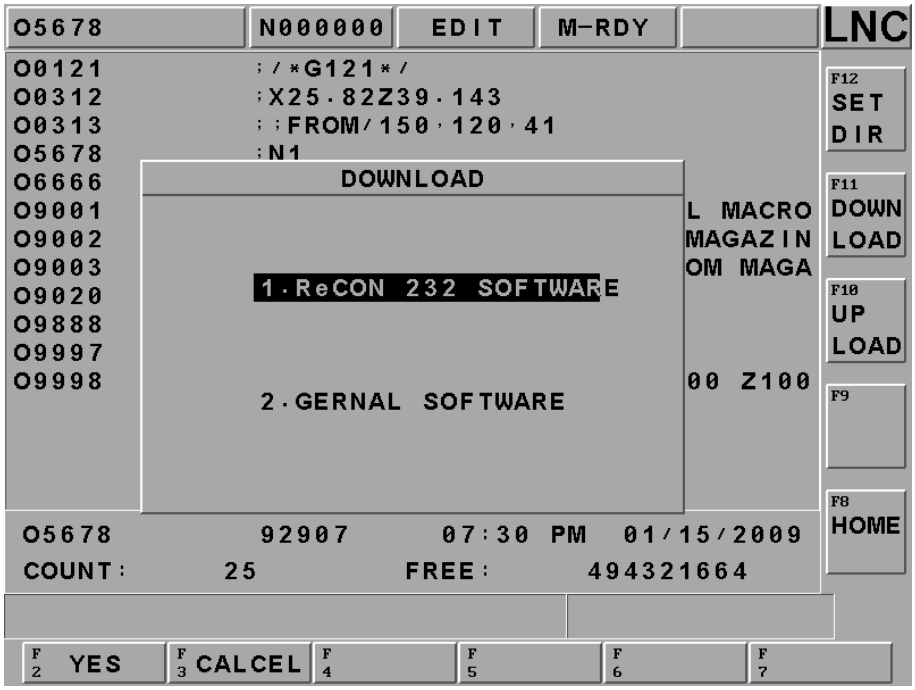
3.4 Communication （RS232）-【 DNC file upload and download functions 】

3.4.1 Path Program Upload & save (using ReCON software)

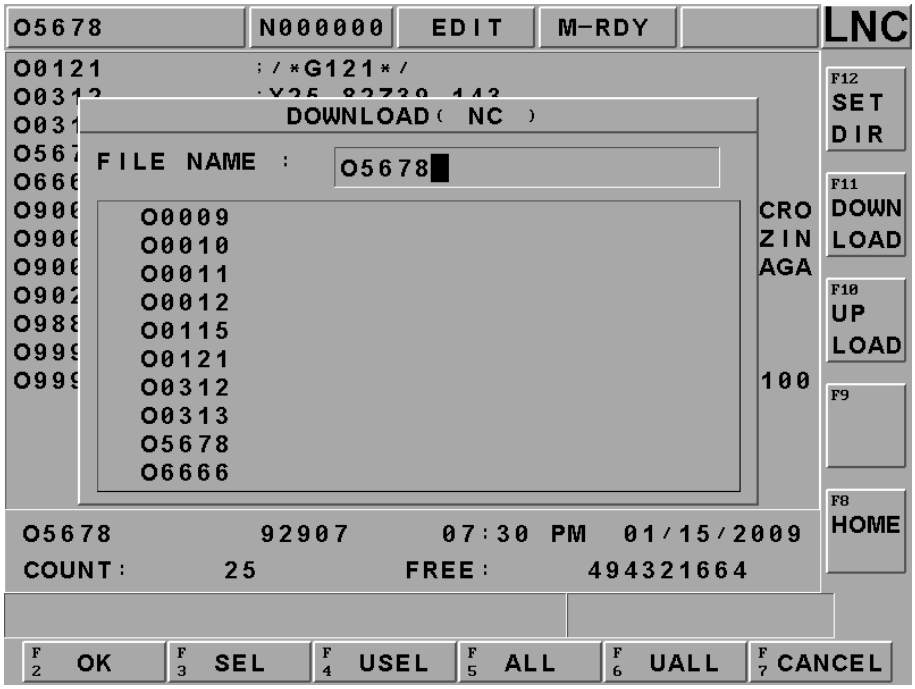
This is a device that could make you transfer the file from PC end to NC end and also save the file at NC end(H.DD/Disk ON Chip/Disk ON Modular), Details as below：

Operation for NC&PC end

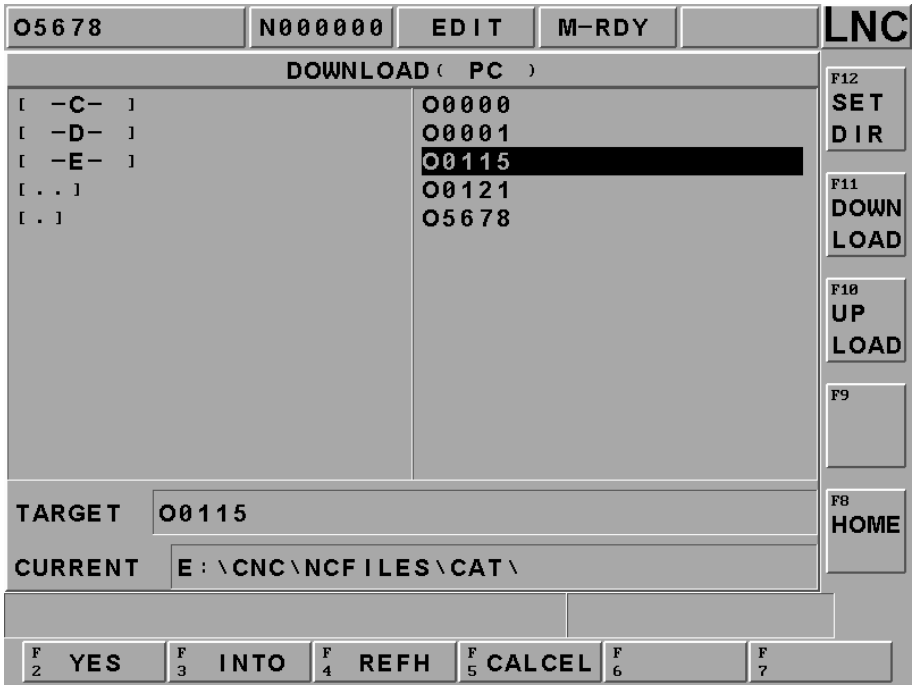
1. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
2. Use PROG group, change to EDIT mode.
3. Go to 【DIRMNG】 and choose the file that you want to transfer to PC end and press [DOWN LOAD]
   Select 1. LNC ReCON 232 software to upload.



