A-LNC Milling Machine Series

Operation Manual

Version: V01.00.003(4408230007)

Leading Numerical Controller

ADVANTECH LNC
Machine Control & Robotics

Advantech-LNC Technology Co., Ltd.

Table of Contents

_Toc448762604

1 CNC operations 1

1.1 Type of operation device 1
1.1.1 M5800 series 1
1.1.2 M6800 series 8
1.2 Screen and function instructions 13
1.3 Monitoring group (MONITOR) 15
1.3.1 Monitoring 15
1.3.2 Coordinate switching 21
1.3.3 Variables 24
1.3.4 Machining settings 26
1.3.5 Machining data 29
1.3.6 Relative coordinates 30
1.3.7 Wear 31
1.3.8 MDI 32
1.3.9 Figures 33
1.3.10 Load 34
1.3.11 Program restart 35
1.4 Program group (PROG) 39
1.4.1 Select a file to be opened 39
1.4.2 Preview 42
1.4.3 Supplementary commands 43
1.4.4 Program editing 44
1.4.5 File management 45
1.4.6 Manufacturer macros 49
1.4.7 Plot settings 54
1.5 Compensation group (OFFSET) 56
1.5.1 Coordinate system 56
1.5.2 Tool management 61
1.5.3 Tool service life 64
1.5.4 Wear management 65
1.5.5 Tool registration 66
1.5.6 Automatic tool alignment 67
1.6 Diagnosis group (DGNOS) 72

1.6.1 Alerts and warnings 72
1.6.2 Ladder diagram 74
1.6.3 IOCSA 75
1.6.4 Timer/counter 76
1.6.5 System information 77
1.6.6 Alert history 78
1.6.7 Operation history 79
1.6.8 R value 80
1.6.9 Waveform monitoring 81
1.7 Maintenance group (MAINTE) 82
1.7.1 User parameters 82
1.7.2 Language settings 83
1.7.3 Network settings 84
1.7.4 Changing identity 85
1.7.5 Hardware contact 86
1.7.6 Parameters 87
1.7.7 Backup 88
1.7.8 System update 90
1.7.9 Tuning functions 91
1.7.10 Period of use 94
1.7.11 Page permissions 96
1.7.12 Change password 97
1.7.13 Date and time 98
1.7.14 Version information 99
1.7.15 Project settings 100
1.8 Usage instructions 101
1.8.1 Opening and editing a file 101
1.8.2 Execute machining 109
1.8.3 Using the MDI function 111
1.8.4 Program restart 113
1.8.5 Automatic tool alignment 116
1.8.6 Network settings and connections 124
1.8.7 System update 127
1.8.8 System data backup - export 131
1.8.9 System data backup - import 135
1.8.10 Preview function 139
1.8.11 Editing and using manufacturer macros 145

1.8.12 Tool offset settings 148
1.8.13 Coordinate system settings 150
1.8.14 Parameter settings 153
1.8.15 File backup - import 158
1.8.16 File backup - export 163
1.8.17 Waveform monitoring function 167
1.8.18 OnLine Help 174
1.8.19 Switching multiple paths 174

2 Control panel operations 176

2.1 Operating panel 176
2.1.1 M5800 series 176
2.1.2 M6800 series 176
2.2 LED signal light (LED SIGNAL) 177
2.3 Axis selection (AXIS SELECTION) 178
2.4 Mode selection (MODE SELECTION) 179
2.5 Spindle operation & spindle speed adjustment 182
2.6 Supplementary function keys 182
2.7 Emergency stop (EMG-STOP) 189
2.8 Program start (CYCLE START) & program pause (FEED HOLD) 190
2.9 Feed rate adjustment 191
2.10 Program protection lock 191
2.11 Seven-segment display of tool number 192
2.12 Power on/off 192

1 CNC operations

1.1 Type of operation device

The operation panel can be divided into the LCD liquid crystal display, MDI data input panel, and OP operation panel. The main function of the MDI data input panel is to allow users to edit or modify a program and set numerical values. The OP operation panel is a control panel for meeting all machining requirements; it is equipped with various switches and function keys, and a pulse generator (hand wheel), etc. The operation panel can have different designs based on different machines, but this system contains a set of standard panels which can be selected by the machine manufacturer.

1.1.1 M5800 series

Keys shown on the LCD liquid crystal display:

Function keys: There are 10 horizontal keys right below the LCD. They allow users to select functions which are shown in the lower part of the screen.

LCD liquid crystal display

Keys shown on the MDI panel:

MDI data input panel

A. CNC function group key:

Name	Function group key	Description
Monitoring group	POS	The group of screens showing various coordinates and machining data.
Program group	PROG	All screens related to program information (in coordination with 1. Editing mode, 2. Memory mode, 3. Manual mode).
Compensation group	OFFSET	Sets the tool offset.
Diagnosis group	DGNOS	Shows real time information from the diagnosis screen.

Name	Function group key	Description
Maintenance group		System related settings.

B. Text symbol and numerical symbol keys:

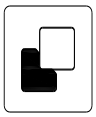
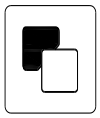
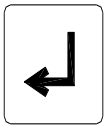
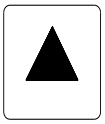
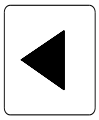
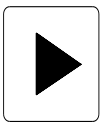
These letters, symbols, and digits are used mainly for program editing and data input. Some of the symbols are minimized below the keys. The SHIFT key and the text symbol key must be pressed simultaneously to use these minimized symbols.

Name	Supplementary editing keys	Description
Letter	X U ～ Z W	There are 26 letter keys from A to Z which can be used for location or argument commands.
Number	0 = ～ 9 ?	There are 10 number keys from 0 to 9 which can be used for numerical values or input data.
Symbol	/	The section requiring diagonal jump during program editing
Symbol	;	1. Pressing this key during program editing will indicate the end of input program section. 2. If this key is placed at the very beginning of a program block, it means this block of code will not be executed.
Symbol	.	The numerical value needs to be separated by a decimal point during program editing.
Symbol	(、) 、< 、> 、 ， 、: 、 & 、* 、? 、[、] 、$ 、% 、 ^ 、! 、@ 、# 、+ 、= 、- 、 _ 、	Symbols to be used during program editing.

C. Supplementary editing keys:

These keys can be used in coordination with cursor and the highlighted row on the screen for program modification, data setting, and page switching.

Name	Supplementary editing keys	Description
System reset	/ / Reset / /	1. Default values of system rest status. 2. Cancel alert after the abnormal situation has been cleared. 3. Cancel the machining process after cycle start. 4. Return the cursor to highlight the program header in editing mode.

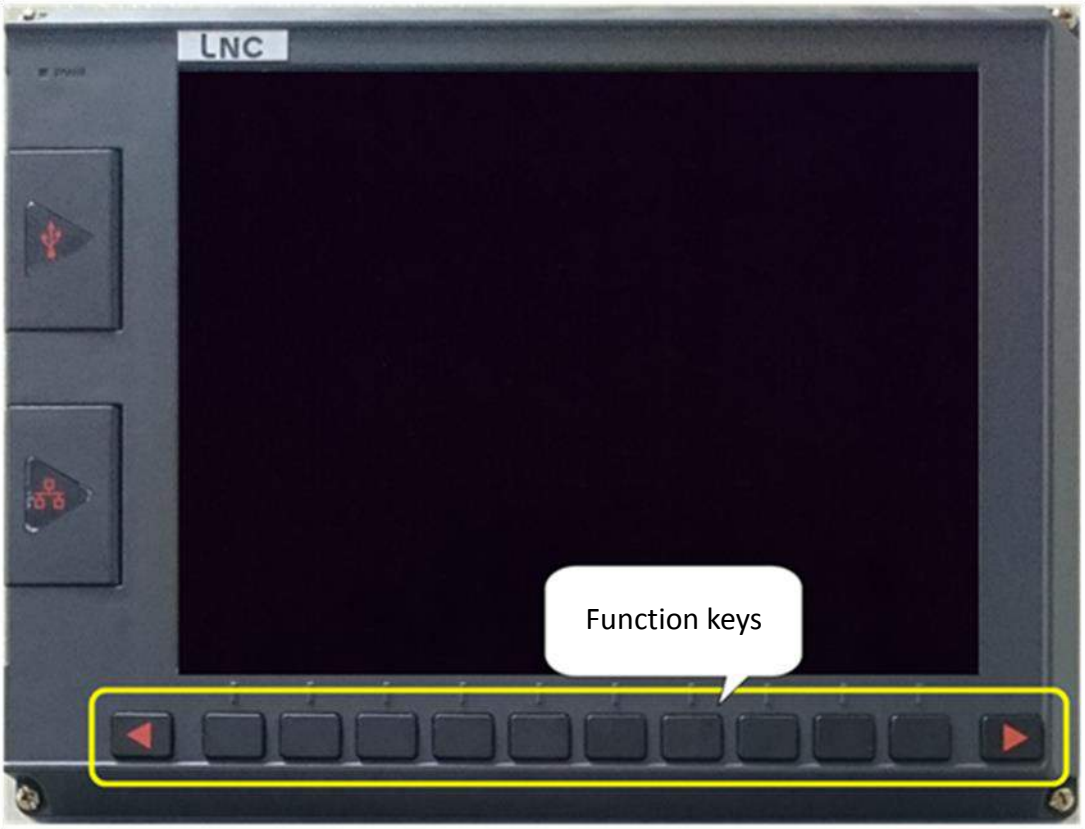
Name	Supplementary editing keys	Description
Go to previous page	 <Page Up>	Field on the screen for navigating to the previous page.
Go to next page	 <Page Down>	Field on the screen for navigating to the next page.
Input	 <Enter>	1. After entering a numerical value in the input area, press <Enter> to store the numerical value into the field. 2. In windows explorer, move the cursor to the program location and press <Enter> to open the file. 3. Press <Enter> in editing mode to insert a blank line.
Moves the cursor up		1. Moves the cursor up while in program editing status. 2. Moves the cursor on this page upward.
Moves the cursor down		1. Moves the cursor down in program editing status. 2. Moves the cursor on this page down.
Moves the cursor left		1. Moves the cursor to the left in program editing status. 2. Moves the cursor on this page to the left.
Moves the cursor right		1. Moves the cursor to the right in program editing status. 2. Moves the cursor on this page to the right.
Whitespace		Enter the whitespace character.
Character shift		It can be used for entering special symbols in combination with symbol/number keys. Please note: Character shift can only be done by simultaneously pressing the SHIFT key and the text symbol key.
Row header position		Return the cursor for highlighted row to the character position at row header during program editing.

Name	Supplementary editing keys	Description
End of row position	End	Return the cursor to the last character position of the selected row during program editing. Please note: it must be used in combination with SHIFT key.
Cancel character	CAN	Cancels the unwanted character at previous position.

1.1.2 M6800 series

Keys shown on the LCD liquid crystal display:

Function keys: There are 12 horizontal keys right below the LCD. They allow users to select functions which are shown in the lower part of the screen.



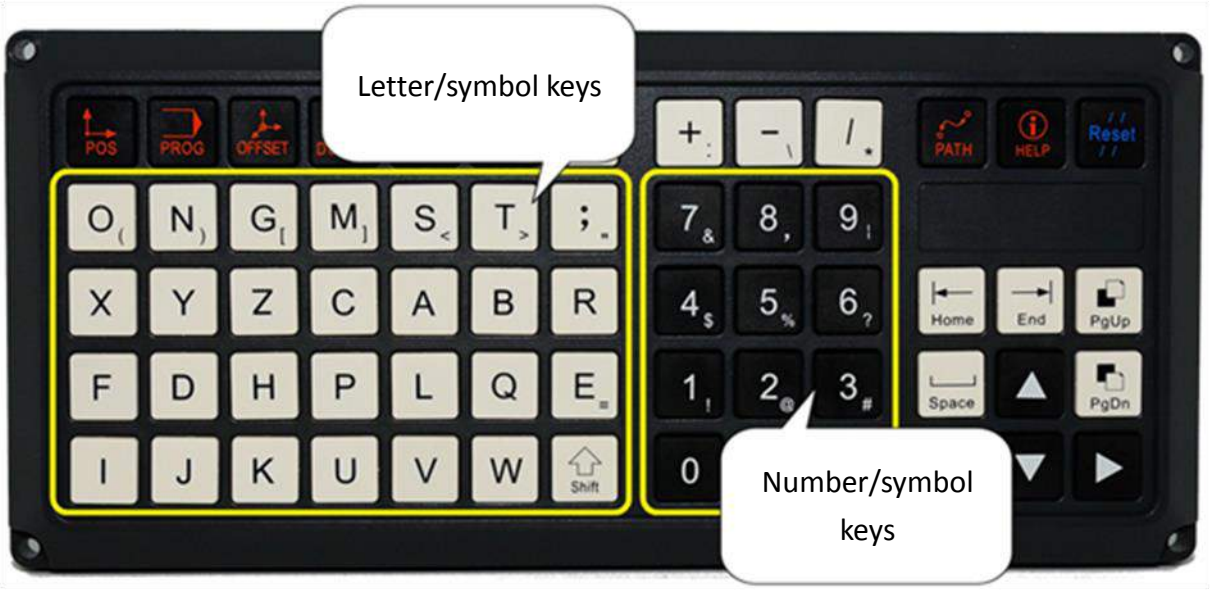
LCD liquid crystal display

Keys shown on the MDI panel:

A. CNC function group keys:

Name	Function group key	Description
Monitoring group	POS	The group of screens showing various coordinates and machining data.
Program group	PROG	All screens related to program information (in coordination with 1. Editing mode, 2. Memory mode, 3. Manual mode).
Compensation group	OFFSET	Sets the tool offset.
Diagnosis group	DGNOS	Shows real time information from the diagnosis screen.
Maintenance group	MAINTE	System related settings.

B. Text symbol and numerical symbol keys:

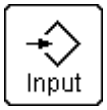


These letters, symbols, and digits are used mainly for program editing and data input. Some of the symbols are minimized below the keys. The SHIFT key and the text symbol key must be pressed simultaneously to use these minimized symbols.

C. Supplementary editing keys:

These keys can be used in coordination with cursor and the highlighted row on the screen for program modification, data setting, and page switching.

Name	Supplementary editing key	Description
System reset	Reset	1. Default values of system rest status. 2. Cancel alert after the abnormal situation has been cleared. 3. Cancel the machining process after cycle start. 4. Return the cursor to highlight the program header in editing mode.
Go to previous page	(PgUp)	1. Field on the screen for navigating to the previous page.
Go to next page	(PgDn)	1. Field on the screen for navigating to the next page.

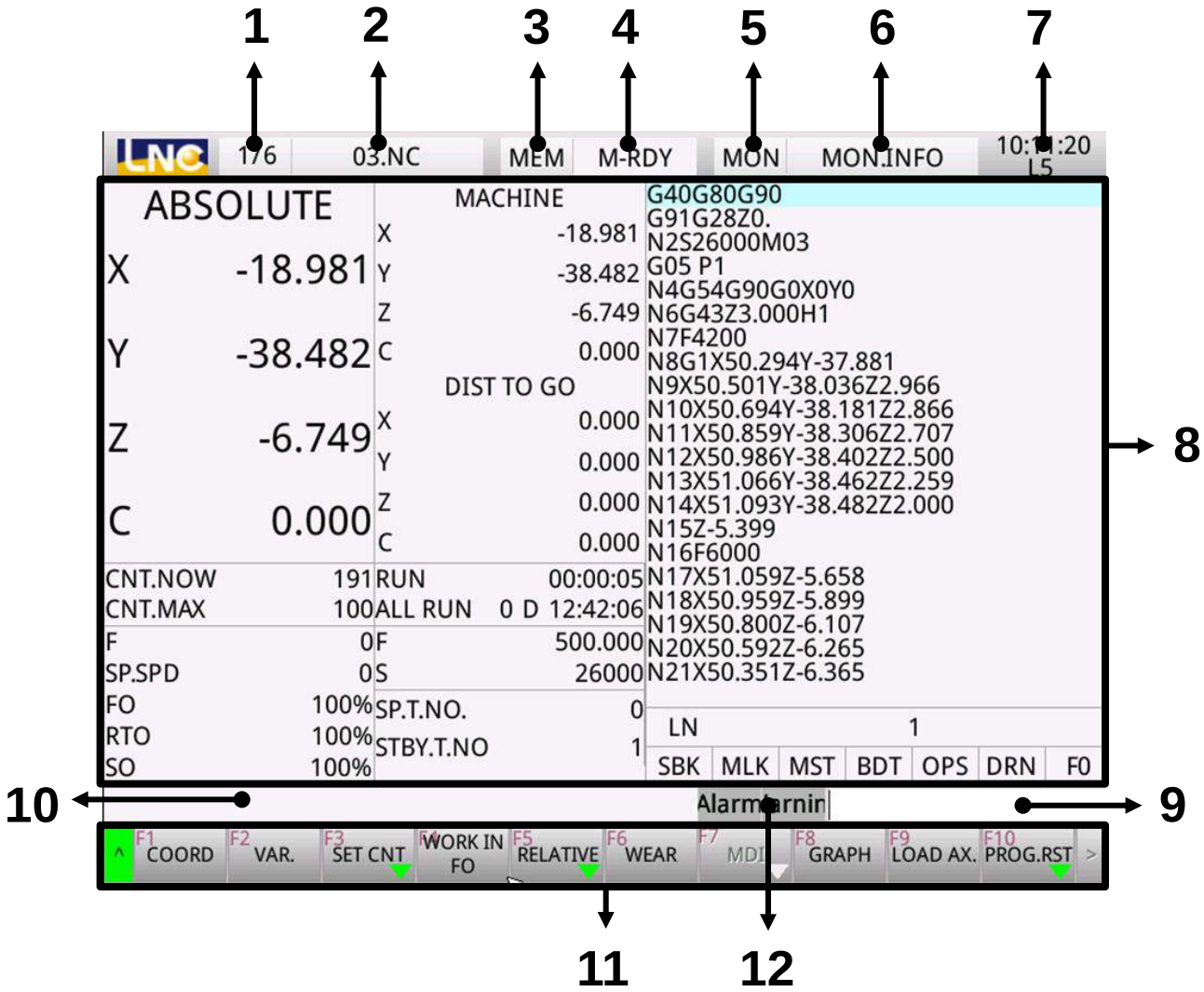
Name	Supplementary editing key	Description
Input		1. After entering a numerical value in the input area, press <Enter> to store the numerical value in the field. 2. After selecting a key in the child window, press <Enter> to confirm. 3. Press <Enter> in editing mode to insert a blank line.
Moves the cursor up		1. Moves the cursor up while in program editing status. 2. Moves the cursor on this page upward.
Moves the cursor down		1. Moves the cursor down in program editing status. 2. Moves the cursor on this page down.
Moves the cursor left		1. Moves the cursor to the left in program editing status. 2. Moves the cursor on this page to the left.
Moves the cursor right		1. Moves the cursor to the right in program editing status. 2. Moves the cursor on this page to the right.
Deletes the character		1. Deletes the unwanted character at the next position.
Whitespace		1. Enter the whitespace character.
Character shift		1. It can be used for entering special symbols in combination with symbol/number keys. Please note: Character shift can only be done by simultaneously pressing the SHIFT key and the text symbol key.
Row header position		1. Return the cursor for highlighted row to the character position at row header during program editing.
End of row position		1. Return the cursor to the last character position of the selected row during program editing.

1.2 Screen and function instructions

This controller can be divided into 5 function groups: Monitoring (MONITOR), program (PROG), offset (OFFSET), diagnosis (DIAGN), and maintenance (MAINTE). In this manual, 【 】represents function keys in the lower part of screen, and < > represents keys on the MDI panel.

Layout of the display screen:

Note: Different models are equipped with different numbers of function keys in the lower part of screen.



1: Path information. (It will only be shown when the multi-path function is available)

Note: numerator is the current path number, and denominator is the total number of paths.

2 : Name of currently assigned program.

Note: it refers to the filename currently being used by the controller.

3 : CNC mode information.

Note: 1. Jog, 2. Reference point, 3. Manual, 4. Incremental, 5. Hand wheel, 6. Automatic

4 : Machine's status information.

Note: 1. Not ready, 2. Ready, 3. Cycle start, 4. Machine paused, 5. Section stopped.

5 : Current function group.

Note: 1. Monitoring, 2. Program, 3. Offset, 4. Diagnosis, 5. Maintenance.

6 : Name of current page.

Note: it refers to the function page currently accessed by the controller.

7 : System time and user level.

Note: it refers to the current user level between L1 to L7.

8 : Range of each screen's display area.

Note: the range of variation to be shown by switching among function pages of each group.

9 : Input area/section.

Note: provided by the controller for users to input values in various fields on each page.

10: Summary information reminder section.

Note: the controller will provide an operation information summary prompt to remind users.

11: Display areas corresponding to the keys on the screen.

Note: this area is to be used for corresponding keys on the screen. If there are keys listed in this manual but they cannot be seen on the controller, it means that current user does not have the proper permission level.

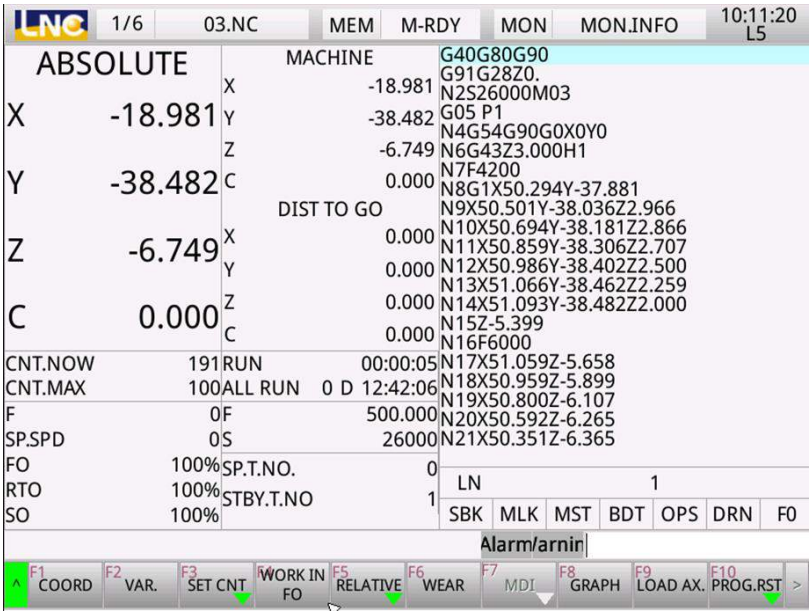
12: Error messages.

Note: Alerts and warnings.

1.3 Monitoring group (MONITOR)

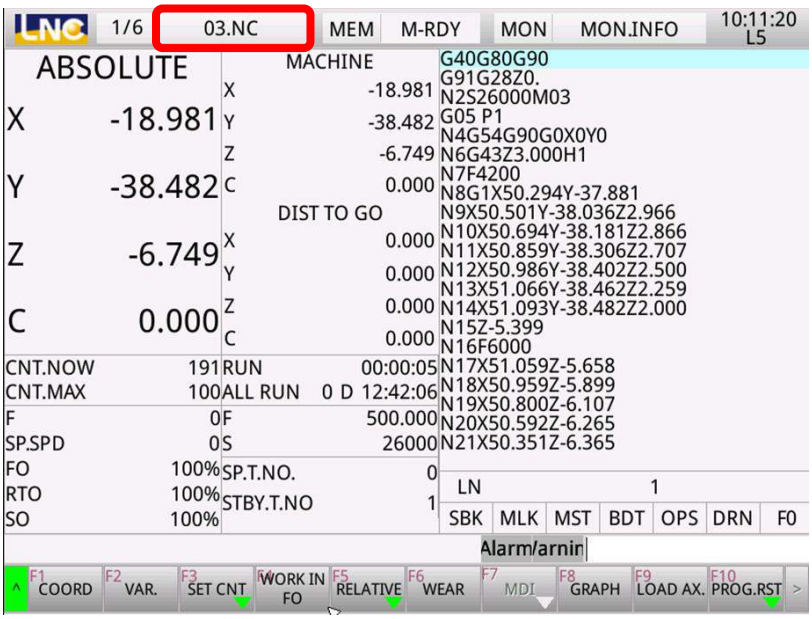
1.3.1 Monitoring

- On the monitoring page, users can view various information such as the filename of the current program, current operating mode, current machine status, has an alert being issued, feeding % status, spindle % status, number of workpieces undergoing the machining process, maximum number of machining pieces, time required for single machining, total time of machining accumulated, and various coordinates.

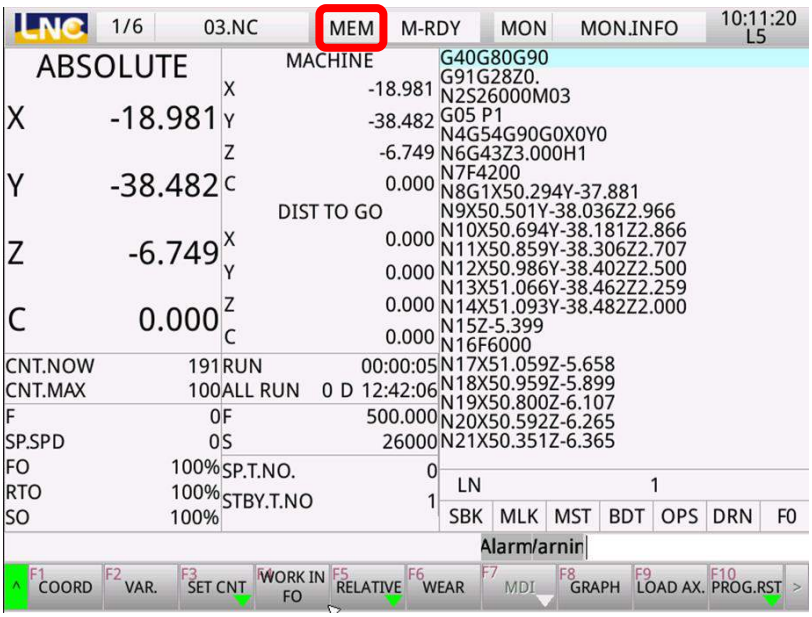


Description of various fields:

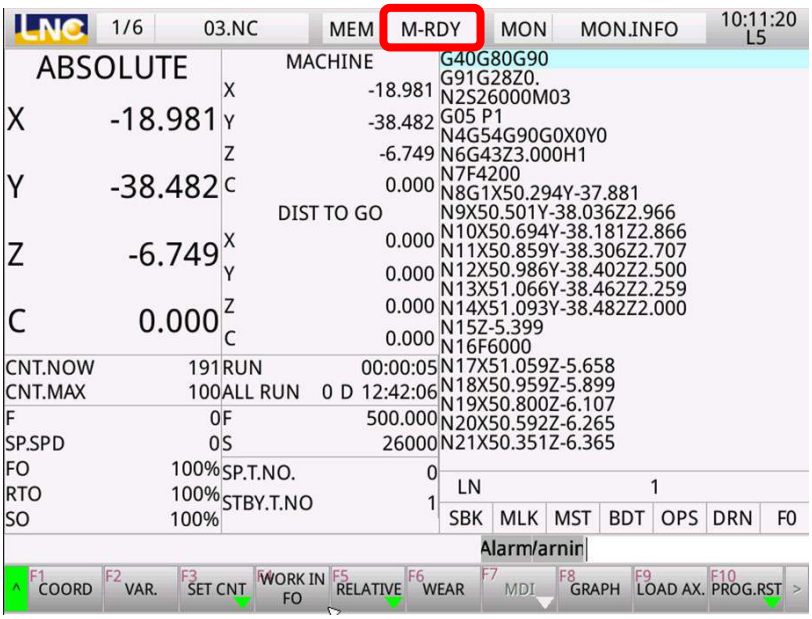
- Users can inspect the current machining program's filename on the monitoring page as shown in the figure below.



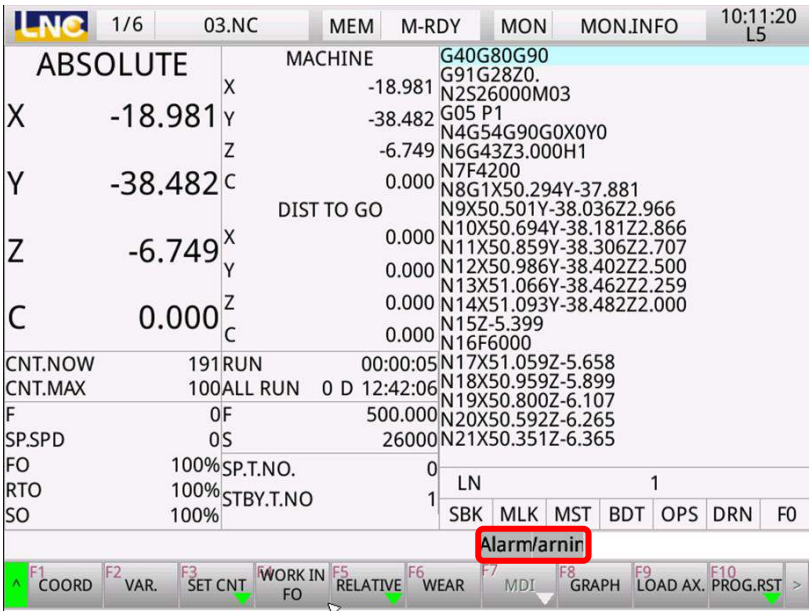
- Users can inspect current status on the monitoring page as shown in the figure below. For example: Jog (JOG), reference point (ZRN), manual (MDI), incremental (INC), hand wheel (MPG), automatic (MEM).



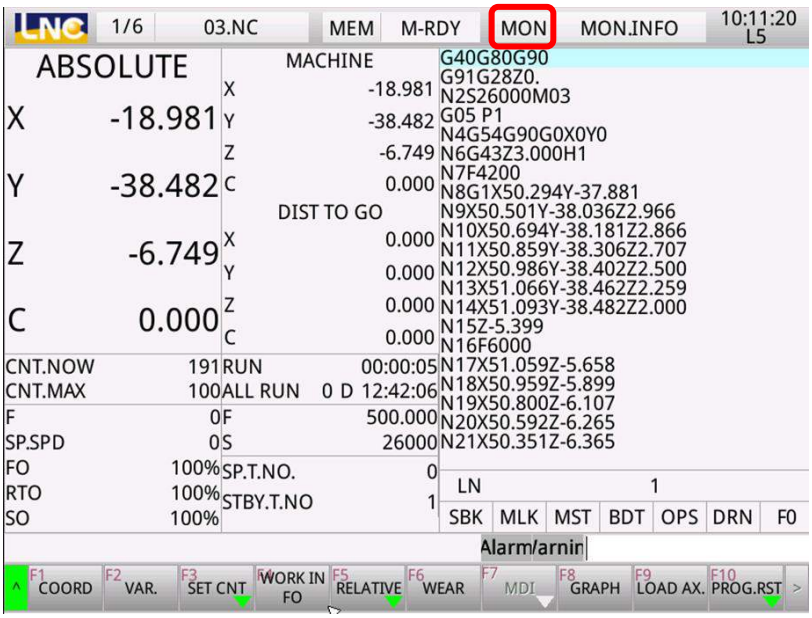
- Users can view whether the current status is ready or other (not ready, machine paused, section stopped) on the monitoring page as shown in the figure below.



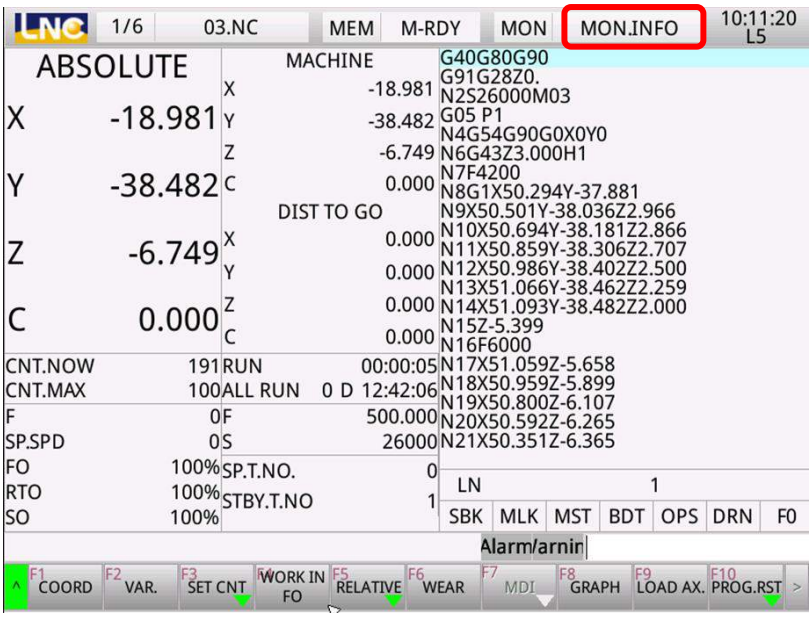
- Users can view whether there is an abnormal situation from the monitoring page as shown in the figure below. For example: When there is insufficient air pressure, insufficient lubricant, or program error, this item will flash to notify users of the current status of machine malfunction.



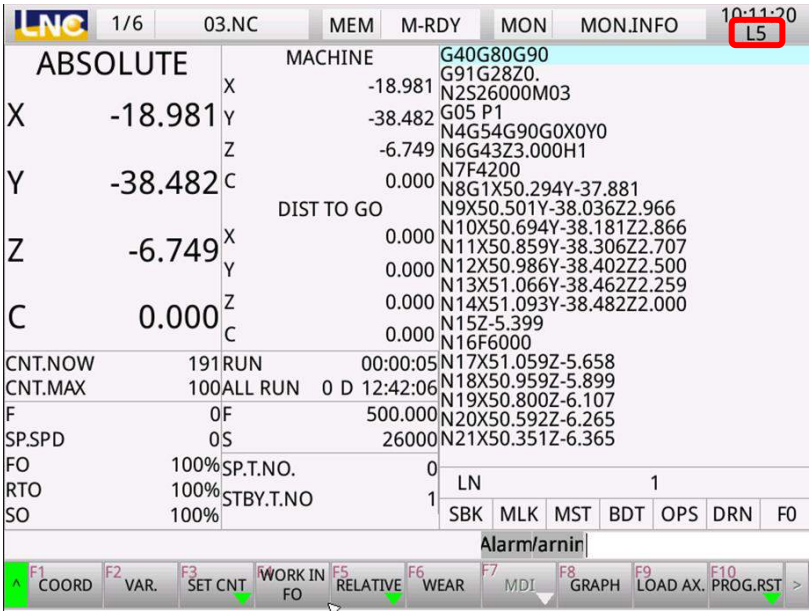
- Users can view the current function group on the monitoring page as shown in the figure below.



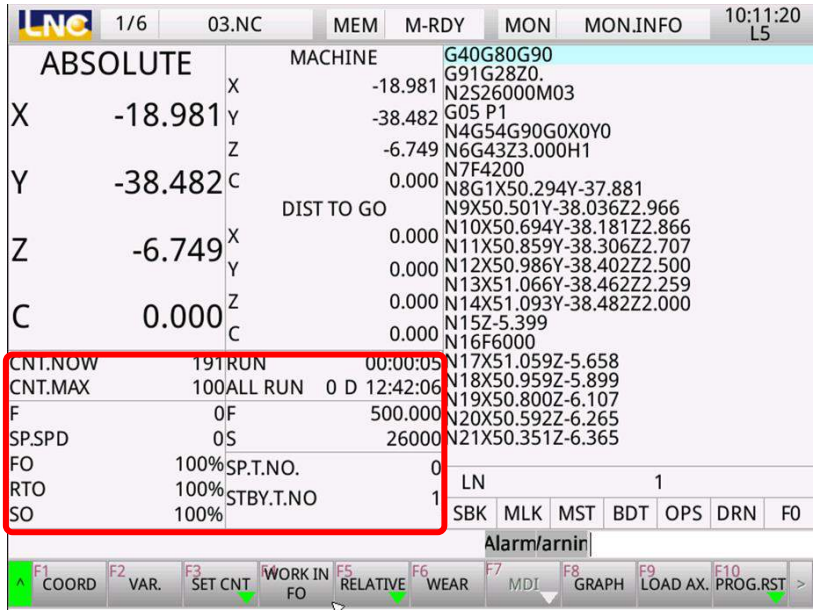
- Users can view the current page's title on the monitoring page as shown in the figure below.



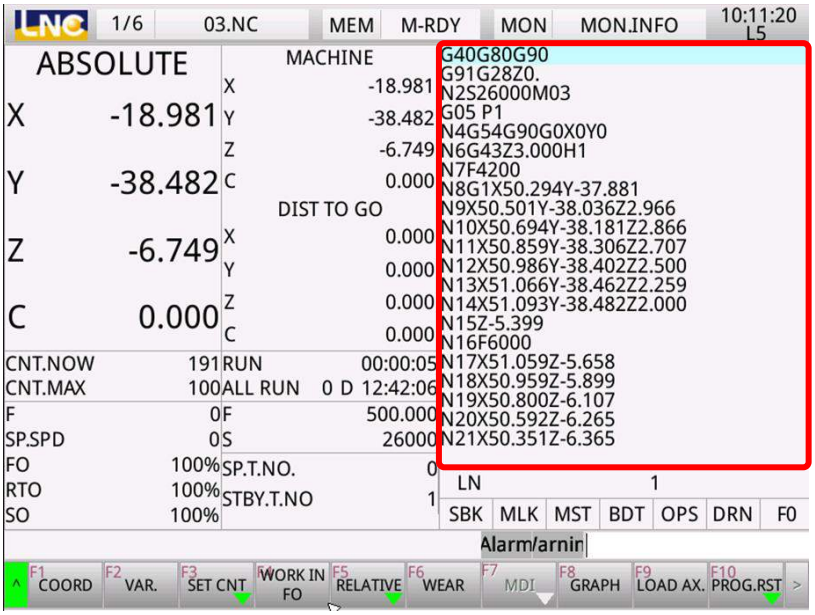
- Users can view the current user level on the monitoring page as shown in the figure below.



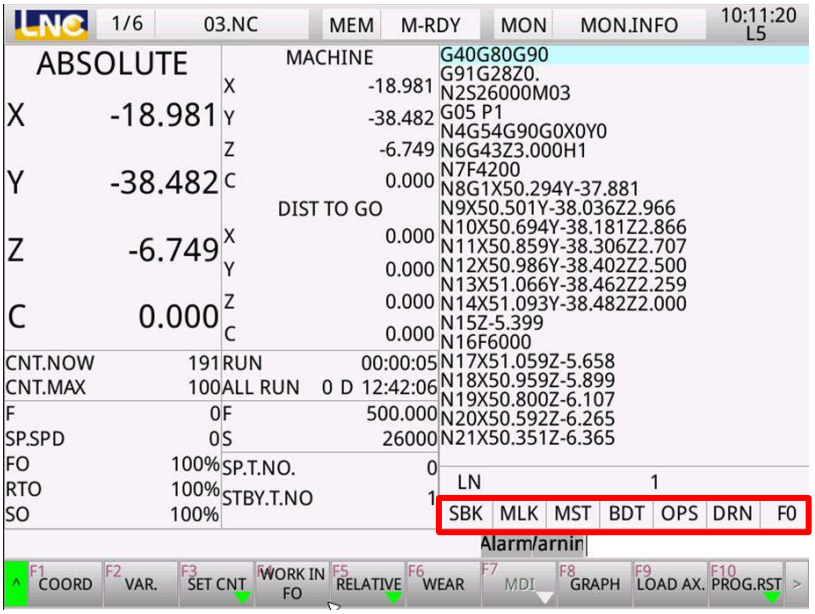
- On the monitoring page, users can view various information such as feed rate, feeding % status, spindle's rotation speed, spindle % status, number of current machining pieces, maximum number of machining pieces, time for single machining, accumulated total machining time, spindle tool number, and standby tool number as shown in the figure below.



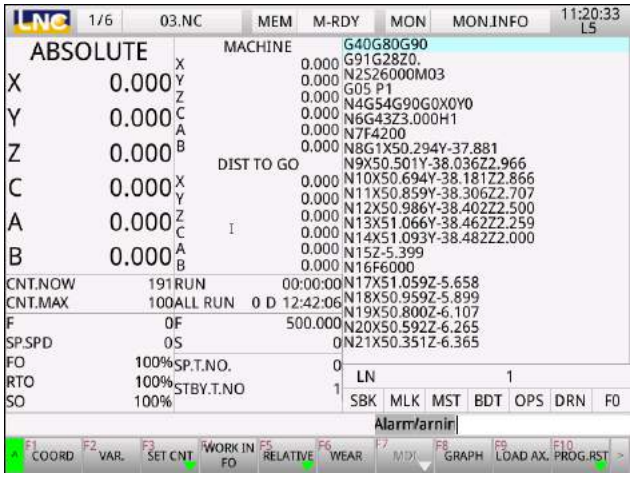
- This is the current program section where the program currently being executed or to be executed is shown, and this screen can allow users to know exactly which line number of the program is being executed currently.

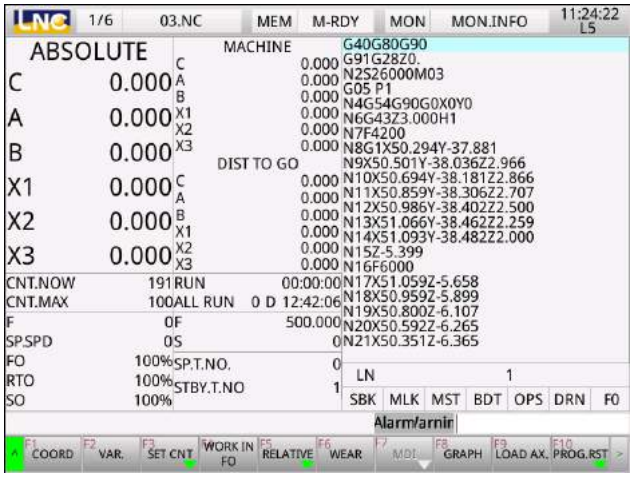


- This is the soft key activation section where activated soft keys are shown; the background color is yellow when activated.



- When the number of axis in use is greater than 6, **<PgUp>** and **<PgDn>** keys on the MDI panel can be used for additional inspections along axial directions as shown in the figure below.



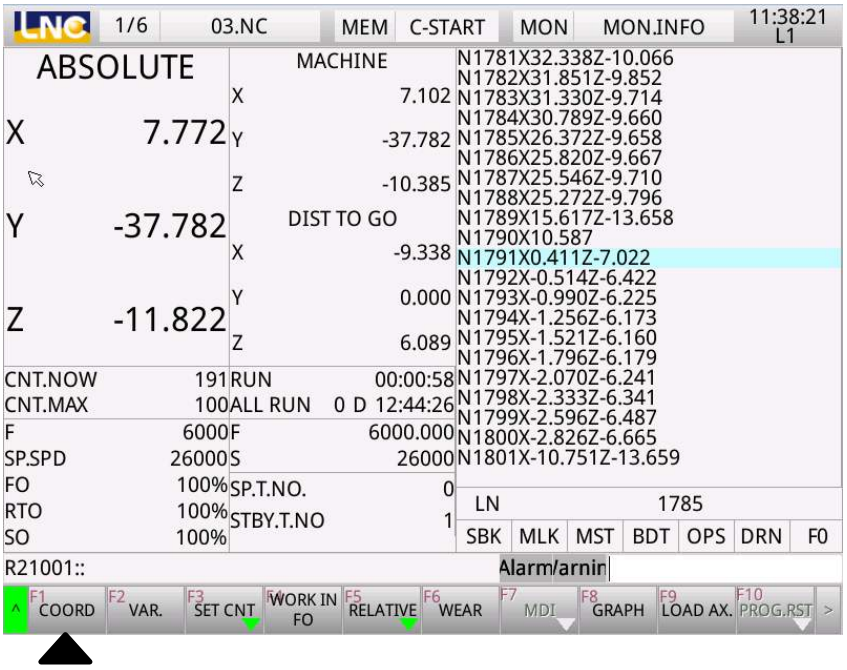


1.3.2 Coordinate switching

(Note: This function key is not available in the M5800 series)

Description of the coordinate switching function:

- Users can use the 【**Coordinate switching**】 key for switching program coordinates, relative coordinates, machine coordinates, and the amount of moves remaining in the main coordinate viewing section.

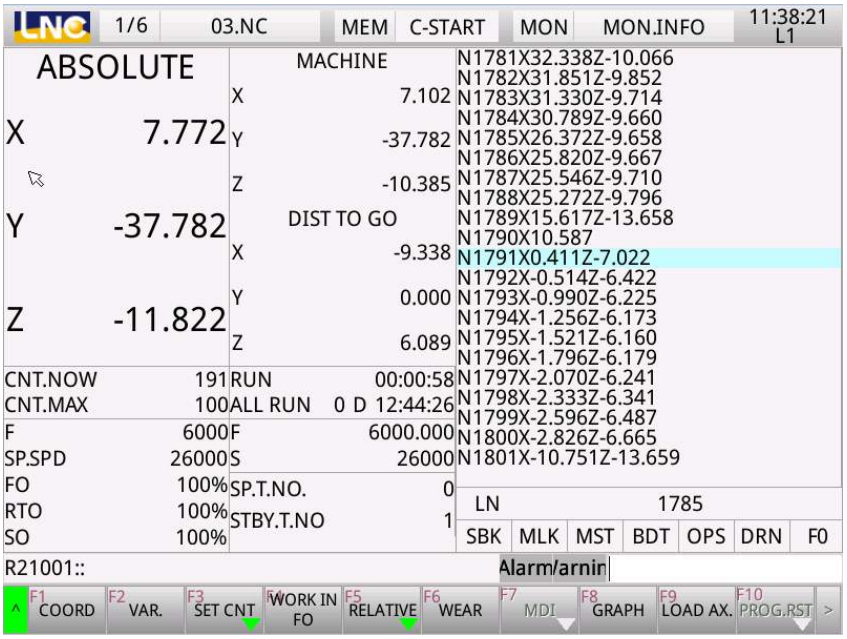


Instructions of coordinate switching:

- Users can used the **<POS>** key on the MDI panel to switch to monitoring group before pressing 【**Coordinate switching**】 to switch program coordinates, relative coordinates, machine coordinates, and the amount of moves remaining.

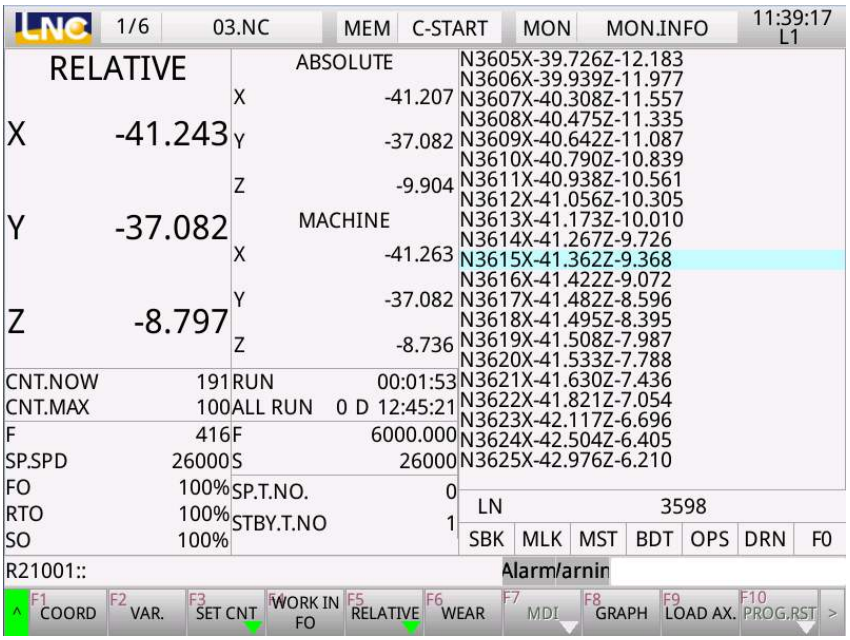
A. Program coordinates

- Program coordinates let users know the tool's exact coordinate position with respect to the program, which shows the distance from tool's current position to program's reference point (workpiece coordinate system).



B. Relative coordinates

- Relative coordinates show the distance from any user defined point to the current position, so users can zero the relative coordinates at any time or directly enter a non-zero value.

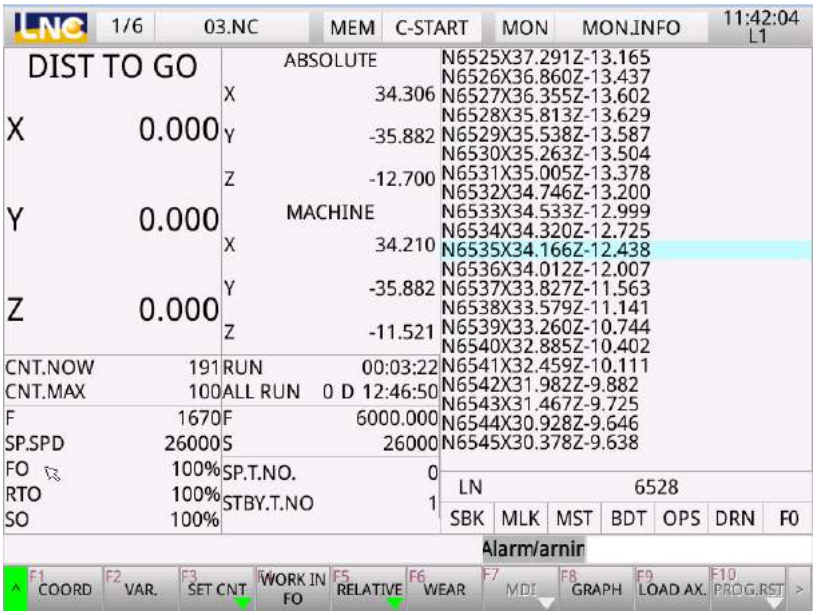


C. Machine coordinates

- Machine coordinates show the relative distance between the current tool position and machine reference point.

D. Amount of moves remaining

- Amount of moves remaining refers to the remaining movement amount before the coordinate point reaches the destination position according to a command issued by the program.



1.3.3 Variables

Description of variable functions:

- They can be used for passing arguments or computing conditional expressions in a macro program.

Using variables:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group before pressing the 【**Variables**】 key to view local and global variables.
- Local variables #1 to #199 can be read/ written by users in a macro program, so their numerical values cannot be entered directly on the human-machine interface.
- Global variables:

  @1 to @999 can be read/written by users inside a macro program.

  @1000 to @1999 are the system macro's global variables reserved for the system.

  @5000 to @5999 are global variables with shared paths which can be edited by users.

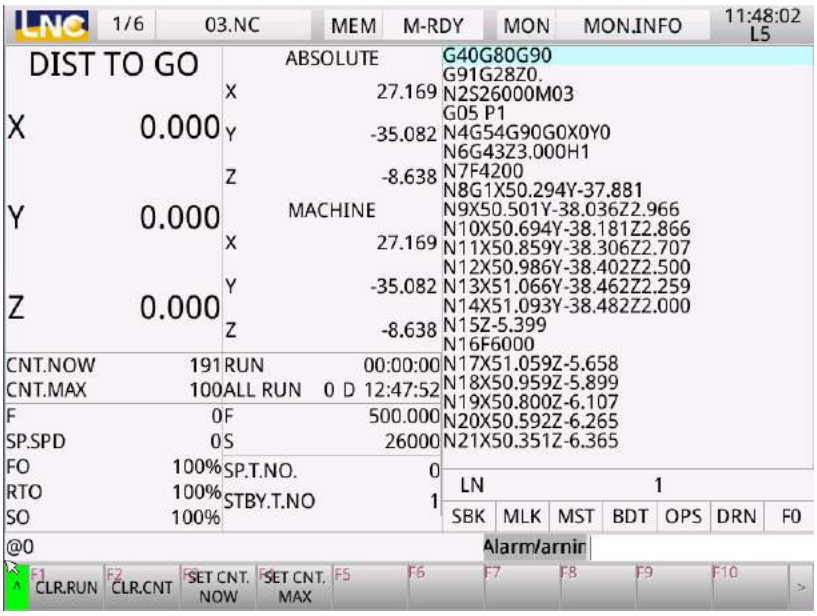
  @6000 to @6999 are global variables with shared paths to be used by the system.
- Users can edit a macro or a program to change the numerical values.

1/6
VARS
MEM
S-STOP
MON
MON.INFO
11:46:22
L5
#LOCAL
@GLOBAL
#1=10
#2=20
#3=30
@1=100
@2=200
@3=300
M00
M30
0 VACANT 0 VACANT
1 10.000 1 100.000
2 20.000 2 200.000
3 30.000 3 300.000
4 VACANT 4 VACANT
5 VACANT 5 VACANT
6 VACANT 6 VACANT
7 VACANT 7 VACANT
8 VACANT 8 VACANT
9 VACANT 9 VACANT
10 VACANT 10 VACANT
11 VACANT 11 VACANT
12 VACANT 12 VACANT
13 VACANT 13 VACANT
LN
7
SBK
MLK
MST
BDT
OPS
DRN
F0
@0
Alarm/arnin
F1 COORD
F2 VAR.
F3 SET CNT
F4 WORK IN FO
F5 RELATIVE
F6 WEAR
F7 MDI
F8 GRAPH
F9 LOAD AX.
F10 PROG.RST

1.3.4 Machining settings

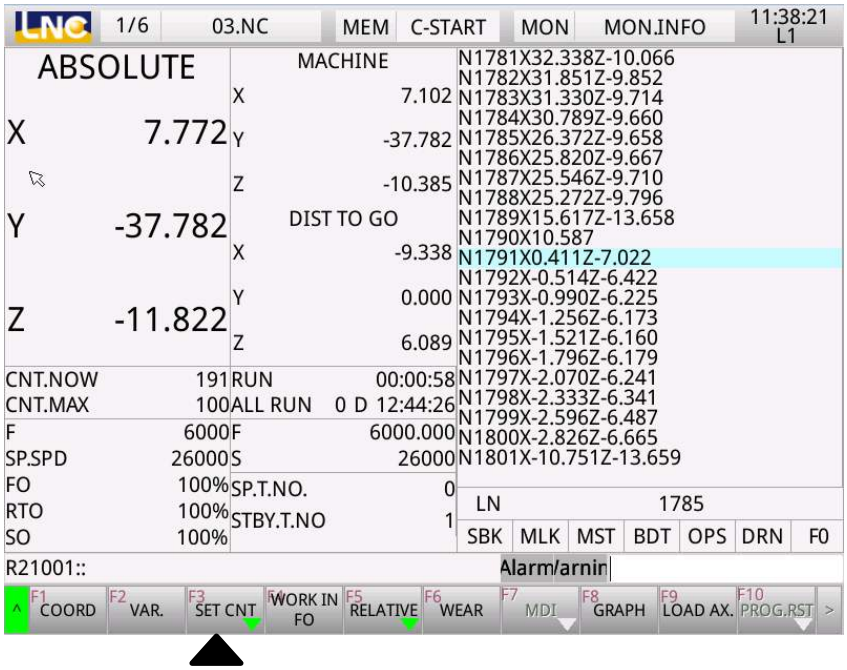
Description of machining settings:

- Users can record the quantity of workpieces to be machined and time required for workpiece machining to have better control over machining time.

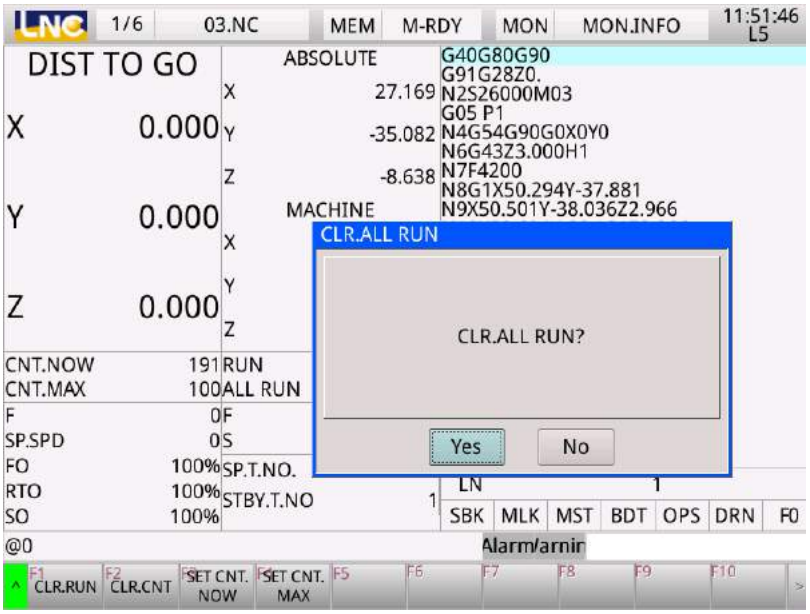


Using machining settings:

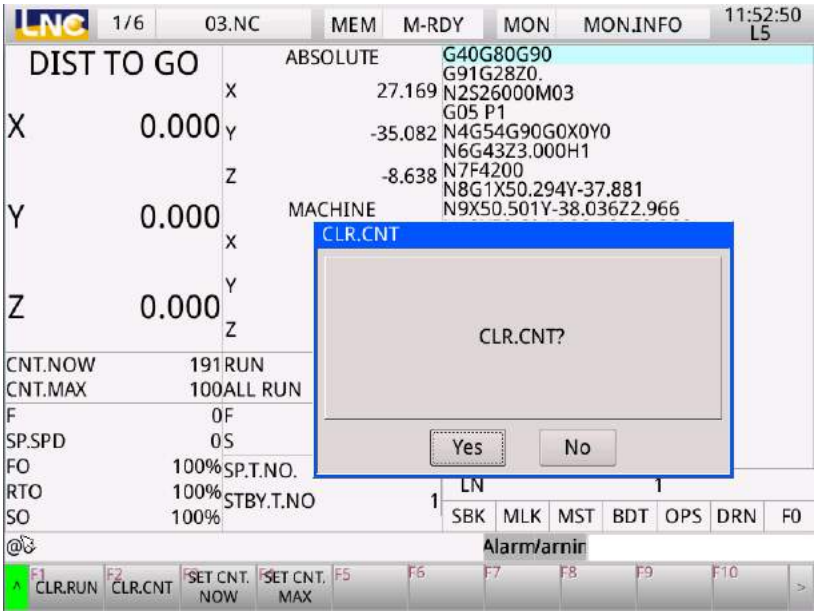
- Users can used the **<POS>** key on the MDI panel to switch to monitoring group before pressing the 【**Machining settings**】 key to enter.



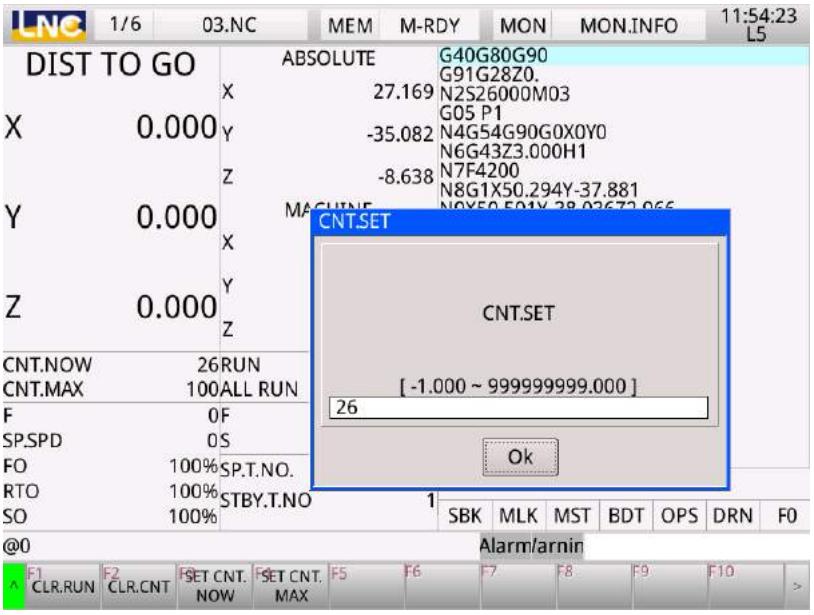
- **Clear accumulated total:** Press the 【**Clear accumulated total**】 key and select **Yes** to clear the accumulated machining time.



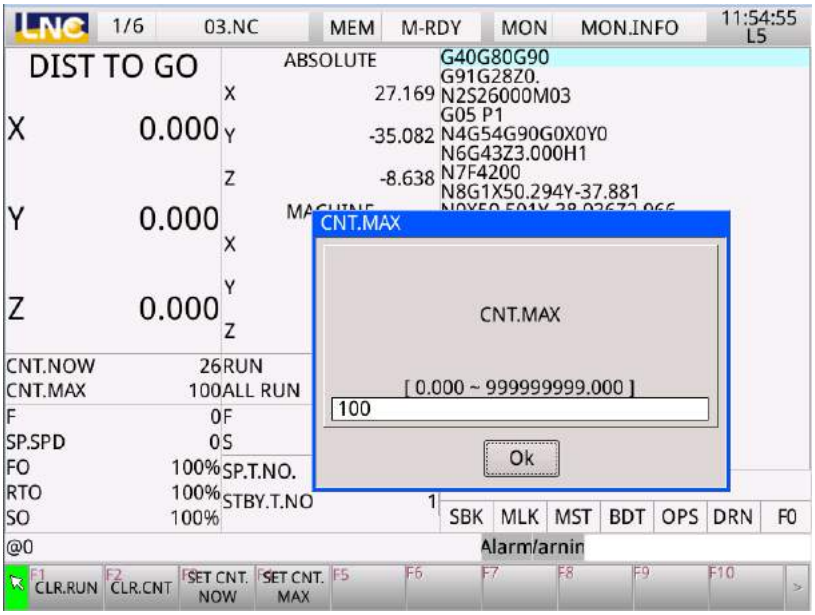
- **Clear the number of workpieces:** Press the 【**Clear the number of workpieces**】 key and select **Yes** to clear the number of workpieces which have already been machined and reset its value to 0.



- **Set the number of workpieces:** Enter a numerical value in the input area and then press the【**Set the number of workpieces**】 key for this value to be the basis for accumulating the number of workpieces.



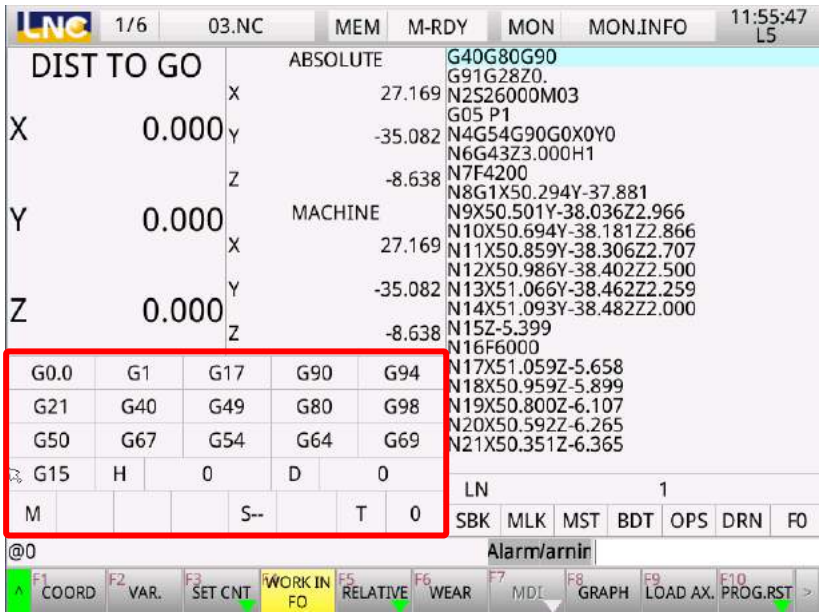
- **Set max value:** Enter a numerical value in the input area and press the 【**Set max value**】 key to set the maximum number of workpieces to be machined. When this number has been reached, an alert will be issued to notify the user.



1.3.5 Machining data

Description of machining data:

- Users can view the group of current G code (G-CODE) and information related to M code, S code, and T code on the monitoring page.



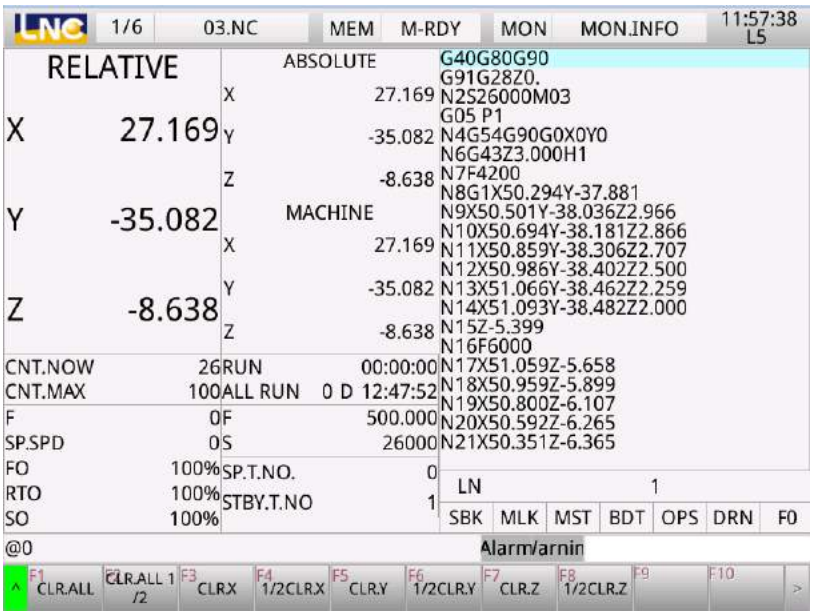
Instructions for using machining information:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group before pressing the **【Machining data】** key to display information related to machining.
- The group of G code (G-CODE) is for showing the current permanent G-CODE. For example: G00 and G01 are in the same group. When G group shows G00, it means coordinate movement by the program will be based on the current G00 status without an existing G00 or G01 command; if the G01 command has been issued by the program, G00 will be replaced by G01 in G group's G-CODE.

1.3.6 Relative coordinates

Description of relative coordinates:

- Relative coordinates show the distance between any user defined point and the current position, such that users can zero relative coordinates at any time or directly enter non-zero numerical values.



Using relative coordinates:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group before pressing the 【**Relative coordinates**】 key to show relative coordinates in the main coordinate viewing area, and to clear relative coordinates of all axial directions or a single axial direction into 1/2 of its numerical value or 0 as needed.

1.3.7 Wear

(Note: This function key is not available in the M5800 series)

Description of the wear function:

- Users can view tool wear information on the monitoring page.

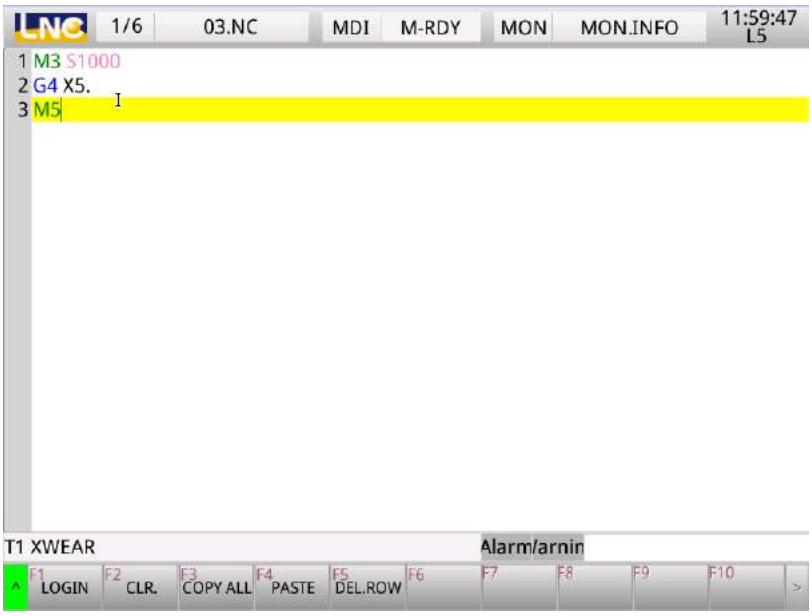
Instructions for using the wear function:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group before pressing the 【**Wear**】 key to show data related to tool wear.

1.3.8 MDI

Description of MDI functions:

- The main function of the MDI data input panel is to allow users to edit/modify program code and set numerical values. Users can also use this page to edit and save simple programs for machine operation.



Instructions for using MDI functions:

- Users should first turn the knob on the OP panel to **<MDI>**, then use the **MDI panel's <POS>** key to switch to monitoring group, pressing the 【**MDI**】 key to enter.

- Users can key in a single action command or simpler program commands on the MDI's editing page, then press the **<Input>** key to switch to the next line to continue entering commands.

Description of relevant keys:

- **Sign-in:** After editing is completed, users can press the 【**Sign-in**】 key before pressing the program start (CYCLE START) key to begin execution.
- **Clear:** Press the 【**Clear**】 key to clear all commands on the editing page.
- **Copy all:** Press the 【**Copy all**】 key to copy all commands currently on the editing page.
- **Paste:** Press the 【**Paste**】 key to paste the copied command.
- **Delete row:** Press the 【**Delete row**】 key to delete the command row where the cursor is currently located.

- **For detailed instructions, please refer to section 1.8 for instructions on Using MDI Functions**

1.3.9 Figures

Description of figures:

- It is for users to view the current machining program's path on the monitoring page.

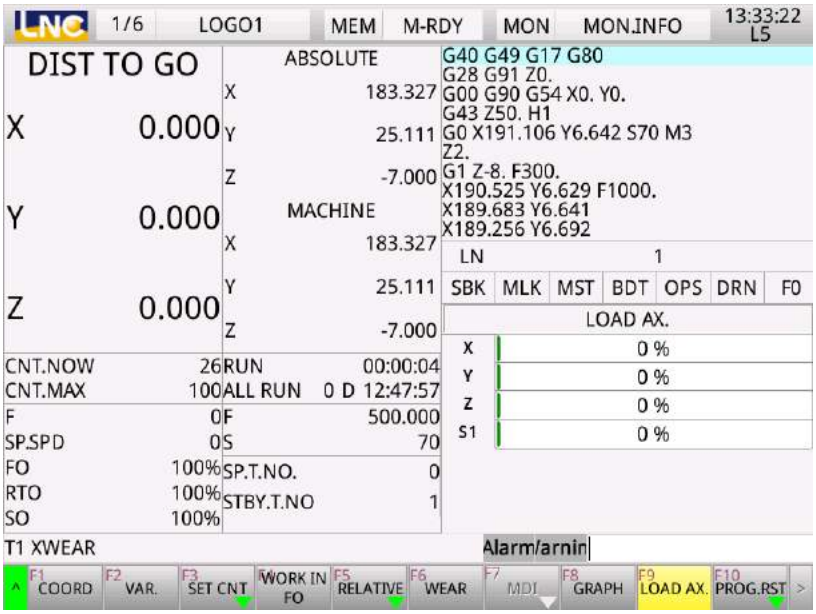
Instructions on how to use figures:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group, then press the 【**Figures**】 key to show the current machining program's path.

1.3.10 Load

Description of the load function:

- It allows users to view load data along each axial direction during machining on the monitoring page.



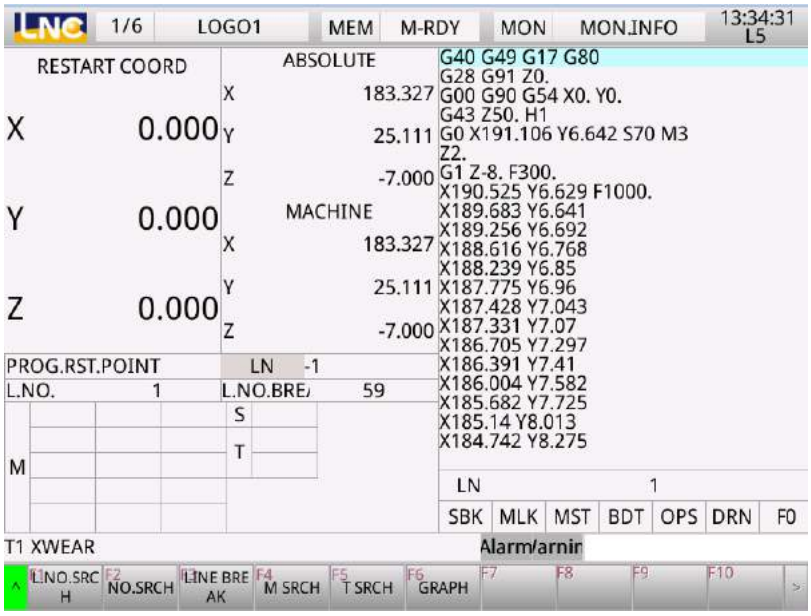
Instructions for using load:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group before pressing the 【**Load**】 key to show the current load for each axial direction.

1.3.11 Program restart

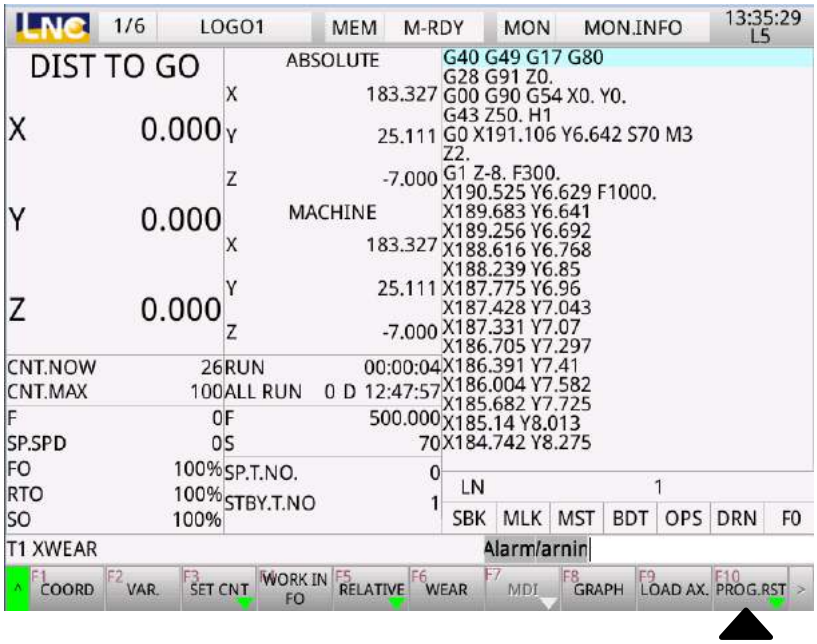
Description of the program restart function:

- When the machining operation is paused due to tool damage, users can record the program line number, change the tool, then use the program restart function to resuming the machining process; this reduces the time required for no-load operations and avoids the situation where the workpiece is not in contact with the tool.



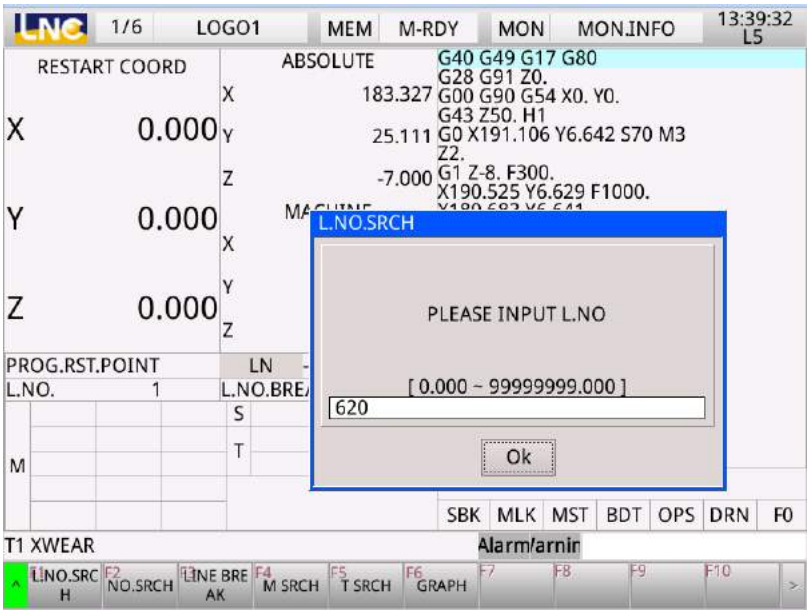
Instructions for program restart:

- Users can use the **<POS>** key on the MDI panel to switch to monitoring group, then press the **【Program restart】** key to enter the program restart page. The program restart function is only available under automatic (MEM) mode.

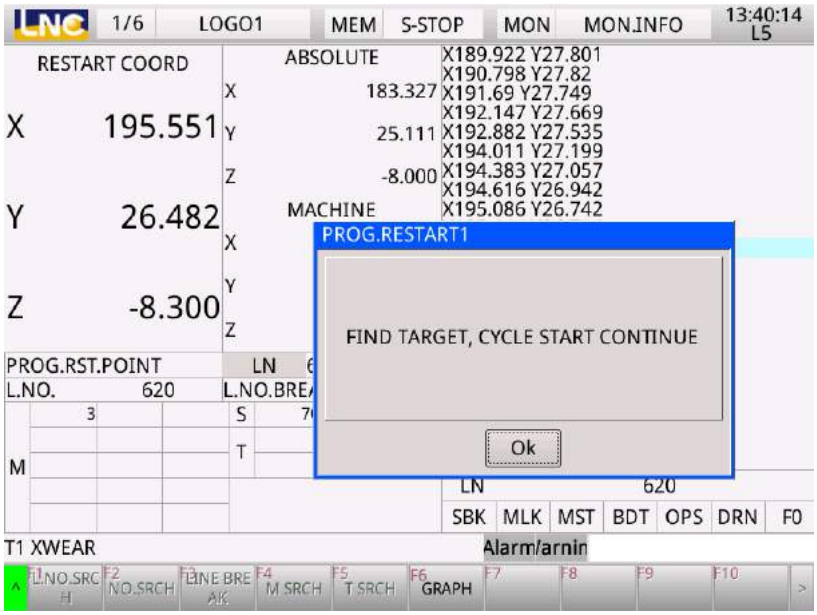


- **Search by line number:** 1. Press the 【**Line number search**】 key.

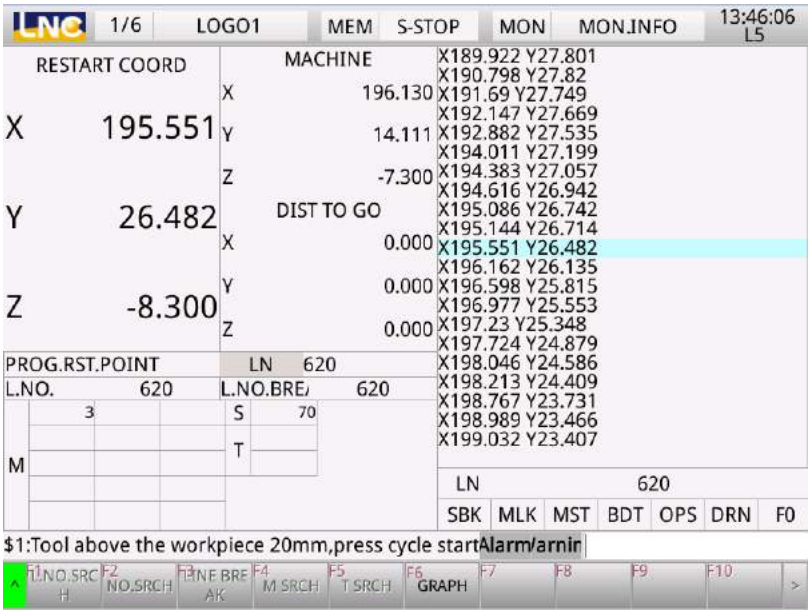
2. Enter the program line number you wish to find, and then press **Confirm** to start searching for that specific line number.



3. After the designated line number is found, a confirmation message window will appear as shown in the figure below.

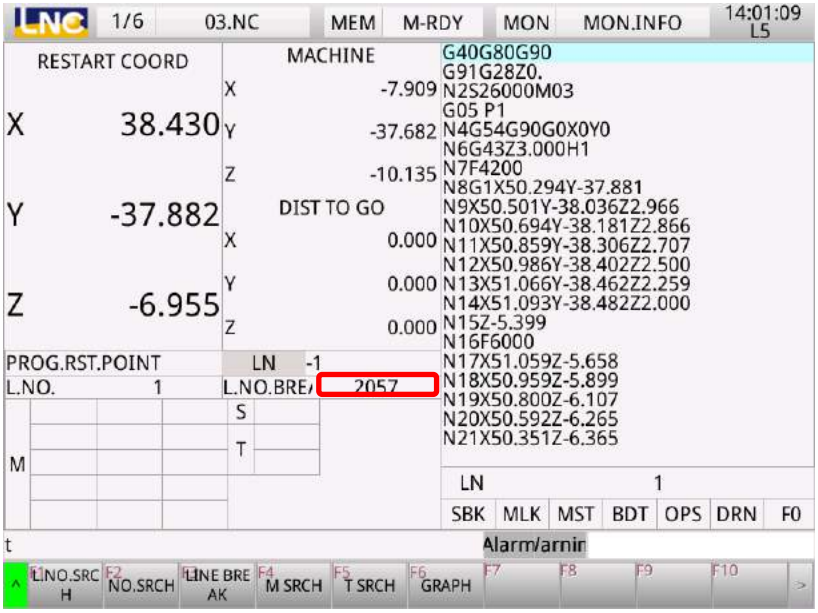


4. Press **Confirm** to close the window and press the program start (CYCLE START) key for the tool to move to the coordinate position of that program block and stop at 20 mm above the workpiece, while a confirmation window will appear (as shown in the figure below). Press **Confirm** to start the tool and continue with machining.



- **Search by serial number:** 1. Press 【**Serial number search**】 key.
  2. Enter the program serial number you wish to find and press **Confirm** to start searching for the specific serial number.
  3. After the designated serial number is found, a confirmation message window will appear indicating that the target has been found.
  4. Press **Confirm** to close the window and then press the program start (CYCLE START) key for the tool to move to the coordinate position of that block and stop at 20 mm above the workpiece; a confirmation window will also pop up. Press **Confirm** to start the tool and continue with machining.

- **Search by line break:** 1. Press **the** 【**Line break search**】 key to start searching for the line number of previous program line break recorded by NC.



2. After the line break's line number is found, a confirmation message window will appear indicating that the target has been found.
3. Press **Confirm** to close the window and then press the program start (CYCLE START) key for the tool to move to the coordinate position of that block and stop at 20 mm above the workpiece; a confirmation window will also pop up. Press **Confirm** to start the tool and continue with machining.

- **Search by M code:** 1. Press the 【**M code search**】 key.
  2. Enter the numerical value of the program M code you wish to find, then press **Confirm** to start searching for the M code.
  3. After the designated M code is found, a confirmation message window will pop up indicating the target has been found.
  4. Press **Confirm** to close the window and then press the program start (CYCLE START) key for the tool to move to the coordinate position of that block and stop at 20 mm above the workpiece; a confirmation window will also pop up. Press **Confirm** to start the tool and continue with machining.

- **Search by T code:** 1. Press the 【**T code search**】 key.
  2. Enter the numerical value of the program T code you wish to find, then press **Confirm** to start searching for the T code.
  3. After the designated T code is found, a confirmation message window will pop up indicating the target has been found.

4. Press **Confirm** to close the window and then press the program start (CYCLE START) key for the tool to move to the coordinate position of that block and stop at 20 mm above the workpiece; a confirmation window will also pop up. Press **Confirm** to start the tool and continue with machining.

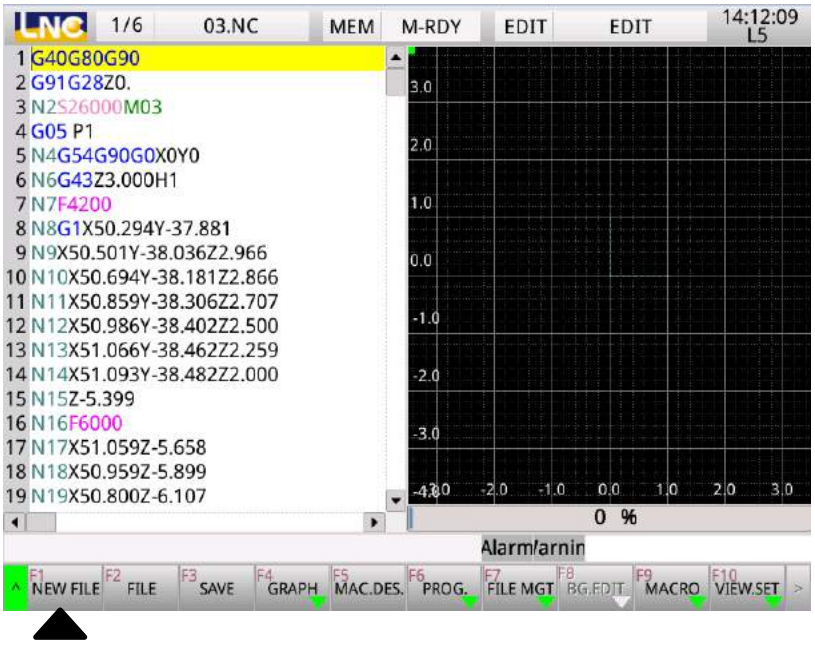
- **For detailed instructions, please refer to section 1.8 for instructions on how to perform a Program Restart**

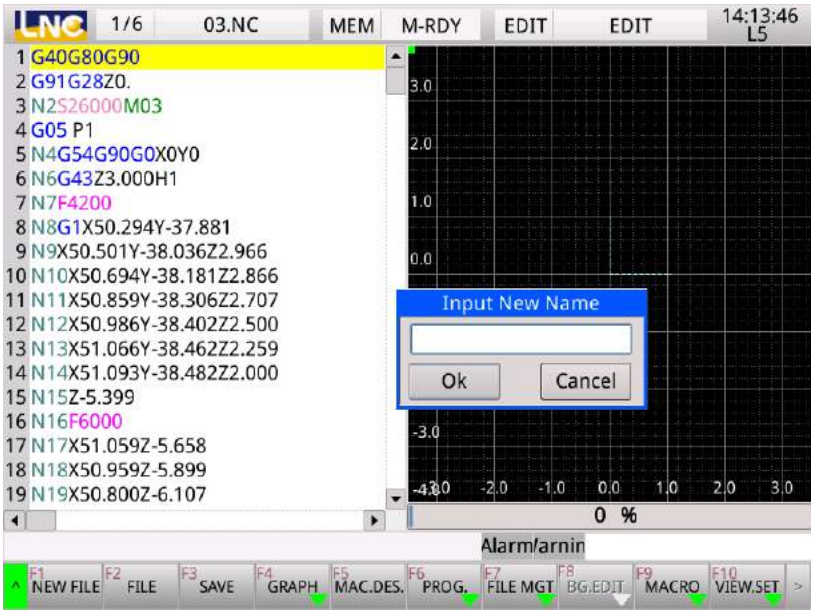
1.4 Program group (PROG)

1.4.1 Select a file to be opened

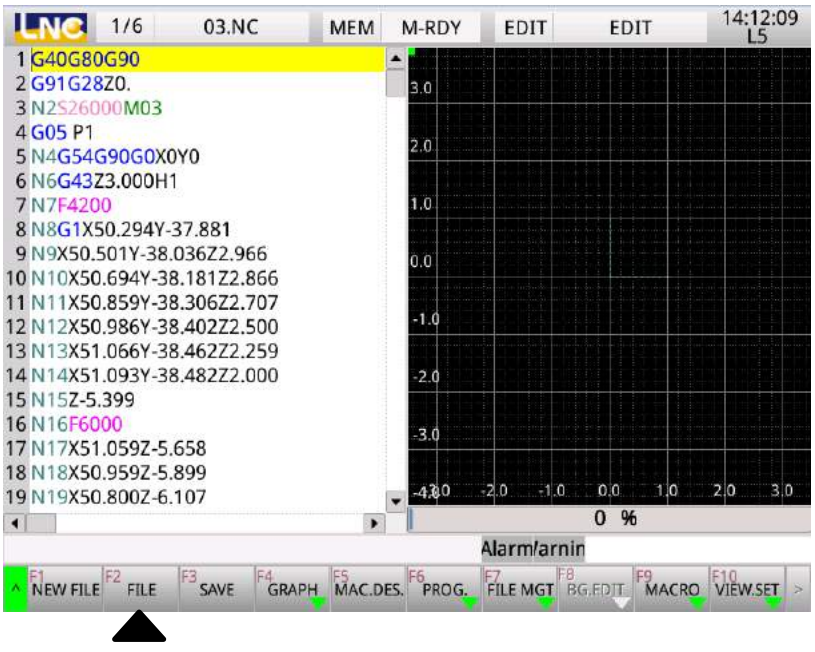
Instructions on file management:

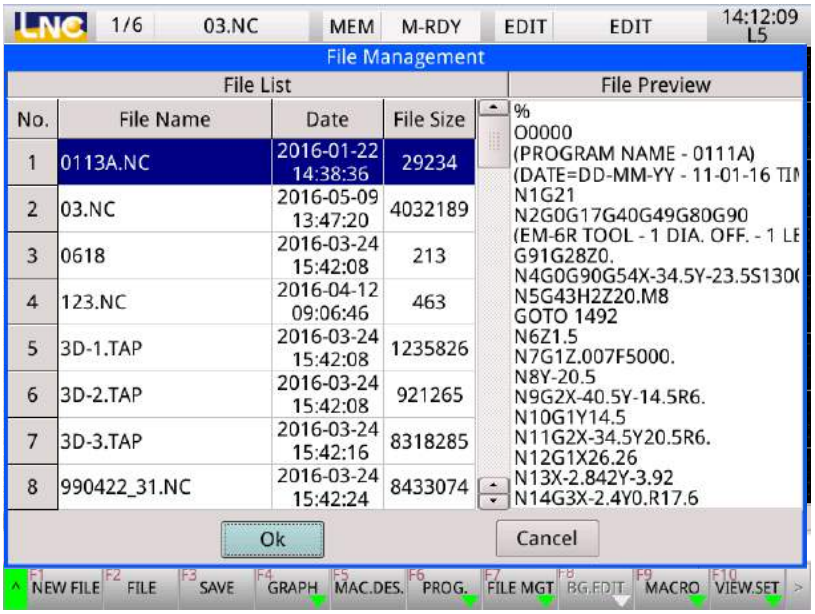
- **Opening a new file:** Users can use the **<PROG>** key on the MDI panel to switch to editing group before pressing the 【**Open new file**】 key for the window requesting a new filename to appear. After entering a new filename and selecting **Confirm**, the open new file operation is completed, or users can select **Cancel** to return to the previous page.