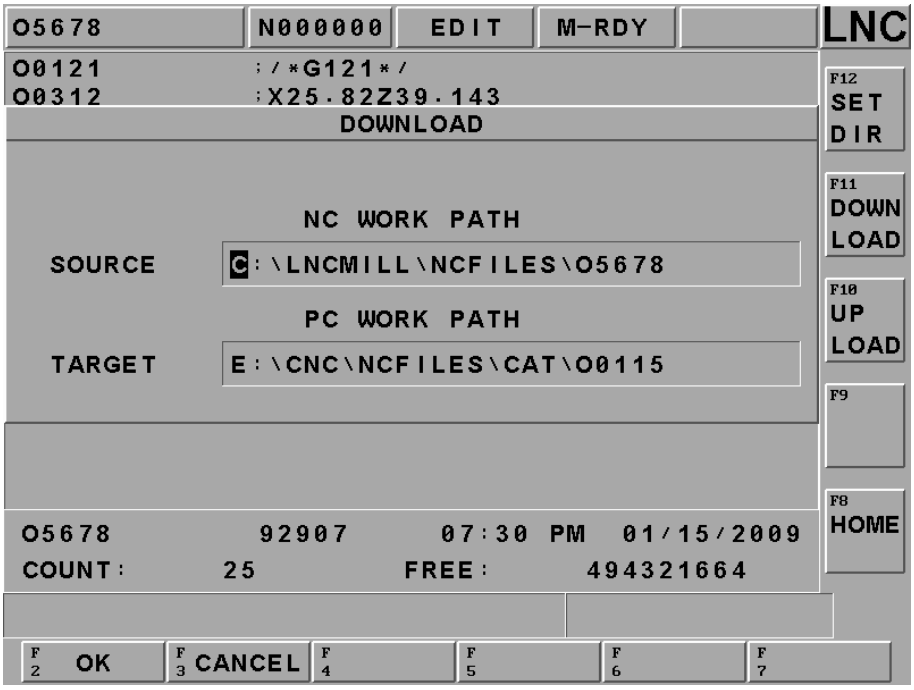
4. Choose the file O5678 you want to download from controller and press OK



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



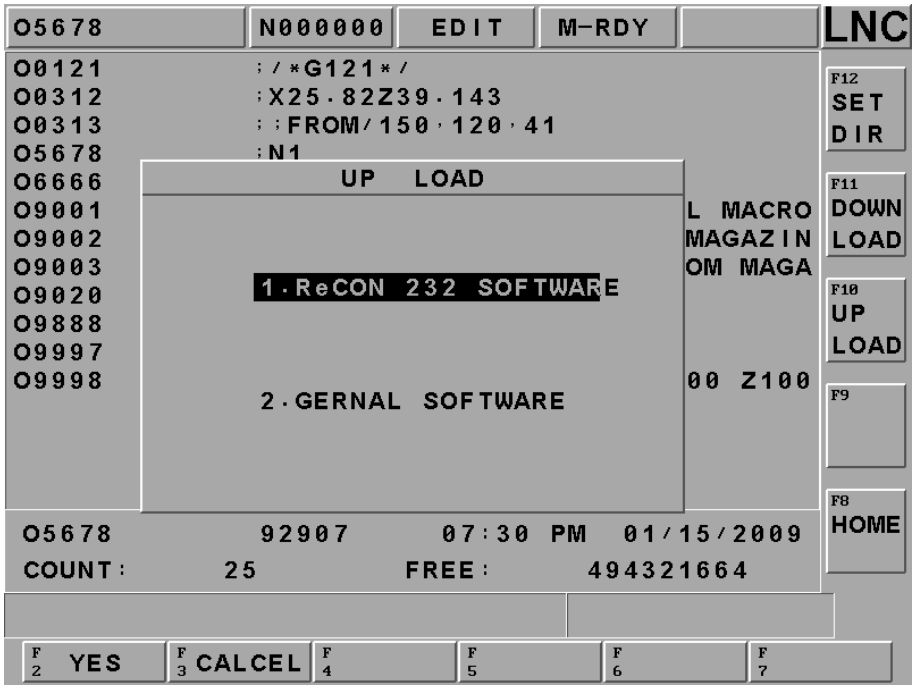
6. Check if file name is ok at NC and PC end, press OK to upload.



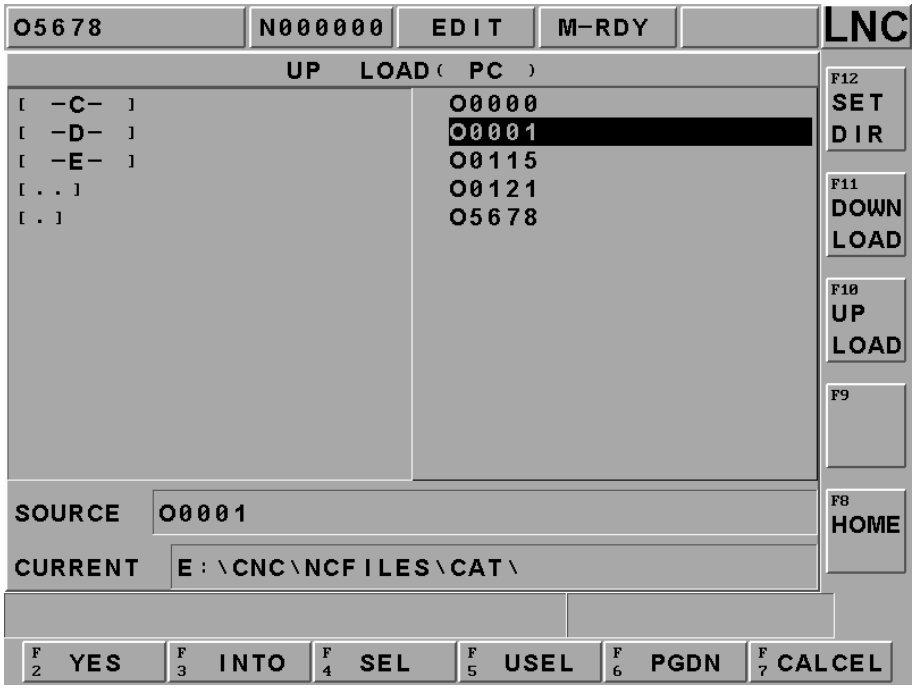
3.4.2 Path Program Upload & save (using ReCON software)

Operation Steps for NC&PC end

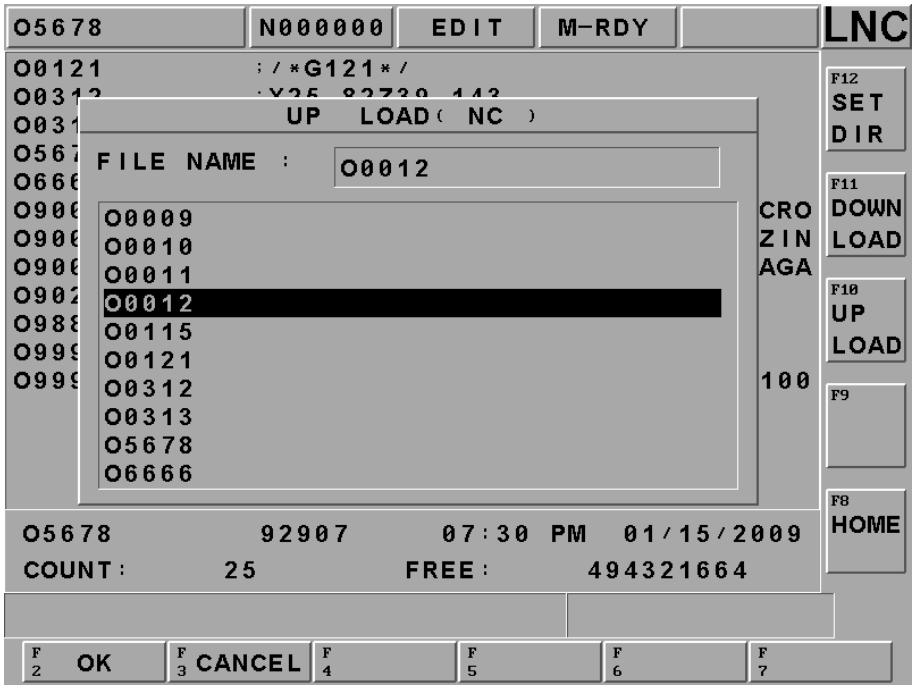
1. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
2. Use PROG group, change to EDIT mode.
3. Go to [DIRMNG] and choose the file that you want to transfer to PC end and press [UP LOAD].
   Select 1. LNC ReCON 232 software to upload.