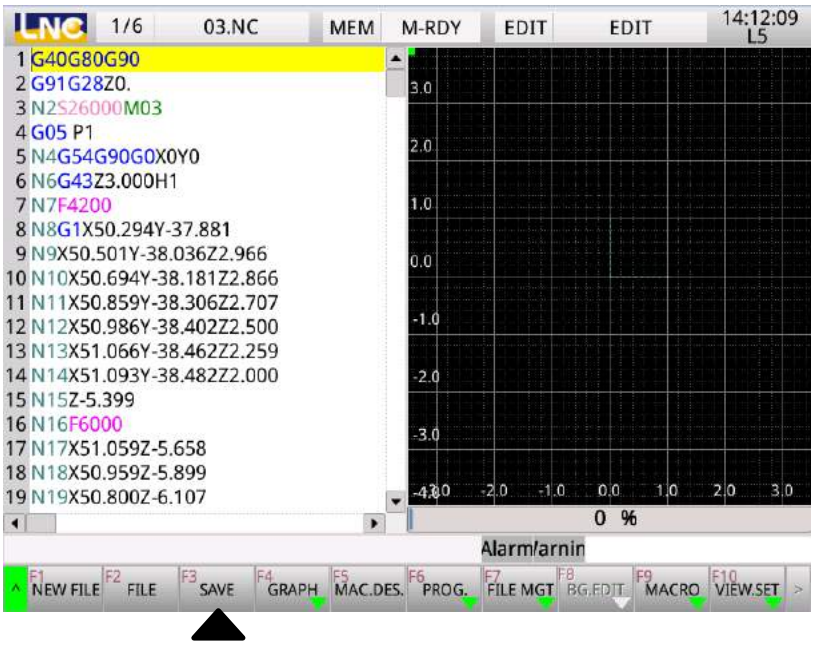


- **Open existing file:** Users can use the **<PROG>** key on the MDI panel to switch to editing group before pressing the 【**Open existing file**】 key to enter the file list. Move the cursor to the filename to be opened and select **Confirm**, the selected file will be opened.





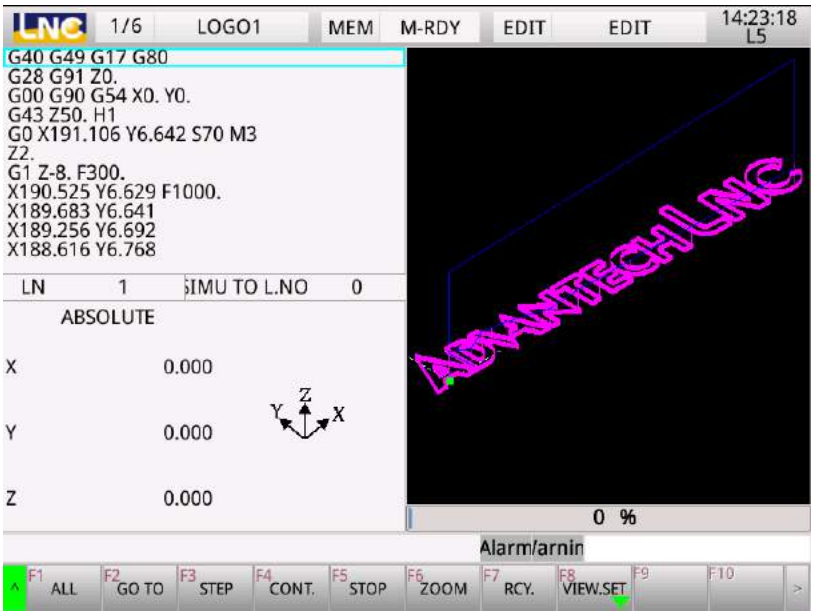
- **Saving a file:** After editing is completed, users can press 【**Save file**】 key to save the file.



1.4.2 Preview

Preview function description:

- Users can view the current machining program and a graphic simulation of the machining path via the preview page. Users can confirm the machining path via this simulated figure.



Instructions for using preview:

- On the preview page, the MDI panel's **<PROG>** key can be used to switch to editing group before pressing the 【**Preview**】 key for using the preview function. All function keys are described below:

  【**Preview all**】: drawing all machining paths of the current program.

  【**Preview to**】: only drawing the machining path up to the current cursor position in the program.

  【**Single step**】: drawing program machining path via a single block execution method.

  【**Continuous**】: continuously drawing program machining paths.

  【**Stop**】: stopping figure preview for the cursor to return to the header position.

  【**Zoom**】: using arrow keys to move the square over the area you wish to magnify, followed by using **<PgUp>** and **<PgDn>** on the MDI panel to adjust the size of square for setting the preview range.

  【**Undo**】: restoring to the preview figure before the zoom operation.

  【**Plot settings**】: setting the viewing angle and range of display during path drawing.

- **For detailed instructions, please refer to section 1.8 for instructions on Preview Function description**

1.4.3 Supplementary commands

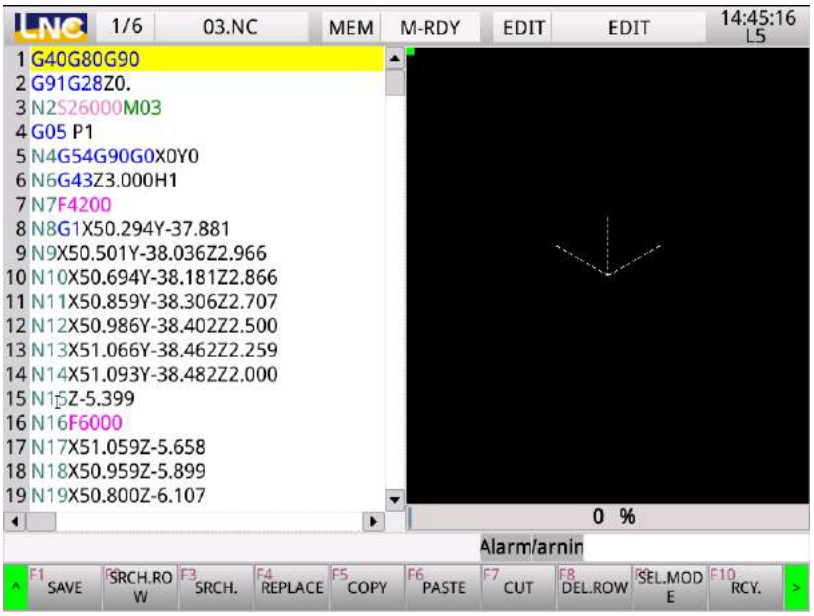
Description of supplementary commands:

- During program editing, users can press the 【**Supplementary commands**】 key to browse the introduction and description of the command at cursor position inside the machining program currently being edited.

1.4.4 Program editing

Description of the program editing function:

- Users can edit the required machining program commands on the program editing page. Characters can be entered at the cursor position, while the **<Input>** key can be used to move to a new line. Users can also use a combination of different function keys including machining program preview, saving the program file, finding a line position in the program, finding characters in the program, replacing characters in the program, selection mode in the program, cut/copy/paste of characters in the program, delete row, undo, and redo.



Instructions for editing a program:

- Users can use the **<PROG>** key on the MDI panel to switch to editing group, followed by pressing the 【**Edit**】 key to access program editing functions.
- Key descriptions which can be accessed by pressing the 【**Edit**】 key are as shown below.

  【**Save file**】: Save the edited program.

  【**Row position**】: Search for a line number in the program.

  【**Find**】: Search for a string or character in the program.

  【**Replace**】: Search for a string or characters in the program and replace it with a new string or character.

  【**Copy**】: Copy the contents of a selected area.

  【**Paste**】: Paste the cut or copied content.

  【**Cut**】: Cut the contents of a selected area.

  【**Delete row**】: Delete the row highlighted by the cursor, and delete the contents in the selected area.

  【**Selection mode**】: Use arrow keys to mark an area after entering selection mode, which can be used in coordination with cut, paste, delete row, and other function keys.

  【**Undo**】: Restore to previous editing status.

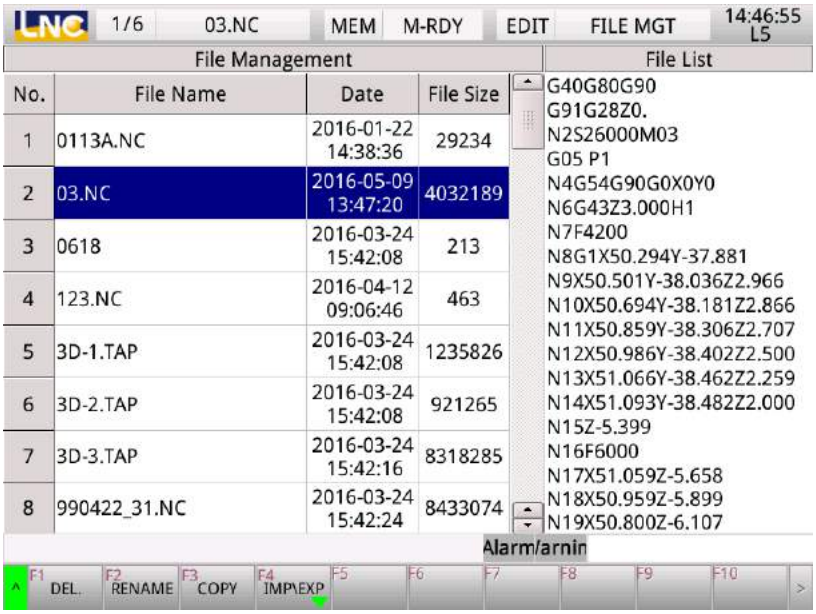
  【**Redo**】: This function is for returning to the action prior to undo.

- **For detailed instructions, please refer to section 1.8 for instructions on how to Open and Edit a File**

1.4.5 File management

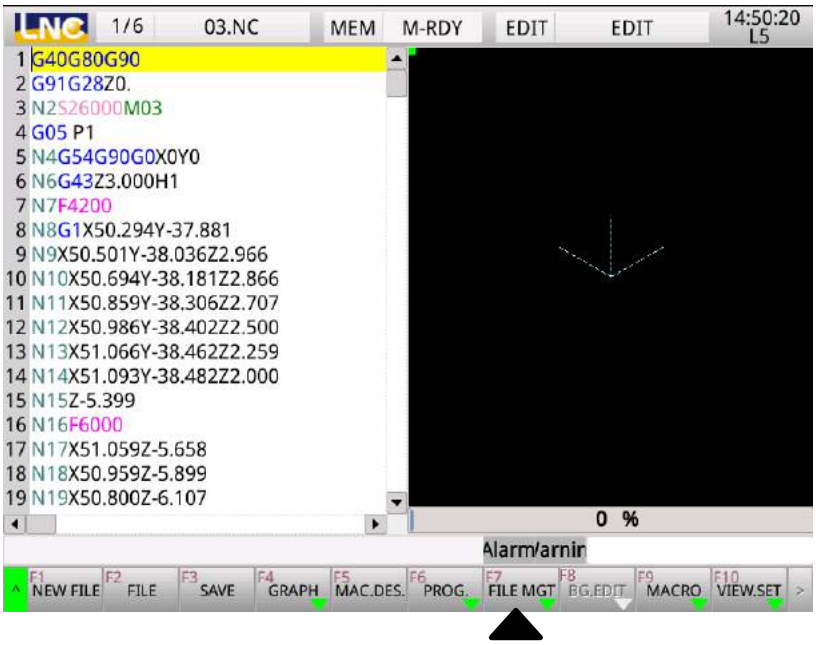
Description of file management functions:

- It allows users to delete, rename, and copy machining programs inside the controller; it can also export machining programs to a USB device or import machining programs from a USB device.

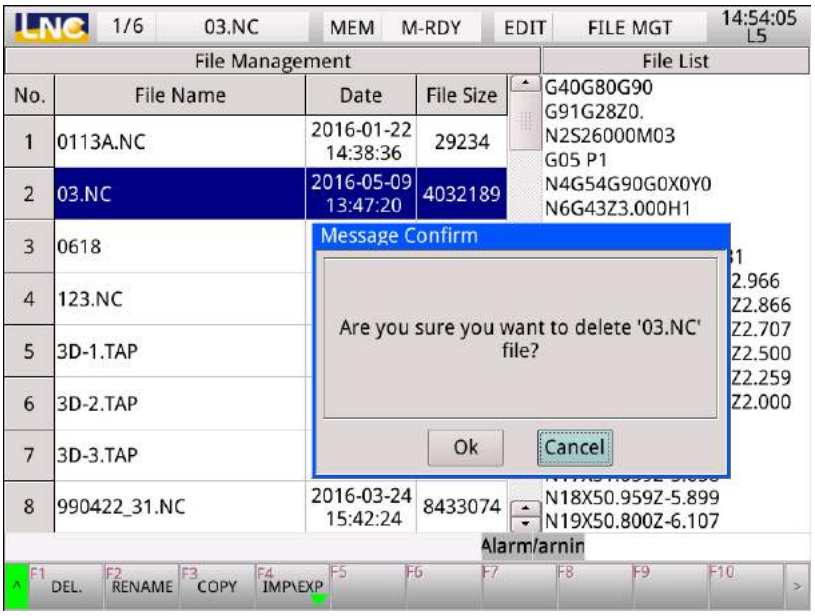


Instructions on file management:

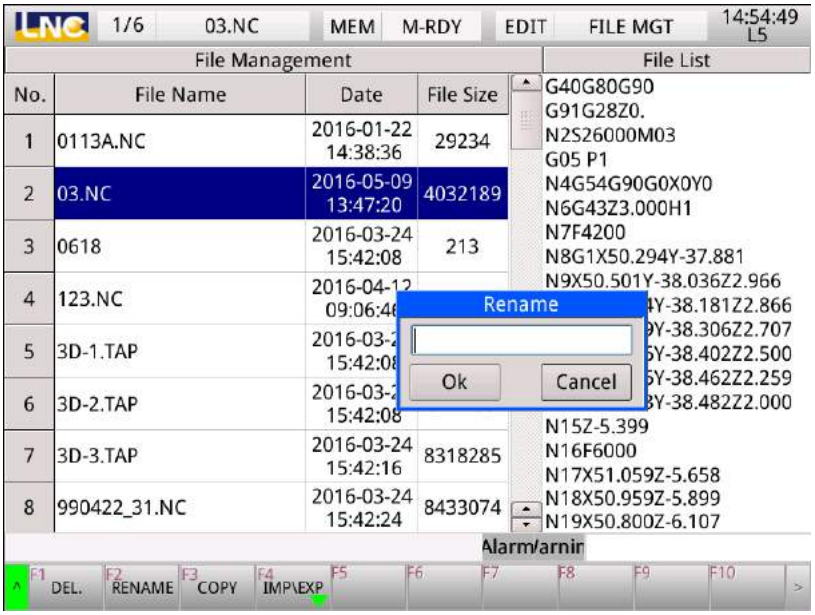
- Users can use the **<PROG>** key on the MDI panel to switch to editing group, then press the **【File management】** key to access the file management page.



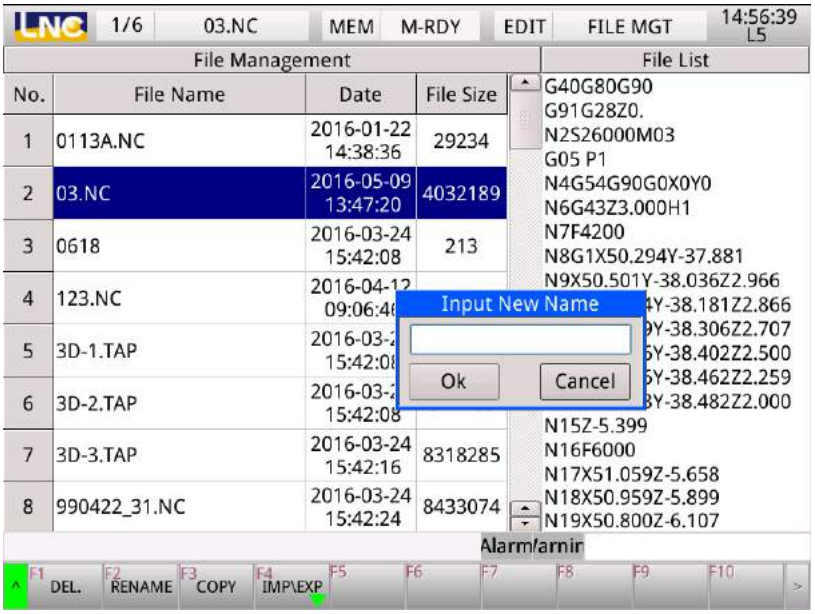
- **Delete:** After moving the cursor to the filename to be deleted, a confirmation message will appear after pressing the 【**Delete**】 key; if this action is confirmed, users should select **Confirm** to complete file deletion or select **Cancel** to return.



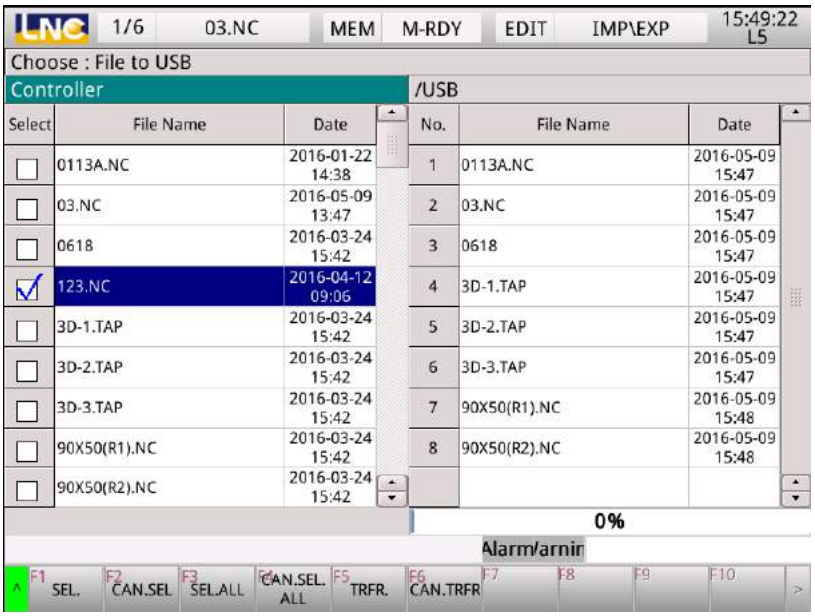
- **Rename:** After moving the cursor to the filename to be renamed, a window requesting a new filename will appear after pressing the 【**Rename**】 key. A new filename should be entered followed by selecting **Confirm** to complete renaming the file, or users can select **Cancel** to return.



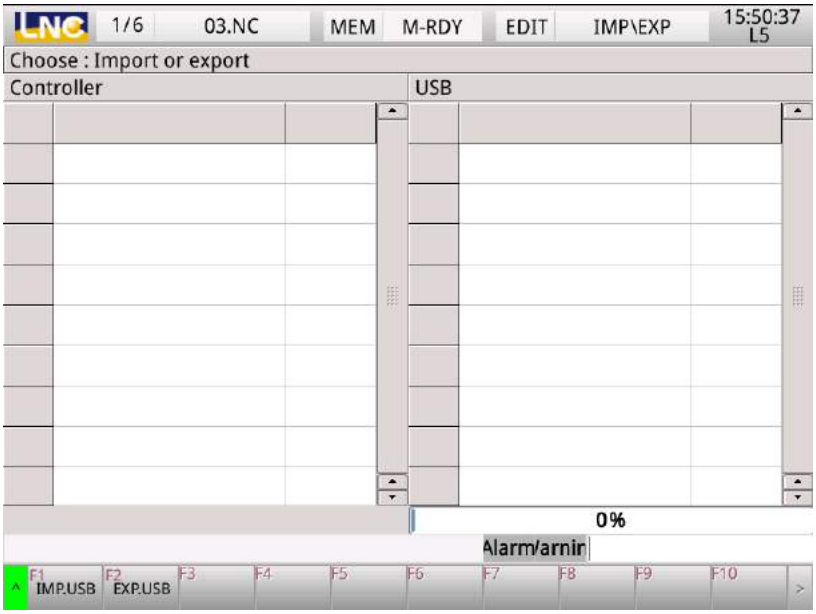
- **Copy:** After moving the cursor to the filename to be renamed, a window requesting a new filename will appear after pressing the 【**Copy**】 key. A new filename for the copied file should be entered, select **Confirm** to complete the file copy operation or select **Cancel** to return.



- **Import\Export:** Export machining programs inside the controller to a USB device, or importing machining programs from a USB device to the controller.



Press **the** 【**Import\Export**】 key to access this function and select [Import from USB] or [Export to USB].



Key function descriptions:

【**Select**】: select a single file.

【**Cancel select**】: cancel the current file selection.

【**Select all**】: select all files.

【**Cancel select all**】: cancel all selected files.

【**Transfer**】: proceed with file import/export.

【**Cancel transfer**】: cancel file transfer of the file to be imported/exported.

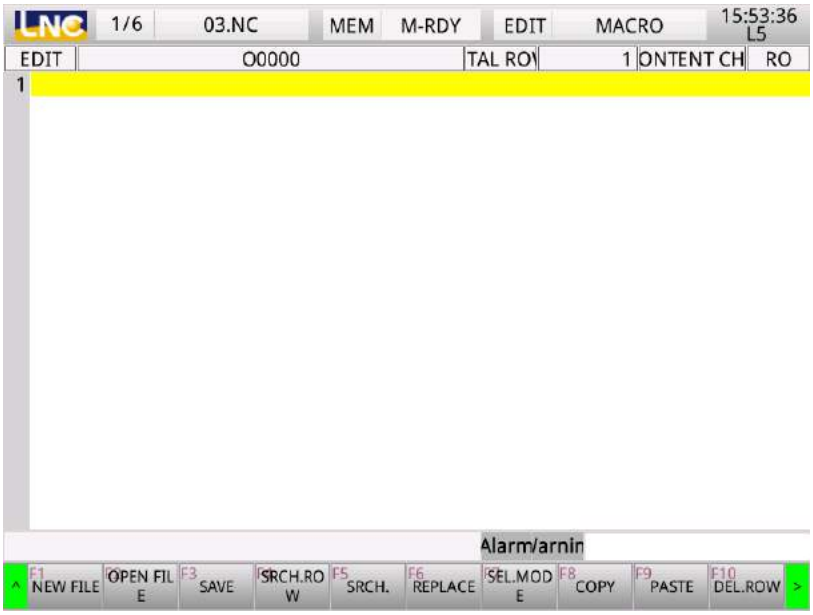
- **For detailed instructions, please refer to section 1.8 for instructions on File Backup - Import and File Backup - Export**

1.4.6 Manufacturer macros

1. Program editing

- Users can edit macros according to their own needs, such as: By editing the square milling program, users can modify it for either square or rectangular workpieces by changing arguments in the macro without editing the program, which saves a lot of time.

  Please note: There are specific regulations associated with the filename, please refer to the section on (Editing and using manufacturer macros).



- Manufacturer macro – users can use the **<PROG>** key on the MDI panel to switch to editing group before pressing 【**Manufacturer macro**】 and 【**Program editing**】 keys to access the program editing page.
- The list below contains the description of each key:

  【**Create a new file**】: Create a new manufacturer macro.

  【**Open file**】: Open a manufacturer macro at the controller end.

  【**Save file**】: Save the modified manufacturer macro.

  【**Row position**】: Search for a line number in the manufacturer macro.

  【**Find**】: Search for a string or character in the manufacturer macro.

  【**Replace**】: Search for a string or character in the manufacturer macro and replace it with a new string or character.

  【**Selection mode**】: Use the arrow keys to mark the area after entering selection mode, which can be used in coordination with cut, paste, delete row, and other function keys.

  【**Copy**】: Copy the contents of a selected area.

  【**Paste**】: Paste the cut or copied content.

  【**Delete row**】: Delete the row highlighted by the cursor, and delete the contents in the selected area.

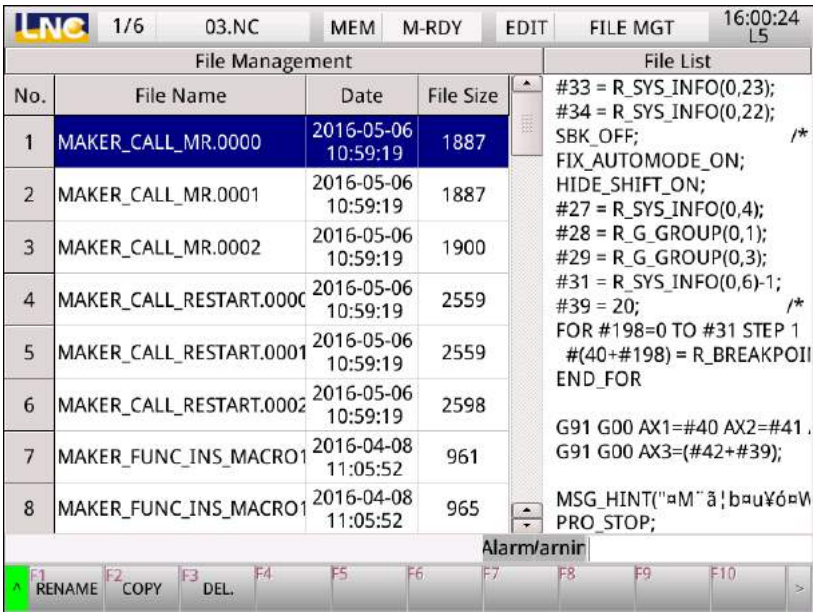
  【**Cut**】: Cut the contents of a selected area.

  【**Undo**】: Restore to previous editing status.

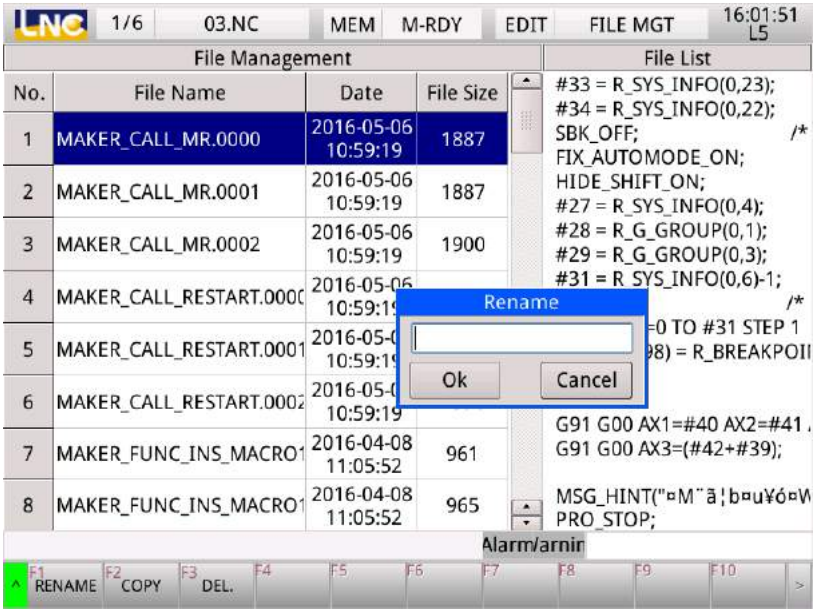
【**Redo**】: This function is for returning to the action prior to undo.

2. File management

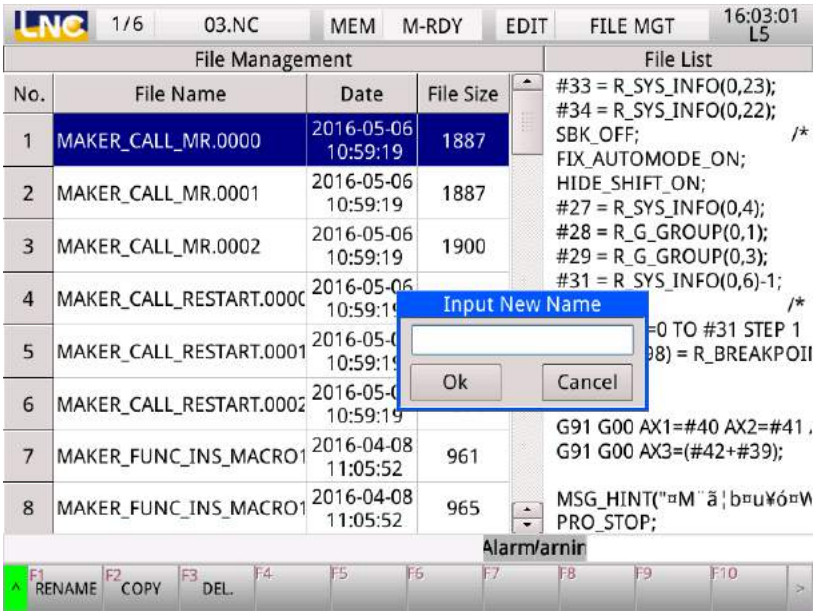
- It can be used to rename, copy, and delete manufacturer macros in the controller.



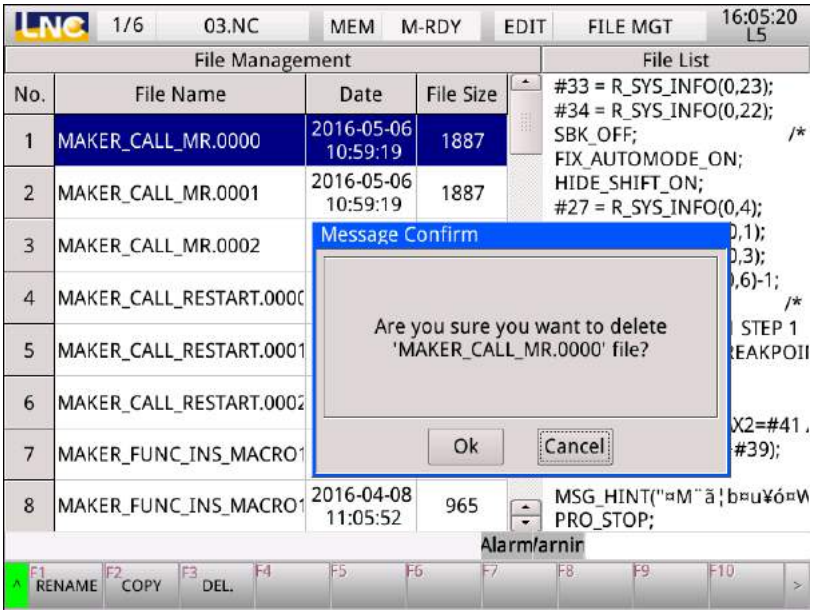
- Manufacturer macro - users can use the MDI panel's **<PROG>** key to switch to editing group before pressing the 【**Manufacturer macro**】 key and the 【**File management**】 key to access the file management page.
- **Rename:** After using cursor to select the filename to be renamed, a window prompting the input of a new file name will be shown by pressing the 【**Rename**】 key. New file name should be entered before selecting **Confirm**, or users can select **Cancel** to return.



- **Copy:** After using the cursor to select the filename to be renamed, a window prompting the input of a new file name will be shown by pressing the 【**Copy**】 key. New file name should be entered before selecting **Confirm**, or users can select **Cancel** to return.



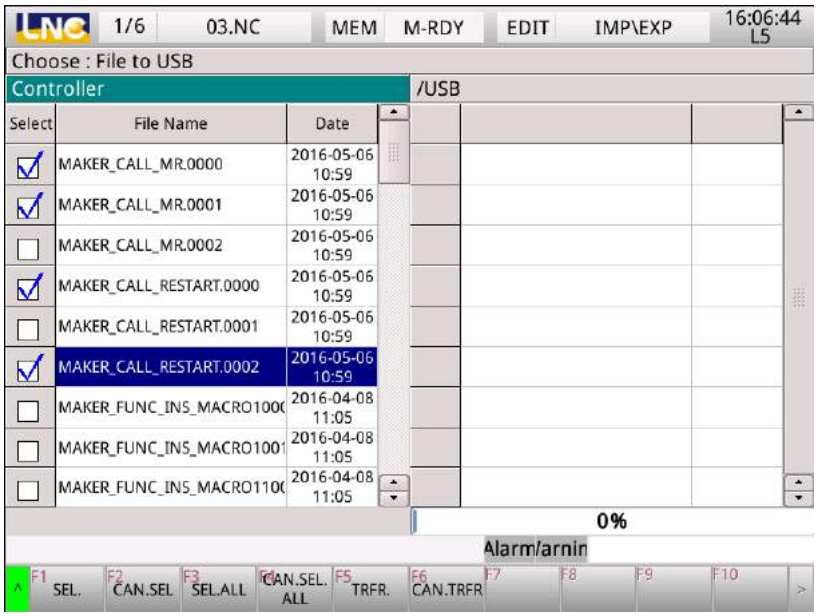
- **Delete:** After using the cursor to select the file to be deleted, a confirmation message will be shown by pressing the 【**Delete**】 key. If this action is confirmed, users should select **Confirm** to continue or **Cancel** to return.



- **For detailed instructions, please refer to section 1.8 for instructions on Editing and Using Manufacturer Macros**

3. Import\Export

- Via a USB device, self-defined macros in the controller can be exported to the USB device, or machining programs in the USB device can be imported to the controller.



- Press the 【**Import\Export**】 key to access this function and then select "Import from USB" or "Export to USB".
- Key function descriptions:

  【**Select**】: select one single macro program file to be imported/exported.

【**Cancel select**】: cancel the current file selection.

【**Select all**】: select all macro files.

【**Cancel select all**】: cancel all selected files.

【**Transfer**】: import/export the macro files.

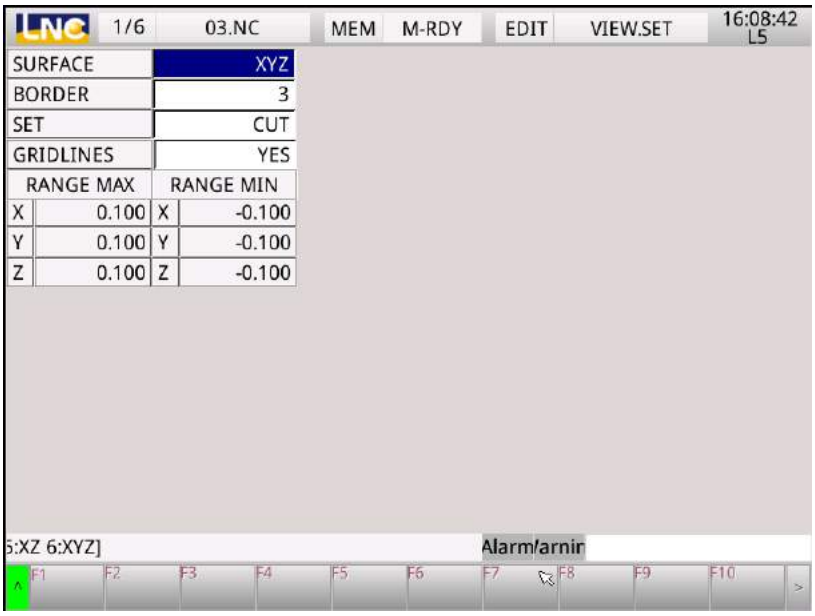
【**Cancel transfer**】: cancel the import/export of macro files.

- **For detailed instructions, please refer to section 1.8 for instructions on File Backup - Import and File Backup - Export**

1.4.7 Plot settings

Description of Plot settings:

- Set the viewing angle and display range during path drawing.



Plot settings instructions:

- Users can use the **<PROG>** key on the MDI panel to switch to editing group before pressing the 【**Plot settings**】 key to enter. Users can select path directions along each axial direction.
- Drawing planes:
  - 0: Drawing the program path by using the XY plane (2D) approach.
  - 1: Drawing the program path by using the YZ plane (2D) approach.
  - 2: Drawing the program path by using the ZX plane (2D) approach.
  - 3: Drawing the program path by using the YX plane (2D) approach.
  - 4: Drawing the program path by using the ZY plane (2D) approach.
  - 5: Drawing the program path by using the XZ plane (2D) approach.
  - 6: Drawing the program path by using XYZ planes (3D) approach.
- Reserved boundaries: Set the preserved boundary values of the path display screen.
- Setting method
  - 0: Manual.
  - 1: Preview result – full stroke.
  - 2: Preview result – cutting stroke.

- Figure grid lines

  0: No.

  1: Yes.

- **For detailed instructions, please refer to section 1.8 for instructions on Preview Function description**

1.5 Compensation group (OFFSET)

1.5.1 Coordinate system

Description of coordinate system functions:

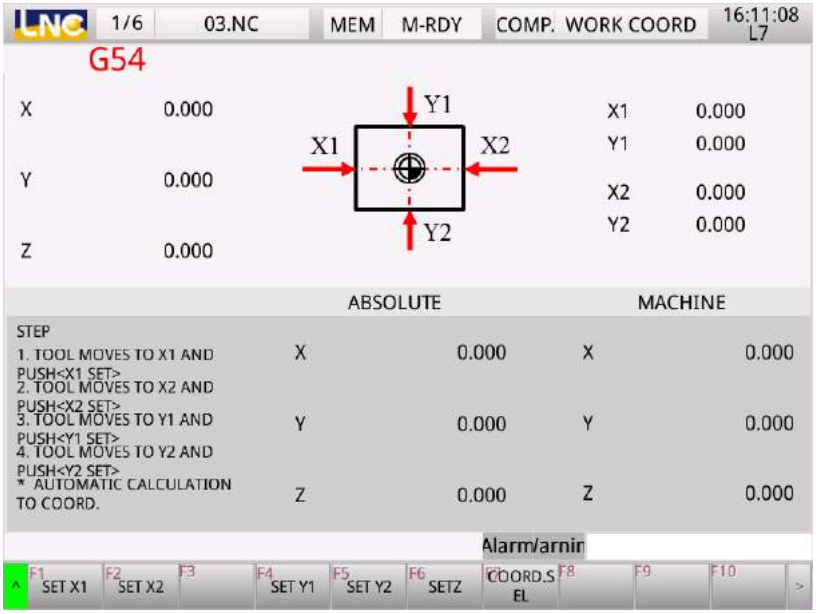
- The controller provides users with 6 sets of workpiece coordinate systems including G54, G55, G56, G57, G58, and G59, as well as 100 sets of extended workpiece coordinate systems from G54P1 to G54P100. Left and right arrow keys on the MDI panel can be used to select the workpiece coordinate system to be set up.

Instructions for using the coordinate system:

- Users can use the **<OFFSET>** key on the MDI panel to switch to compensation group before pressing the **【Coordinate system】** key to access the coordinate system page.
- After entering the coordinate system, the cursor should be moved to the coordinate system to be set up such as G54 to G59, G54P1 to G54P100. The controller provides three options for users to quickly set up coordinate values in accordance with the properties of the workpiece.

1. Center of a rectangle

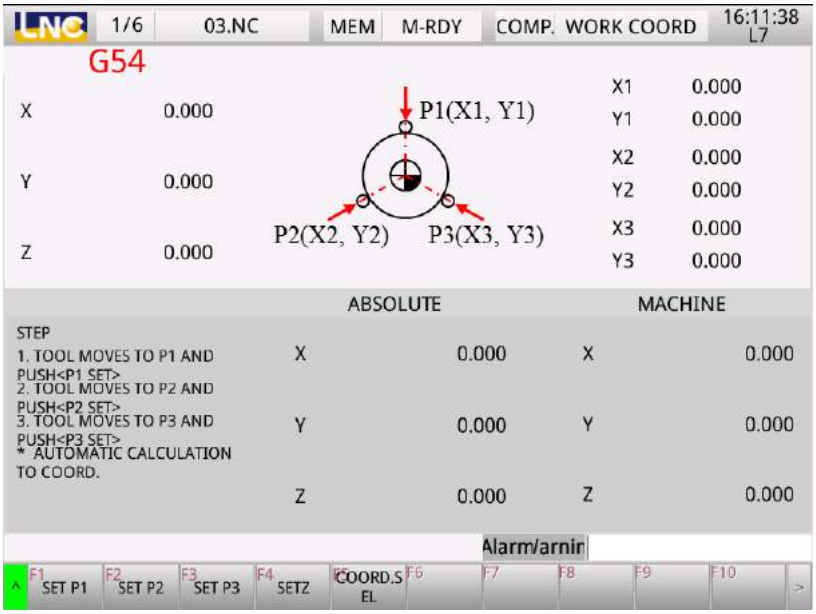
- It is for users to set up the center point of a rectangular shaped workpiece.



- Press the 【**Rectangle**】 key to enter the rectangular workpiece coordinate system settings screen, press 【**Coordinate selection**】 to enter the coordinate system you wish to set up, and then allow the measuring instrument to come in contact with the workpiece in accordance with the drawing showing the center point of a rectangle. 【**Set X1**】 should be pressed after contacting X1, 【**Set X2**】 should be pressed after contacting X2, 【**Set Y1**】 should be pressed after contacting Y1, 【**Set Y2**】 should be pressed after contacting Y2, and the controller will automatically derive the center of a rectangular workpiece.

2. Center of a circle

- It is for users to set up the center point of a circular shaped workpiece.



- Press the 【**Circle**】 key to enter the circular workpiece coordinate system settings screen, press 【**Coordinate selection**】to enter the coordinate system you wish to set up, and then allow the measuring instrument to come in contact with the workpiece in accordance with the drawing showing the center point of a circle. 【**Set P1**】 should be pressed after contacting P1, 【**Set P2**】 should be pressed after contacting P2, 【**Set P3**】 should be pressed after contacting P3, and then the controller will automatically derive the center of a circular workpiece based on the positions of these three points.

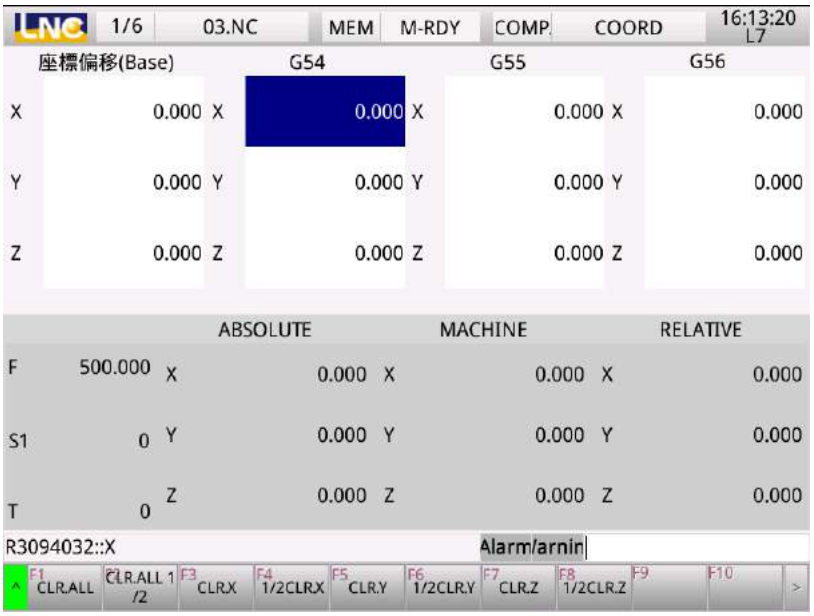
3. Instruction input (Set all, set X, set Y, set Z, ...)

- It is for users to set up workpiece coordinates (non-square, non-circular).

- Press the 【**Guided input**】 key to enter the settings page of guided input for the workpiece coordinate system, allow the measuring instrument to be in contact with the workpiece and then press 【**Set X**】, 【**Set Y**】, and 【**Set Z**】 keys for the controller to set up each axis according to the current contact position of the measuring instrument. Users can also press the 【**Set all**】 key to set the machine coordinate values of each axis on the current machine as the work coordinate system at current cursor position.

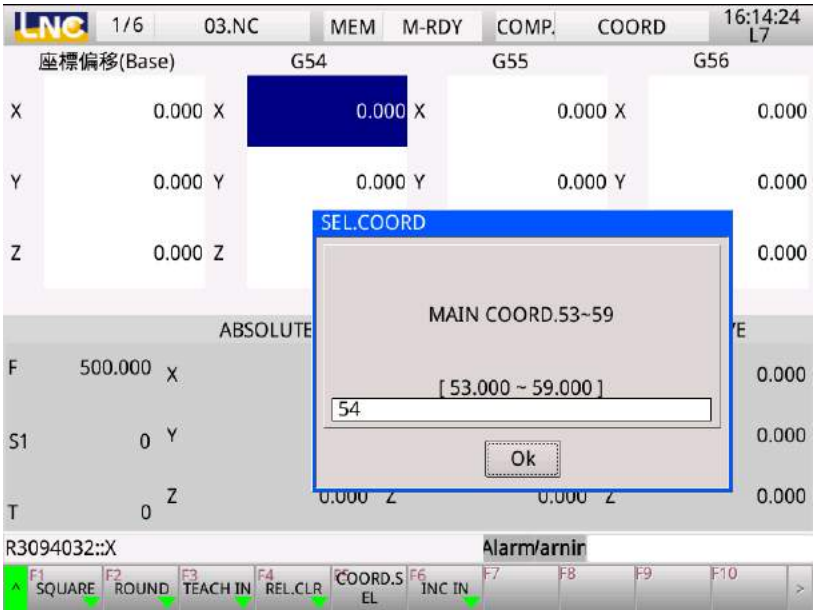
4. Clear relative

- It is for users to clear the relative coordinates of all axial directions or one single axial direction into 1/2 the numerical value or 0, when necessary.



5. Coordinate selection

- It is for users to quickly search for a coordinate system. Press the 【**Coordinate selection**】 key to enter the screen for searching workpiece coordinate systems, enter the number of the coordinate system into the dialog box and press **Confirm** for the cursor to move to that coordinate system. It can facilitate search convenience, but the sub-coordinate system can only be used by G54.