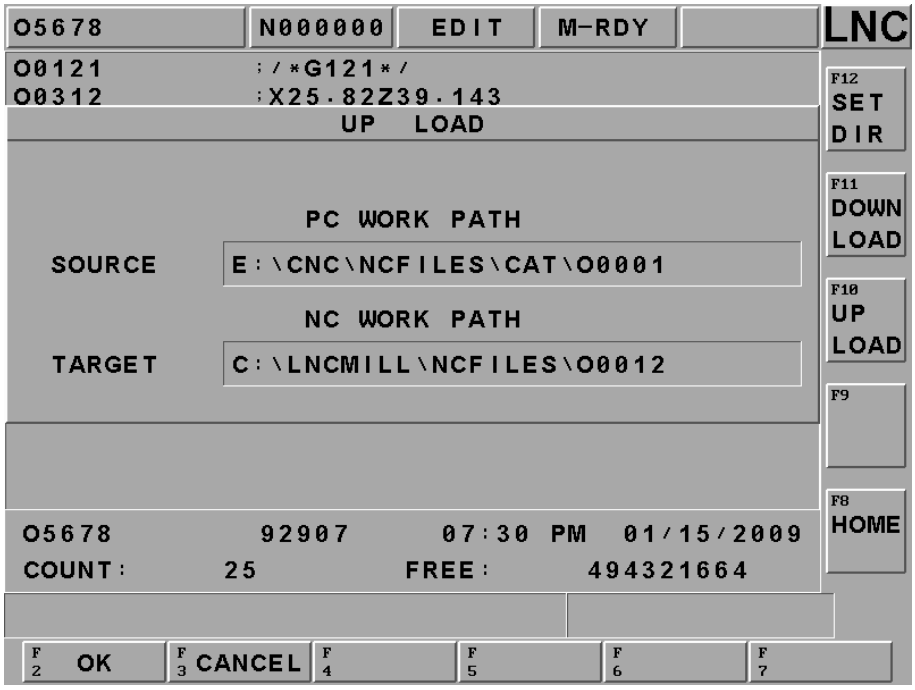
4. Select save path and upload name at PC end.



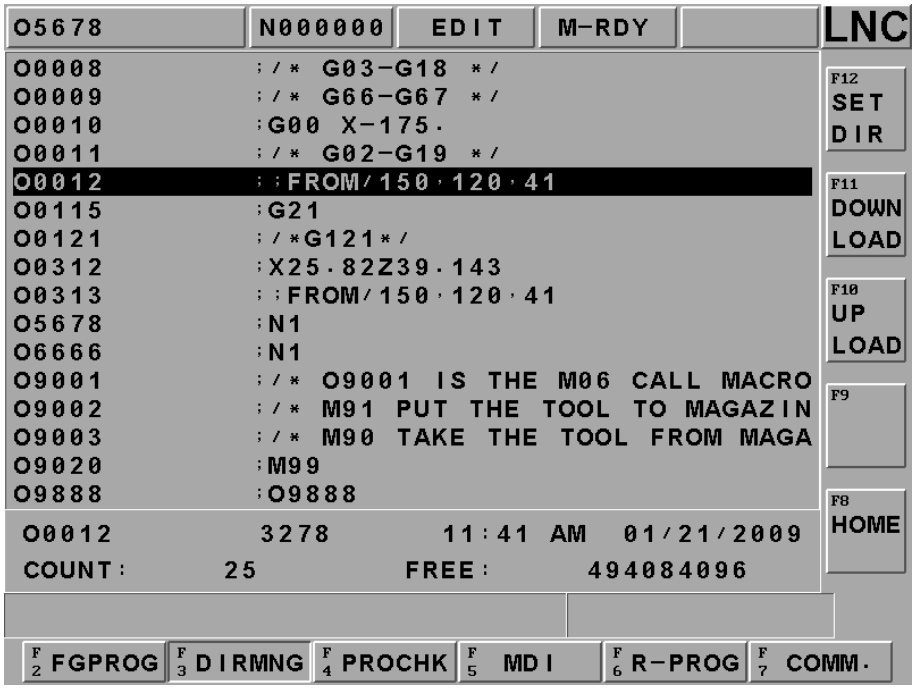
5. At NC end, you could input new name or save at the same file.



6. Confirm and press [OK]



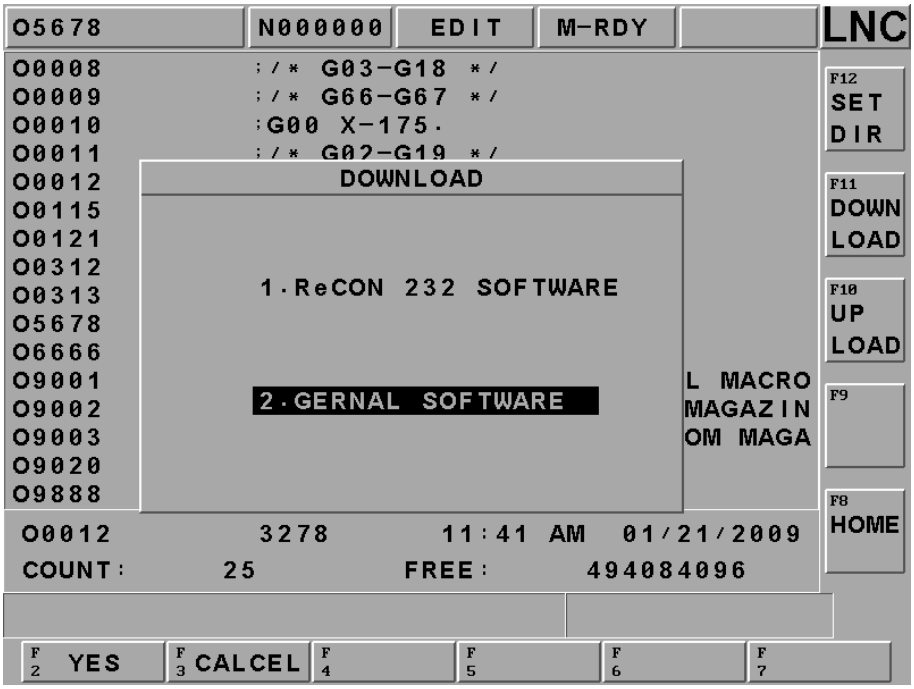
7. Upload file successful.



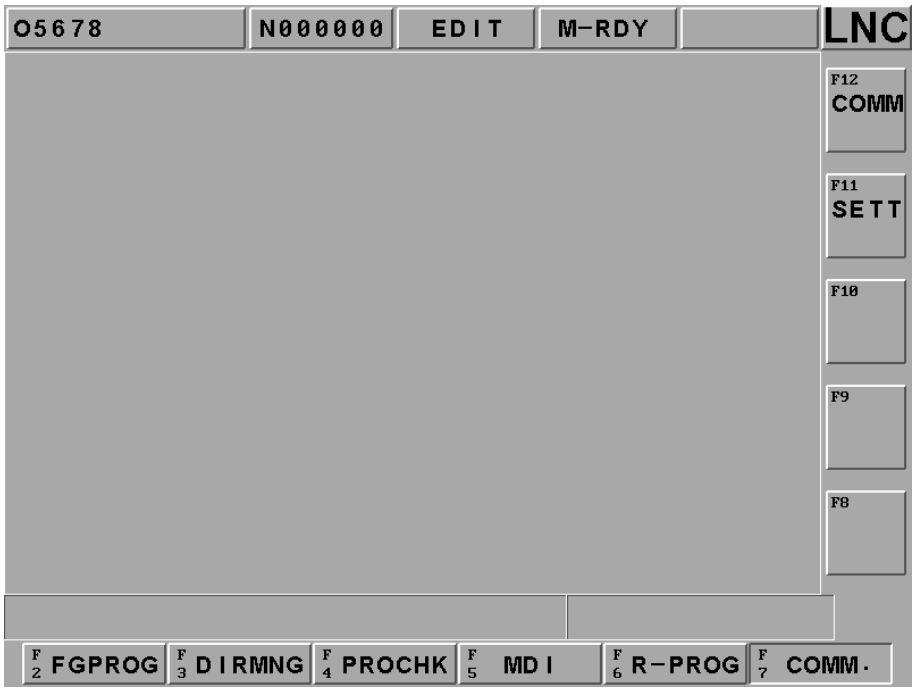
3.4.3 Path Program Upload & Save (using general software)

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].

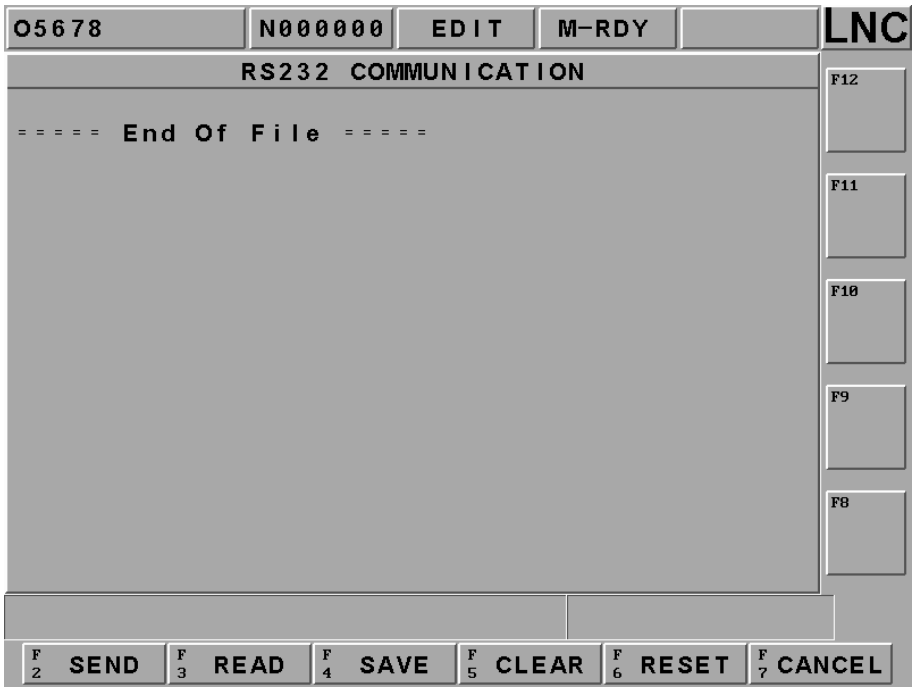
   Select 1. General software to upload.



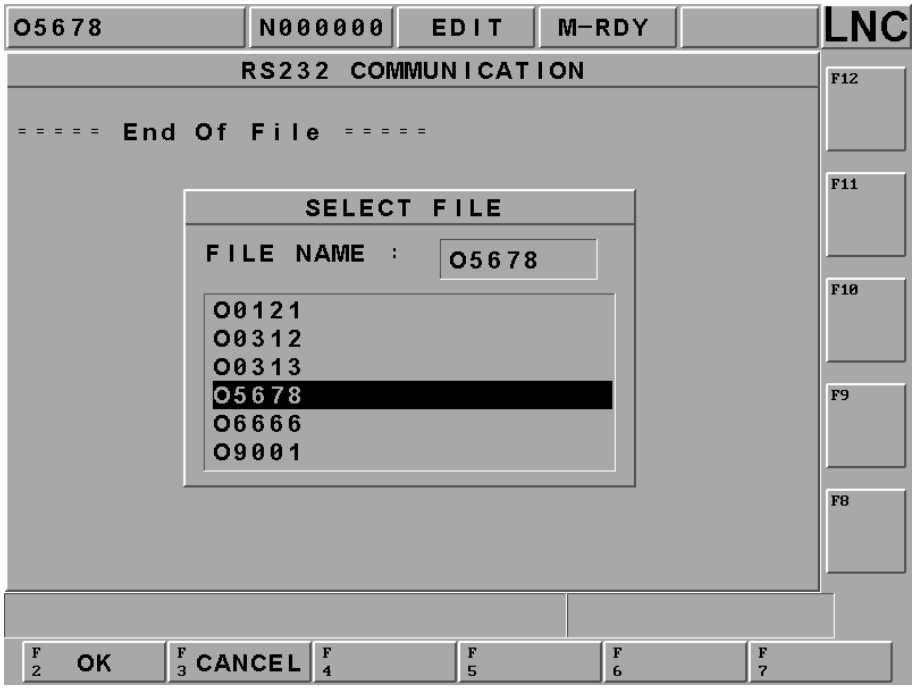
4. Go to Communication page and press [COMM] at the down right side.



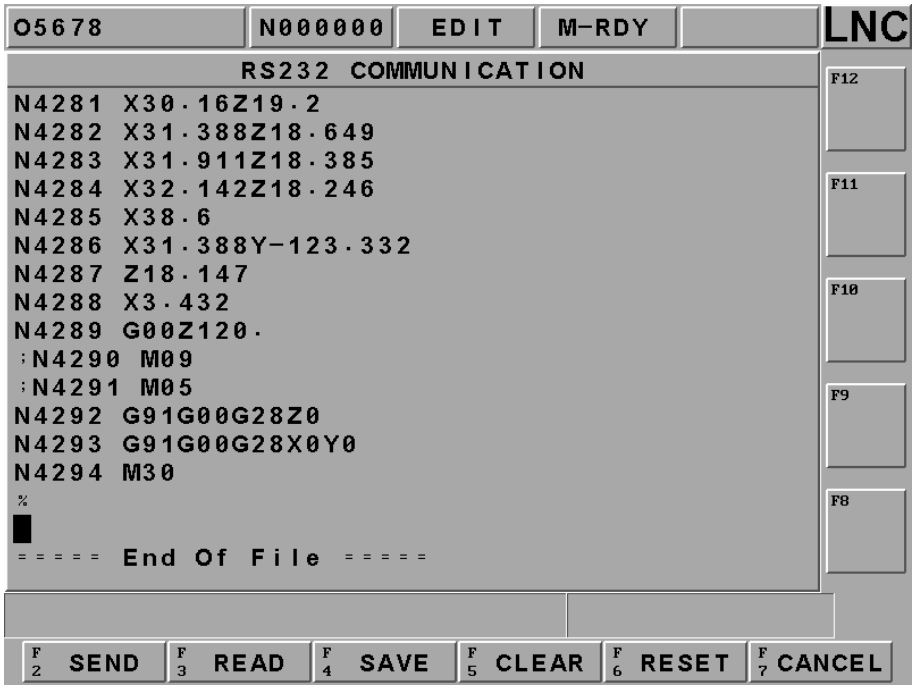
5. Go to RS232 COMMUNICATION file download page.