- **For detailed instructions, please refer to section 1.8 for instructions on Coordinate System Settings**

1.5.2 Tool management

Description of tool management functions:

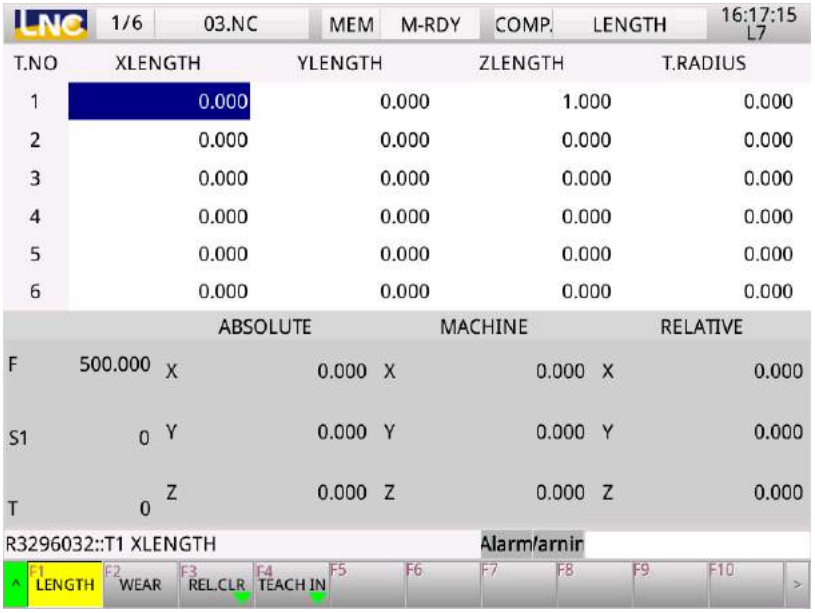
- Users can manually enter a numerical value in accordance with different types of tools via the tool offset screen.

Instructions for using the tool offset:

- Users can use the **<OFFSET>** key on the MDI panel to switch to compensation group, then press the **【Tool management】** key to access the tool offset page.

1. Tool length offset

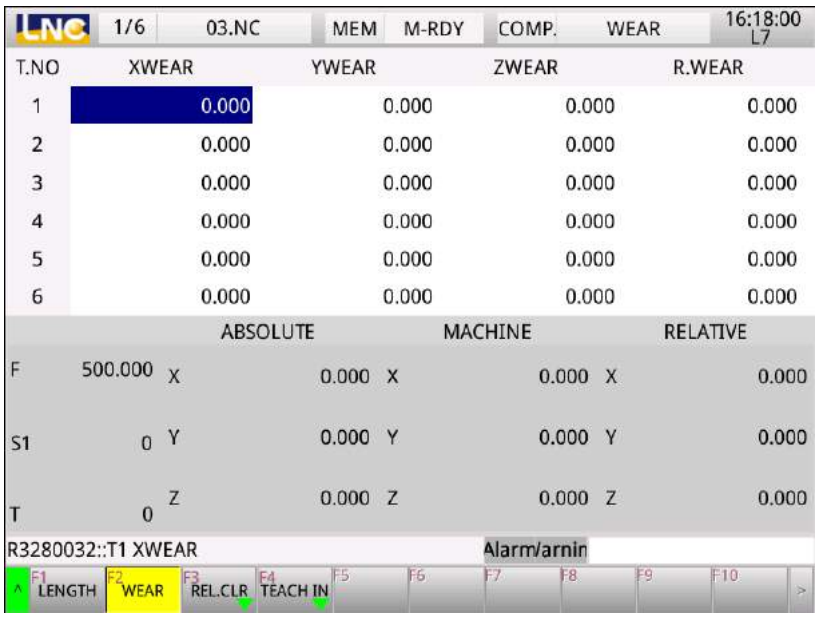
- Press the **【Length】** key to access the tool length offset page.
- There are a total of 99 sets of settings available for tool length offset, and **<PgUp>** and **<PgDn>** keys can be used to switch the settings screen.
- Use arrow keys to move the cursor to X-length, Y-length, Z-length, and tool radius for the tool number you wish to set up, enter a set value in the input area and press the **<Input>** key to enter the numerical value into the controller. (This is an absolute setting)
- It can be used in coordination with the **【Guided input】** key for changing settings.



- **For detailed instructions, please refer to section 1.8 for instructions on how to change Tool Length Offset Settings**

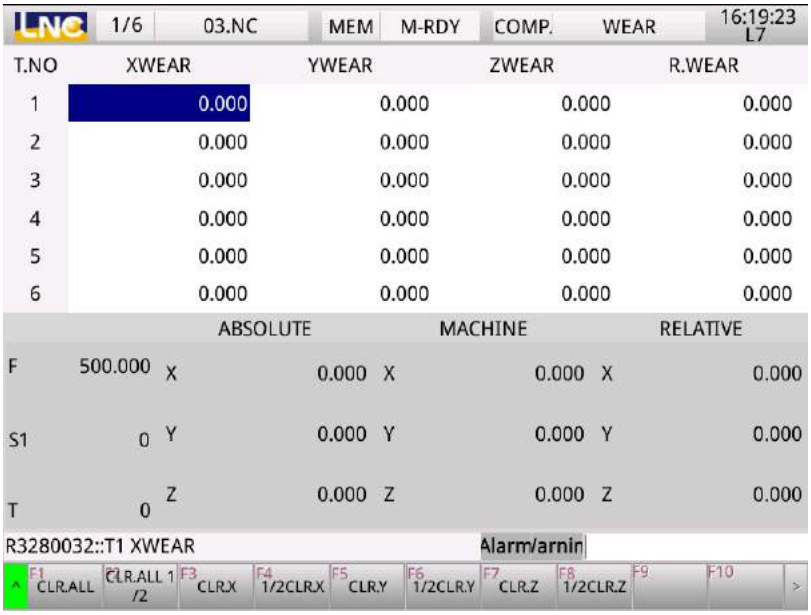
2. Tool wear offset

- Press the 【**Wear**】 key to enter tool wear offset page.
- There are a total of 99 sets of settings available for tool wear offset, and **<PgUp>** and **<PgDn>** keys can be used to switch the settings screen.
- Use arrow keys to move the cursor to X-wear, Y-wear, Z-wear, and wear radius for the tool number you wish to set up, enter a set value into the input area and press the **<Input>** key to enter the numerical value into the controller. (This is an incremental setting)



3. Clear relative

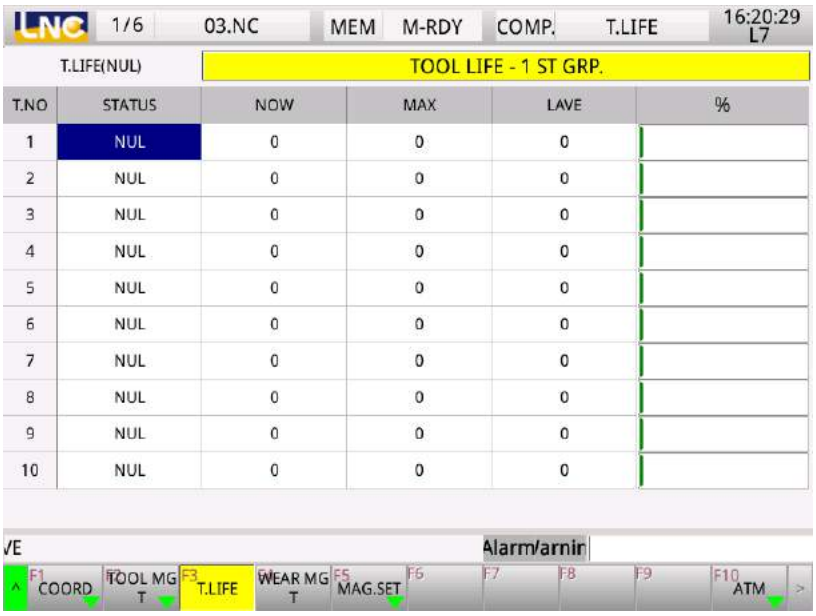
- It is for users to clear the relative coordinates of all axial directions or one single axial direction into 1/2 the numerical value or 0, when necessary.



1.5.3 Tool service life

Description of tool service life functions:

- For users to manage a tool's usage status, and use settings such as "remaining service life reminder" and "maximum service life" to issue alerts or warnings to remind users.



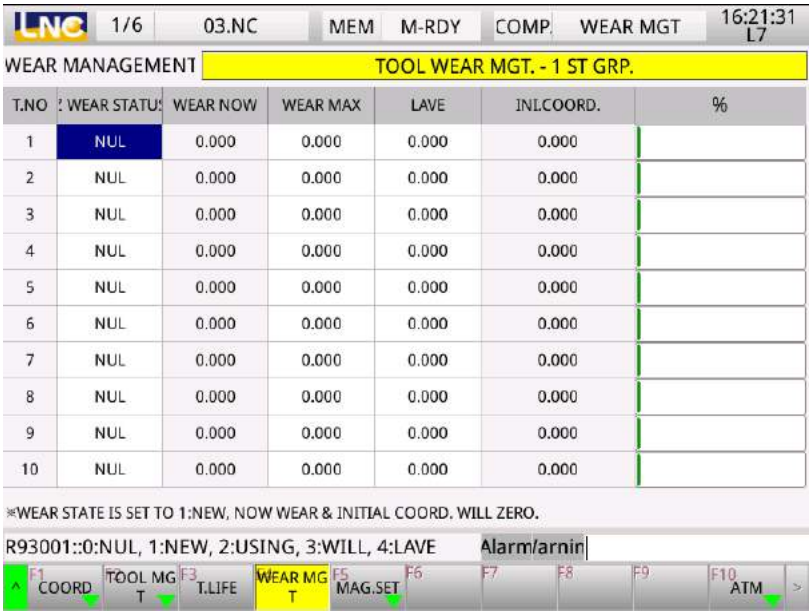
Instructions for using tool service life functions:

- "Tool status" field: It can be divided into five types of status including "Not used", "New tool", "Used", "Approaching", and "Reached". If the field is reset as a "New tool", the numerical value of the "Current service life" field will be set as 0.
- "Current service life" field: The total number of times a tool has been used.
- "Maximum service life" field: The maximum number of times a tool has been used. When the numerical value in the "Current service life" field is equal to the numerical value in the "Maximum service life" field, the "Tool status" field will change into "Reached", and the system will issue an alert.
- "Remaining service life reminder" field: It is for reminding users that the number of times this tool has been used has reached the set value. When the numerical value of "Maximum service life" minus the numerical value of "Current service life" is equal to or less than the numerical value of "Remaining service life reminder", the "Tool status" field will become "Approaching", and the system will issue an alarm.

1.5.4 Wear management

Description of wear management functions:

- For users to manage tool wear status, and users can set reminders by issuing alerts or warnings via settings such as "wear reminder" and "maximum Z axis wear" in coordination with the M50 command.
- The numerical value of wear measured by executing the M50 command will be filled into the "Current Z axis wear" field of the tool number corresponding to the current spindle tool number.



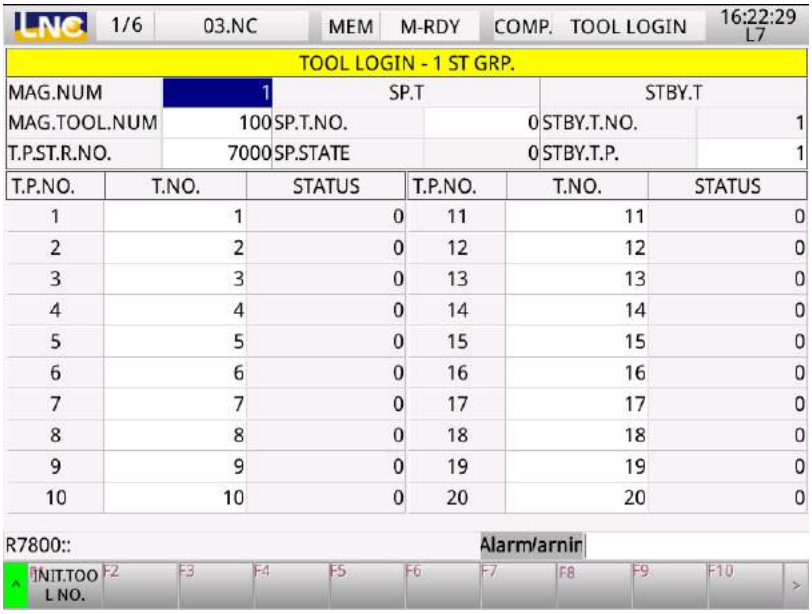
Instructions for using wear management:

- "Tool wear status on the Z axis" field: It can be divided into five types of status including "Not used", "New tool", "Used", "Approaching", and "Reached". If the field is reset as a "New tool", the numerical values of "Current Z axis wear" and "Initial measured tool length coordinate" will be set as 0.
- "Current wear on the Z axis" field: Current wear on the tool.
- "Maximum Z axis wear" field: Maximum amount of wear on a tool. When the numerical value in the "Current Z axis wear" field is equal to the "Maximum Z axis wear" field, then "Tool Z axis wear status" will change into "Reached" and the system will issue an alert.
- "Wear reminder" field: It will reminder users when the amount of tool wear remaining has reached the set value. When the numerical value of "Maximum Z axis wear" minus the numerical value of "Current Z axis wear" is equal to or less than the numerical value of "Wear reminder", the "Tool status" field will change to "Approaching" and the system will issue an alarm.
- The "Initially measured tool length coordinate" field: When the "Tool Z axis wear status" is set as a "New tool", the numerical value measured when the M50 command is executed for the first time will be filled into that field.

1.5.5 Tool registration

Description of the tool registration function:

- Users can inquire the tool number (T code) corresponding to the tool pocket number currently on the tool magazine via the tool registration page. This function is only available for machines equipped with tool magazines.



Instructions for using tool registration:

- Users can use the **<OFFSET>** key on the MDI panel to switch to compensation group, then press the **【Tool registration】** key to access the tool registration page.
- Field descriptions:

  "Quantity of tool magazines": sets and displays the total quantity of tool magazines in the machine.

  "Quantity of tools in the tool magazine": displays the total quantity of tools inside a tool magazine.

  "Initial R number of tool pocket": sets and records the R value number of the tool pocket number. (The range of R value is between R7000 to R7399.)

  "Spindle tool number": sets and displays the tool number currently on the spindle.

  "Spindle tool status": displays the current status of spindle tool.

  "Standby tool number": displays the tool number currently in standby mode.

  "Standby tool pocket": sets and displays the tool pocket number currently in standby mode.

- Key function descriptions:

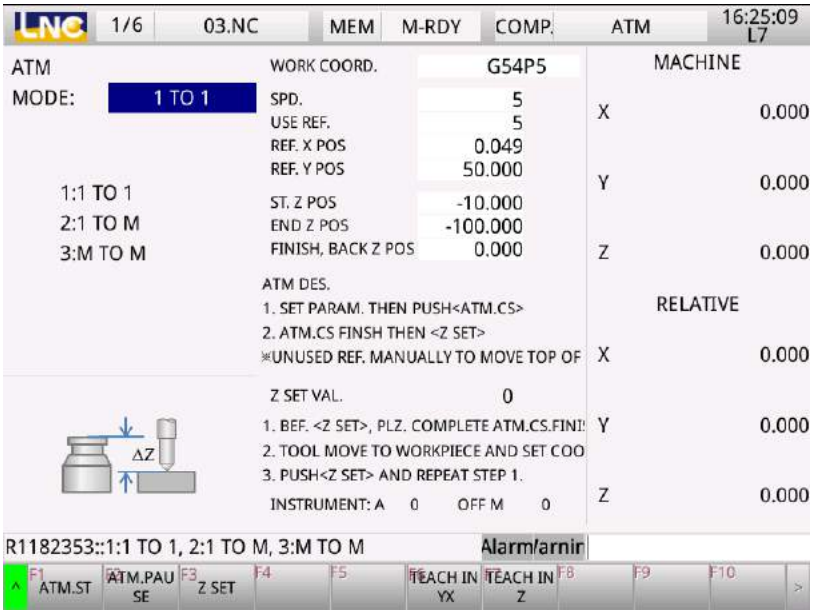
  【Arrange tool】: this function key can be used to re-arrange the tool number sequence.

1.5.6 Automatic tool alignment

1.5.6.1 CNC\woodworking machine & applicable industries

Description of the automatic tool alignment function:

- With the automatic tool alignment function, the tool alignment instrument in the machine will measure the distance between tool tip and the tool alignment instrument, and this distance will be automatically filled into the designated work coordinate system and tool offset table in accordance with different tool alignment methods (single tool/single workpiece, single tool/multiple workpieces, and multiple tools/multiple workpieces) as the basis for tool length offset during machining.



Instructions for using automatic tool alignment:

- Users can use the **<OFFSET>** key on the MDI panel to switch to compensation group before pressing the **【Automatic tool alignment】** key to access the automatic tool alignment page.
- Field descriptions: (Different tool alignment modes will require slightly different field settings)

  "Tool alignment mode": 1. Single tool/single workpiece, 2. Single tool/multiple workpieces, 3. Multiple tools/multiple workpieces.

Single tool/single workpiece:

"Work coordinate system": sets the work coordinate system for saving tool length after automatic measurement.

"Measurement speed": sets the feed rate (mm/min) during measurement, which is generally between 100 and 300.

"Whether to use a reference point": when 0 is selected (not using a reference point), the tool alignment procedure will be carried out based on current X/Y coordinates; when 1 is selected (a reference point is used), Z axis will return to the machine reference point, then it will

be moved to the X/Y coordinate positions of the tool alignment instrument, dropped down to the position of tool alignment starting point Z before the tool alignment procedure can start.

"Tool alignment instrument's X axis coordinate reference point": sets the machine coordinate of the X axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment instrument's Y axis coordinate reference point": sets the machine coordinate of the Y axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment starting point Z": sets the machine coordinate for starting tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Lowest machine coordinate of the Z axis": sets the machine coordinate of the lower limit position for tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Return to Z's safety point after tool alignment": sets the machine coordinate of the point to return to after completing tool alignment on the A axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Blowing air M": sets the M code for blowing air during tool alignment.

"Shut off air M": sets the M code for shutting down air after tool alignment is completed.

Single tool/multiple workpieces:

"Measurement speed": sets the feed rate (mm/min) during measurement, which is generally between 100 and 300.

"Whether to use a reference point": when 0 is selected (not using a reference point), the tool alignment procedure will be carried out based on current X/Y coordinates; when 1 is selected (a reference point is used), Z axis will return to the machine reference point, then it will be moved to the X/Y coordinate positions of the tool alignment instrument, dropped down to the position of tool alignment starting point Z before the tool alignment procedure can start.

"Tool alignment instrument's X axis coordinate reference point": sets the machine coordinate of the X axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment instrument's Y axis coordinate reference point": sets the machine coordinate of the Y axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment starting point Z": sets the machine coordinate for starting tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Lowest machine coordinate of the Z axis": sets the machine coordinate of the lower limit position for tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Return to Z's safety point after tool alignment": sets the machine coordinate of the point to return to after completing tool alignment on the A axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Set coordinate system of Z axis gap": set the work coordinate system of Z axis gap. The 【Z gap】 key can be pressed to set the gap value to the designated coordinate system.

"Blowing air M": sets the M code for blowing air during tool alignment.

"Shut off air M": sets the M code for shutting down air after tool alignment is completed.

Multiple tools/multiple workpieces:

"Tool number": sets the tool number to be saved in the tool offset page after automatically measuring the tool length.

"Is automatic tool change required": sets the automatic tool change command. (0: No 1: M6Tx 2: TxM6)

"Measurement speed": sets the feed rate (mm/min) during measurement, which is generally between 100 and 300.

"Whether to use a reference point": when 0 is selected (not using a reference point), the tool alignment procedure will be carried out based on current X/Y coordinates; when 1 is selected (a reference point is used), Z axis will return to the machine reference point, then it will be moved to the X/Y coordinate positions of the tool alignment instrument, dropped down to the position of tool alignment starting point Z before the tool alignment procedure can start.

"Tool alignment instrument's X axis coordinate reference point": sets the machine coordinate of the X axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment instrument's Y axis coordinate reference point": sets the machine coordinate of the Y axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

"Tool alignment starting point Z": sets the machine coordinate for starting tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Lowest machine coordinate of the Z axis": sets the machine coordinate of the lower limit position for tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Return to Z's safety point after tool alignment": sets the machine coordinate of the point to return to after completing tool alignment on the A axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

"Set coordinate system of Z axis gap": set the work coordinate system of Z axis gap. The 【Z gap】 key can be pressed to set the gap value to the designated coordinate system.

"Blowing air M": sets the M code for blowing air during tool alignment.

"Shut off air M": sets the M code for shutting down air after tool alignment is completed.

- Key function descriptions:

  【Start tool alignment】: this function key is for carrying out automatic tool alignment.

  【Pause tool alignment】: this function key can be used to pause automatic tool alignment.

  【Z gap】: this function key is used for setting the distance between surface of tool instrument and surface of the workpiece.

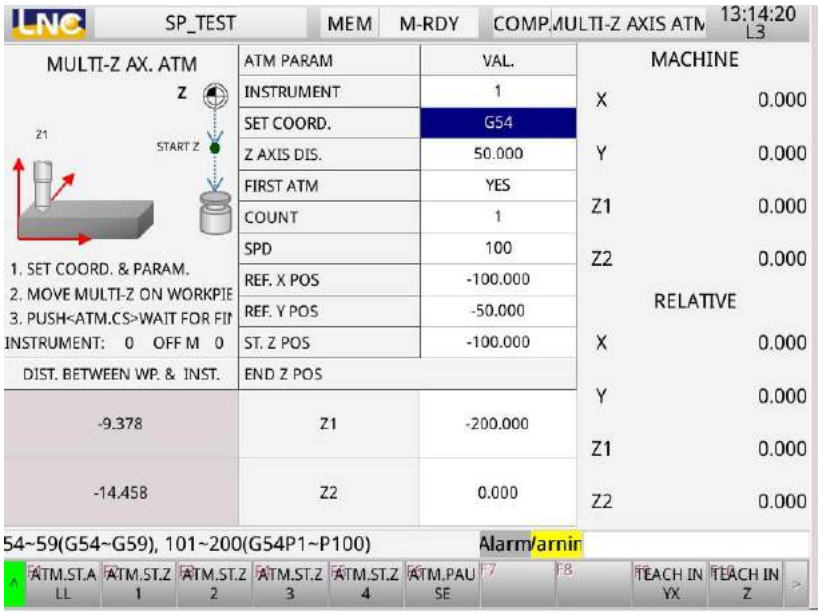
  【XY axis guide】: this function key can be used to set the XY axis' machine coordinate value after the tool is moved to the reference point.

  【Z axis guide】: this function key is for setting the current machine coordinate of the Z axis.

1.5.6.2 Glass edge milling machine, applicable industries

Description of the automatic tool alignment function:

- With the automatic tool alignment function, the tool alignment instrument in the machine will measure the distance between tool tip and the tool alignment instrument, and this distance will be automatically filled into the designated work coordinate system as the basis for tool length offset during the machining process.



Instructions for using automatic tool alignment:

- Users can use the **<OFFSET>** key on the MDI panel to switch to compensation group before pressing the **【Automatic tool alignment】** key to access the automatic tool alignment page.

- Field descriptions:

  "Set coordinate system": sets the work coordinate system for saving the tool length after taking automatic measurement.

  Corresponding numerical values are to be entered in the input areas as settings. 54 to 59 (G54 to G59), 101 to 200 (G54P1 to G54P100).

  "Spacing between multiple spindles": sets the distance between spindles.

  "First tool alignment": sets whether it is the first tool alignment. (0: No 1: Yes)

  "Number of tool alignment": set the number of tool alignment. (0 to 10)

  "Tool alignment speed": set the feed speed (mm/min) during tool alignment, which is generally 100 to 300.

  "X axis machine coordinate of the tool alignment instrument": sets the machine coordinate of X axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

  "Tool alignment instrument's Y axis machine coordinate": sets the machine coordinate of the Y axis above the tool alignment instrument. Press 【XY guide】 for guided input after the tool is moved to the designated position.

  "Tool alignment starting point Z's machine coordinate": sets the machine coordinate for starting tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

  "machine coordinate of Z's lowest tool alignment point": sets the machine coordinate of the lower limit position for tool alignment on the Z axis. Press【Z axis guide】 for guided input after moving the cursor to this field.

- Key function descriptions:

  【Start tool alignment】: this function key is for carrying out automatic tool alignment.

  【Pause tool alignment】: this function key can be used to pause automatic tool alignment.

  【XY axis guide】: this function key can be used to set the XY axis' machine coordinate value after the tool is moved to a position above the tool alignment instrument.

  【Z axis guide】: this function key is for setting the current machine coordinate of the Z axis.

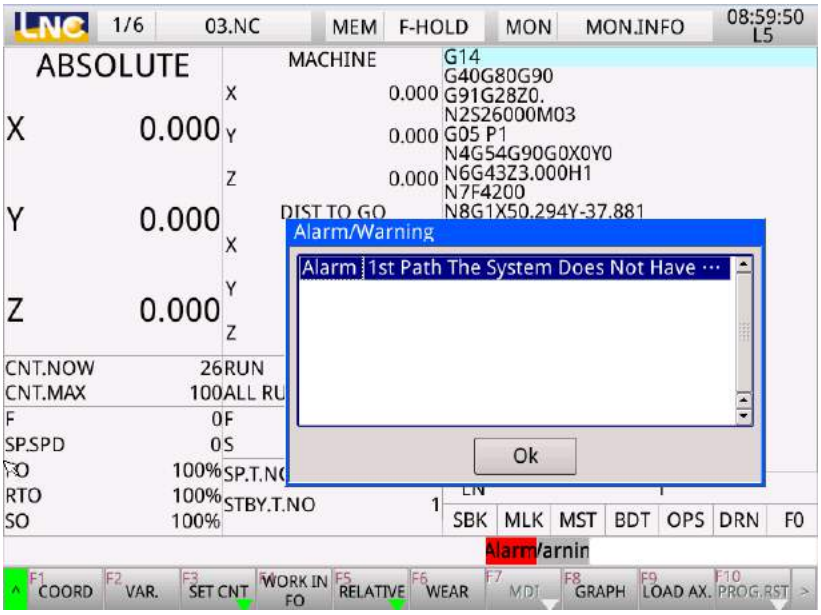
- **For detailed instructions, please refer to section 1.8 for instructions on how to use Automatic Tool Alignment**

1.6 Diagnosis group (DGNOS)

1.6.1 Alerts and warnings

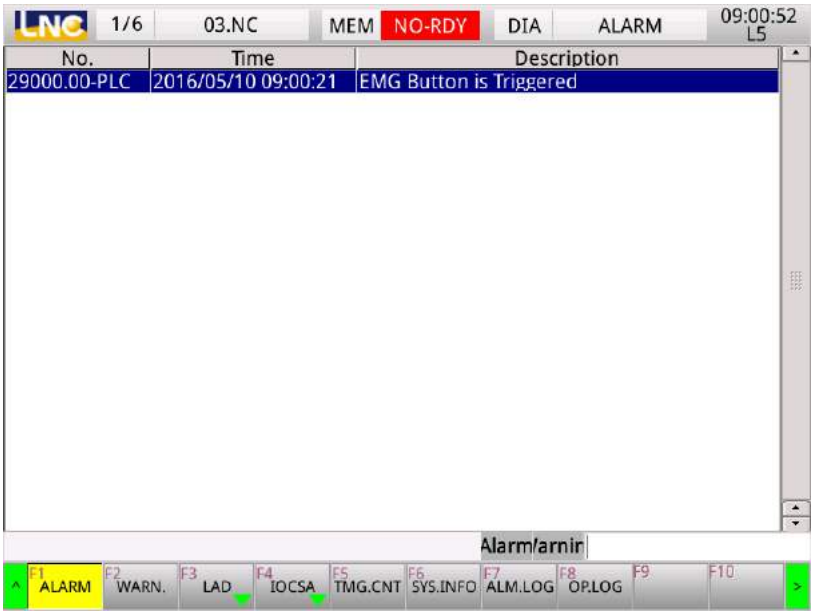
Description of alert/warning functions:

- In case of a problem, abnormality, and error during system operation, or the wrong G-CODE was used during program editing, an alert will be issued and system operation will be suspended immediately. A small window containing an alert/warning message will appear from any given page. After the alert/warning has been resolved, the **<Reset>** key must be pressed to clear the alert/warning.

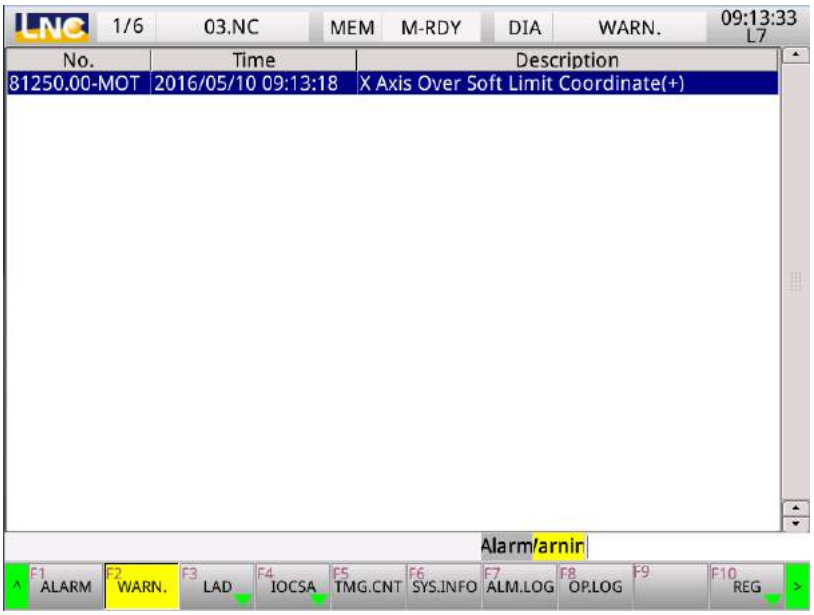


Instructions for alerts/warnings:

- Alert page: After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the **【Alert】** key or the **<ALARM>** key on the MDI panel can be pressed to access this page.



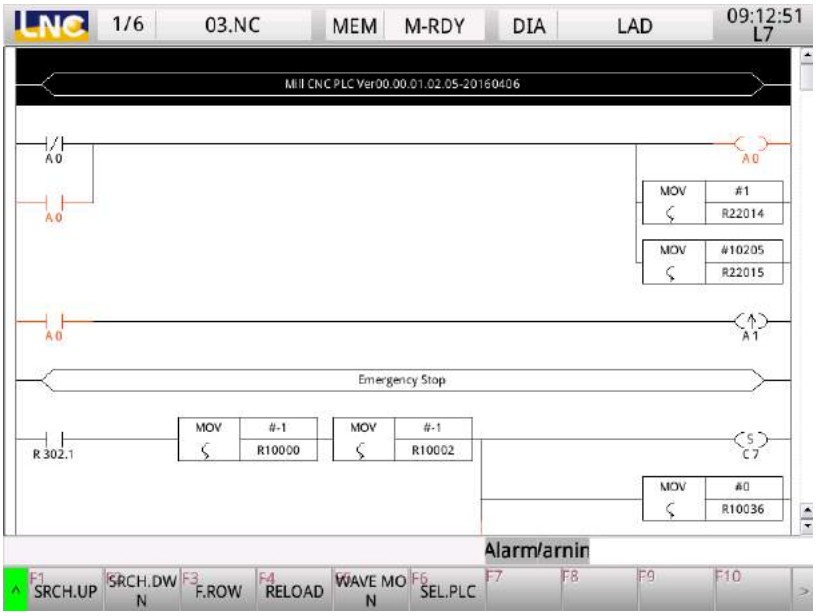
- Warning page: After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the【**Warning**】key can be pressed to access this page.



1.6.2 Ladder diagram

Description of ladder diagram functions:

- A ladder diagram is the corresponding diagram between keys on the machine and functional operations; this page is to be used by system designers and maintenance personnel. Users can only view this page; its contents cannot be modified.



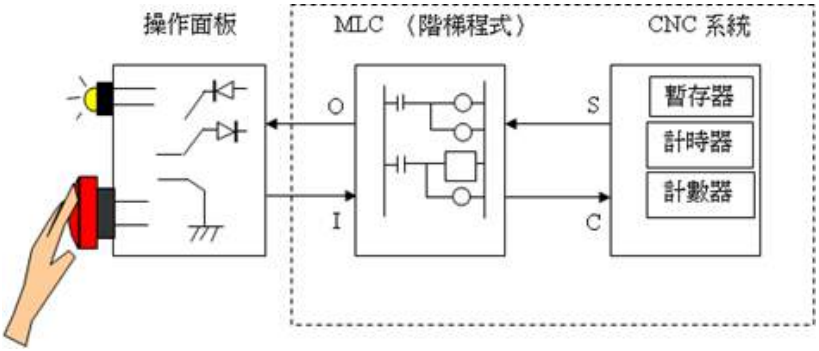
Instructions for using the ladder diagram:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the **【Ladder diagram】** key can be pressed to access the ladder diagram page.
- The device code to be found can be entered in the input area (for example: I12, A113, T1...), and the **【Search upward】** and **【Search downward】** keys can be used to find the position screen corresponding to this code. Press the **【First line】** key to return the cursor to the file header position.
- After a new ladder diagram file is imported, the **【Reload】** key on the human machine interface should be pressed to update the diagram.

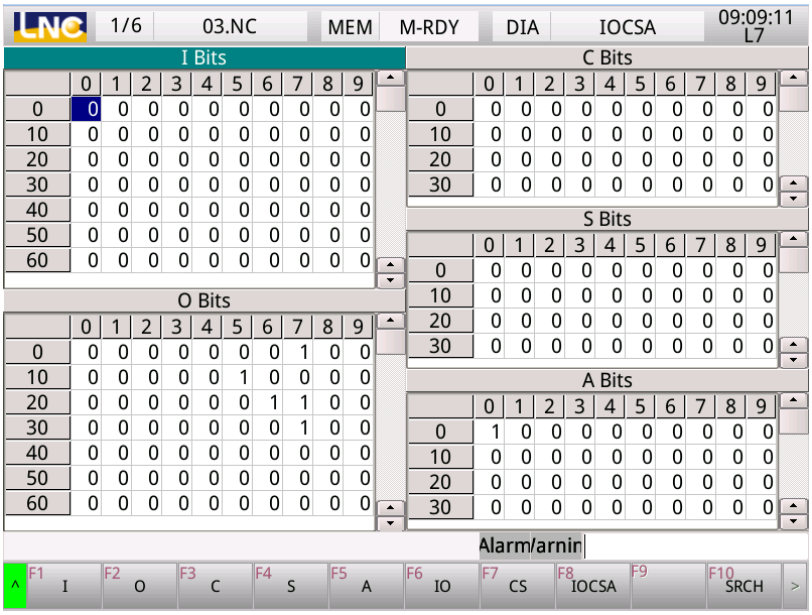
1.6.3 IOCSA

Description of the IOCSA function:

- It is used to examine the system's I/O status and internal status.



- The contact points at I, O, C, S, and A of the PLC represent machine functions corresponding to keys on the machine. This page is for system designers and maintenance personnel, as well as for users to inquire and browse the switching actions of contacts points I, O, C, S, and A.



Instructions for using IOCSA:

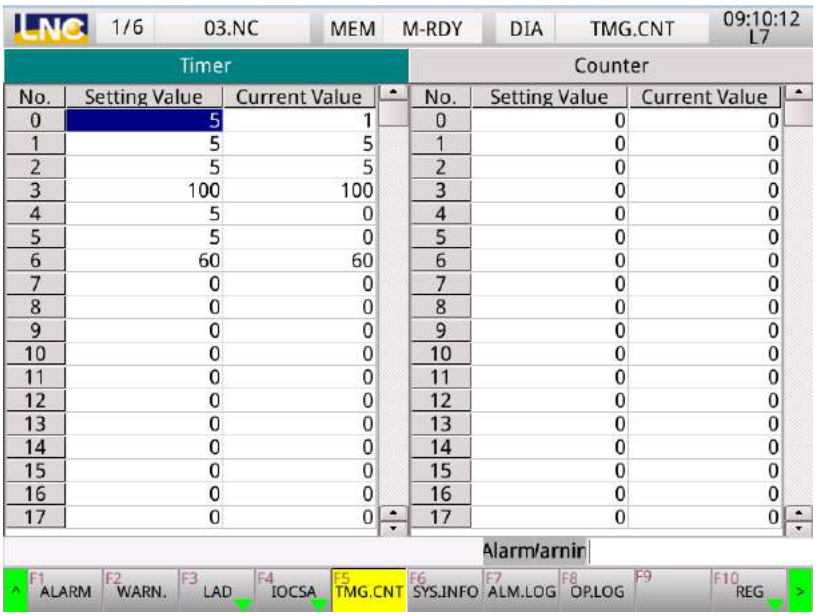
- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the **【IOCSA】** key should be pressed to access the IOCSA page.
- Press the **【 I 】** key, **【 O 】** key, **【 C 】** key, **【 S 】** key, **【 A 】** key, **【 IO 】** key, **【 CS 】** key, **【IOCSA】** key, and then press the **【Find】** key to enter the number to be found in the input section, press **Confirm** to switch to the position screen corresponding to that number.

- When any bit of IOCSA is selected, the bit function's description will be displayed in the message display section located near the lower left corner of the screen.

1.6.4 Timer/counter

Description of the timer/counter function:

- When the timer/counter function is used by system startup, program execution, and the PLC, this page can be used for inquiring and browsing current operating status of the timer/counter.



Instructions for using the timer/counter:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the 【**Timer/counter**】 key can be pressed to access the timer/counter page.
- Viewers can change views by using the arrow keys on the MDI panel, or use the **<PgUp>** and **<PgDn>** keys for quickly switching pages.

1.6.5 System information

Description of system information:

- It is for examining whether the system is in normal operation.
- It can be used to examine information related to machine coordinates, encoder, servo errors, OPR grid, final output command, and amount of memory remaining.

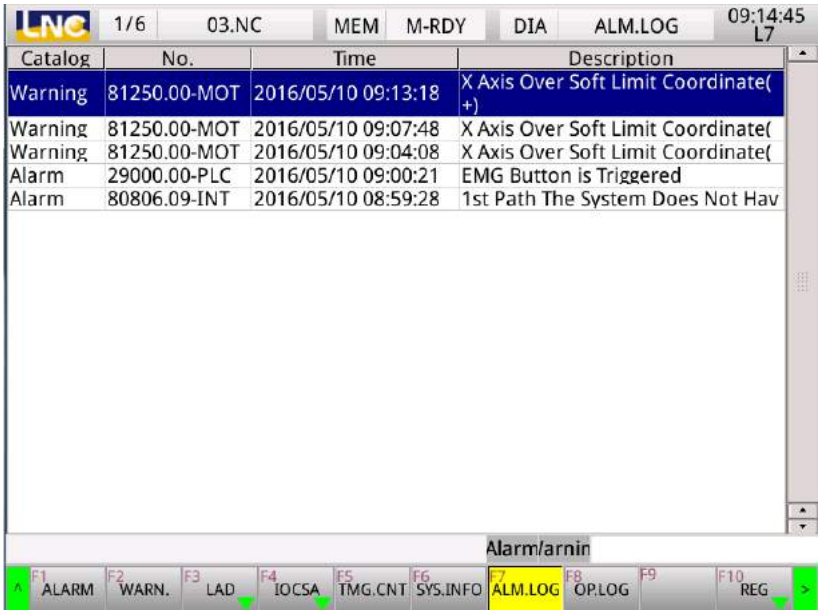
Instructions for using system information:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the 【**System information**】 key can be pressed to access the system information page.

1.6.6 Alert history

Description of the alert history function:

- The controller will record all alerts and warnings issued previously in accordance with their sequence of occurrence so that the machine factory and controller maker can easily review previous problems.



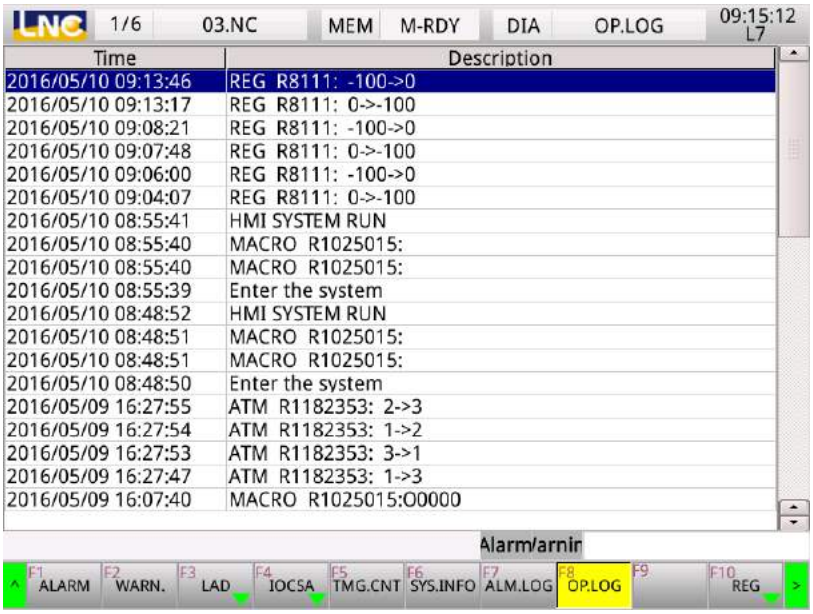
Instructions for using alert history:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the **【alert history】** key can be pressed to access the alert history page.
- Users can change views by using the arrow keys on the MDI panel, or use the **<PgUp>** and **<PgDn>** keys for quickly switching pages.

1.6.7 Operation history

Description of operation history function:

- When the system is turned on, controller actions will be recorded by the system for machine designers and system maintenance personnel to examine its operation sequence.



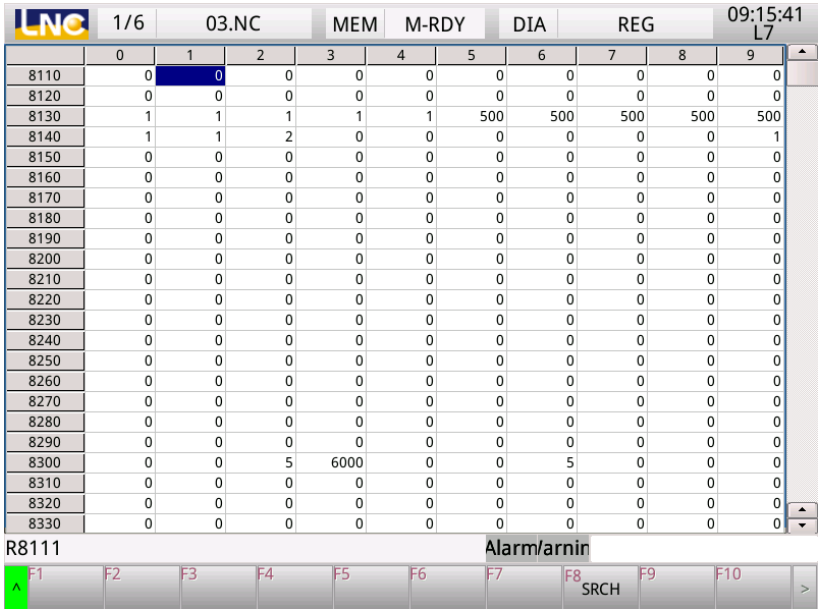
Instructions for using operation history:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the 【**Operation history**】 key can be pressed to access the operation history page.
- Users can change views by using the arrow keys on the MDI panel, or use the **<PgUp>** and **<PgDn>** keys for quickly switching pages.

1.6.8 R value

Description of the R value function:

- The R value is the screen for system maintenance variables; this function is to be used by system designers and maintenance personnel. Users can only view this page, its contents cannot be modified.



	0	1	2	3	4	5	6	7	8	9
8110	0	0	0	0	0	0	0	0	0	0
8120	0	0	0	0	0	0	0	0	0	0
8130	1	1	1	1	1	500	500	500	500	500
8140	1	1	2	0	0	0	0	0	0	1
8150	0	0	0	0	0	0	0	0	0	0
8160	0	0	0	0	0	0	0	0	0	0
8170	0	0	0	0	0	0	0	0	0	0
8180	0	0	0	0	0	0	0	0	0	0
8190	0	0	0	0	0	0	0	0	0	0
8200	0	0	0	0	0	0	0	0	0	0
8210	0	0	0	0	0	0	0	0	0	0
8220	0	0	0	0	0	0	0	0	0	0
8230	0	0	0	0	0	0	0	0	0	0
8240	0	0	0	0	0	0	0	0	0	0
8250	0	0	0	0	0	0	0	0	0	0
8260	0	0	0	0	0	0	0	0	0	0
8270	0	0	0	0	0	0	0	0	0	0
8280	0	0	0	0	0	0	0	0	0	0
8290	0	0	0	0	0	0	0	0	0	0
8300	0	0	5	6000	0	0	5	0	0	0
8310	0	0	0	0	0	0	0	0	0	0
8320	0	0	0	0	0	0	0	0	0	0
8330	0	0	0	0	0	0	0	0	0	0

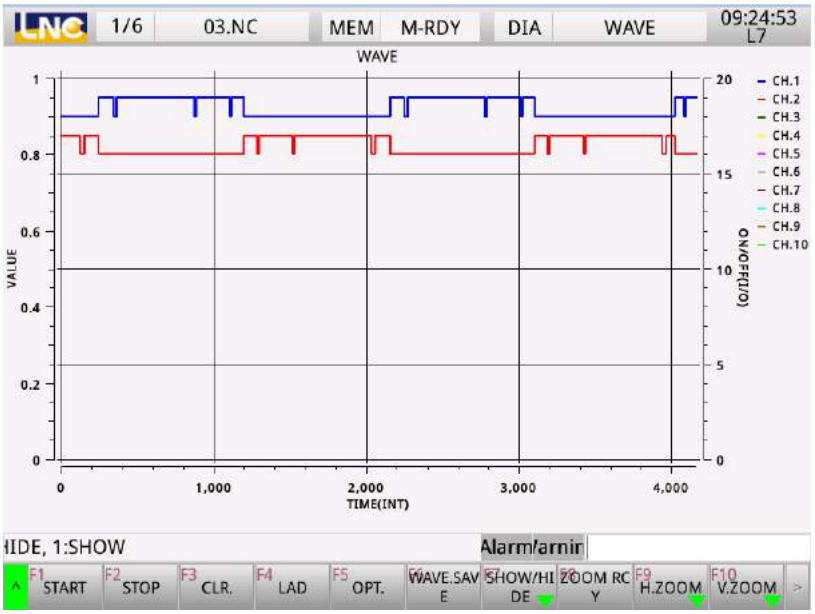
Instructions for using the R value:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the 【**R value**】 key can be pressed to access the R value page.
- After entering the R value number you wish to find in the input section, press the 【**Find**】 key and the cursor will be moved to the position corresponding to that number.