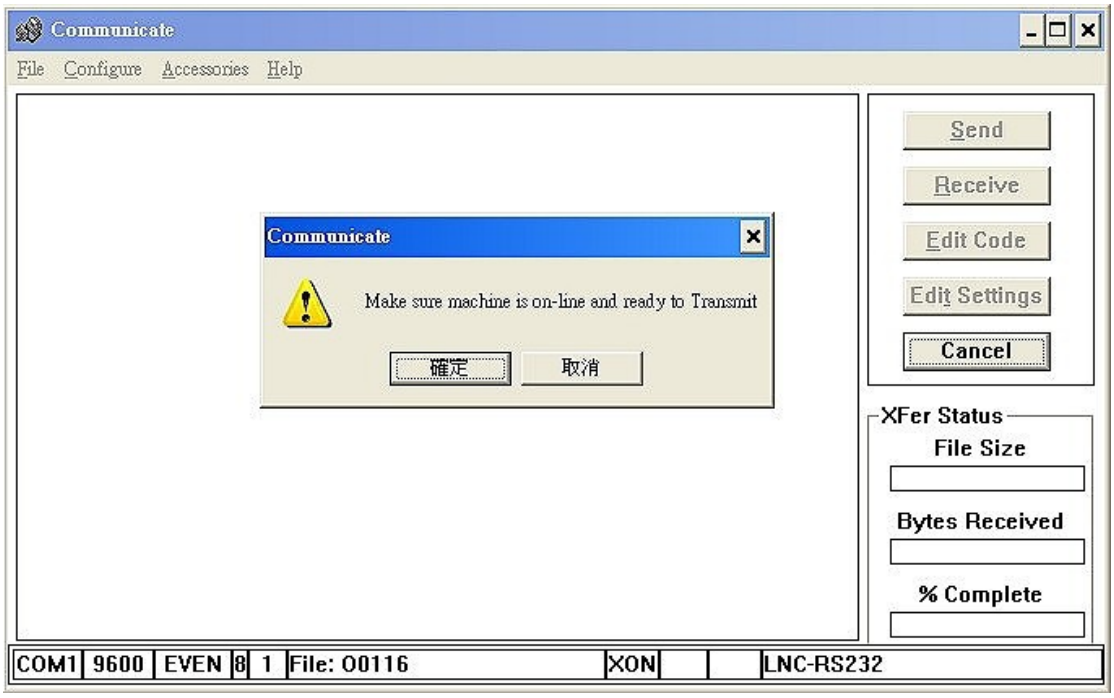
6. Press [SEND], start to transfer files from NC end to PC end, select name and press [OK]



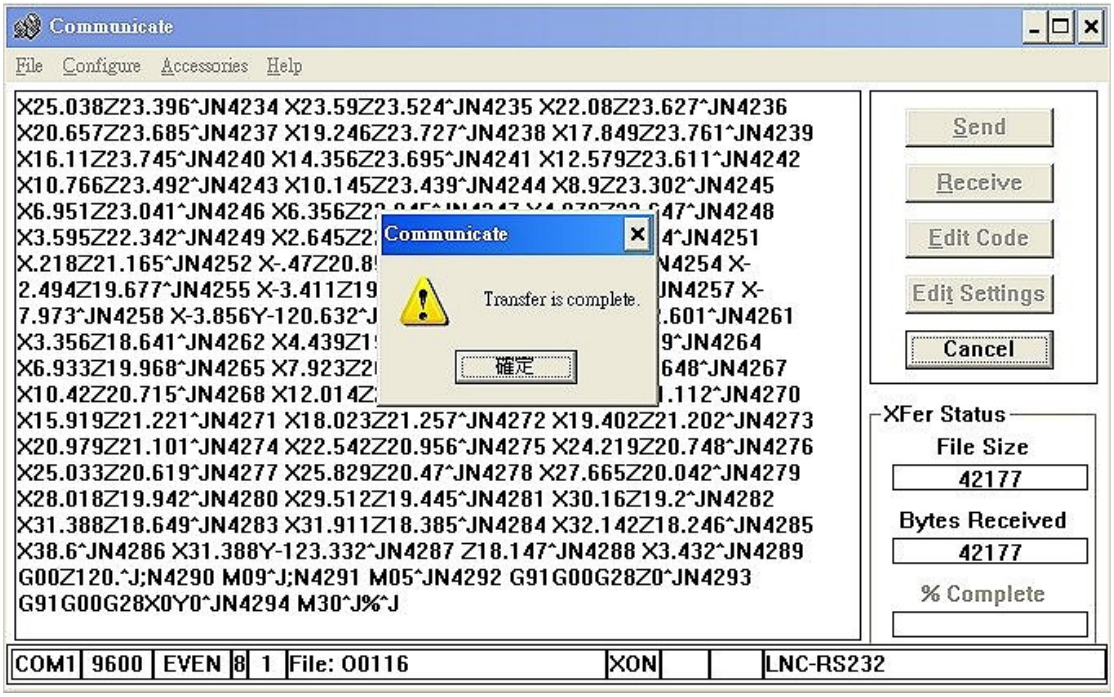
7. NC end's file will start to transfer to PC end.



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

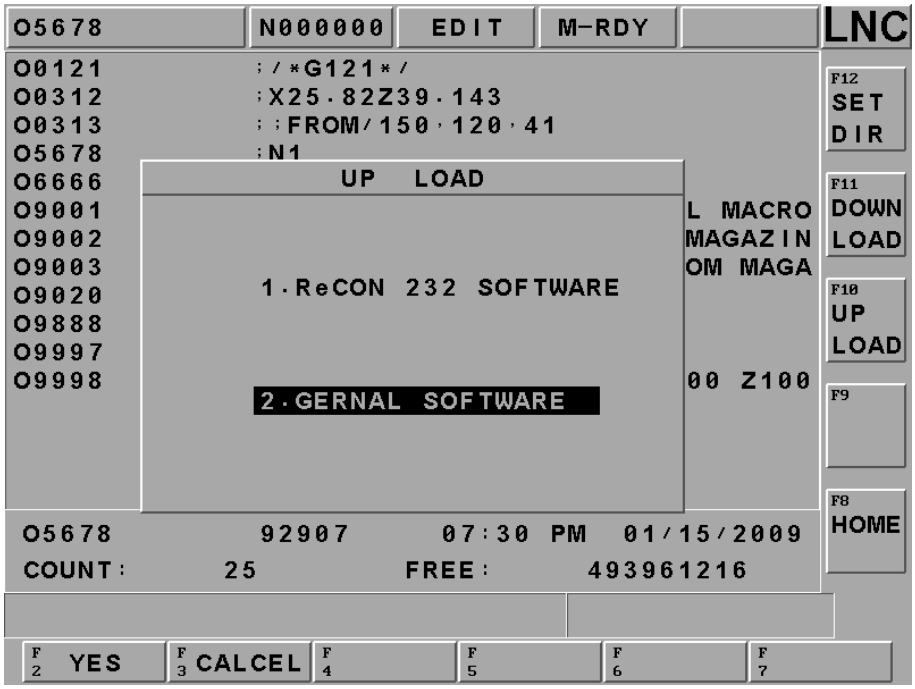


9. After transfer is completed, press [OK]

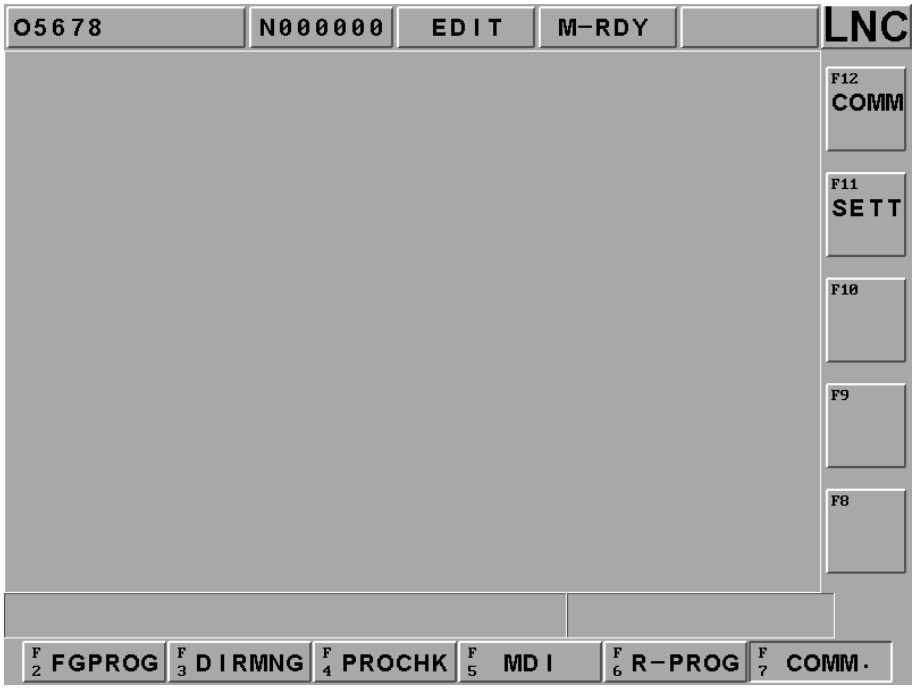


3.4.4 Path Program Upload & Save (using general software)

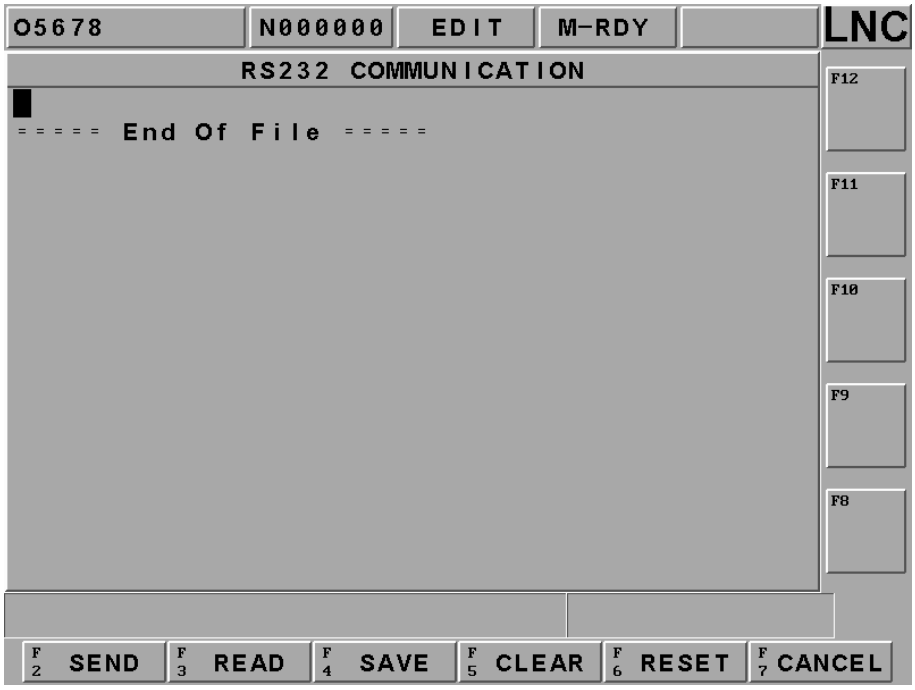
1. Edit RS232 setting at controller and PC end, and make sure they are coordinate.
2. Use PROG group, change to EDIT mode.
3. Go to 【DIRMNG】 and choose the file that you want to transfer to PC end and press [UP LOAD].
   Select 1. General software to upload.



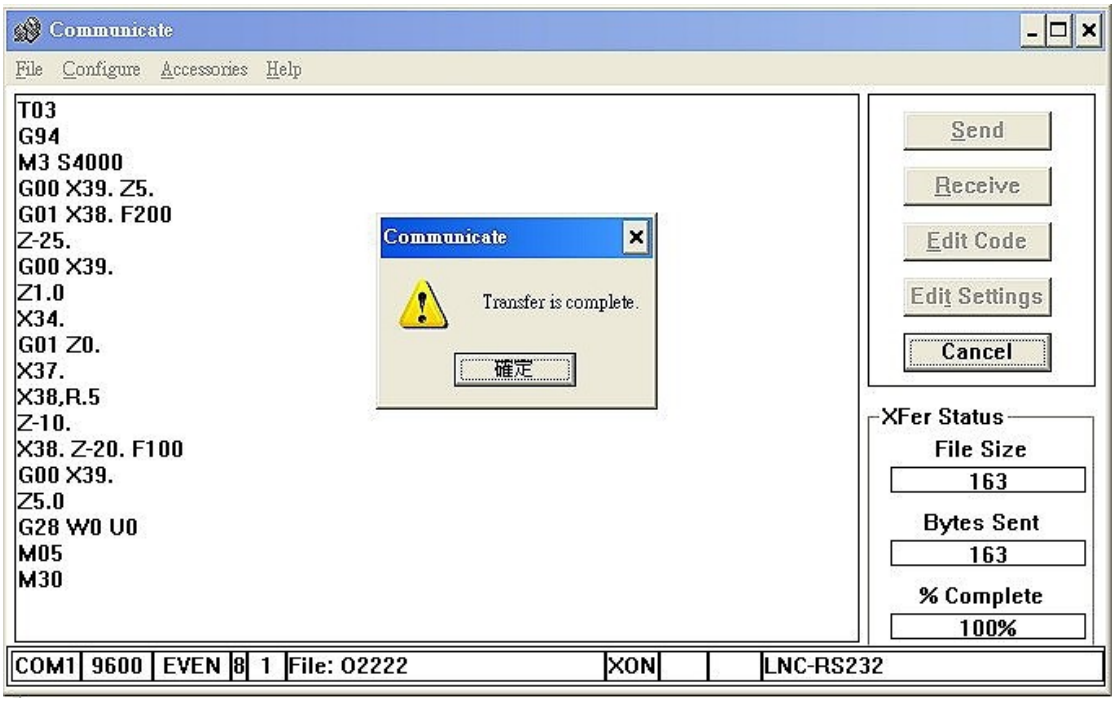
4. Go to communication page and press [COMM] at the down right side.



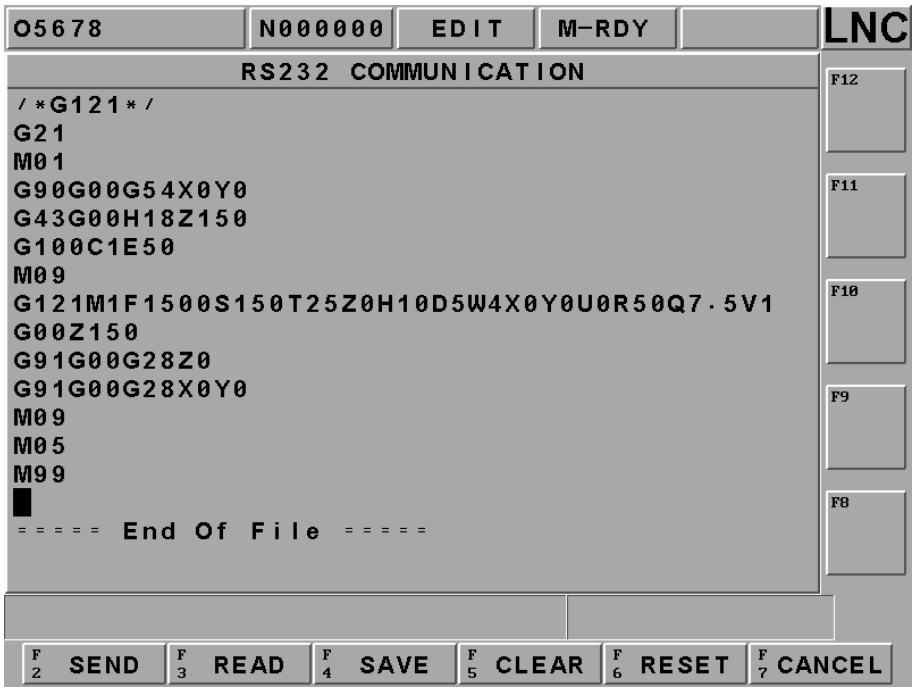
5. Go to RS232 communication.