1.6.9 Waveform monitoring

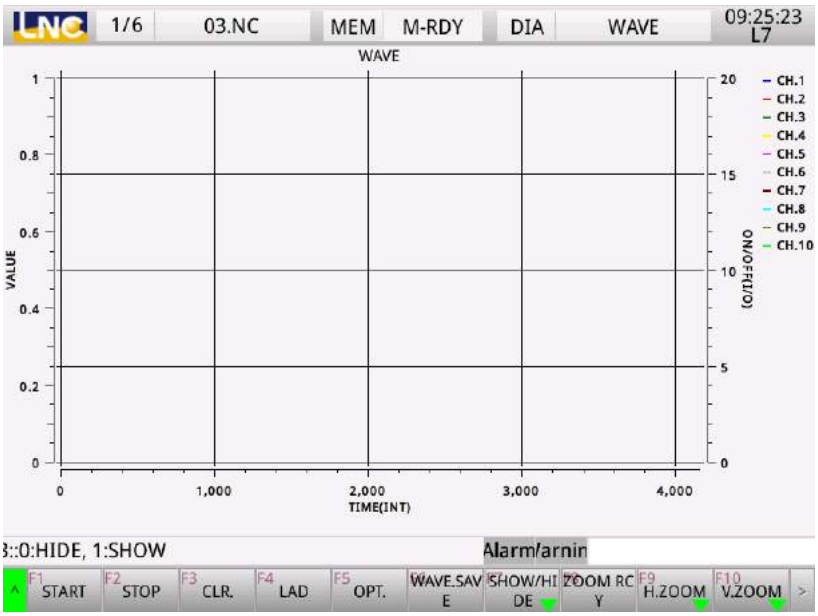
Description of the waveform monitoring function:

- The waveform monitoring page is for showing the operation wave patterns of contacts I, O, C, S, A, R, TM, CT corresponding to the machine. This page is only for designers, system maintenance personnel, and users to make inquiries and browse.



Instructions for using waveform monitoring:

- After using the **<DGNOS>** key on the MDI panel to switch to diagnosis group, the 【 > 】 key can be pressed, followed by pressing the 【**Waveform monitoring**】 key to access the waveform monitoring page.



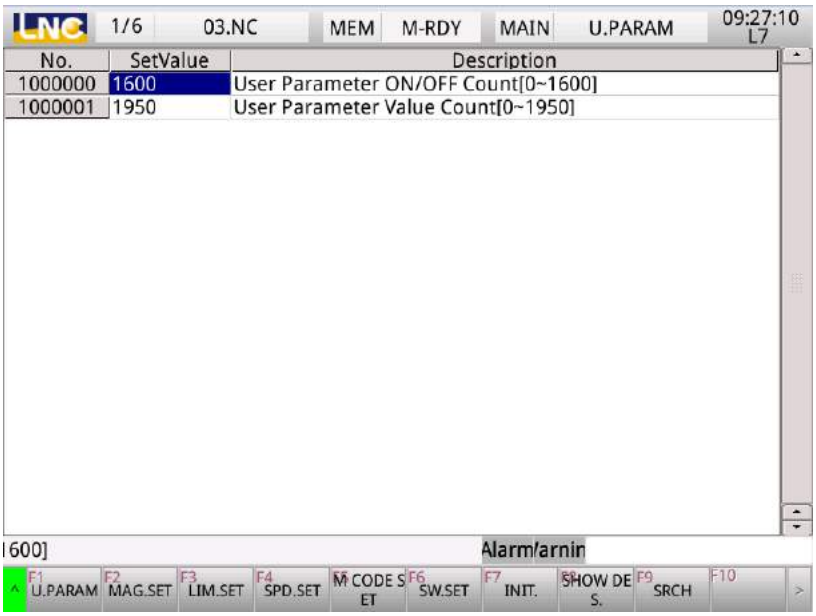
- **For detailed instructions, please refer to section 1.8 for instructions on the Waveform Monitoring Function**

1.7 Maintenance group (MAINTE)

1.7.1 User parameters

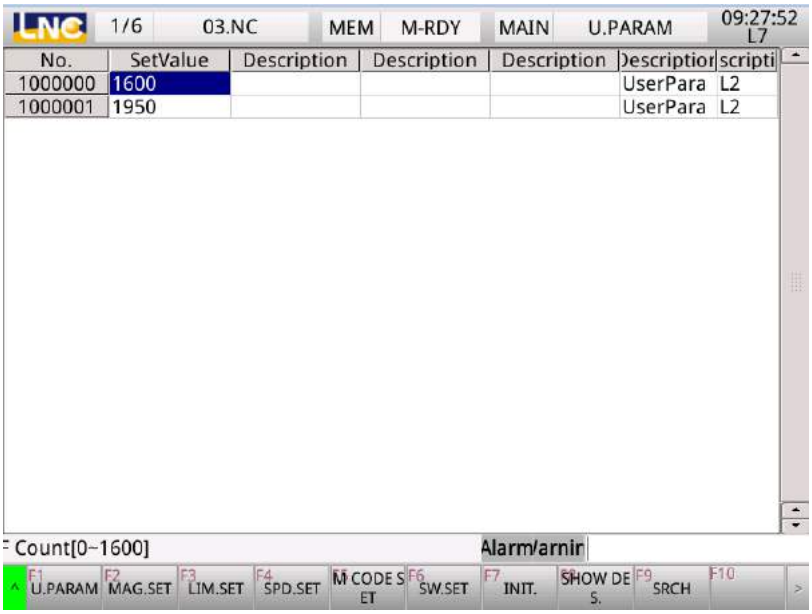
Description of user parameters:

- It is for setting parameters related to PLC, axial direction software limit, RAPID, JOG, and hand wheel for a dry run.



Instructions for using user parameters:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**User parameters**】 key can be pressed to access the user parameters page.
- After moving the cursor to the parameter number you wish to modify, a numerical value can be entered in the input section before pressing the **<Input>** key to complete the input procedure. The controller also provides initialization functions. The 【**Initialization**】 key can be pressed to restore parameters to their default values; as for the search function, the parameter number you wish to find should be entered into the input area, then press the 【**Find**】 key to quickly search and move the cursor to the position of that number.

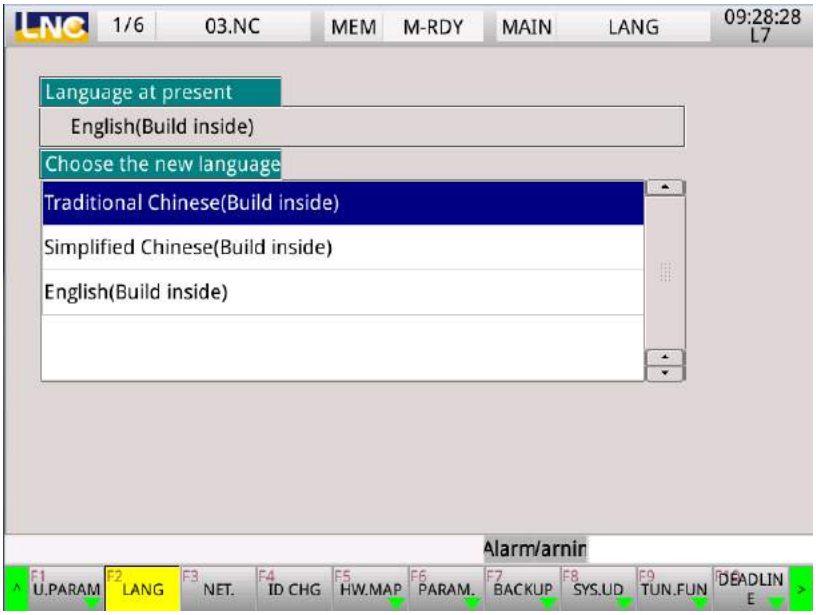


【**Display instruction**】: instruction for opening and canceling parameter display on the page.

1.7.2 Language settings

Description of language setting functions:

- There are three different languages available in the controller for users to choose from.



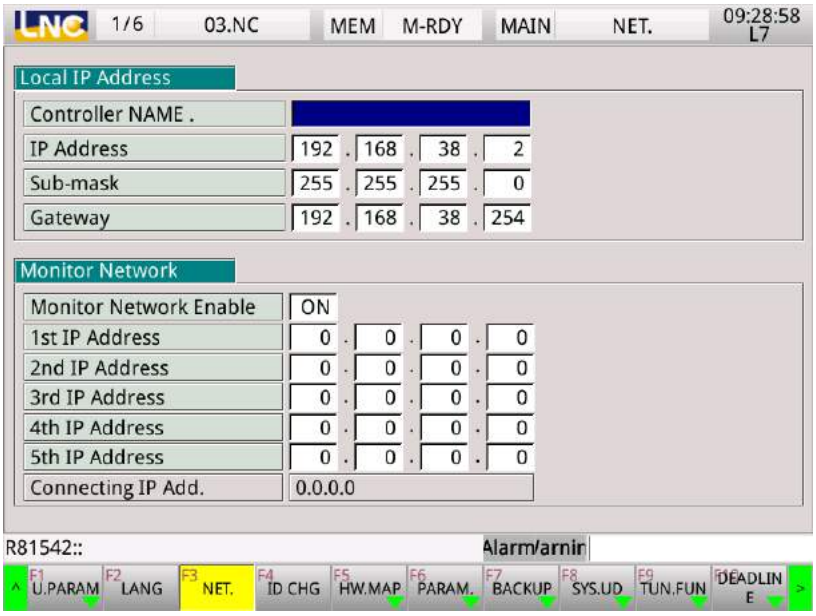
Instructions on using language settings:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**Language setting**】 key can be pressed to access the language setting page.
- After moving the cursor to the language to be replaced, press the **<Input>** key to change to the selected language.

1.7.3 Network settings

Description of network settings:

- A PC can be connected to the controller via ReCON related software by changing network settings, such that the PC can upload/download machining files used by the controller as well as the software update function via corresponding software.



Instructions on using network settings:

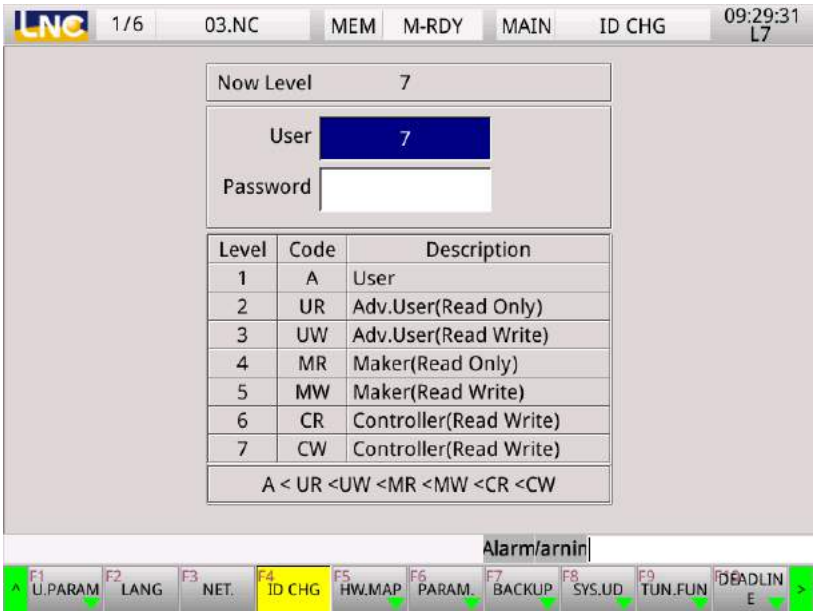
- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Network settings】** key can be pressed to access the network settings page.
- When setting the IP address, make sure that the first three regional addresses of the PC and controller are identical.

- **For detailed instructions, please refer to section 1.8 for instructions on configuring Network Settings and Connections**

1.7.4 Changing identity

Description of the identity change function:

- There are 7 permission types associated with this controller in accordance with different user levels, with each permission having access to different function settings. Usually the machine is set at the permission of a general user upon delivery in order to avoid inadvertent changes made by users.



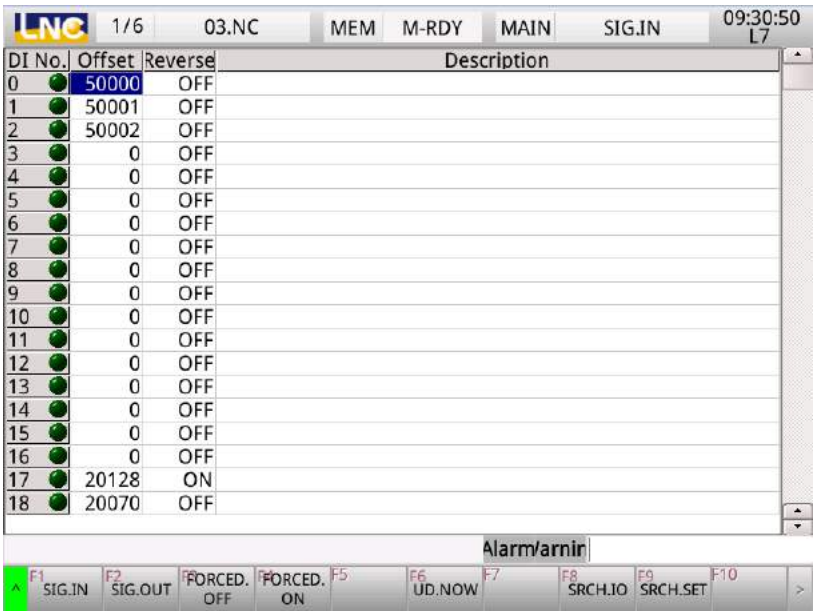
Instructions for changing identity:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**Change identity**】 key can be pressed to access the identity change page.
- When making permission changes, the cursor should be moved to the "User" field and the level code (1 to 7) should be entered in the input area before pressing the **<Input>** key, then move the cursor to the "Password" field to enter the password, the **<Input>** key can be pressed to change the current permission level. Users can check the current permission field or the upper right corner of the screen to see if the change was made successfully.

1.7.5 Hardware contact

Description of the hardware contact function:

- Users can use this function to examine the I/O signal status of hardware connected externally to the controller.



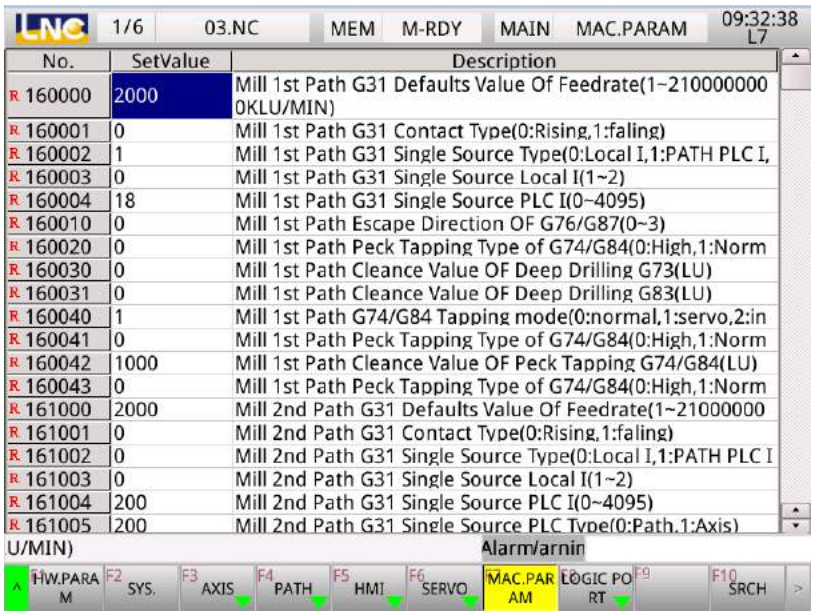
Instructions for using hardware contact:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Hardware contact】** key can be pressed to access the hardware contacts page.
- **【Signal input】**: examine the status of signal input at point I.
- **【Signal output】**: examine the status of signal output at point O.
- **【Force close】**: forcibly set the I/O point signal status to close, and then press **【Update now】** after the setting is completed.
- **【Force open】**: forcibly set the I/O point signal status to open, and then press **【Update now】** after the setting is completed.
- **【Update now】**: update signal status.
- **【Find】**: enter the parameter number you wish to find in the input area, then press the **【Find】** key to quickly search and move the cursor to the position of that number.

1.7.6 Parameters

Description of parameters:

- It is mainly for setting the controller's internal parameters.



Instruction for using parameters:

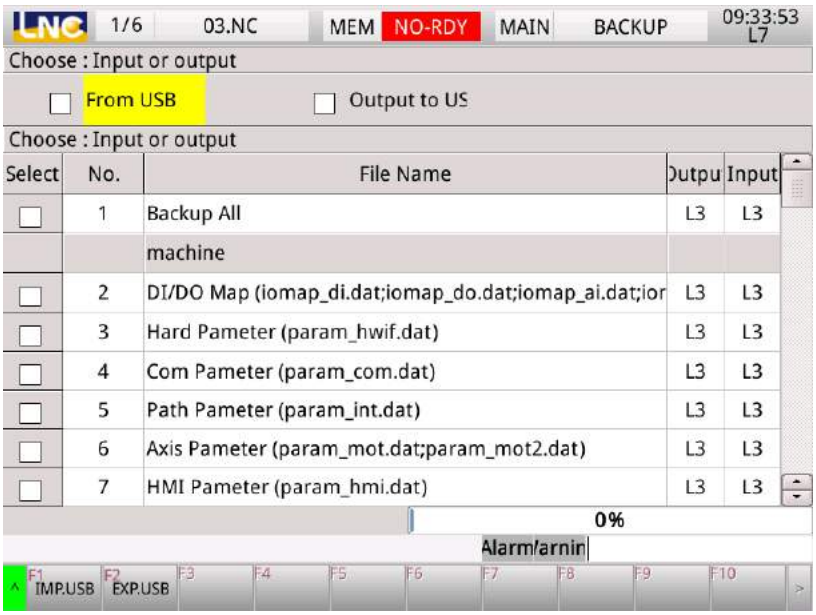
- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**Parameters**】 key can be pressed to access the parameters page.
- After moving the cursor to the parameter number you wish to modify, a numerical value can be entered before pressing the **<Input>** key to complete the input procedure. The controller also provides default values. The 【**Default**】key can be pressed to restore parameters to their default values; as for the search function, the parameter number you wish to find should be entered into the input area, then press the 【**Find**】 key to quickly search and move the cursor to the position of that parameter number.
- The 【 > 】 key can be pressed before pressing the 【**Reset all to default**】 to restore all parameters to their default values.

- **For detailed instructions, please refer to section 1.8 for instructions on changing Parameter Settings**

1.7.7 Backup

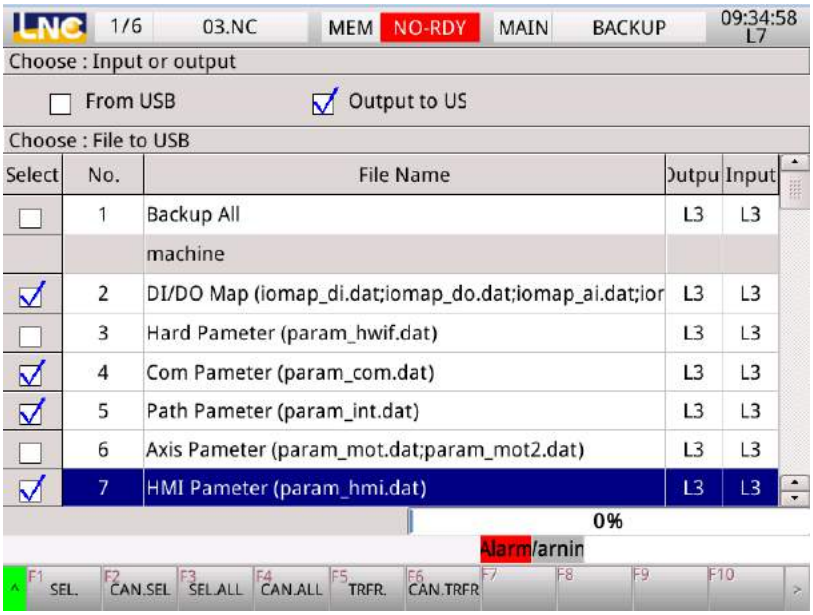
Description of the backup function:

- System parameters, macro parameters, OP parameters, ladder diagrams, and user parameters can be exported to a USB device, or data in a USB drive can be imported into the controller.



Instructions on using backup:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Backup】** key can be pressed to access the backup page.
- **【Import from USB】**: import data from a USB device to controller.
- **【Export to USB】**: export system data from controller to a USB device.



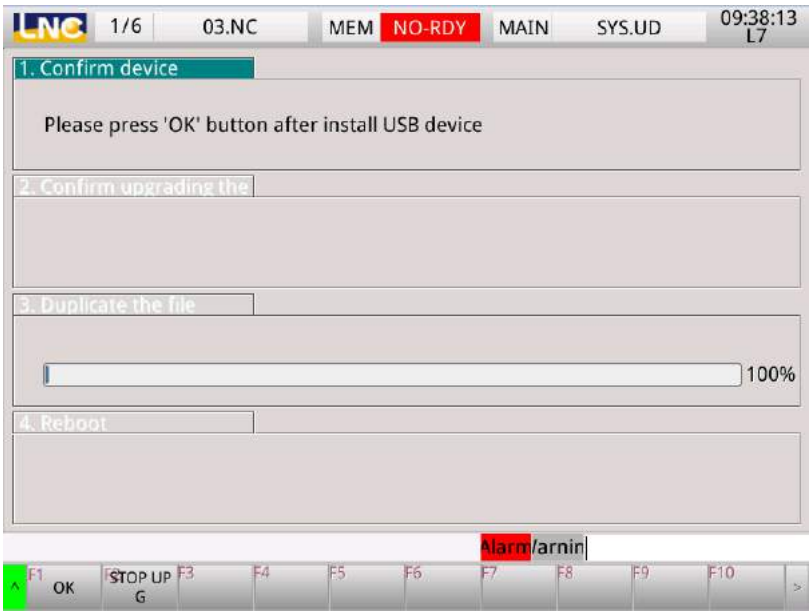
- 【**Select**】: select the file to be imported/exported.
- 【**Cancel select**】: cancel the current file selection.
- 【**Select all**】: select all files.
- 【**Cancel select all**】: cancel all selected files.
- 【**Transfer**】: proceed with file import/export.
- 【**Cancel transfer**】: cancel file transfer of the file to be imported/exported.

- **For detailed instructions, please refer to section 1.8 for instructions on System Data Backup-Import and System Data Backup-Export**

1.7.8 System update

Description of the system update function:

- The controller can perform system software update via a USB device.



Instructions for system update:

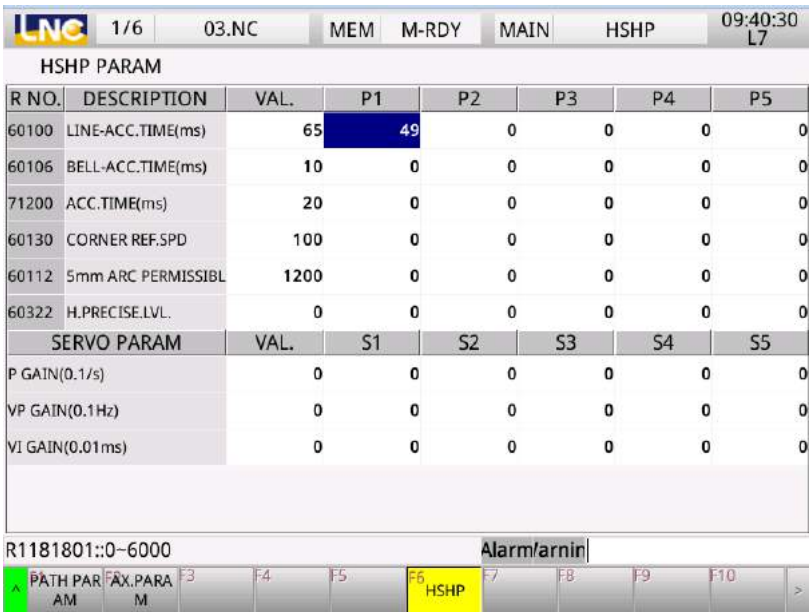
- After using the **<MAINTE>** on the MDI panel to switch to maintenance group, the 【**System update**】 key can be pressed to access the system update page.
- After entering the system update page, the USB drive containing system software should be inserted into the USB interface on the control panel, then select the 【**Confirm**】 key for the system to read the USB device. After selecting the folder where update file is located, the update version will be displayed on the screen. After confirmation has been given, system update can be completed by following instructions on the screen. The system will be automatically restart after update, and system update will be completed.

- **For detailed instructions, please refer to section 1.8 for instructions on how to perform a System Update**

1.7.9 Tuning functions

Description of high-speed, high-precision functions:

- It is for users to set short-cut parameter keys in accordance with various curved surfaces and machining requirements.
- Users can take into consideration machining speed and precision requirements in accordance with the type of workpiece to be machined before changing the settings.
- The numerical values of these settings include linear acceleration/deceleration time, bell-type acceleration/deceleration time, post acceleration/deceleration time, turning reference speed, and 5 mm arc permissible speed.

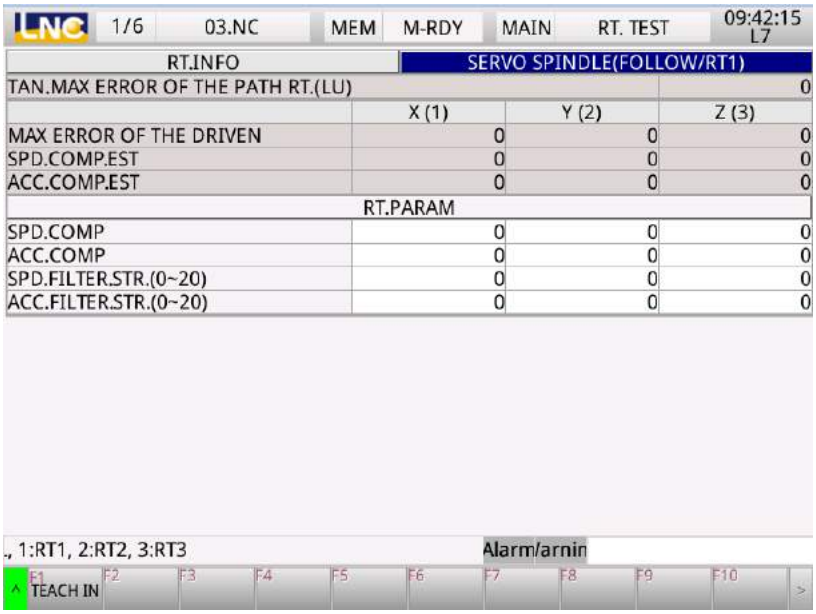


Instructions for using high-speed, high-precision functions:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Tuning functions】** key should be pressed before pressing the **【 High-speed/high-precision 】** key to access the high-speed/high-precision page.
- This function is for users to set up machining parameters. Users can choose to place more emphasis on machining speed or precision in accordance with the type of workpiece to be machined before changing the settings. The numerical values of these settings include linear acceleration/deceleration time, bell-type acceleration/deceleration time, post acceleration/deceleration time, turning reference speed, and 5 mm arc permissible speed.
- Arrow keys on the MDI panel can be used to move the cursor to the corresponding field, and a numerical value can be entered into the input area before pressing the **<Input>** key to complete the setting.

Description of the rigid tapping tuning function:

- It is for users to view rigid tapping information and configure relevant parameters.



Instructions for using rigid tapping tuning:

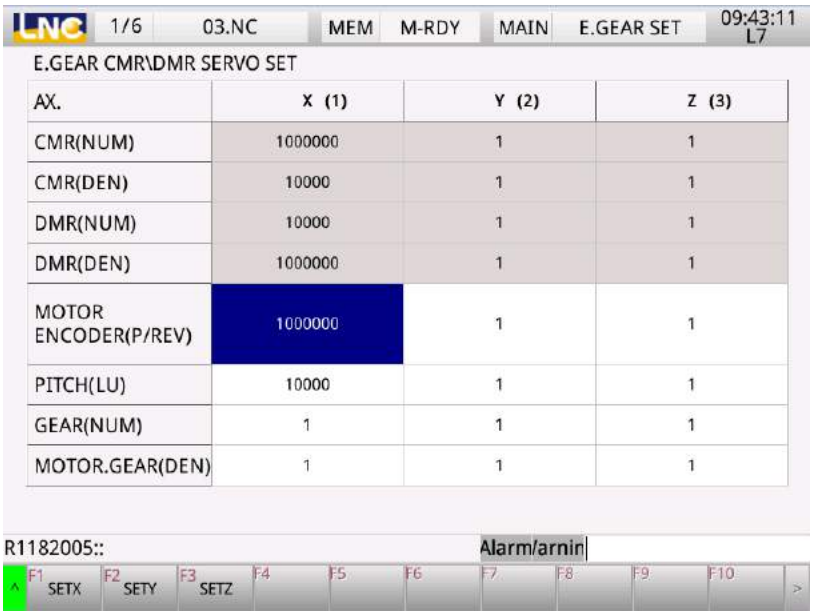
- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Tuning functions】** key should be pressed before pressing the **【Rigid tapping tuning】** key to access the rigid tapping tuning page.
- **【Guided input】**: sets the value of estimated measurement into the offset field.
- **【Rigid tapping program】**: set parameters related to rigid tapping test program.
- **【Start recording】**: start executing rigid tapping program and making a drawing.

Description of the C\DMR function:

- It is for users to conveniently set electronic gear ratio which can be divided into the two parts: servo axis and spindle.

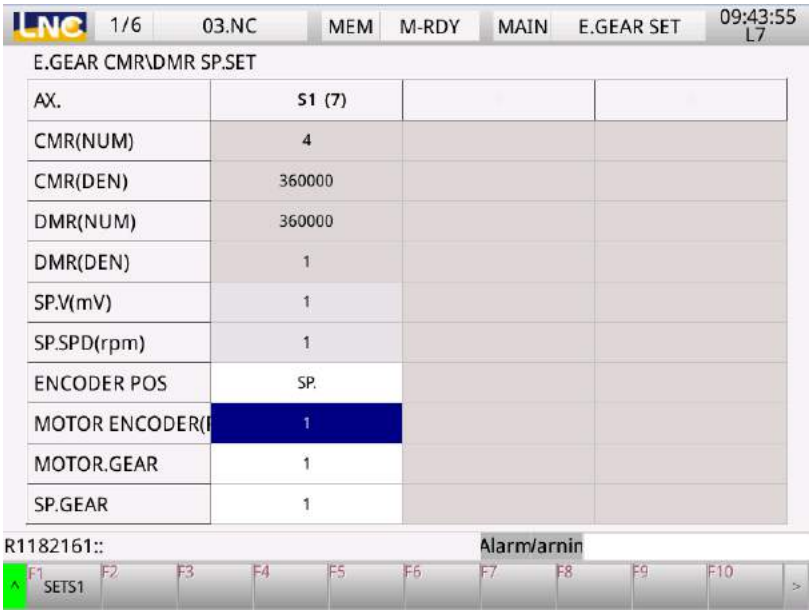
Instructions on using C\DMR:

1. Servo axis setting



- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**Tuning functions**】 key should be pressed before pressing the 【**C/DMR**】 key to access the page via the 【**Servo axis**】 key.
- After setting the fields for "Motor encoder", "Screw rod pitch", "Number of gear teeth on the screw rod side", and "Number of gear teeth on the motor side", the corresponding axial setting keys such as 【**Set X**】, 【**Set Y**】, and 【**Set Z**】 keys can be pressed to modify the current C\DMR values of the numerator and denominator.

2. Spindle setting

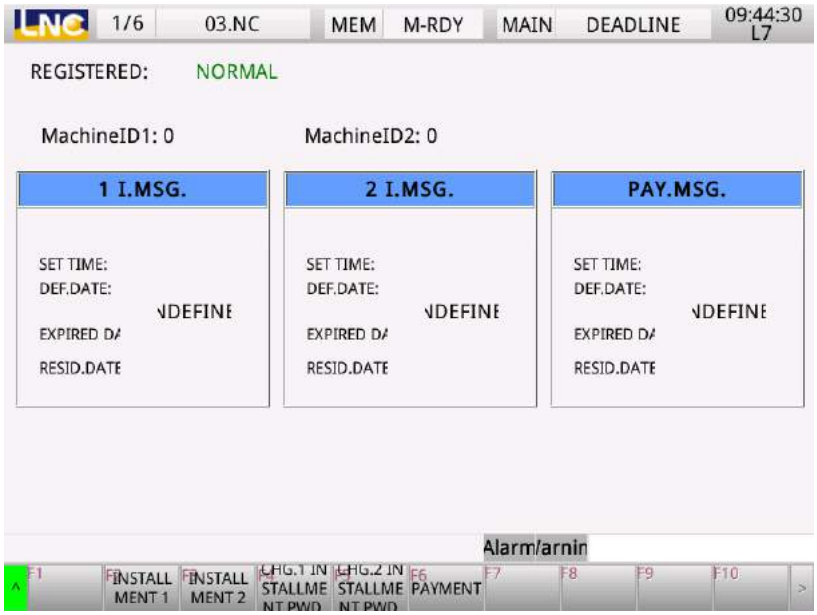


- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the **【Tuning functions】** key should be pressed before pressing the **【C/DMR】** key to access the page via **【Spindle】** key.
- After setting the fields of "Encoder position", "Motor encoder", "Number of gear teeth on the motor side", and "Number of gear teeth on the spindle side", the corresponding spindle setting keys such as **【Set S1】** and **【Set S2】** keys can be pressed to modify the current C\DMR values of the numerator and denominator.

1.7.10 Period of use

Description of the period of use function:

- This function allows machine factory/distributor to track and limit a customer's use of the controller, which can be used for handling periodic payments and removing the lock on payment by installments.



- If the usage limit has been reached, machining activation (CYCLE START) will not work. If (CYCLE START) is pressed when the usage limit has been reached, the system will issue an alert message stating that "the system's usage period has expired, please contact your supplier".

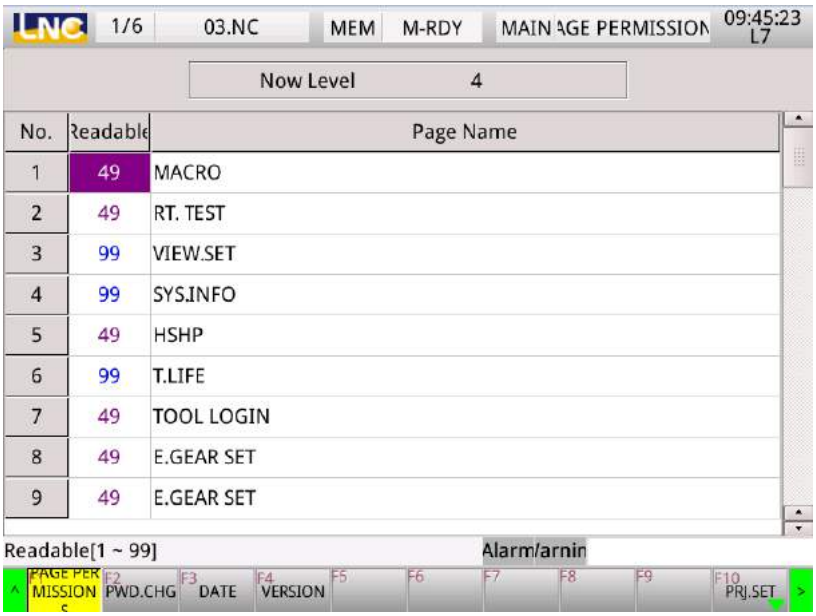
Instructions on usage period:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【**Service life**】 key can be pressed to access the service life page.

1.7.11 Page permissions

Description of the page permission function:

- The function screen which can be viewed or set up by users at various levels can be determined by the display permission and setting permissions displayed on this page.



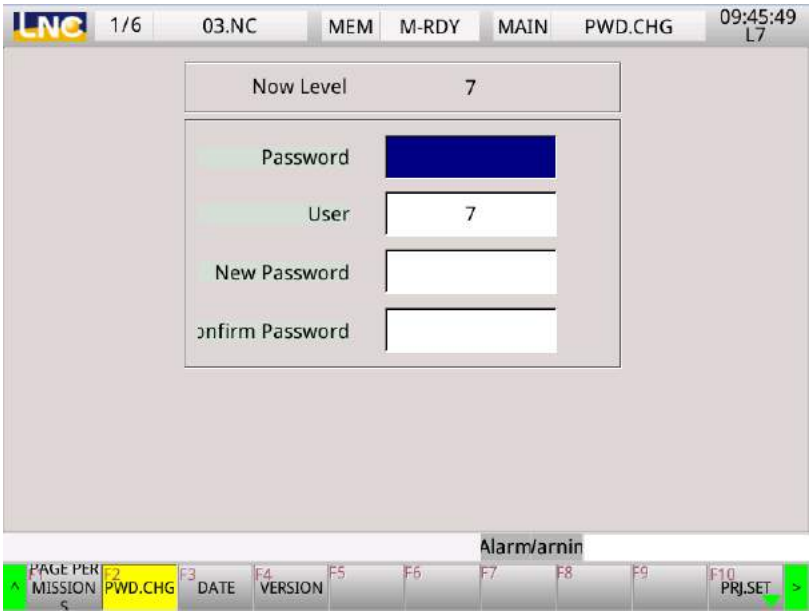
Instructions for setting up page permissions:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the【 > 】key can be pressed to access the page permission function via the 【**Page permission**】 key.
- After moving the cursor to the item number you wish to change, the permission code should be entered in the input field before pressing the **<Input>** key to complete the change.

1.7.12 Change password

Description of the change password function:

- It is for changing the password for accessing the current level The change password function is restricted to only allow users with higher permission levels to modify passwords for users with lower permission levels. This page will only be shown to users at this permission level.



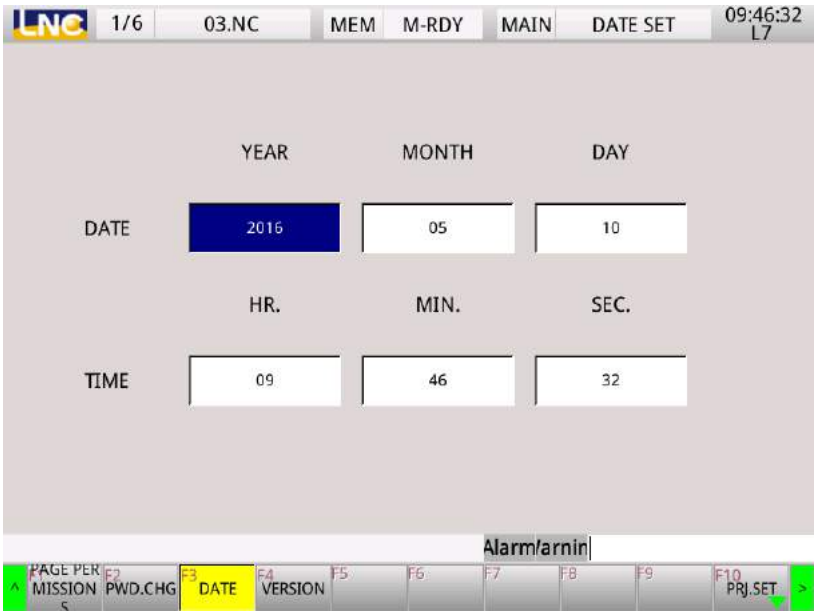
Instructions for changing password:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, pressing the 【 > 】 key to access the change password page via the 【**Change password**】 key.
- The password should be entered into the input area, press the **<Input>** key and confirm the password to complete the procedure.

1.7.13 Date and time

Description of date and time functions:

- It is for displaying the date and time of the controller system.



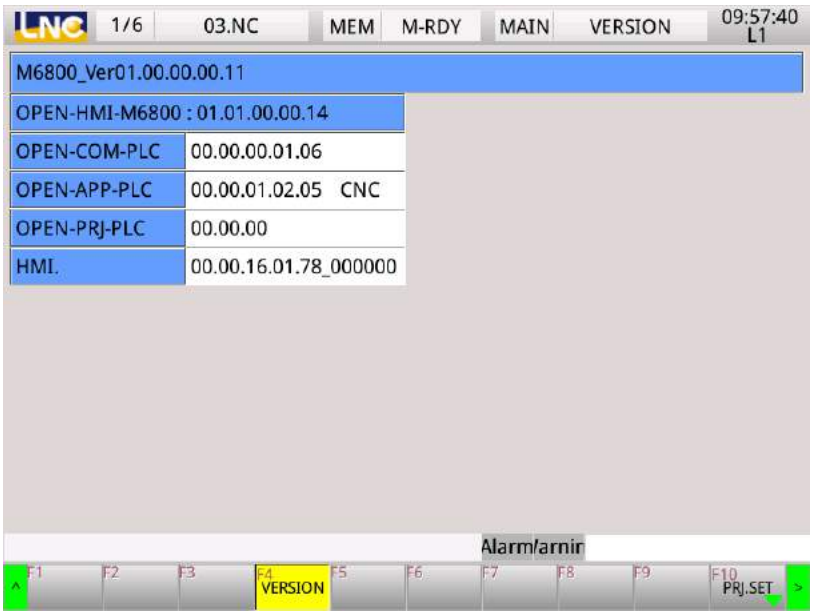
Instructions for setting date and time:

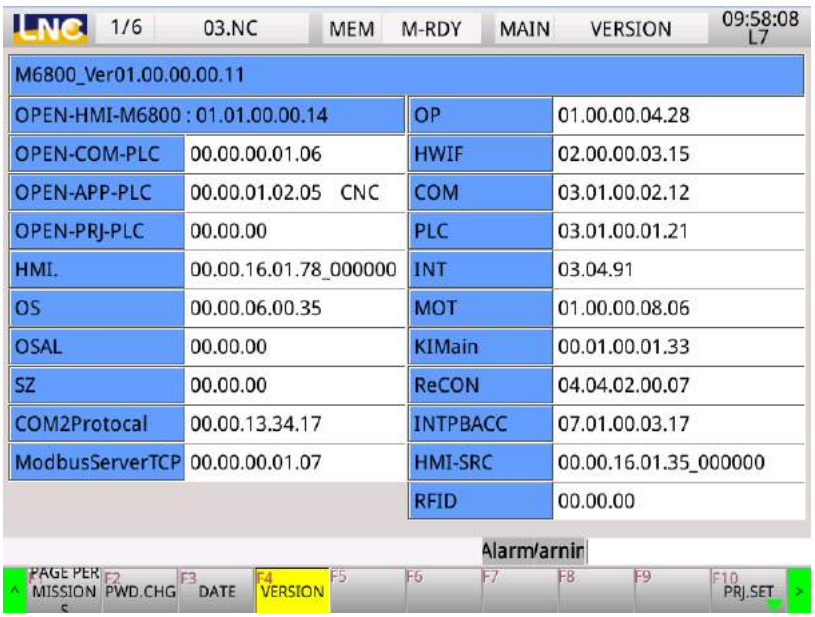
- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the【 **>** 】key can be pressed to access the date and time page via the 【**Date and time**】 key.

1.7.14 Version information

Description of the version information function:

- It is for displaying version information related to the system's hardware and software. This page can be used by designers as well as machine and system maintenance personnel for maintenance and inspection. Users can find out their software version from this page.
- Different information is displayed for users of different levels.





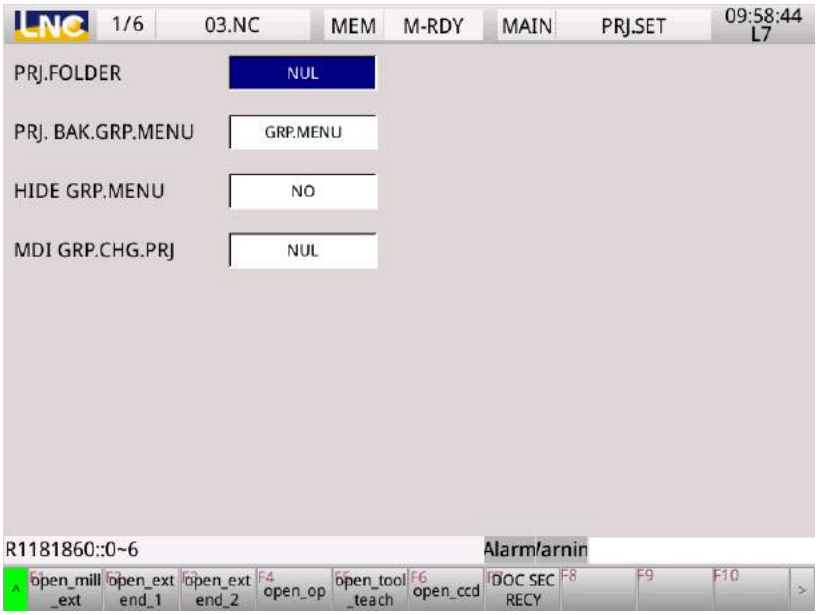
Instructions for accessing version information:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【 > 】key can be pressed to access the version information page via the 【**Version information**】 key.