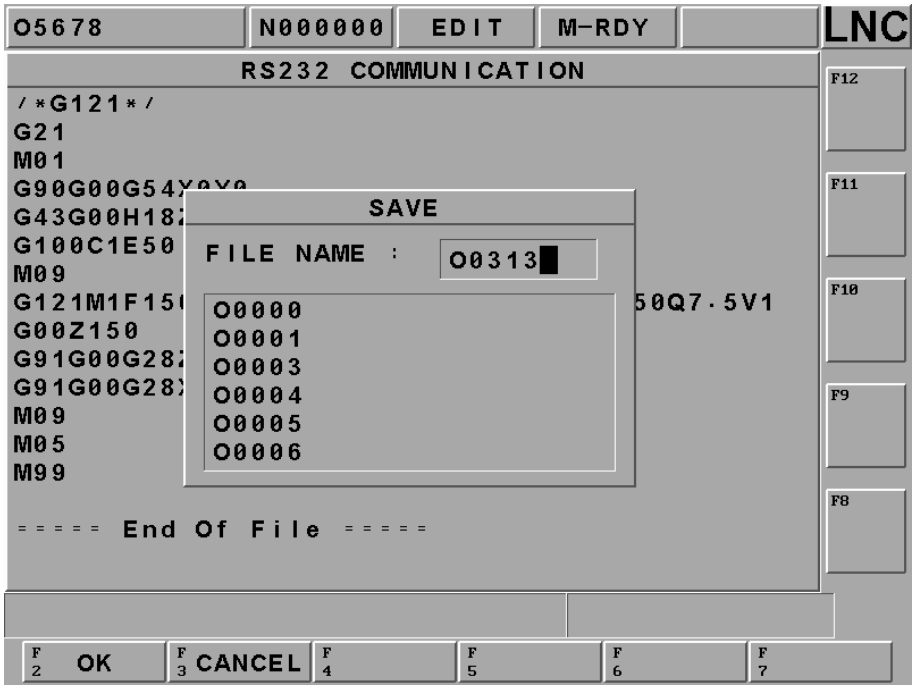
6. After communicate setting at PC end, open an old file for transferring, press [Send], confirm to upload file, press [OK], upload complete .



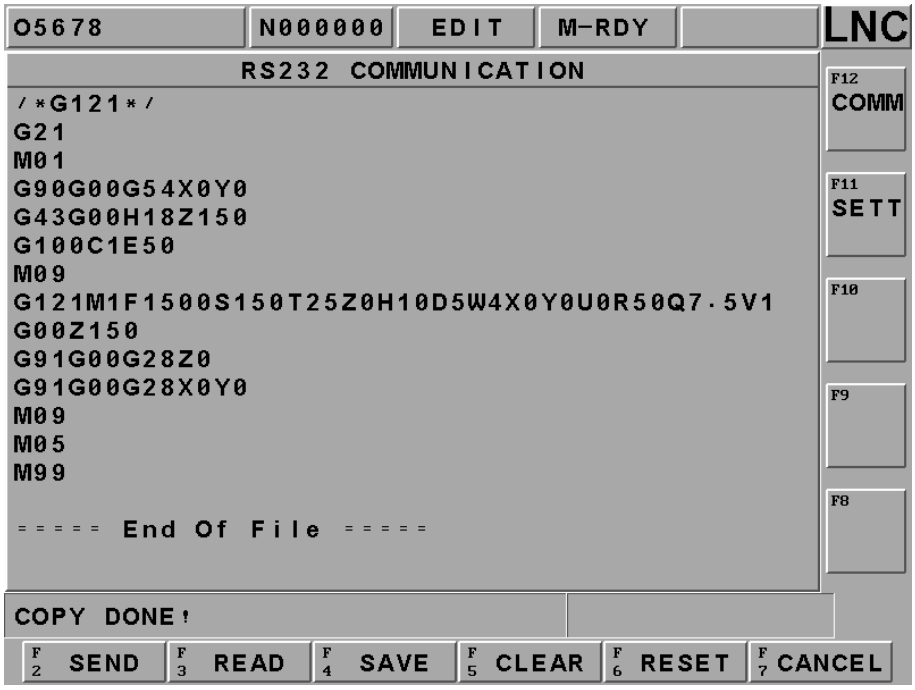
7. PC end will start to receive the files from NC end automatically.



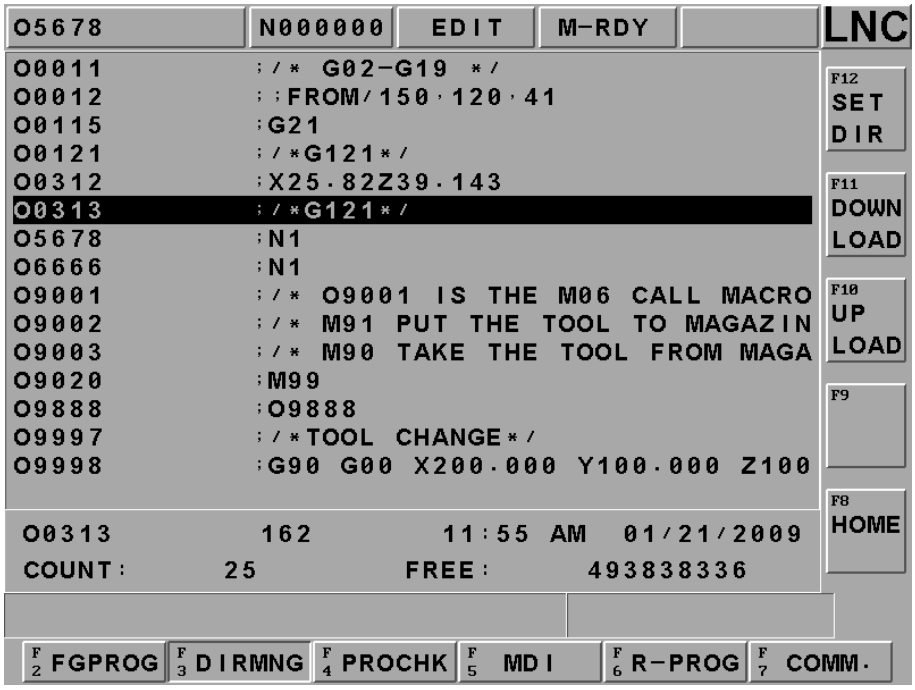
8. After receiving, press SAVE with a new name or SAVE at the same file.