1.7.15 Project settings

Description of the project setting function:

- In response to various industry demands, this system allows users to customize extra functions in addition to the general functions, which can be executed in coordination with project settings.



Instructions for configuring project settings:

- After using the **<MAINTE>** key on the MDI panel to switch to maintenance group, the 【 > 】key can be pressed to access the project settings page via the 【**Project settings**】 key.
- Description of various fields:
    1. **Subproject folder:** It is used for setting the subproject to be directly executed after machine startup. If this field is set, the home page will directly jump to the subproject page after system startup. (0 to 6, 0: None 1: open_mill_ext 2: open_ext_end_1 3: open_ext_end_2 4: open_op 5: open_tool_teach 6: open_ccd)

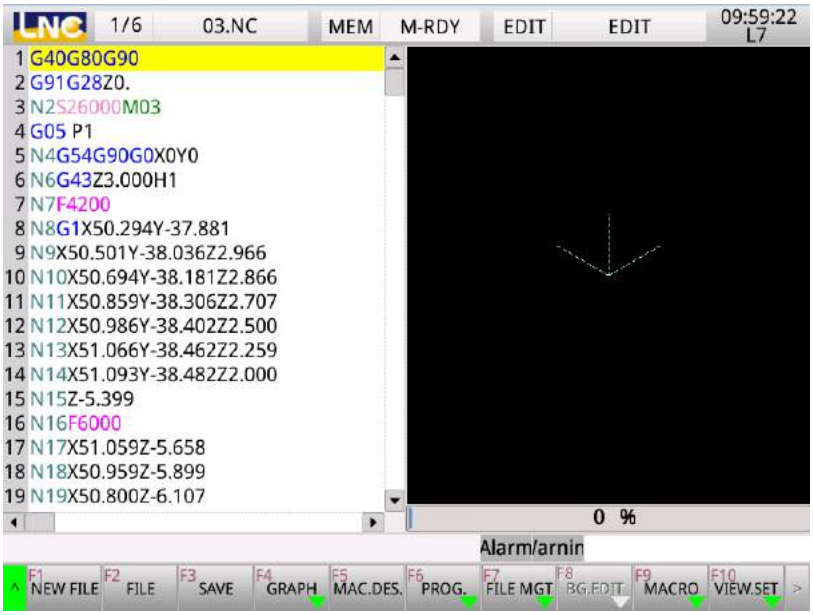
       If this field is set as 0, the subproject can be accessed by pressing the function key corresponding to the folder name in which the subproject is placed.
    2. **Subproject - return to group key:** It is for setting the key menu level which can be reached by pressing the return key on the subproject's key menu. (-1 to 5, -1: self-defined for the project 0: Group key menu 1: Monitoring group 2: Program group 3: compensation group 4: Diagnosis group 5: Maintenance group)
    3. **Hide group key menu:** It is for setting whether users can return to the group key menu level by pressing the return key. (0 to 1, 0: No 1: Yes)

4. **MDI group key replacement project:** It is for setting the group key to be replaced. For example: when it is set as 1, the <POS> key on the MDI panel can be pressed to show the subproject page rather than the monitoring group's page. (0 to 5, 0: None 1: Monitoring group 2: Program group 3: compensation group 4: Diagnosis group 5: Maintenance group)

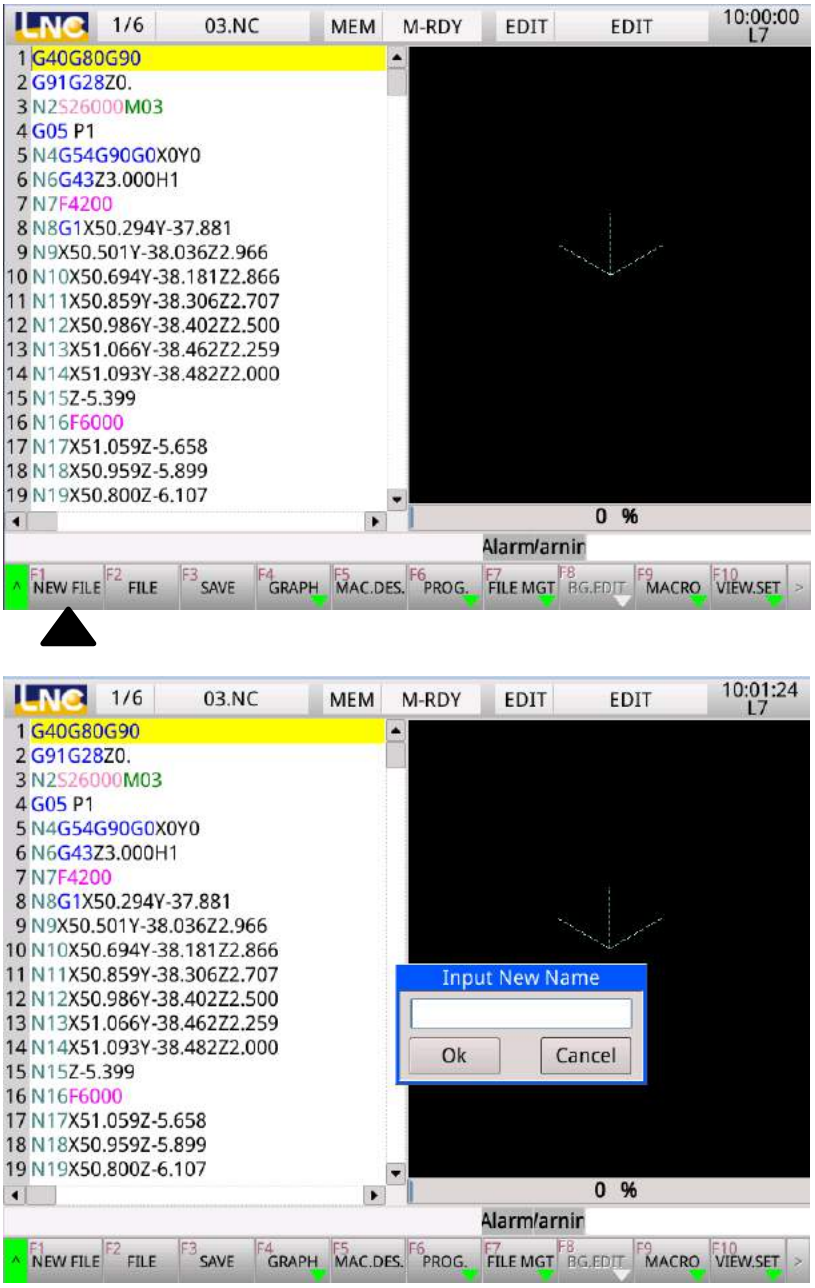
1.8 Usage instructions

1.8.1 Opening and editing a file

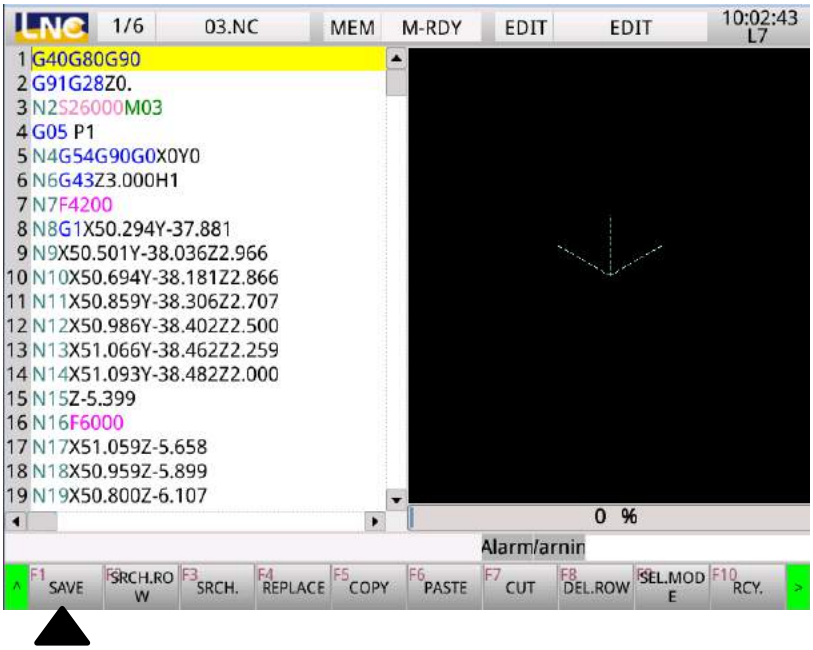
- The **<PROG>** key on the controller's MDI panel can be pressed to switch the controller's human machine page to program editing page. This partial function can be divided into modification of an original file and opening a new file.
- For editing current program, please press the 【**Edit**】 key under the "Ready" status in coordination with the auxiliary edit function key.



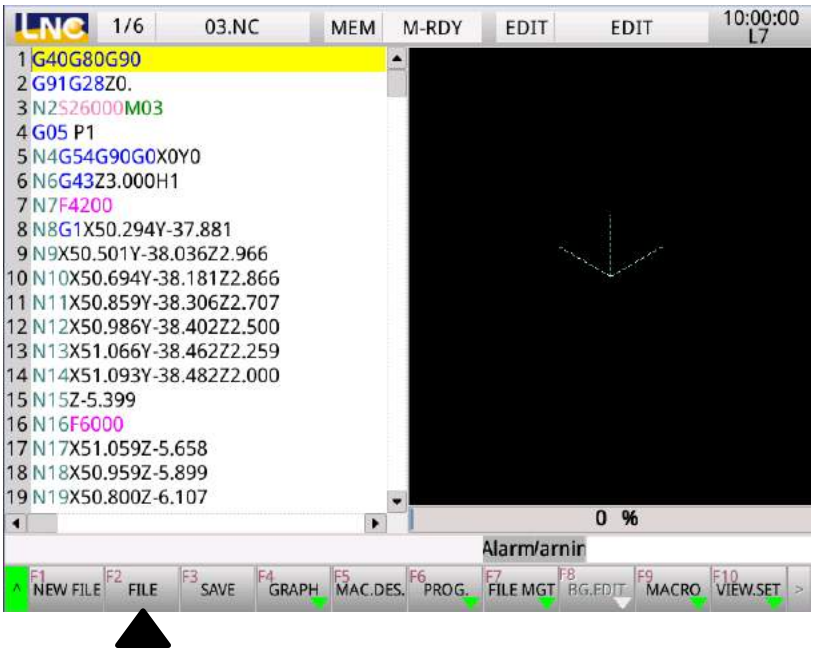
- For opening a new file, the【**Open new file**】key can be pressed to show a dialog window for entering a new file name, and then select **Confirm** to enter the editing screen (as shown in the figure below). By selecting **Cancel**, the dialog window for entering a new file name will be closed.



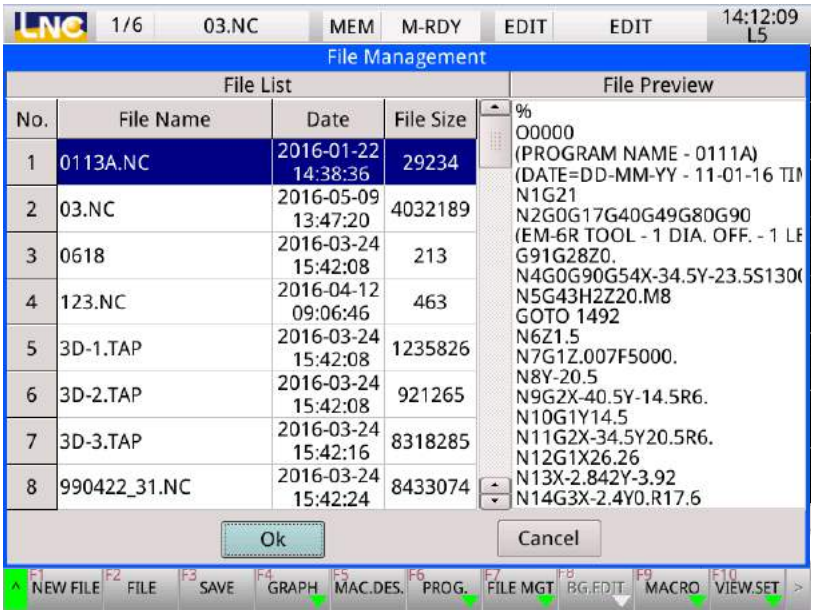
- After the completion of the open a new file procedure, the machining program to be executed can be edited in the editing window, or the 【**Edit**】 key can be pressed in coordination with supplementary editing function key. The **<Input>** key can be pressed during the editing process to start a new line.
- After editing is completed, the 【**Save**】 key can be pressed to save your progress.



- For modifying other existing files, the 【 ∧ 】 key or the **<PROG>** key on the MDI panel can be used to return to the human machine page keys as shown in the figure above, press the 【**Open existing file**】 key to access the file list page.



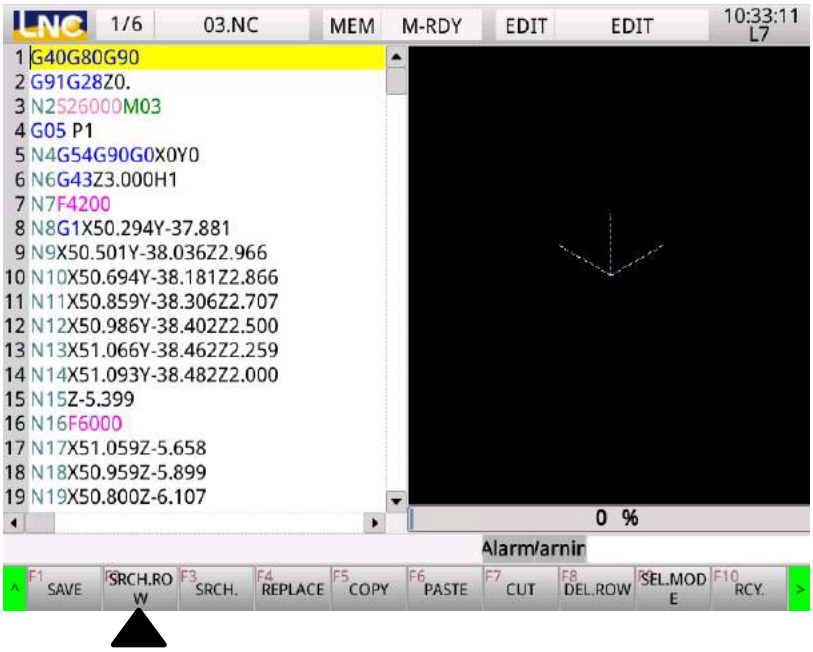
- After using arrow keys on the MDI panel to select the file to be modified, select **Confirm** to enter the file editing page to make changes.

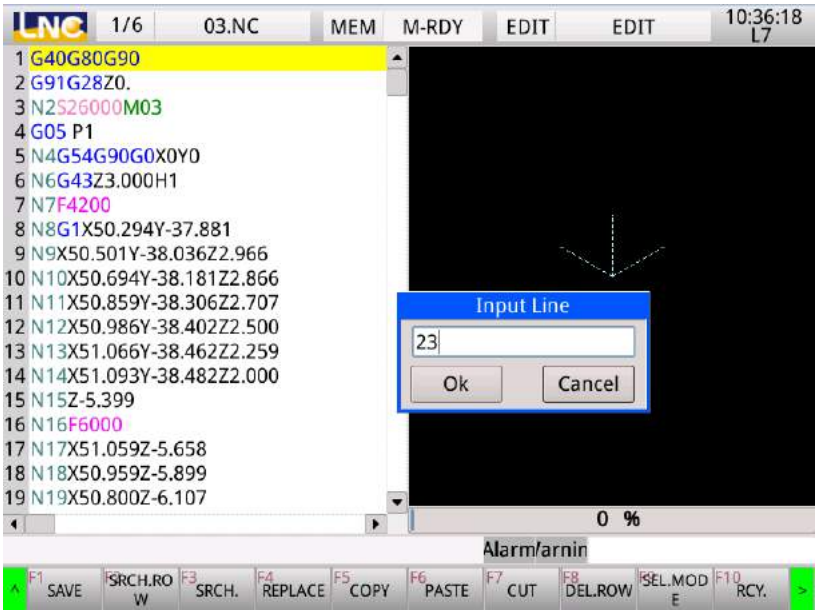


Various functions which can be used during editing are shown below:

Row position》

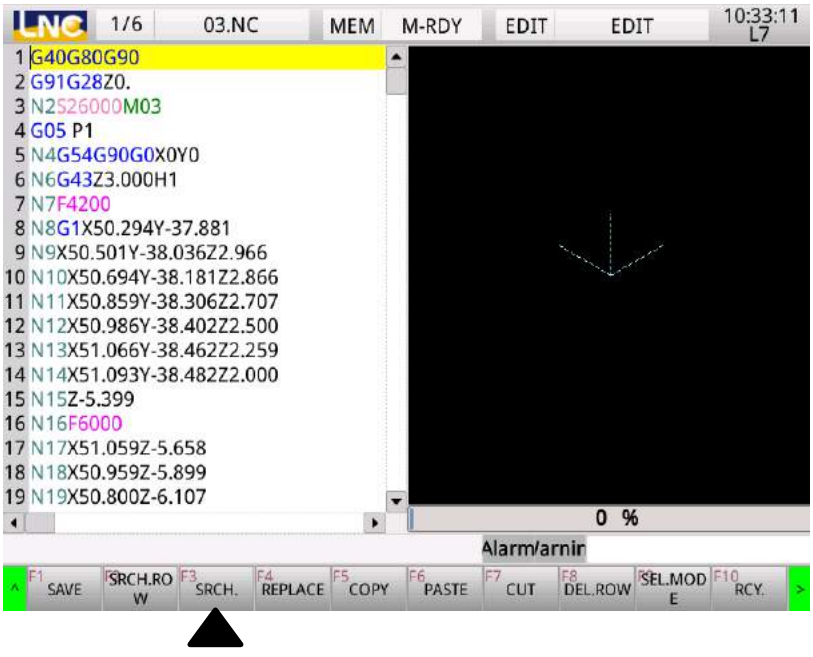
- For direct modification of a certain line within the program, the 【**Row position**】 key can be pressed to show a row position window, and then a line number can be entered before selecting **Confirm** for the cursor to move to the single block of that line.

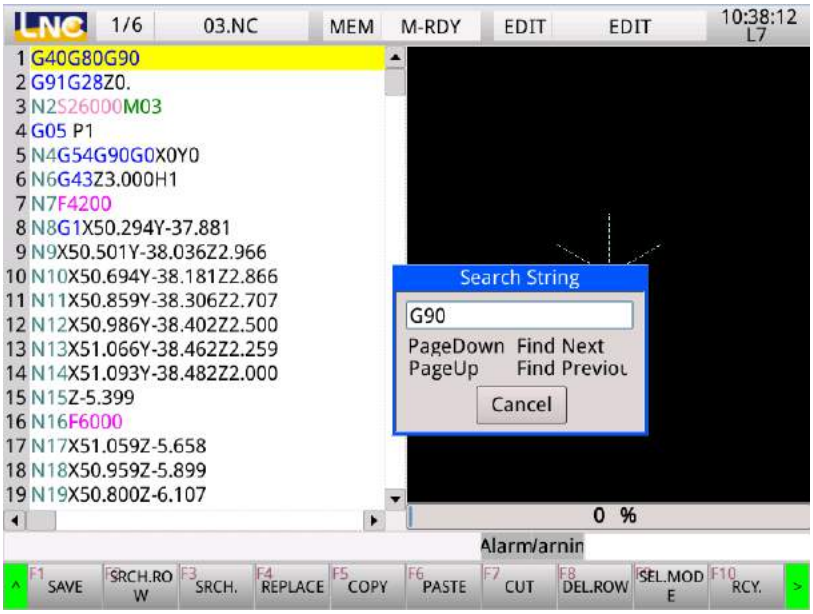




Find》

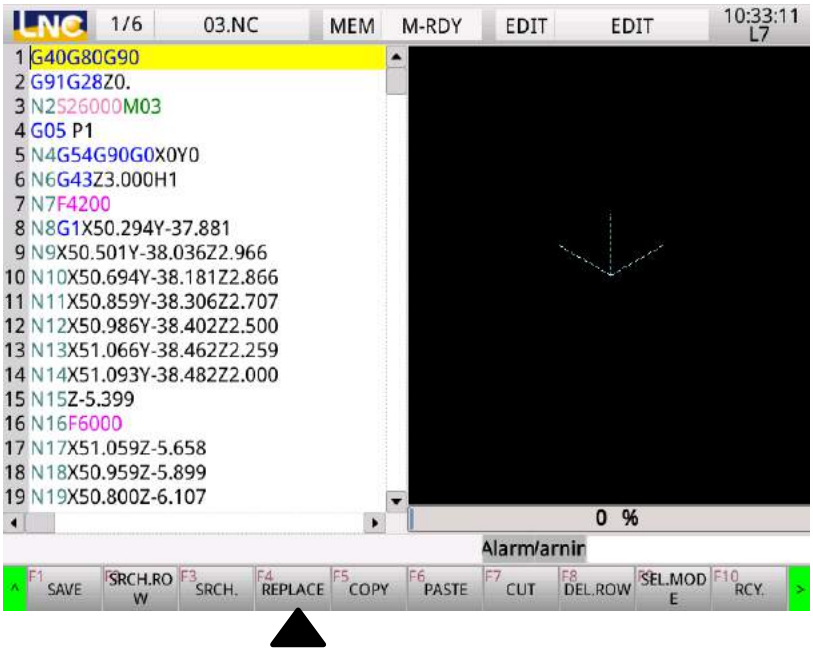
- For finding any characters/strings in the program, the【**Find**】key can be pressed to display a search window for entering the characters/string you wish to find, then **<PgDn>** or **<PgUp>** keys can be used for upward or downward searching; or select **Cancel** to close this search window.

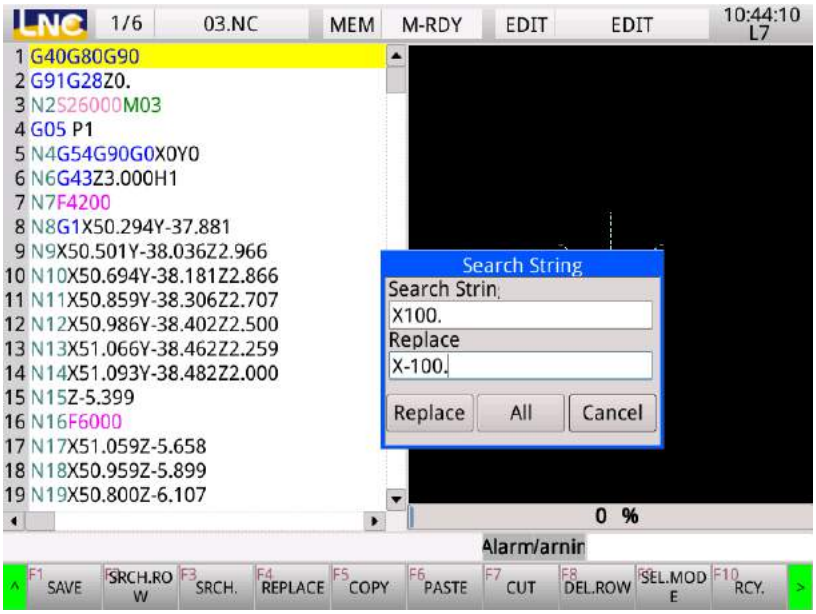




Replace》

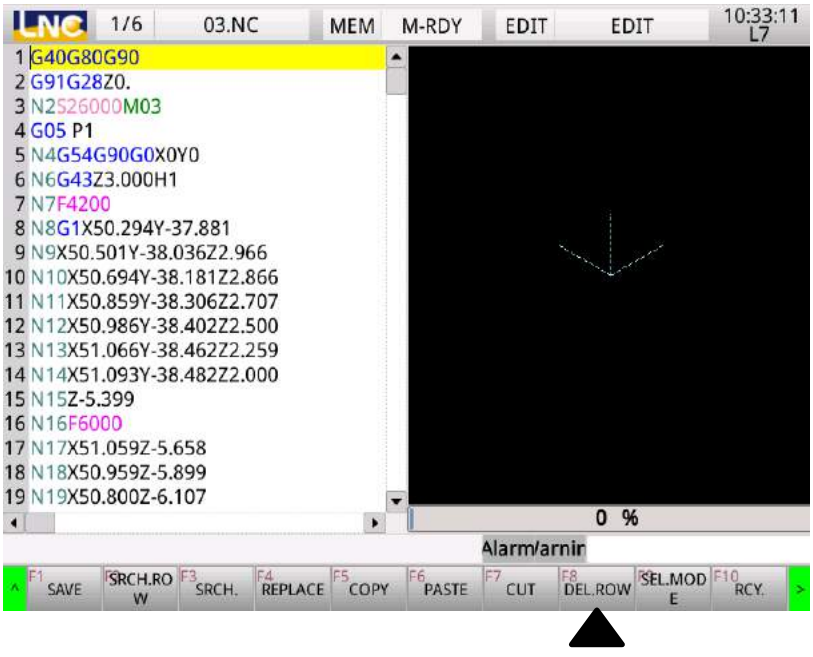
- Program editing also provides the function to replace a character/string. A text replacement window will be displayed after pressing the 【**Replace**】 key, and the character/string to be modified can be entered in the "Find string" field before pressing arrow keys on the MDI panel to move the cursor to the "Replace string" field; where the character/string can be entered.
- For example, if X100. is to be replaced by X125., selecting **Replace** will only replace that particular X100. with X125., and selecting **Replace all** will replace all X100. in the program with X125. The text replacement window can be closed by selecting **Cancel**.





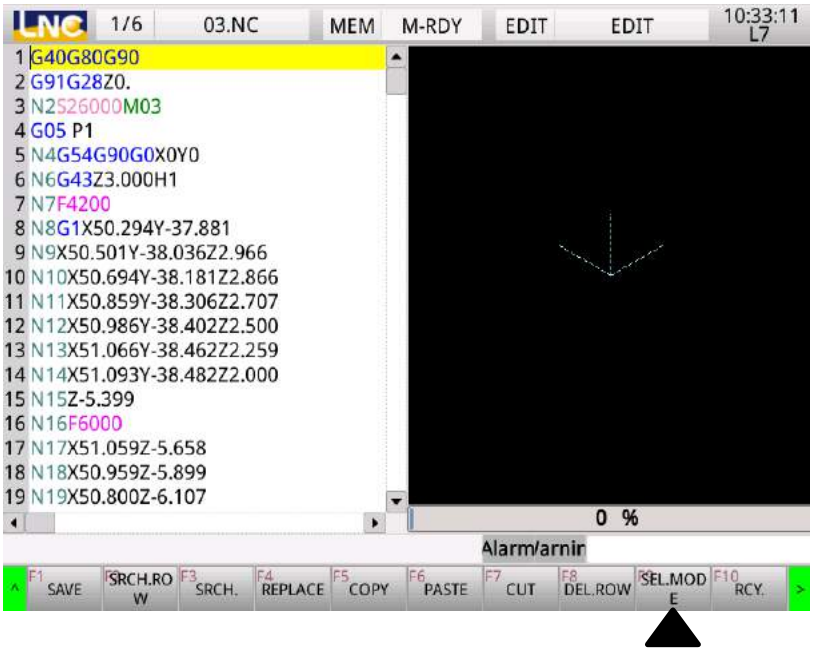
Delete row》

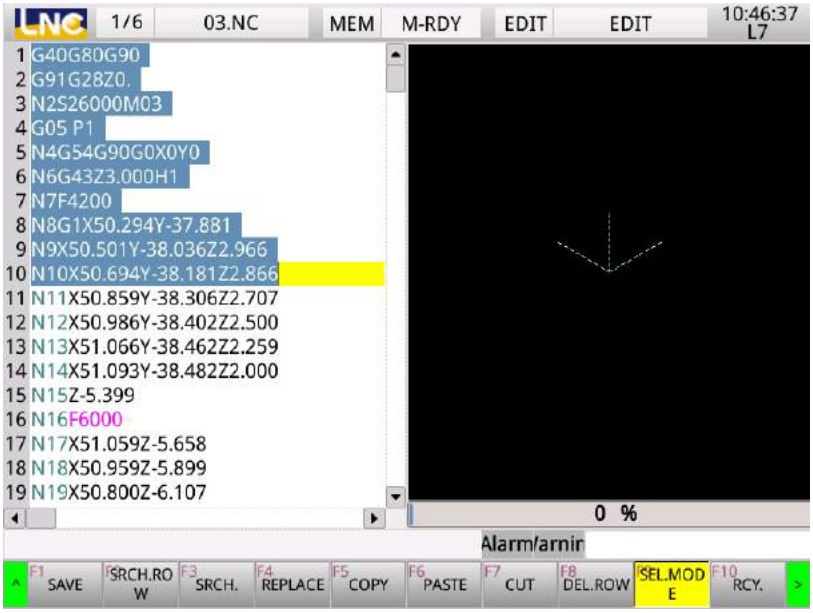
- For deleting an entire line of command in the program, the cursor should be moved to the line number to be deleted before pressing the 【**Delete row**】 key, a confirmation window will appear, simply select **[Yes]** to complete the process.



Selection mode》

- For selecting the range of command in the program, the 【**Selection mode**】 key can be pressed while using arrow keys on the MDI panel to select the desired range. Press the 【**Selection mode**】 key again in order to use 【**Cut**】 or 【**Copy**】 to copy the selected range.





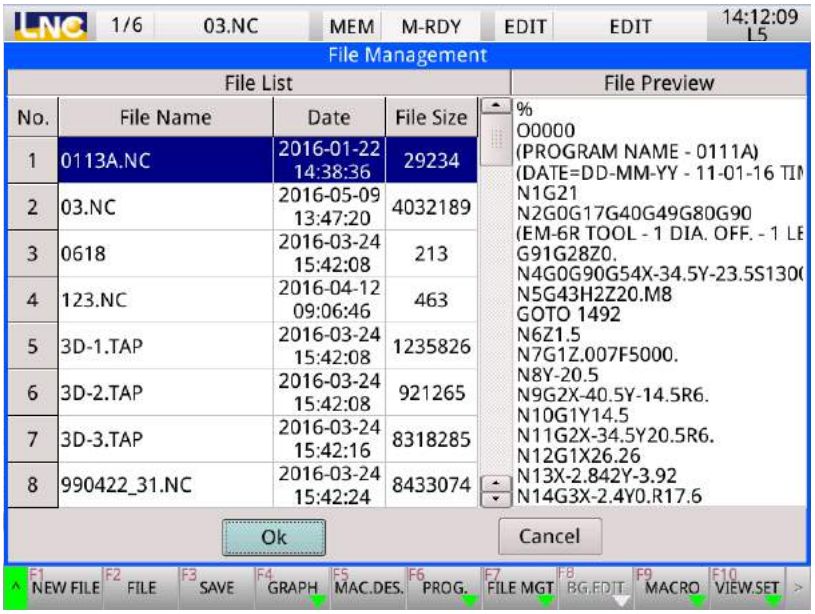
- Take "Copy" as an example. After selecting a desired range, the 【**Copy**】 key should be pressed before using arrow keys to move the cursor to the location you wish to paste the content, and then press 【**Paste**】 to complete the copy process.

Undo and Redo》

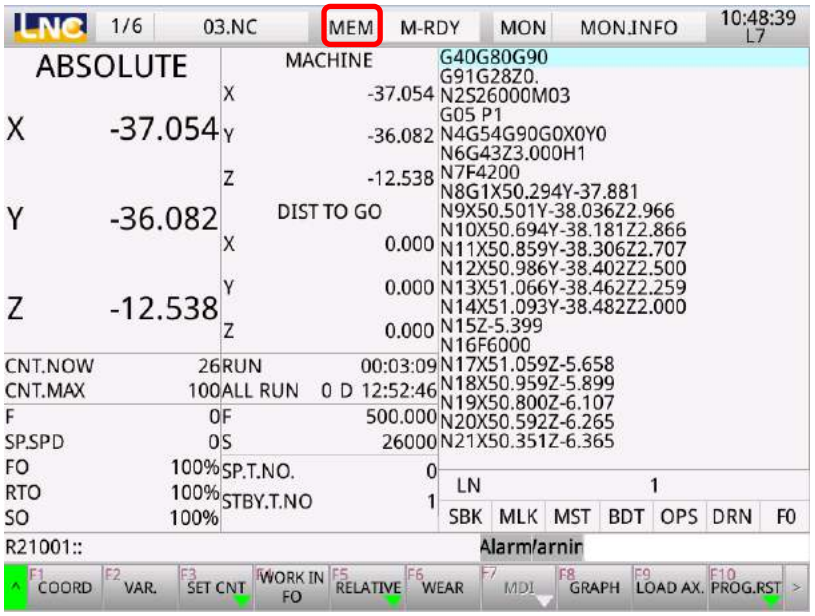
- The 【**Undo**】 key can be pressed to restore the deleted or modified command. The 【 **>** 】 key can be pressed before selecting the 【**Redo**】 key to reapply previous action.

1.8.2 Execute machining

- After starting up the machine and entering the system or releasing the emergency stop (EMG), the **<ZRN>** key on the controller's OP panel should be pressed to execute the reference point return procedure.
- Press the **<PROG>** key, followed by pressing the 【**Open existing file**】 key to enter the file list page as shown in the figure below.



- Arrow keys on the MDI panel can be used to select the machining program, press **Confirm** to open the machining program or open a new file directly for editing the machining program.
- Set the coordinate system in accordance with the machining program (please refer to the description of coordinate system settings).
- Set the tool offset value in accordance with the machining program (please refer to the description of tool offset settings).
- To avoid damages caused by tool and workpiece colliding with each other, the **<ZRN>** key should be pressed to execute the reference point return procedure for the Z axis.
- Press the **<MEM>** key after the reference point return procedure is completed in order to switch the system to memory mode (as shown in the figure below), and then press the **<Reset>** key to return the cursor to the starting row of the program.



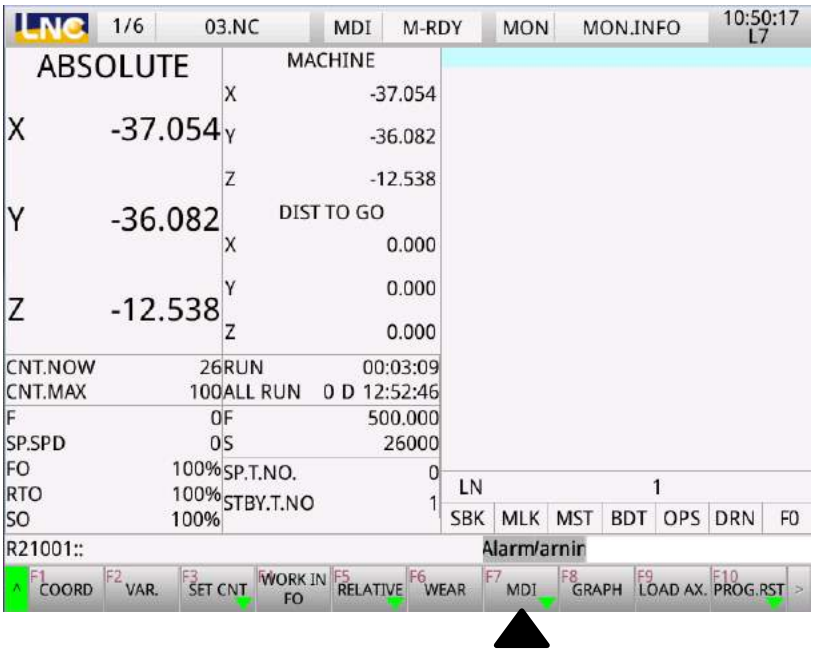
- Press the program start 〔CYCLE START〕 key to start the machining process.

1.8.3 Using the MDI function

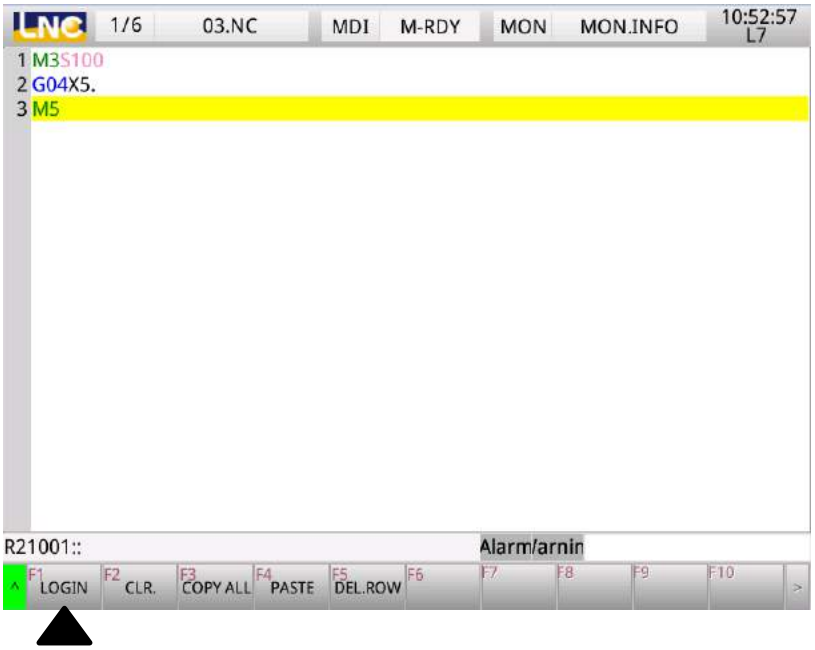
- Turn the knob on OP panel to MDI mode.

MDI

- After pressing the **<POS>** key on the controller's MDI panel to switch controller's human machine page to monitoring page, the 【**MDI**】 key on the controller human machine page should be pressed to confirm that the status mode is MDI before editing.



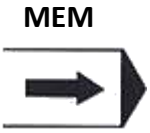
- During editing, the **<Input>** key can be pressed to start editing a new line; the 【**Sign-in**】 key should be pressed after editing is completed.



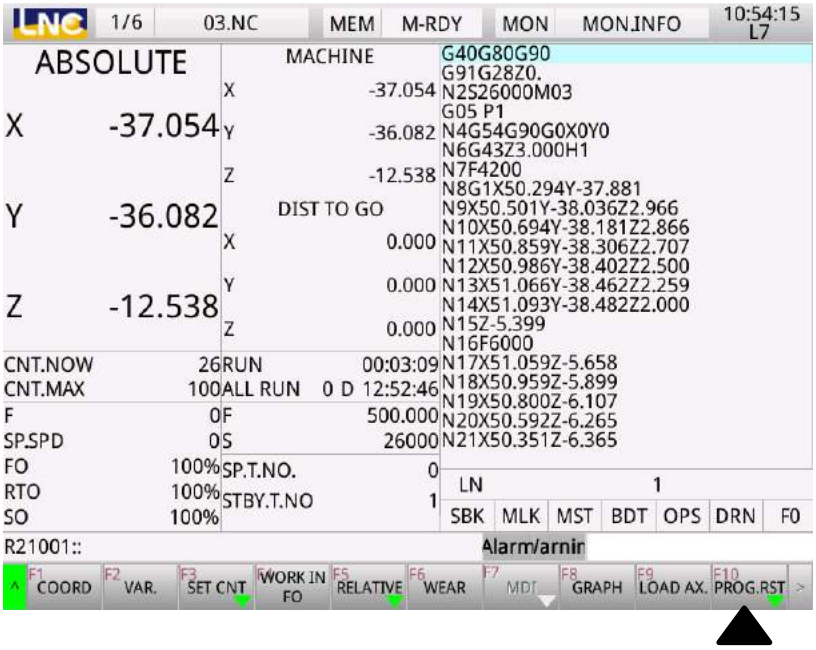
- Press the program start 〔CYCLE START〕 button to execute this function.

1.8.4 Program restart

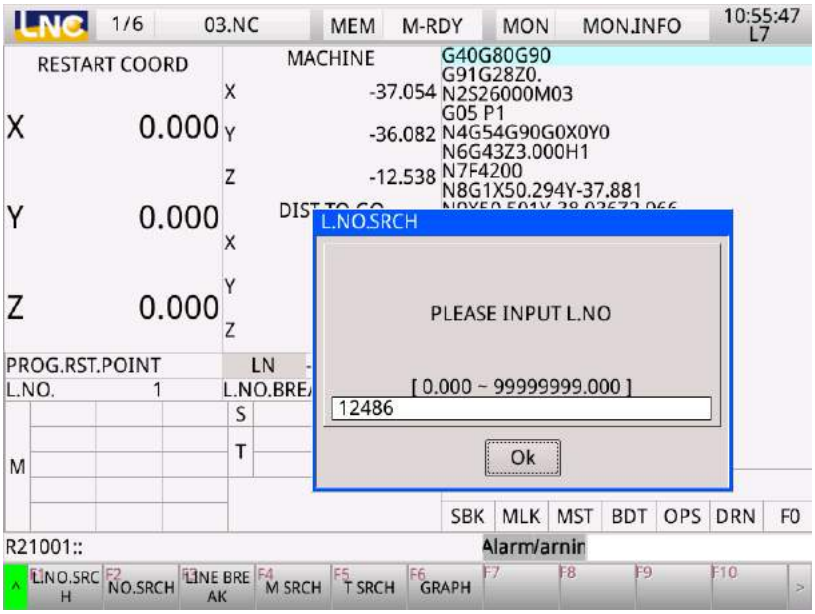
- Turn the knob on the OP panel to MEM mode under the ready state.

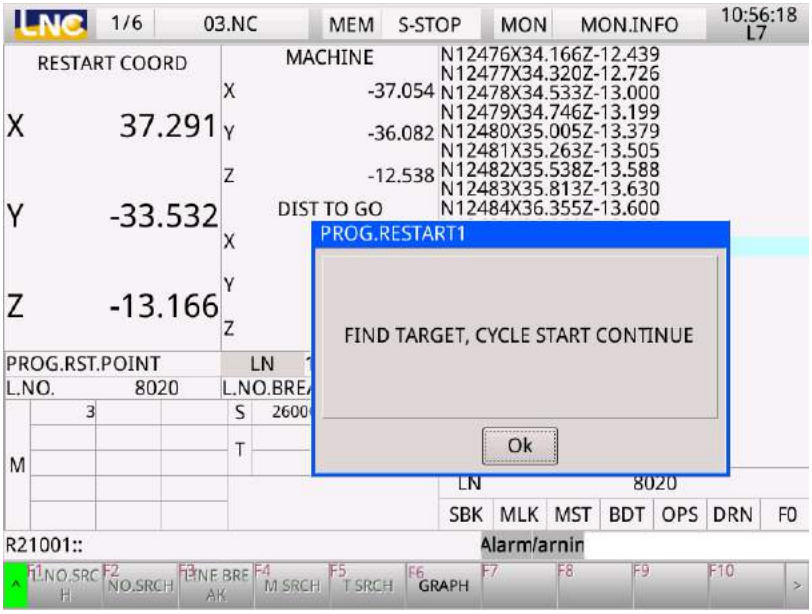


- Press the 【**Program restart**】 key on controller human machine page.

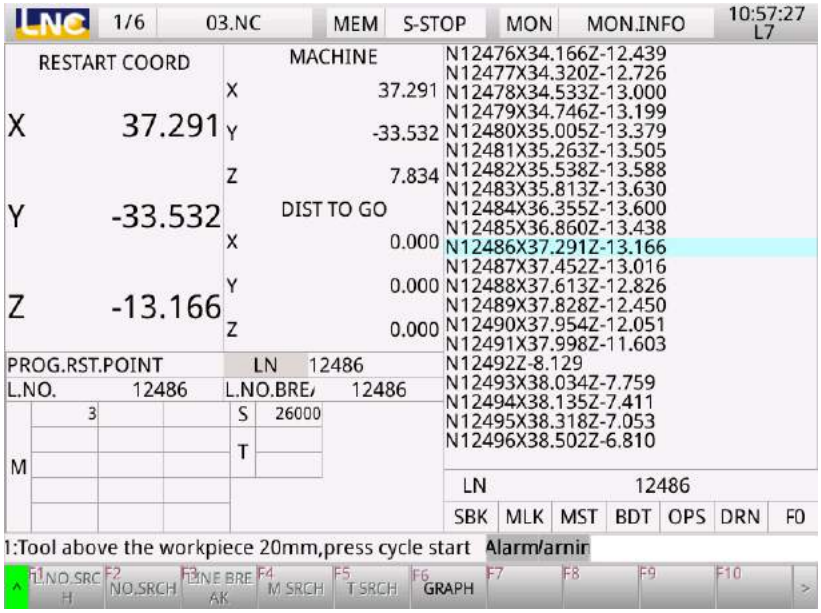


- To search for a given line number, the 【Find line number】 (or press the 【Find serial number】 key to find a given serial number, and so on) key should be pressed to display a search window. The line number you wish to find should be entered, then press the **<Input>** key to start searching for the line number of the restart point. After the target has been found, a confirmation window will appear on the human machine page (as shown in figure below) with the system under the state of section stop. Press the **Confirm** key to complete this process.





- And then press the program start〔CYCLE START〕button for the tool to be moved to the coordinate position of that program block and stopped 20 mm above the workpiece before a confirmation window appears. Press the **Confirm** key start the tool and continue with the machining process.


1/6
03.NC
MEM
S-STOP
MON
MON.INFO
10:57:27
L7
RESTART COORD
X 37.291
Y -33.532
Z -13.166
MACHINE
X 37.291
Y -33.532
Z 7.834
DIST TO GO
X 0.000
Y 0.000
Z 0.000
PROG.RST.POINT
LN 12486
L.NO. 12486
L.NO.BRE/ 12486
M 3
S 26000
T
N12476X34.166Z-12.439
N12477X34.320Z-12.726
N12478X34.533Z-13.000
N12479X34.746Z-13.199
N12480X35.005Z-13.379
N12481X35.263Z-13.505
N12482X35.538Z-13.588
N12483X35.813Z-13.630
N12484X36.355Z-13.600
N12485X36.860Z-13.438
N12486X37.291Z-13.166
N12487X37.452Z-13.016
N12488X37.613Z-12.826
N12489X37.828Z-12.450
N12490X37.954Z-12.051
N12491X37.998Z-11.603
N12492Z-8.129
N12493X38.034Z-7.759
N12494X38.135Z-7.411
N12495X38.318Z-7.053
N12496X38.502Z-6.810
LN 12486
SBK MLK MST BDT OPS DRN F0
1:Tool above the workpiece 20mm,press cycle start
Alarm/arnir
F1 L.NO.SRCH
F2 NO.SRCH
F3 LINE BREAK
F4 M SRCH
F5 T SRCH
F6 GRAPH
F7
F8
F9
F10

1.8.5 Automatic tool alignment

1.8.5.1 CNC\woodworking machine & applicable industries

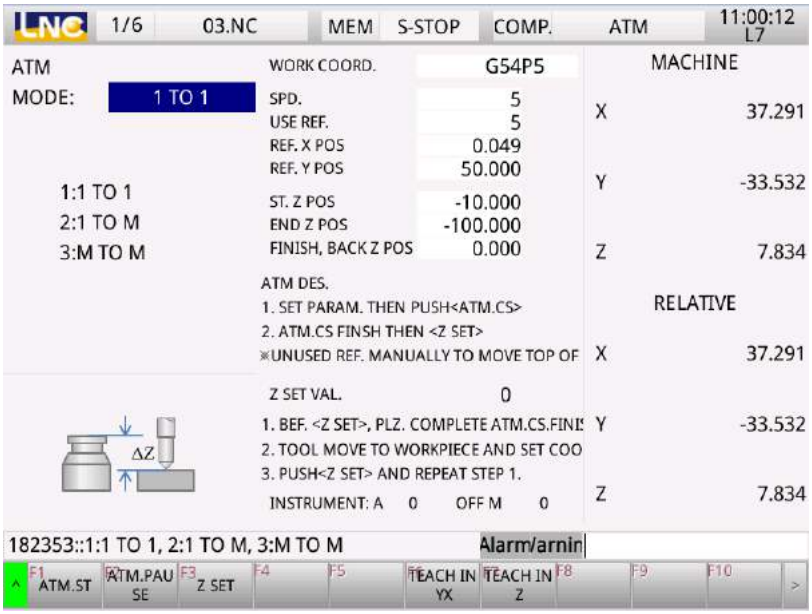
The automatic tool alignment function provides three types of tool alignment methods: "Single tool/single workpiece", "Single tool/multiple workpieces", and "Multiple tools/multiple workpieces". Different tool alignment methods can be selected with respect to different machining applications.

The numerical value for tool offset will be automatically inserted in accordance with the selected tool alignment approach:

1. Tool offset: Z length offset.
2. Workpiece coordinate system Z axis (G54 to G59, G54P1 to P100 coordinate systems).
3. Workpiece coordinate system offset value (00 coordinate system).

《Single tool/single workpiece》

For the single tool/single workpiece method, the tool length of automatic tool alignment will be saved into the Z axis coordinate of the designated work coordinate system, and the gap value between the tool alignment instrument and workpiece surface should be saved into the 00 coordinate system in accordance with Z axis gap settings. The operation is described below:

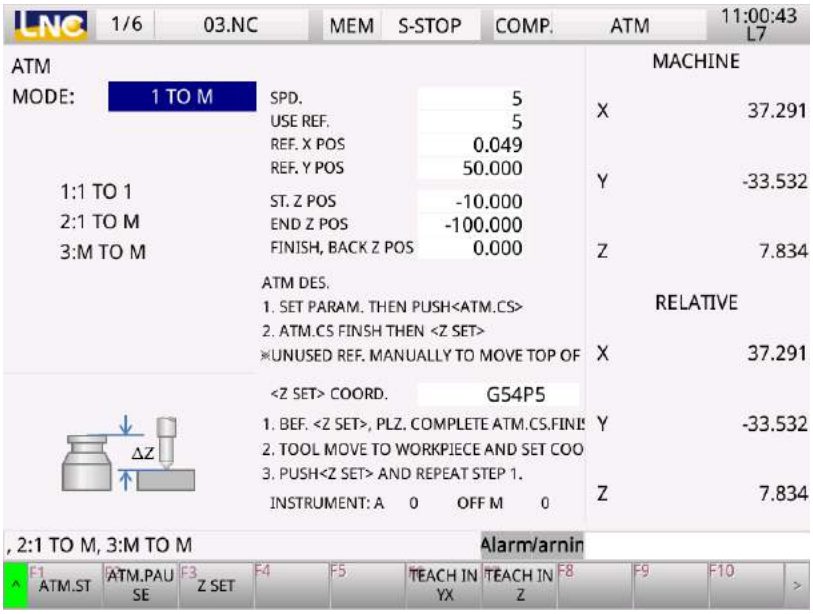


- Move the cursor to the tool alignment mode field, enter the value 1 in the input area and press the **<Input>** key to set the mode as single tool/single workpiece.
- Set "work coordinate system": Move the cursor to the "Work coordinate system" field, enter a numerical value (from 54 to 59) in the input area and press the **<Input>** key. If the extended coordinate system G54 is being used, the "Work coordinate system" field should be set as G54 before moving the cursor to the next field (extended coordinate system), enter a numerical value (1 to 100) in the input area and pressing the **<Input>** key.
- Set "Measurement speed": Move the cursor to "Measurement speed", enter a numerical value in the input field and then press the **<Input>** key.

- Set "Whether to use a reference point": Move the cursor to the "Whether to use a reference point" field, enter a numerical value (from 0 to 1) in the input field and press the **<Input>** key. If a reference point is to be used, it should be switched to **hand wheel mode** (MPG), and the tool should be moved to the position above the tool alignment instrument. After confirming that the tool tip is right above the tool alignment instrument, the cursor should be moved to the "Tool alignment instrument's X coordinate reference point" field, press the **【XY axis guide】** key to set up both "Tool alignment instrument's X coordinate reference point" and "Tool alignment instrument's Y coordinate reference point". If a reference point is not used, please move the tool to the position above the tool alignment instrument manually.
- Replace the current tool with the longest tool, use the hand wheel to move the tool (and stop when the tool tip is at a certain distance away from the tool alignment instrument) while making sure the tool tip is not in contact with the tool alignment instrument. Move the cursor to the "Tool alignment starting point Z" and press the **【Z axis guide】** key for guided input.
- Replace the current tool with the shortest tool, use the hand wheel to move the tool such that the tool tip is deviated from the tool alignment instrument by a small distance (not directly above the tool alignment instrument), move the tool tip to a location at around 2 mm (not too much) lower than the normal position of the tool alignment instrument and stop right here. The cursor should be moved to the "Z axis lowest machine coordinate" field before pressing the **【Z axis guide】** for guided input. (With the protective mechanism, when the tool alignment instrument has malfunctioned, the Z axis will not be moved further downward)
- Set the numerical value for the "Return to safety point Z after tool alignment" field.
- Move the tool to a proper height (higher than the tool alignment instrument) and press the **【Start tool alignment】** key to execute the automatic tool alignment procedure. The system will fill in the designated coordinate system with the tool length compensation value after completing automatic tool alignment.
- Switch to **hand wheel mode** (MPG), move the tool to a position above the **workpiece** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the 00 coordinate system.

《Single tool/multiple workpieces》

For the single tool/multiple workpieces method, the tool length of automatic tool alignment will be saved into the 00 coordinate system, and the gap value between the tool alignment instrument and workpiece surface should be saved into the designated work coordinate system in accordance with Z axis gap settings. The operation is described below:



- Move the cursor to the tool alignment field, enter the value 2 in the input area and press the **<Input>** key to set the mode as single tool/multiple workpieces.
- Set "Measurement speed": Move the cursor to "Measurement speed", enter a numerical value in the input field and then press the **<Input>** key.
- Set "Whether to use a reference point": Move the cursor to the "Whether to use a reference point" field, enter a numerical value (from 0 to 1) in the input field and press the **<Input>** key. If a reference point is to be used, it should be switched to **hand wheel mode** (MPG), and the tool should be moved to the position above the tool alignment instrument. After confirming that the tool tip is right above the tool alignment instrument, the cursor should be moved to the "Tool alignment instrument's X coordinate reference point" field, press the **【XY axis guide】** key to set up both "Tool alignment instrument's X coordinate reference point" and "Tool alignment instrument's Y coordinate reference point". If a reference point is not used, please move the tool to the position above the tool alignment instrument manually.
- Replace the current tool with the longest tool, use the hand wheel to move the tool (and stop when the tool tip is at a certain distance away from the tool alignment instrument) while making sure the tool tip is not in contact with the tool alignment instrument. Move the cursor to the "Tool alignment starting point Z" and press the **【Z axis guide】** key for guided input.
- Replace the current tool with the shortest tool, use the hand wheel to move the tool such that the tool tip is deviated from the tool alignment instrument by a small distance (not directly above the tool alignment instrument), move the tool tip to a location at around 2 mm (not too much) lower than the normal position of the tool alignment instrument and stop right here. The cursor should be moved to the "Z axis lowest machine coordinate" field before pressing the **【Z axis guide】** for guided input. (With the protective mechanism, when the tool alignment instrument has malfunctioned, the Z axis will not be moved further downward)
- Set the numerical value for the "Return to safety point Z after tool alignment" field.

- Move the tool to a proper height (higher than the tool alignment instrument) and press the **【Start tool alignment】** key to execute the automatic tool alignment procedure. The system will fill in coordinate system 00 with the tool length compensation value after completing automatic tool alignment.
- Set "Gap setting for the work coordinate system": Move the cursor to the "Work coordinate system" field, enter a numerical value (from 54 to 59) in the input area and press the **<Input>** key. If the extended coordinate system G54 is being used, the "Work coordinate system" field should be set as G54 before moving the cursor to the next field (extended coordinate system), enter a numerical value (1 to 100) in the input area and pressing the **<Input>** key.
- Switch to **hand wheel mode** (MPG), move the tool to a position above the **workpiece** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the designated coordinate system.

Note: The approach for setting the gap of each workpiece is as shown below:

For example: There are two workpieces; one is based on the G55 coordinate system (workpiece A) and the other is based on the G54P100 coordinate system (Workpiece B)

Set the gap value for workpiece A:

i. Move the cursor to the "Work coordinate system" field, enter the value 55 in the input area and press the **<Input>** key.

ii. Switch to **hand wheel mode** (MPG), move the tool to the position above **workpiece A** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the G55 coordinate system.

Set the gap value for workpiece B:

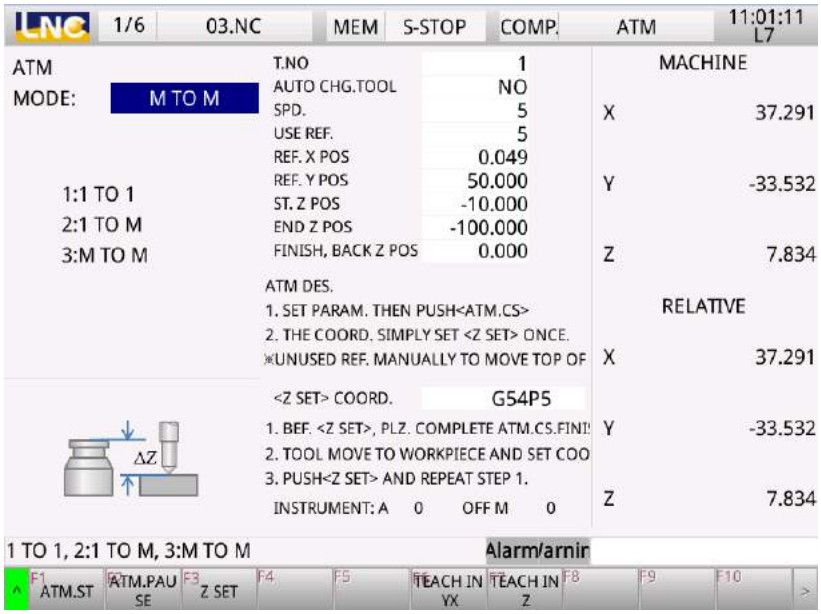
i. Move the cursor to the "Work coordinate system" field, enter the value 54 in the input area and press the **<Input>** key.

ii. Move the cursor to the next field (extended coordinate system), enter the value 100 in the input area and press the **<Input>** key.

iii. Switch to **hand wheel mode** (MPG), move the tool to the position above **workpiece B** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the G54P100 coordinate system.

《Multiple tools/multiple workpieces》

For the multiple tools/multiple workpieces method, the tool length of automatic tool alignment will be saved into Z length offset of the tool length offset number corresponding to the tool, and the gap value between the tool alignment instrument and workpiece surface should be saved into the designated work coordinate system in accordance with Z axis gap settings. The operation is described below:



- Move the cursor to the tool alignment mode field, enter the value 3 in the input area and press the **<Input>** key to set the mode as single tool/multiple workpieces.
- Set "Tool number": Move the cursor to the "Tool number" field, enter a numerical value (from 1 to 99) in the input area and press the **<Input>** key.
- Set "Is automatic tool change required": Move the cursor to the "Is automatic tool change required" field, enter a numerical value (from 0 to 1) in the input field and press the **<Input>** key.
- Set "Measurement speed": Move the cursor to "Measurement speed", enter a numerical value in the input field and then press the **<Input>** key.
- Set "Whether to use a reference point": Move the cursor to the "Whether to use a reference point" field, enter a numerical value (from 0 to 1) in the input field and press the **<Input>** key. If a reference point is to be used, it should be switched to **hand wheel mode** (MPG), and the tool should be moved to the position above the tool alignment instrument. After confirming that the tool tip is right above the tool alignment instrument, the cursor should be moved to the "Tool alignment instrument's X coordinate reference point" field, press the **【XY axis guide】** key to set up both "Tool alignment instrument's X coordinate reference point" and "Tool alignment instrument's Y coordinate reference point". If a reference point is not used, please move the tool to the position above the tool alignment instrument manually.
- Replace the current tool with the longest tool, use the hand wheel to move the tool (and stop when the tool tip is at a certain distance away from the tool alignment instrument) while making sure the tool tip is not in contact with the tool alignment instrument. Move the cursor to the "Tool alignment starting point Z" and press the **【Z axis guide】** key for guided input.
- Replace the current tool with the shortest tool, use the hand wheel to move the tool such that the tool tip is deviated from the tool alignment instrument by a small distance (not directly above the tool alignment instrument), move the tool tip to a location at around 2 mm (not too much) lower than the normal position of the tool alignment instrument and stop right here. The cursor should be moved to the "Z axis lowest machine

coordinate" field before pressing the 【**Z axis guide**】 for guided input. (With the protective mechanism, when the tool alignment instrument has malfunctioned, the Z axis will not be moved further downward)

- Set the numerical value for the "Return to safety point Z after tool alignment" field.
- Move the tool to a proper height (higher than the tool alignment instrument) and press the 【**Start tool alignment**】 key to execute the automatic tool alignment procedure. The system will fill in the tool length compensation value into Z length offset of the tool length offset number corresponding to the tool after completing automatic tool alignment.
- Set "Gap setting for the work coordinate system": Move the cursor to the "Work coordinate system" field, enter a numerical value (from 54 to 59) in the input area and press the **<Input>** key. If the extended coordinate system G54 is being used, the "Work coordinate system" field should be set as G54 before moving the cursor to the next field (extended coordinate system), enter a numerical value (1 to 100) in the input area and pressing the **<Input>** key.
- Switch to **hand wheel mode** (MPG), move the tool to a position above the **workpiece** such that the tool tip is just touching the surface of workpiece, and then press the 【**Z gap**】 key to set the gap value into the designated coordinate system.

Note: The setting method for tool length compensation values for each tool and the gap value for each workpiece are as follows:

For example: There are two workpieces, one is based on the G55 coordinate system (workpiece A) and the other is based on the G54P100 coordinate system (workpiece B). There are two tools T1 and T2 corresponding to tool offsets H1 and H2, and it also uses the automatic tool change (M6Tx) function.

Set the tool length compensation value for tool T1:

i. Move the cursor to the "Tool number" field, enter the value 1 in the input area and press the **<Input>** key.

ii. Move the cursor to the "Is automatic tool change required" field, enter the value 1 in the input area and press the **<Input>** key.

iii. Move the tool to a proper height (higher than the tool alignment instrument) and press the 【**Start tool alignment**】 key to execute the automatic tool alignment procedure. The system will conduct the automatic tool alignment procedure after automatically changing tools at T1. The system will then fill in the tool length compensation value into the Z length offset of tool length offset number 1 corresponding to the tool after completing automatic tool alignment.

Set the tool length compensation value for tool T2:

i. Move the cursor to the "Tool number" field, enter the value 2 in the input section and press the **<Input>** key.

ii. Move the cursor to the "Is automatic tool change required" field, enter the value 1 in the input area and press the **<Input>** key.

iii. Move the tool to a proper height (higher than the tool alignment instrument) and press the 【Start tool alignment】 key to execute the automatic tool alignment procedure. The system will conduct the automatic tool alignment procedure after automatically changing tools at T2. The system will then fill in the tool length compensation value into the Z length offset of tool length offset number 2 corresponding

to the tool after completing automatic tool alignment.

Set the gap value for workpiece A:

i. Move the cursor to the "Work coordinate system" field, enter the value 55 in the input area and press the **<Input>** key.

ii. Switch to **hand wheel mode** (MPG), move the tool to the position above **workpiece A** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the G55 coordinate system.

Set the gap value for workpiece B:

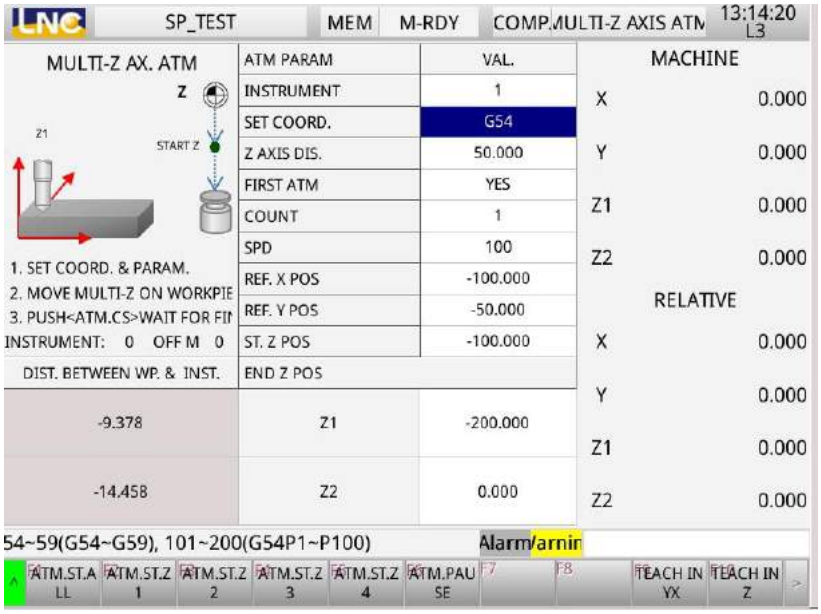
i. Move the cursor to the "Work coordinate system" field, enter the value 54 in the input area and press the **<Input>** key.

ii. Move the cursor to the next field (extended coordinate system), enter the value 100 in the input area and press the **<Input>** key.

iii. Switch to **hand wheel mode** (MPG), move the tool to the position above **workpiece B** such that the tool tip is just touching the surface of workpiece, and then press the **【Z gap】** key to set the gap value into the G54P100 coordinate system.

1.8.5.2 Glass edge milling machine, applicable industries

The tool length of automatic tool alignment should be saved into the Z axis coordinate of the designated work coordinate system. The operation is described below:



- Setting up the "set coordinate system": Move the cursor to the "Set coordinate system" field, enter a numerical value (from 54 to 59) in the input area and press the **<Input>** key. If the extended coordinate system G54 is required, a numerical value (from 101 to 200) should be entered in the input area before pressing the **<Input>** key.

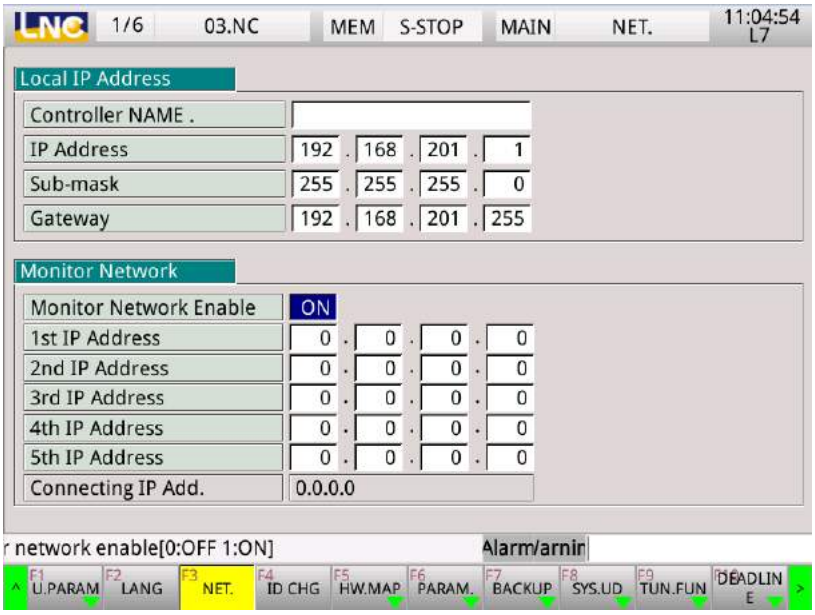
- Setting the "space between multiple spindles": If there are more than two spindles, this field should be set up in accordance with the actual distance between spindles in the machine.
- Setting up the "first tool alignment": If it is the first tool alignment, the cursor should be moved to the "First tool alignment" field, and a numerical value should be entered in the input area before pressing the **<Input>** key.
- Setting the "number of tool alignment": It is for setting the number of times tool alignment needs to be executed. Move the cursor to the "Number of tool alignment" field, enter a numerical value in the input area and press the **<Input>** key. (0 to 10)
- Setting "tool alignment speed": Move the cursor to the "Tool alignment speed" field, enter a numerical value in the input field and press the **<Input>** key.
- Switch to **hand wheel mode** (MPG) and move the tool to a position above the tool alignment instrument. After confirming that the tool is right above the tool alignment instrument, the cursor should be moved to the "X axis machine coordinate of the tool alignment instrument" field, press the **【XY axis guide】** key to set up both "X axis machine coordinate of the tool alignment instrument" and "Y axis machine coordinate of the tool alignment instrument".
- Use the hand wheel to move the tool (and stop when the tool tip is at a certain distance away from the tool alignment instrument) while making sure the tool tip is not in contact with the tool alignment instrument. Move the cursor to the "Tool alignment starting point machine coordinate Z" and press the **【Z axis guide】** key for guided input.
- Use the hand wheel to move the tool such that the tool tip is deviated from the tool alignment instrument by a small distance (not directly above the tool alignment instrument), move the tool tip to a lower position by around 2 mm (not too much) than the normal position of the tool alignment instrument and then stop right here. Move the cursor to the "machine coordinate of Z's lowest tool alignment point" field then press the **【Z axis guide】** for guided input. (With the protective mechanism, when the tool alignment instrument has malfunctioned, the Z axis will not be moved further downward)
- Move the tool to a proper height (higher than the tool alignment instrument) and press the **【Start tool alignment】** key to execute the automatic tool alignment procedure. The system will fill in the designated coordinate system with the tool length compensation value after completing automatic tool alignment.

1.8.6 Network settings and connections

- After pressing the **<MAINTE>** key on the controller's MDI panel, the 【**Network settings**】 key on the controller's human machine page can be pressed to enter the network settings page as shown in the figure below.

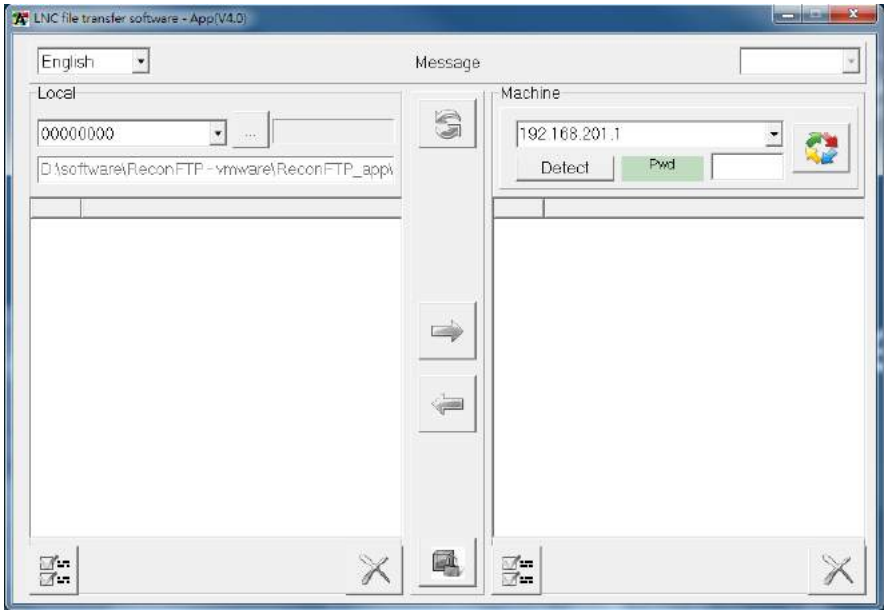
- After pressing and moving the cursor to the corresponding field, and entering the IP address in the input area, the **<Input>** key should be pressed to show the entered numbers at the cursor's position.

- After the IP address setting of this machine is completed, the cursor can be moved to online monitoring to select whether to start the online monitoring function, and its setting code will be displayed in the message prompt area. Enter 1 in this input field and press the **<Input>** key to set it as "Open".



- Open ReconFTP_user as shown in the figure below.

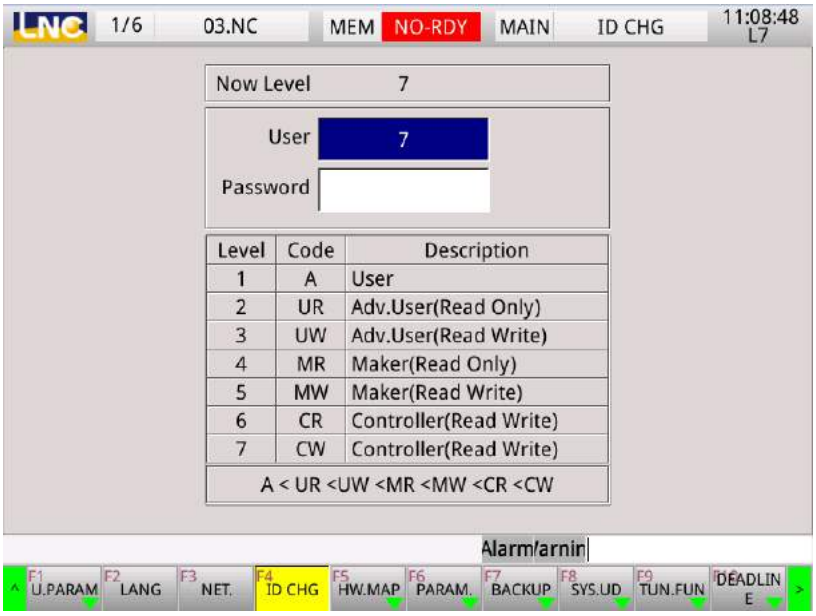
- Enter the IP address and password before pressing to connect as shown in the figure below.

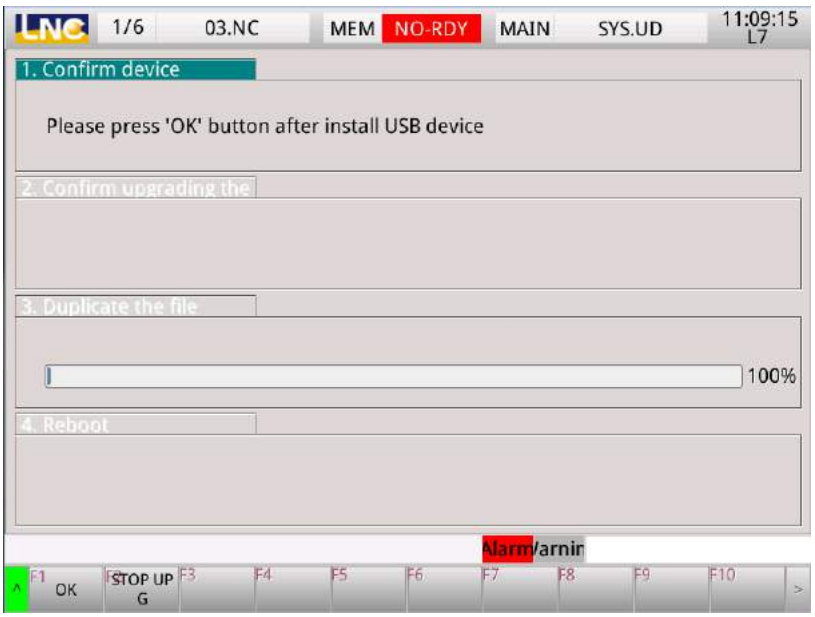


- If the connection needs to be interrupted, press again.

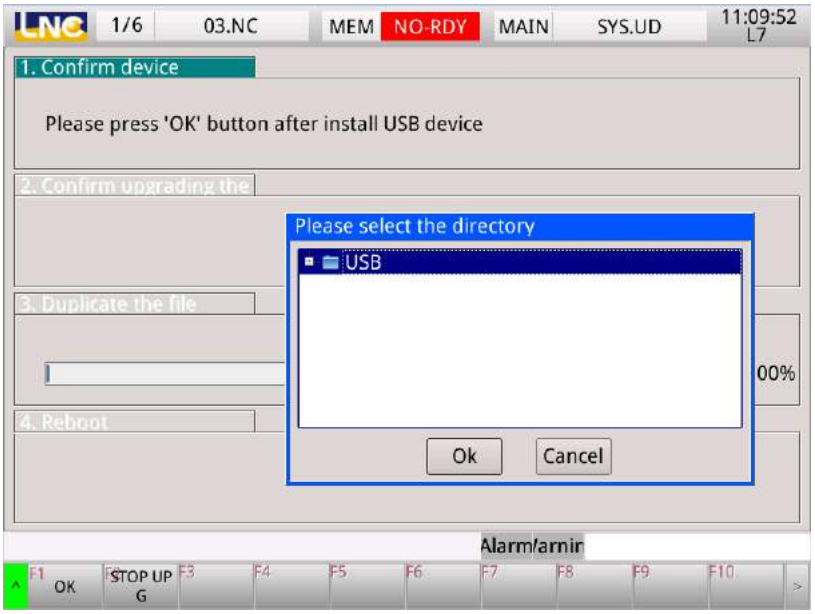
1.8.7 System update

- Due to safety concerns, the emergency stop 〔EMG-STOP〕 button on the controller's OP panel must be pressed for the machine to be in "Not ready" state before system update can be executed.
- After pressing the **<MAINTE>** key on the controller's MDI panel, the **【System update】** key on the controller's human machine page can be pressed to enter the system update page.

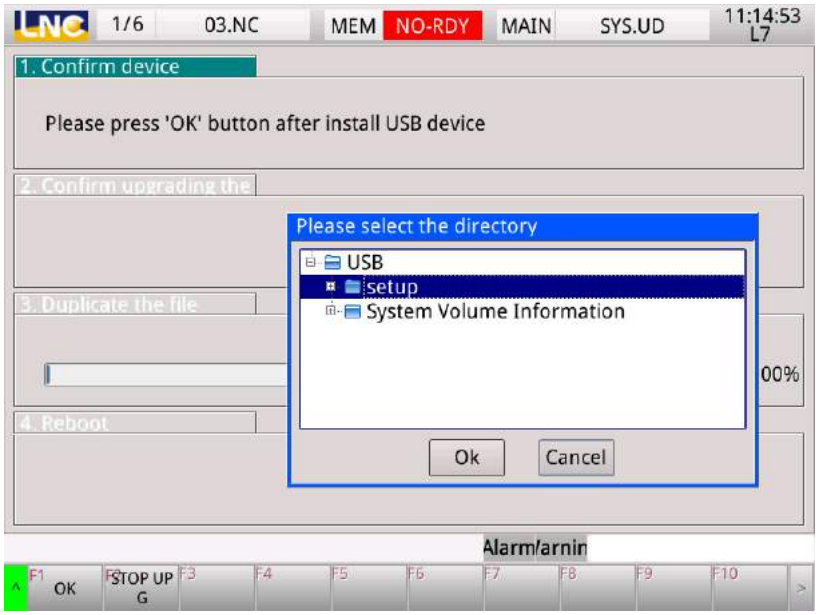




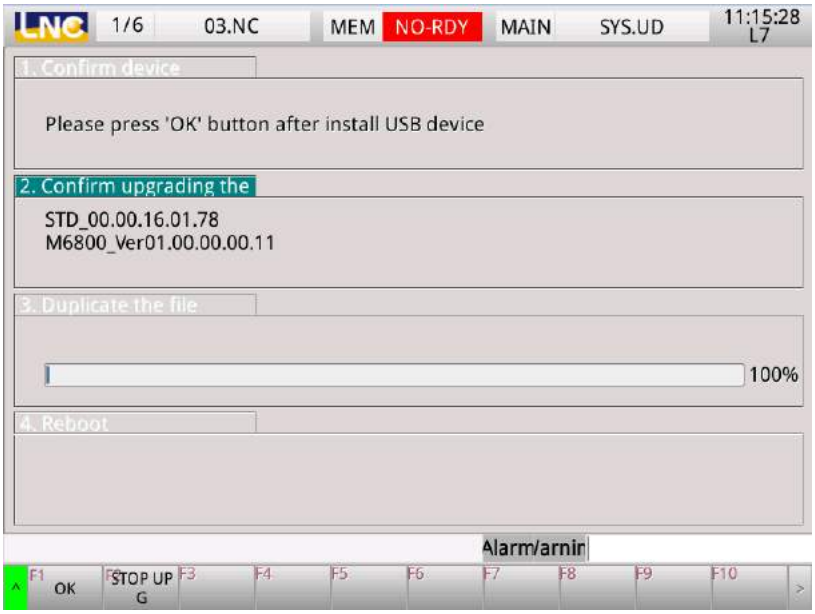
- After inserting a USB device with update files, the 【**Confirm**】 key on the controller's human machine page can be pressed for the directory selection window to appear.



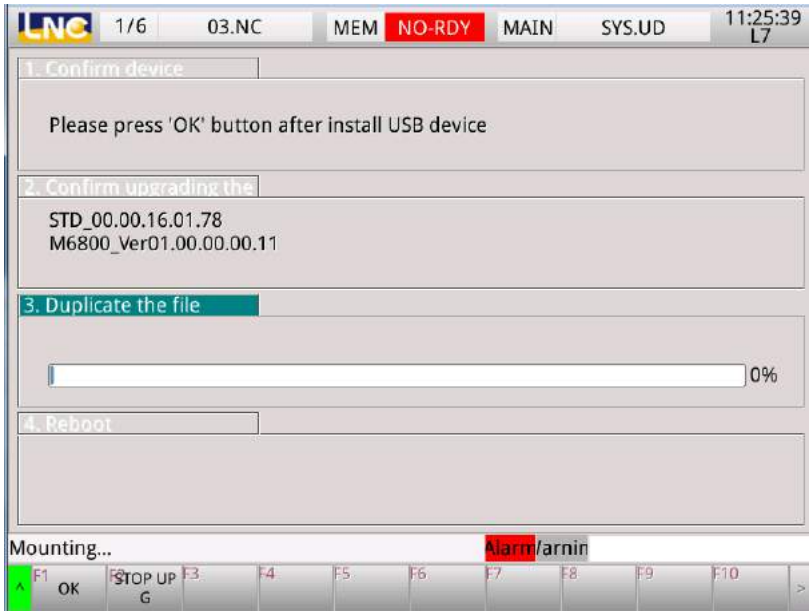
- The ▶ key on the MDI panel can be used to open a subdirectory.



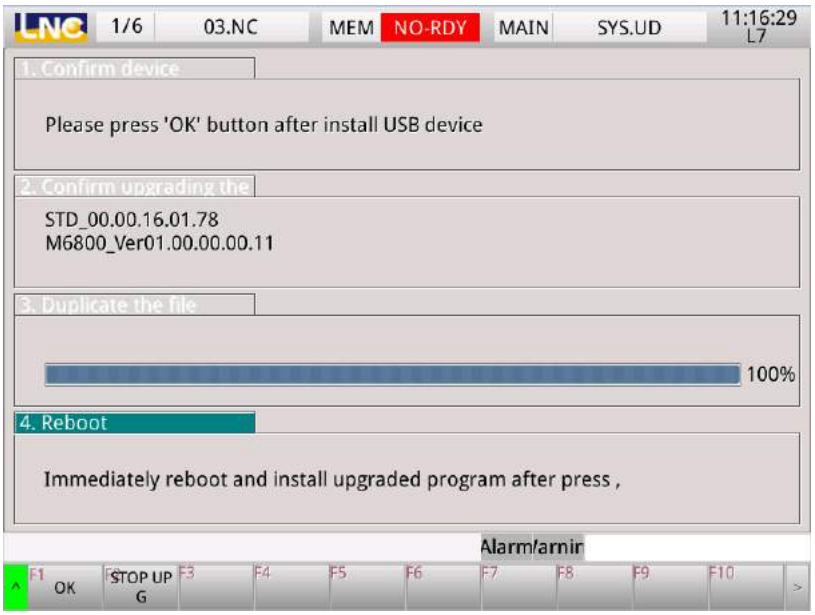
- The ▲ ▼ keys can be used to select the folder for saving the update file, and the **<Input>** key can be pressed before selecting **Confirm** to enter the "Confirm upgrade version" stage.



- The 【**Confirm**】 key can be pressed to enter the "Copy file" stage (as shown in the figure below).



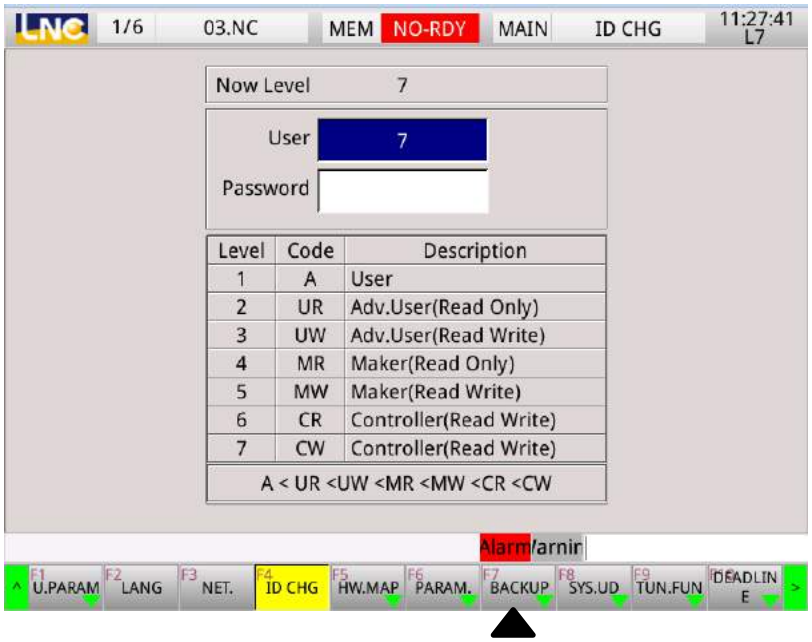
- After the file copy operation is 100% complete, it will enter the "Reboot" stage (as shown in the figure below).

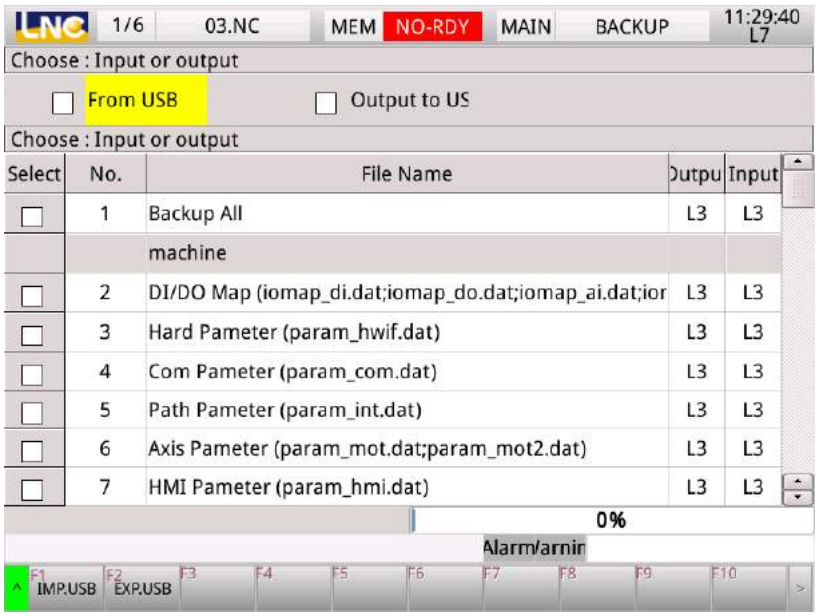


- After pressing the 【**Confirm**】 key, wait for the controller to reboot automatically and complete the system update process.

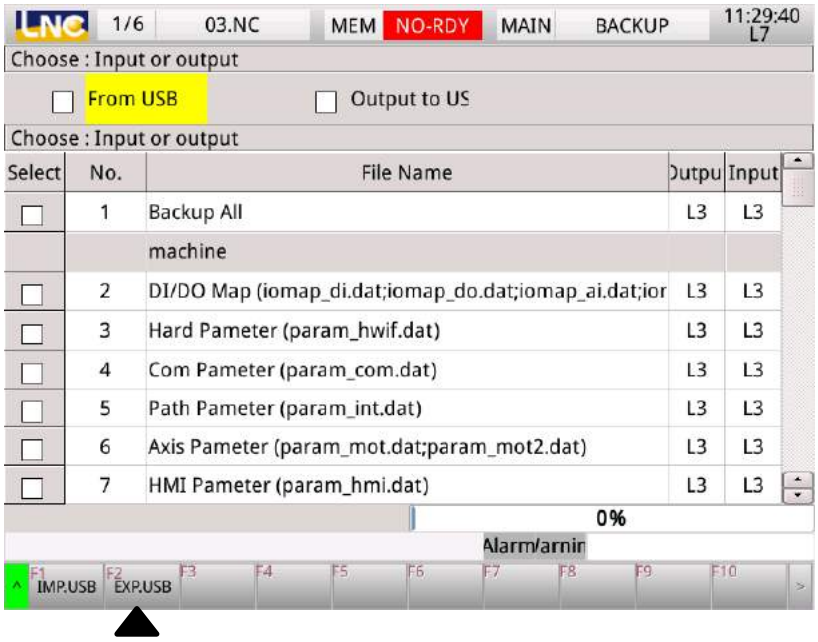
1.8.8 System data backup - export

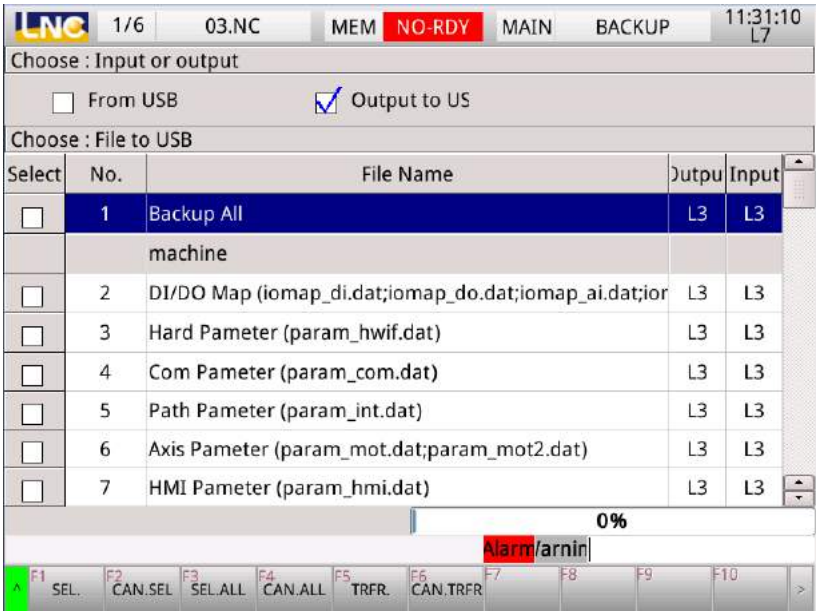
- This system data backup function can only be accessed by users with 〔Advanced user〕 permission or above, which is L3.
- Due to safety concerns, the emergency stop 〔EMG-STOP〕 button on the controller's OP panel must be pressed for the machine to be in "Not ready" status before the backup function can be executed.
- After inserting a USB device and pressing the **<MAINTE>** key on the controller's MDI panel, the **【Backup】** key on the controller's human machine page can be pressed to enter the system data backup page.