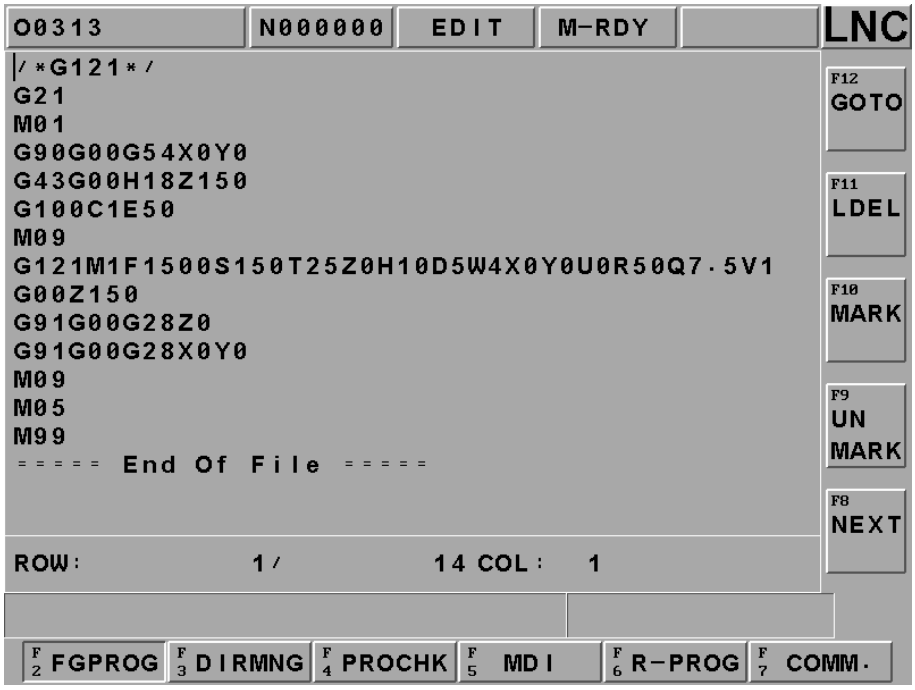
9. Press OK, Copy complete.



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.

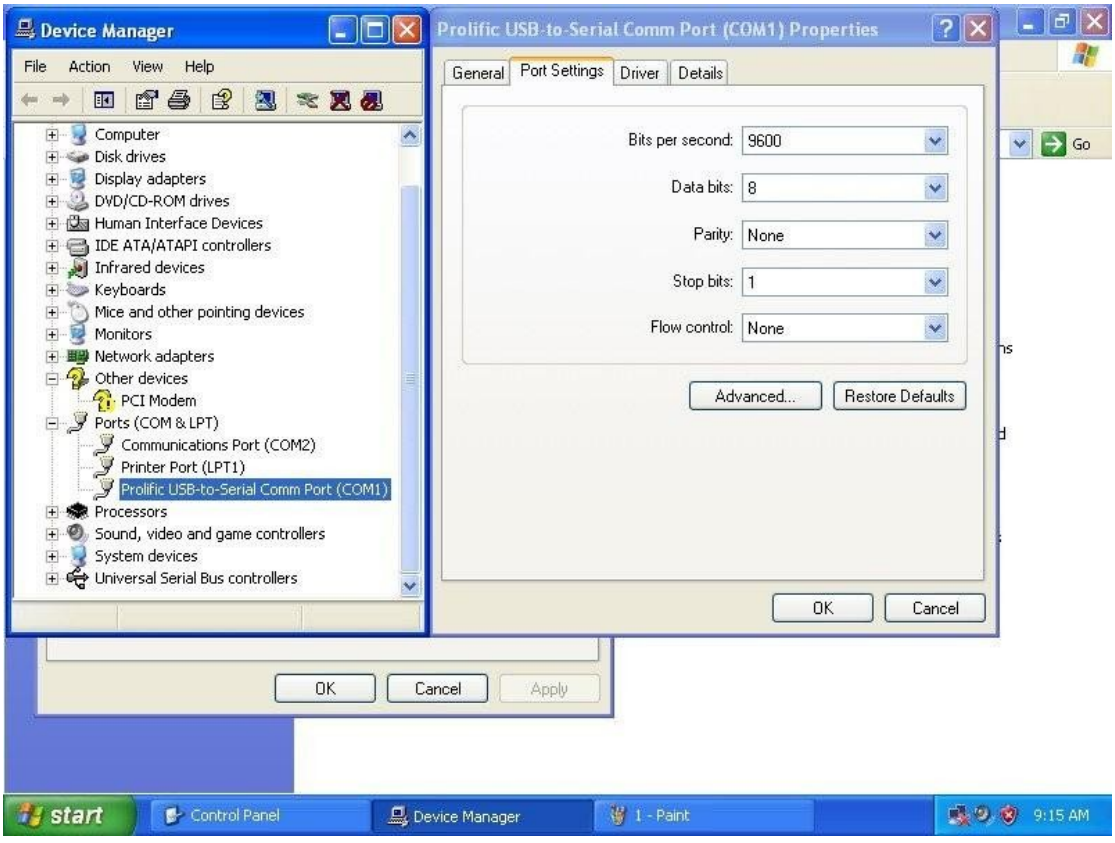


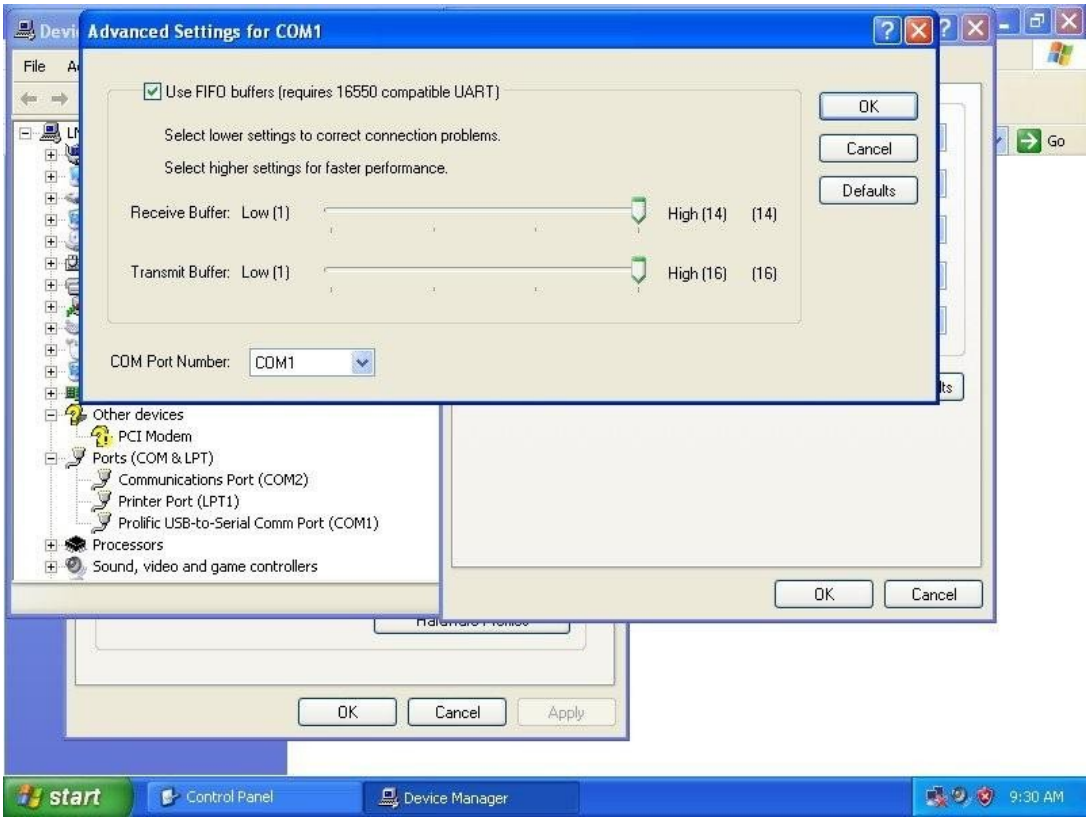
3.5 Setting & Attention for USB to RS232 Port

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
OK
Cancel
Apply
start
Control Panel
Device Manager
9:30 AM