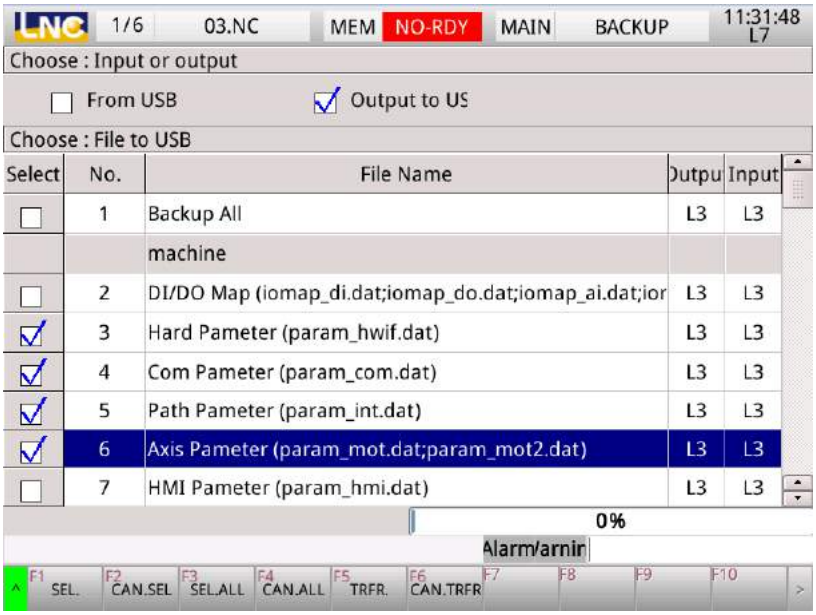


- Press the 【**Export to USB**】 key as shown in the figure below.

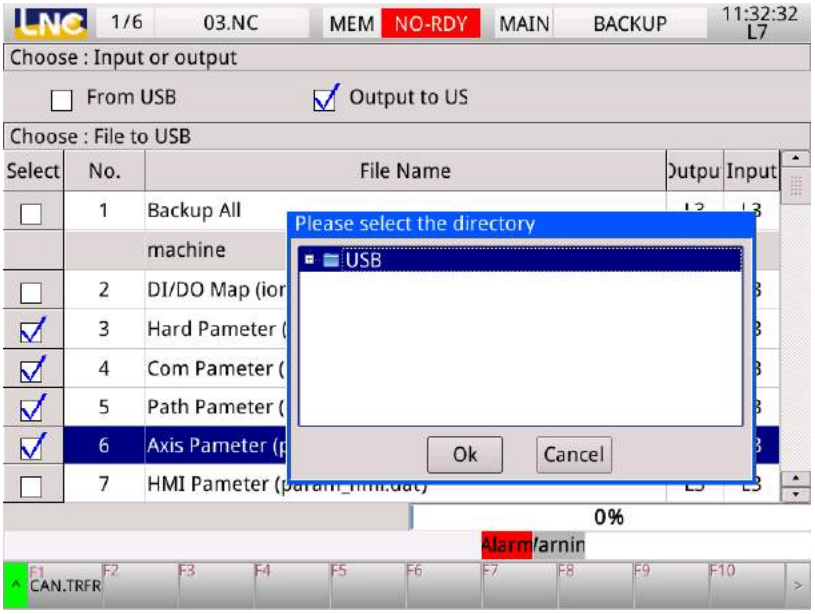




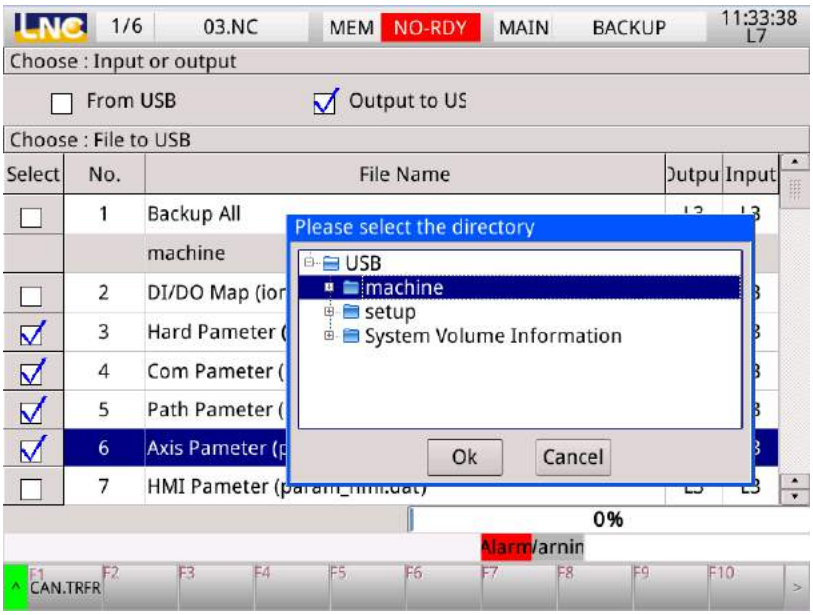
- Move the cursor to select the file to be exported and press the【**Select**】key on the controller's human machine page; the selected file will be checked (as shown in the figure below). Or the 【**Cancel select**】 key can be pressed to cancel the selection. In addition, the 【**Select all**】 key on the controller's human machine page can be pressed to directly backup all files.

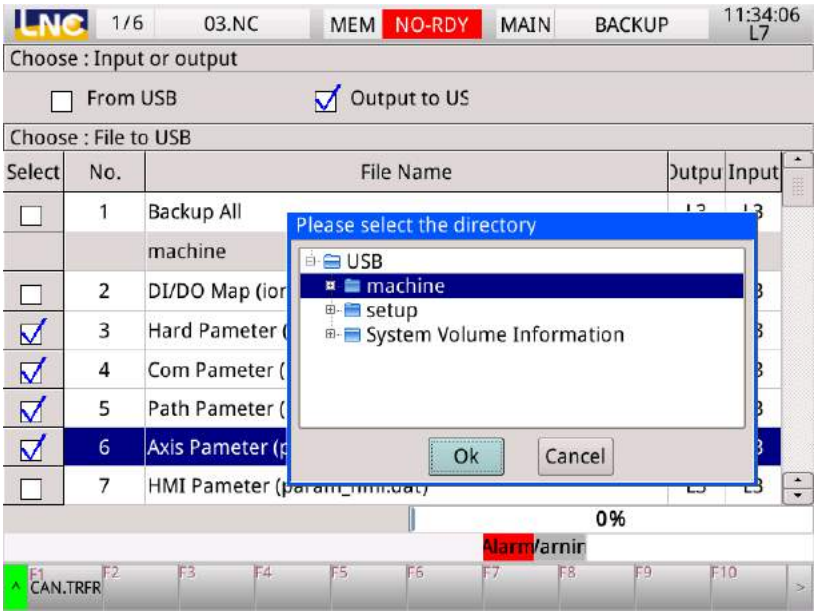


- After pressing the 【**Transfer**】 key on the controller's human machine page, a window for directory selection will appear (as shown in the figure below).



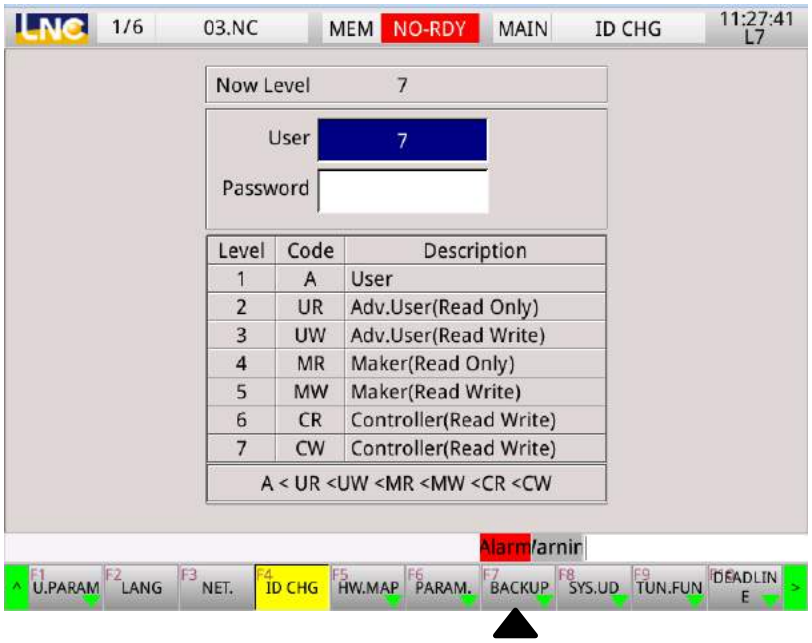
- Use the ▶ key to open a subdirectory, and then use ▲ ▼ keys to select the folder to be exported. Press the **<Input>** key and select **Confirm** to start the transfer.

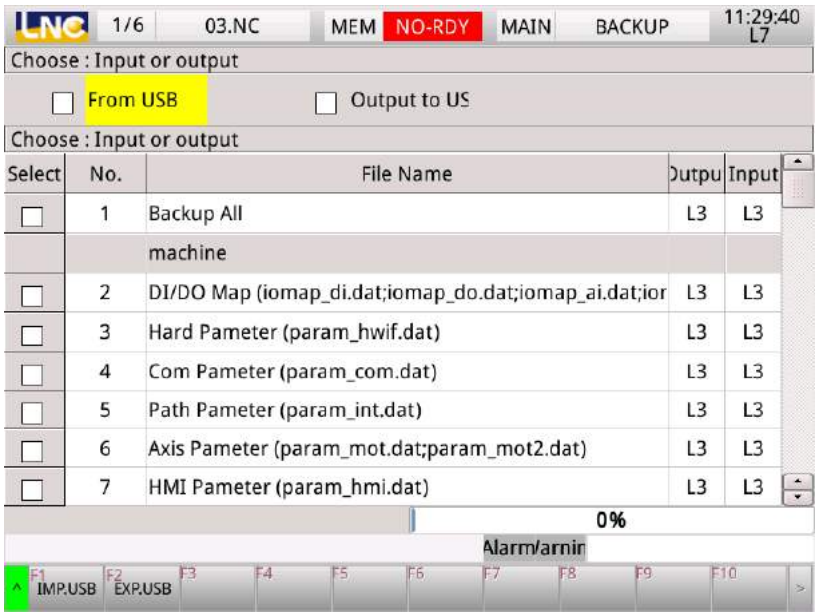




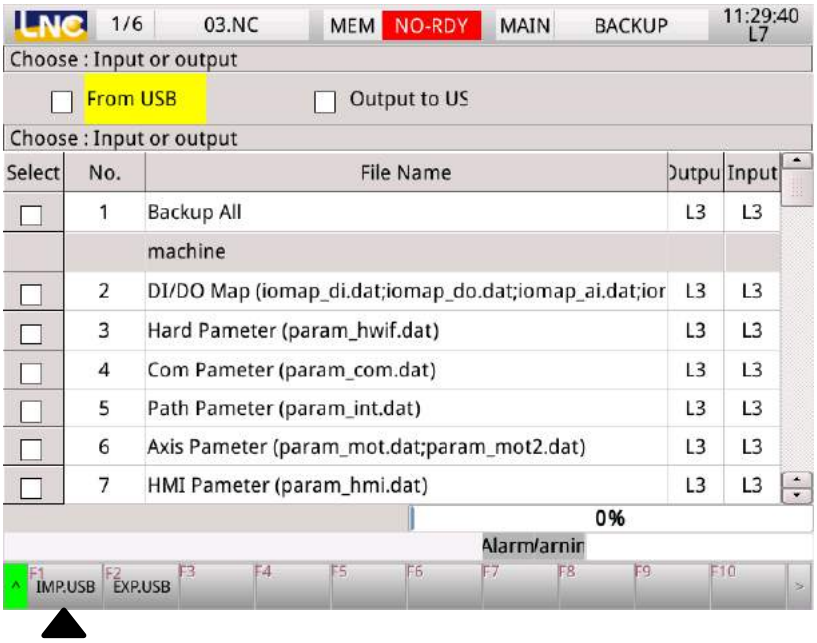
1.8.9 System data backup - import

- This system data backup function can only be accessed by users with 〔Advanced user〕 permission or above, which is L3.
- Due to safety concerns, the emergency stop 〔EMG-STOP〕 button on the controller's OP panel must be pressed for the machine to be in "Not ready" status before the backup function can be executed.
- After pressing the **<MAINTE>** key on the controller's MDI panel, the **【Backup】** key on the controller's human machine page can be pressed to enter the backup page.

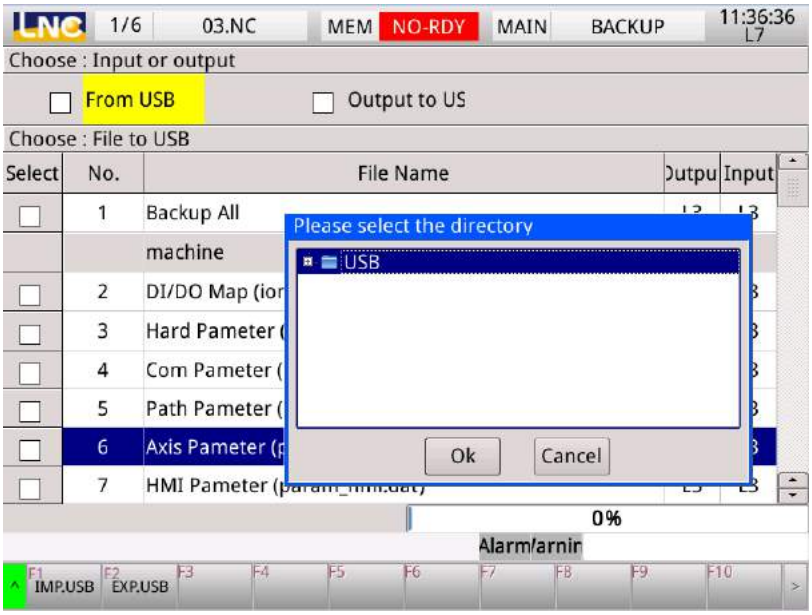




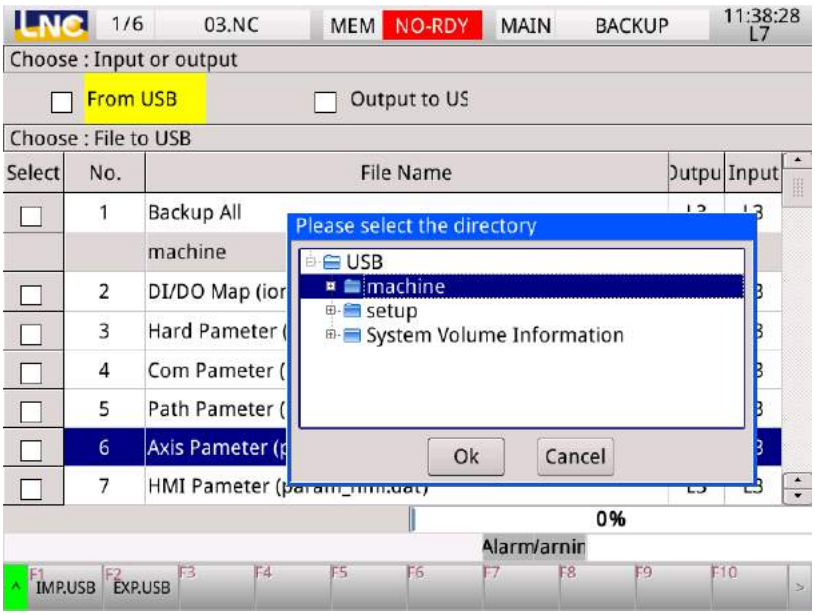
- Press the 【**Import from USB**】 key as shown in the figure below.

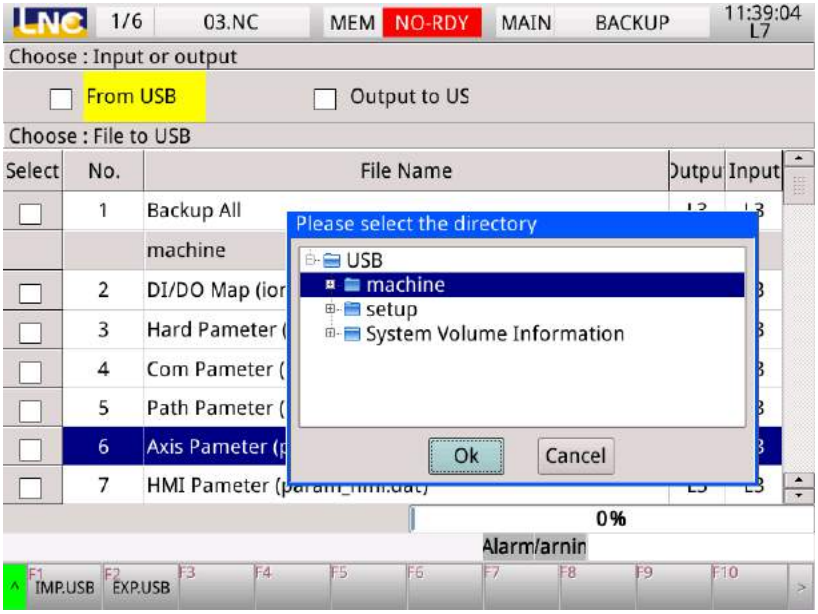


- At this moment, a window for directory selection will appear.

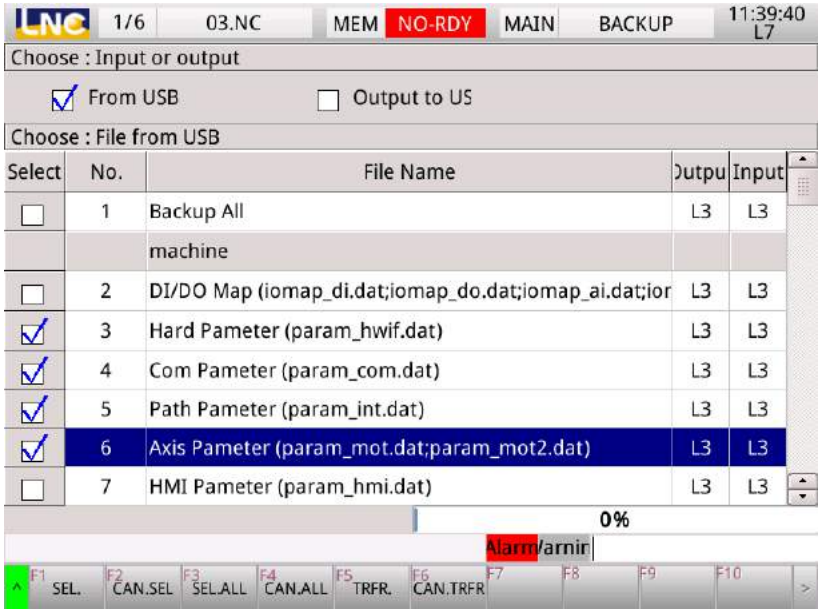


- Use the ▶ key to open a subdirectory, and then use ▲ ▼ keys to select a folder for saving the imported file. Press the **<Input>** key and select **Confirm**.

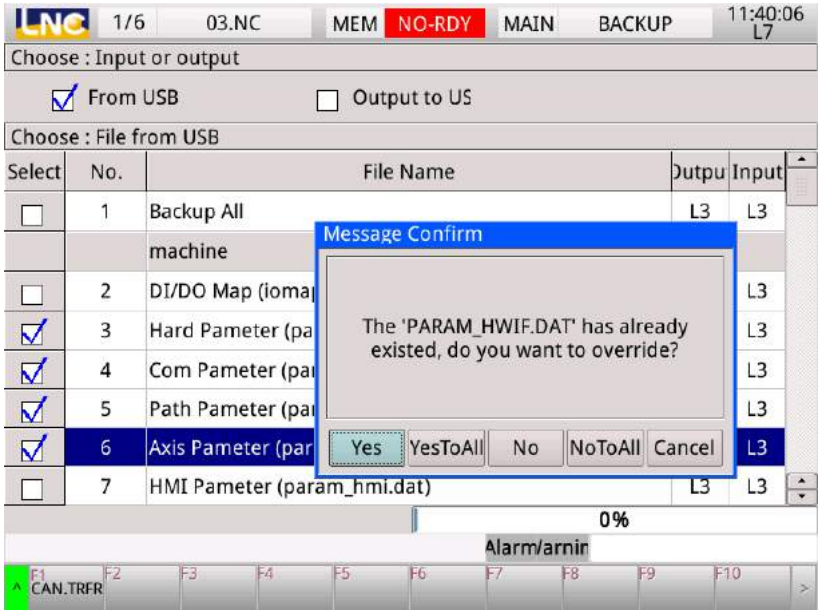




- Use ▲ ▼ keys to move the cursor to select the file to be imported, press the 【**Select**】 key and the selected file will be checked (as shown in the figure below). Or the【**Cancel select**】key can be pressed to cancel the selection. In addition, the 【**Select all**】 key on the controller's human machine page can be pressed to directly import all files.

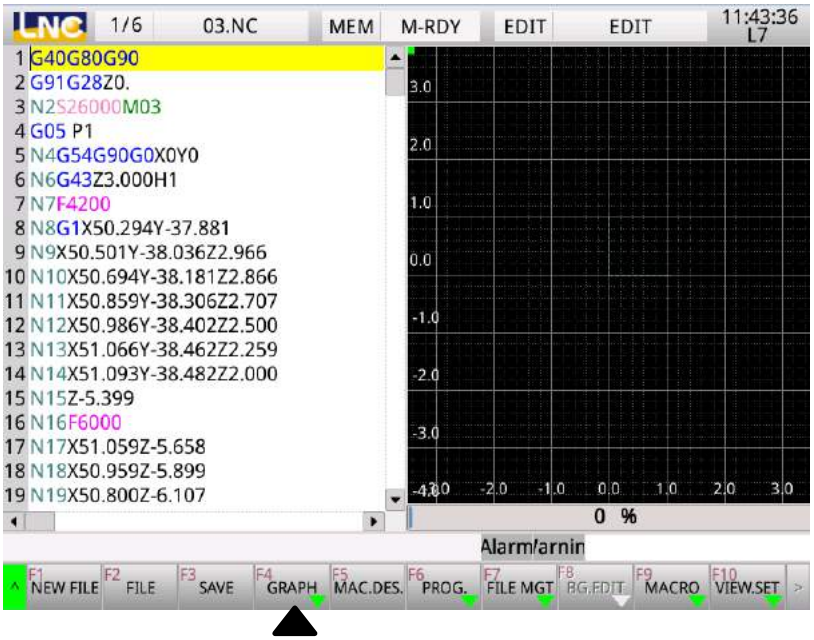


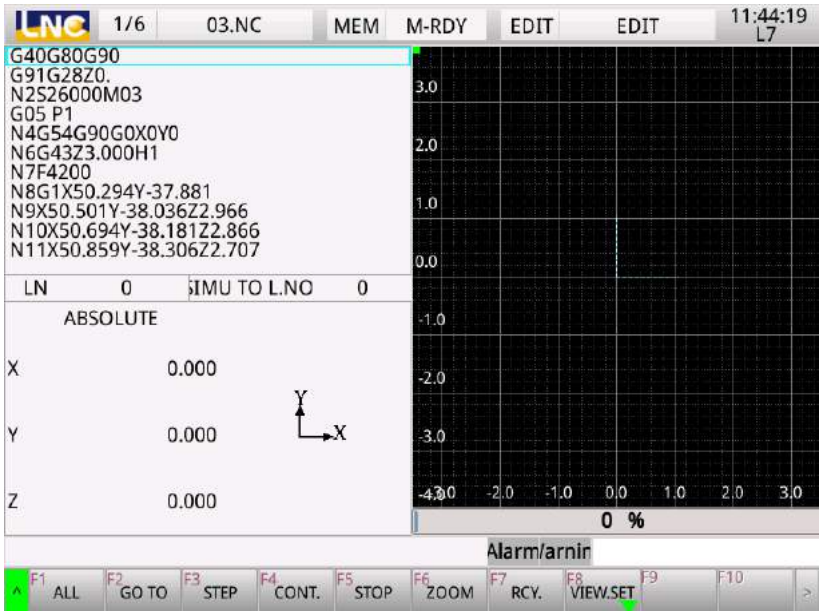
- If a file already exists in the controller when the 【**Transfer**】 key is pressed, a confirmation message will appear (as shown in the figure below), and users can select **Yes** or **Yes to all** to overwrite the existing file. If the wrong folder was selected, a "file type error" will be shown in the message prompt area.



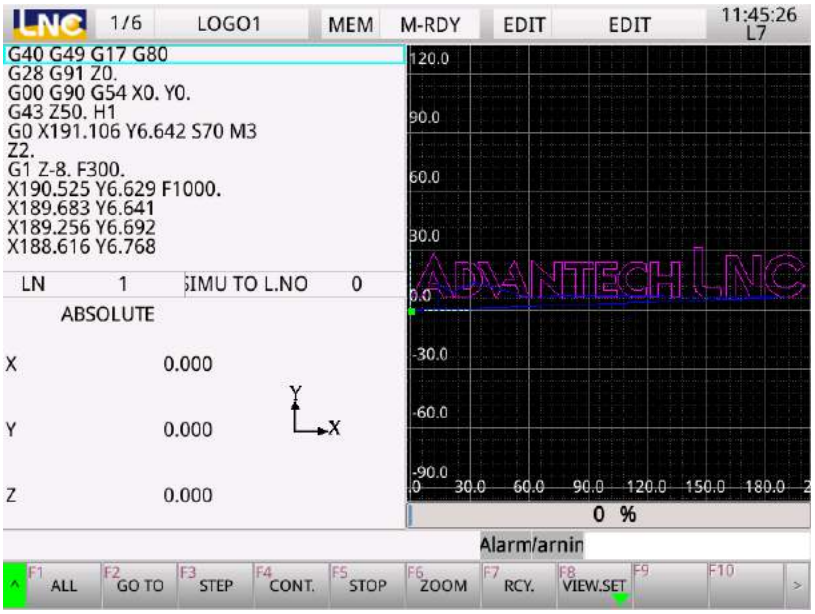
1.8.10 Preview function

- While under the ready status, the **<PROG>** key on the MDI panel can be pressed to switch the controller's human machine page to the program editing page, and the 【**Preview**】 key on the human machine page can be pressed to enter the preview page as shown in the figure below.





- The 【**Preview all**】 key can be pressed to preview the path (or press the 【**Preview to**】 key to preview the program block at the cursor's current position) as shown in the figure below.

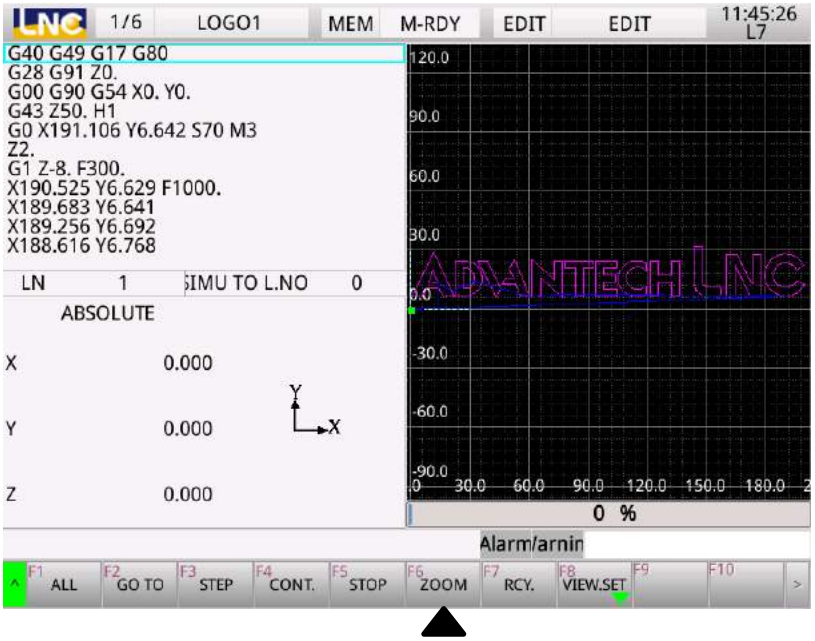


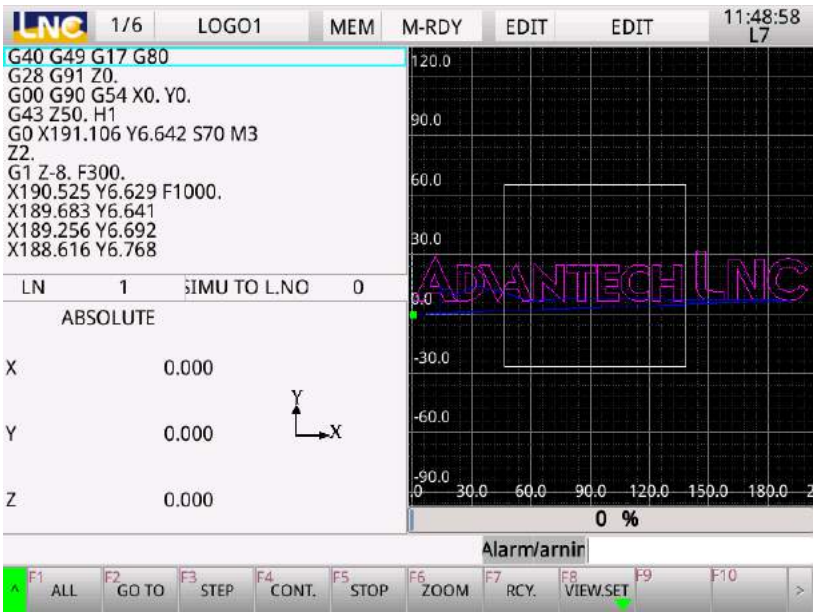
- To start a new preview, the 【**Stop**】 key can be pressed to clear the preview path.

- There are two preview approaches: "single step" and "continuous".
  1. By pressing the 【**Single step**】 key on the human machine page, the program path of that program block will be drawn, and the program path of the next block will be drawn by pressing the 【**Single step**】 key again. Program paths can be drawn by repeating this process as shown in the figure below. If the 【**Stop**】 key is pressed during the preview process, all paths drawn are cleared.

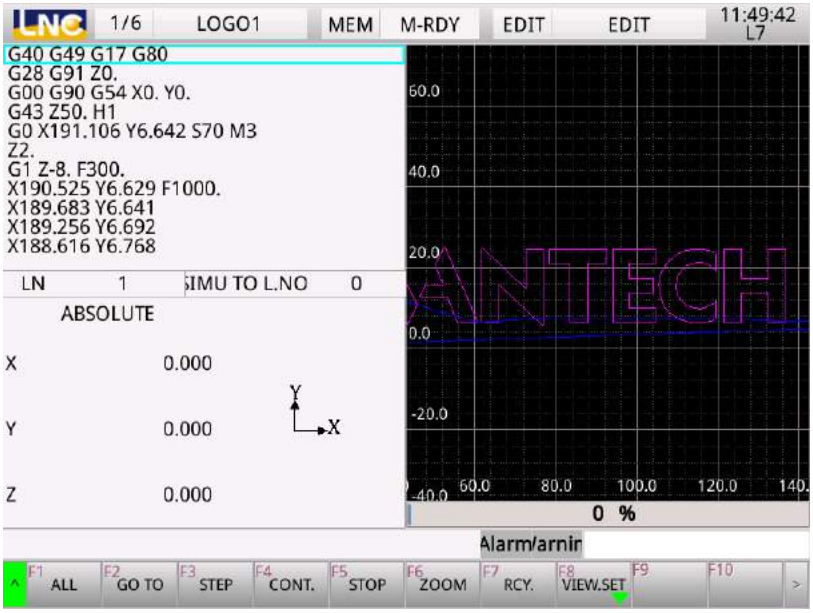
  1.
  2. If the 【**Continuous**】 key is pressed, the program path will be drawn continuously along with the program block. If the 【**Stop**】 key on the human machine page is pressed during preview, all paths drawn will be cleared.

- For magnifying a certain area of the path, the 【**Zoom**】 key on the human machine page can be pressed for a white frame to appear. Arrow keys on the MDI panel can be used to move this frame, and **<PgUp>** and **<PgDn>** keys on the MDI panel can be used to determine the size of the area you wish to select.

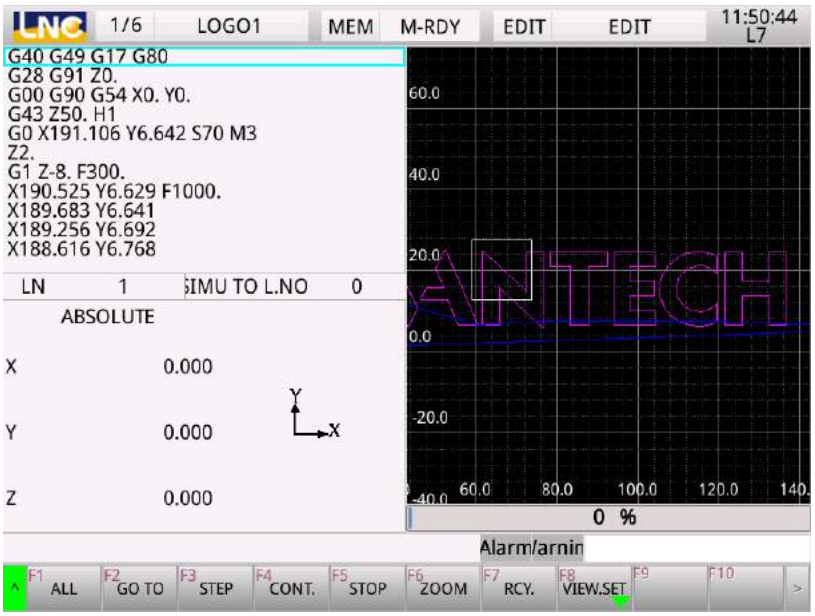


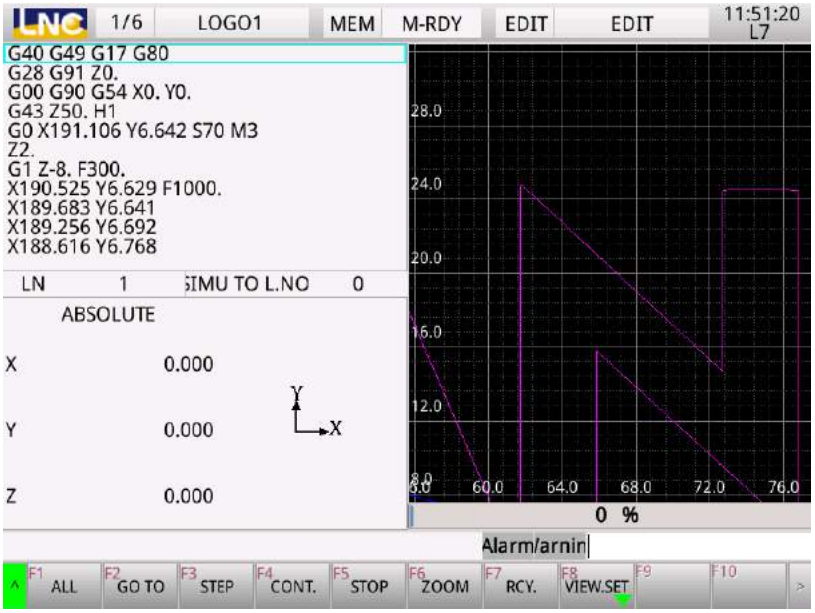


- After selecting the area to be magnified, press the 【**Zoom**】 key again and the selected area enclosed inside the white frame will be magnified as shown in the figure below. The 【**Restore**】 key on human machine page can be pressed to cancel magnification.

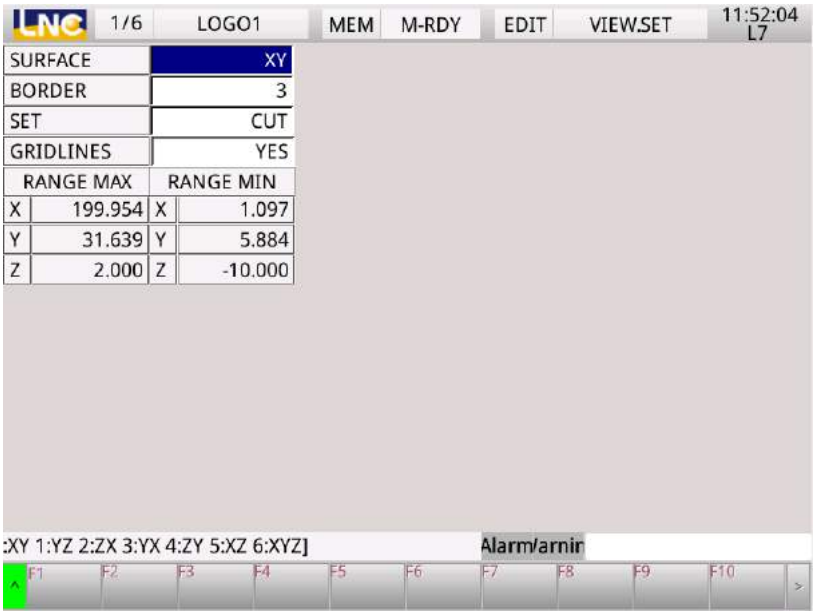


- By pressing the 【**Zoom**】 key again on the human machine page, a certain area of the currently magnified path drawing can be selected to be magnified once again as shown in the figure below.





- The 【**Plot settings**】 key on the human machine page can be pressed to enter the screen as shown in the following figure.



- If the coordinate viewing angle used for displaying the path screen is to be changed, arrow keys on the MDI panel can be used to move the cursor to the "Drawing plane" field (as shown in the figure above); a value should be entered in the input area before pressing the **<Input>** key to complete the setting. (0: XY, 1: YZ, 2: ZX, 3: YX, 4: ZY, 5: XZ, 6: XYZ)

- Reserved boundary values of the path screen display can be set up by using arrow keys on the controller's MDI panel to move the cursor to "Reserved boundaries".
- If the drawing range of the path screen display is to be changed, the cursor can be moved to the "Setting method" field (as shown in the figure below); a set value should be entered in the input area to complete the setting.

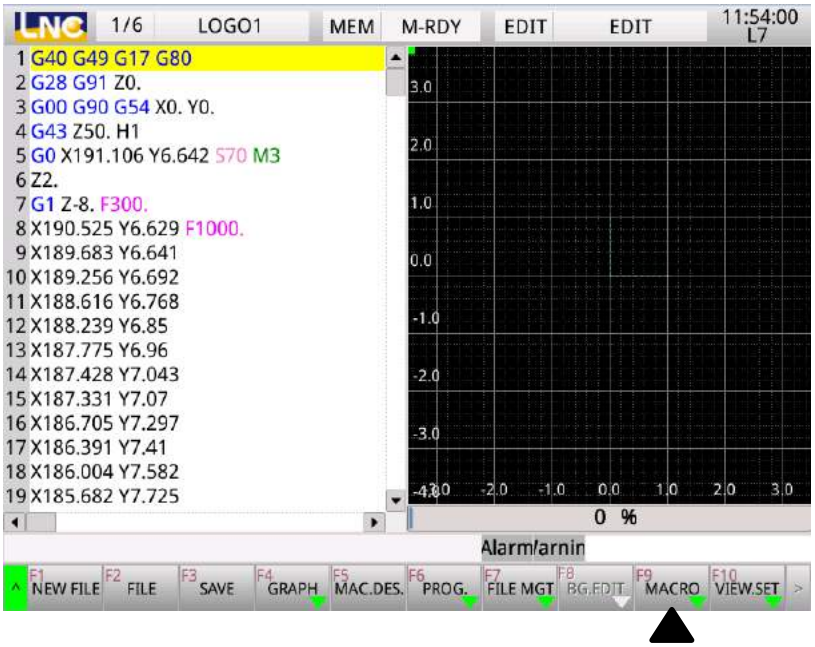
  0: Manual. The range of a drawing preview is defined by the maximum and minimum values from manual setting.

  1: Preview result - full stroke (including movement path). The range of a drawing preview is defined by the maximum and minimum values of the program's machining path.

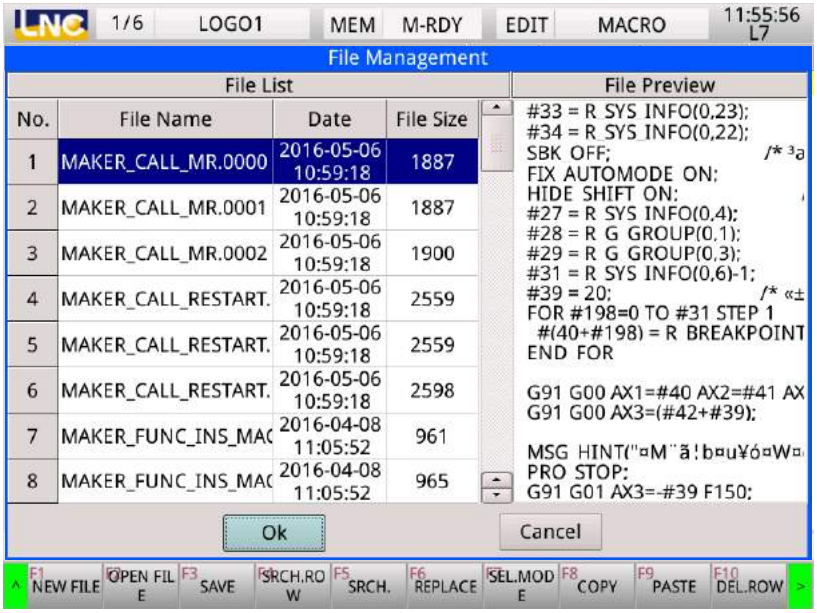
  2: Preview result - cutting stroke (only the cutting path is included). The range of a drawing preview is defined by the maximum and minimum values of the program's cutting path.
- When the "Setting method" field is set to 0, the cursor can be moved to the "Maximum value in range" and "Minimum value in range" fields, and the maximum and minimum values of the of X, Y, and Z axis ranges can also be set up.
- If grid lines are required for viewing the figure, it can be set up in the "Figure grid lines" field. (0: No, 1: Yes)

1.8.11 Editing and using manufacturer macros

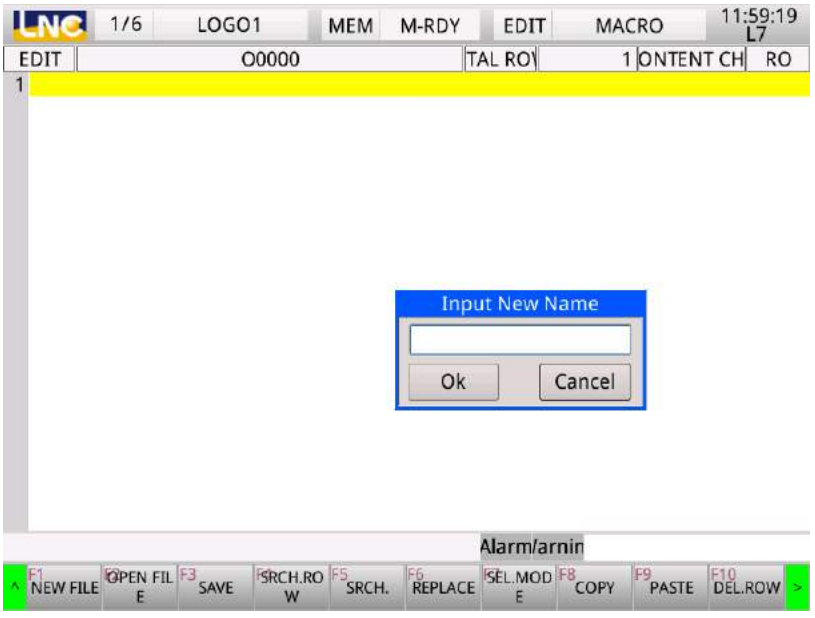
- This operation can only be executed by users with the permission level of { Machine factory (read and write) } or above, which is L5.
- After pressing the **<PROG>** key on the controller's MDI panel to switch the controller's human machine page to the program editing page, press the **【Manufacturer macro】** key to enter the manufacturer macro page.



- Press the 【**Program edit**】 key and the 【**Open file**】 key to enter the file list page as shown in the figure below.



- Press the 【**Open new file**】 key for a dialog window requesting a new filename to appear (please refer to the file naming regulations in the table below). After a new filename has been entered, select **Confirm** to enter the editing screen. By selecting **Cancel**, the dialog window for entering a new file name will be closed.

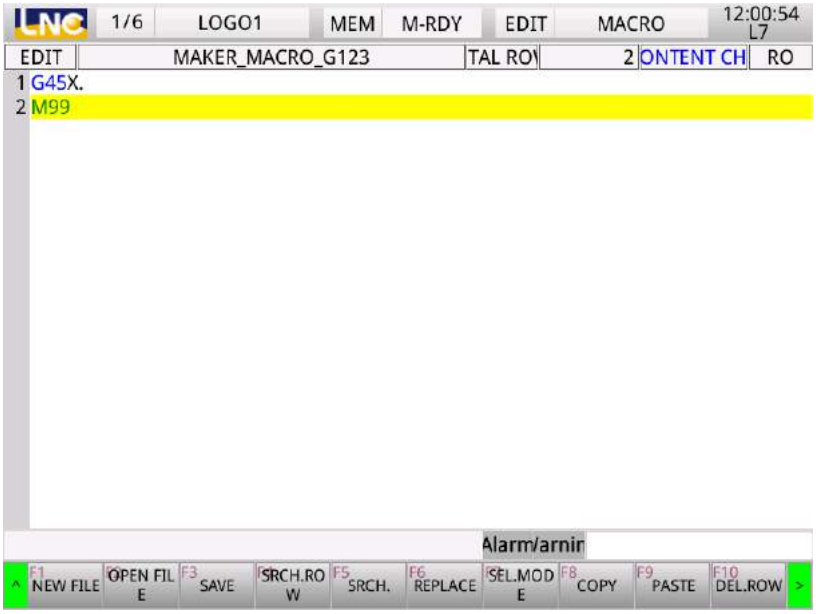


Naming regulations

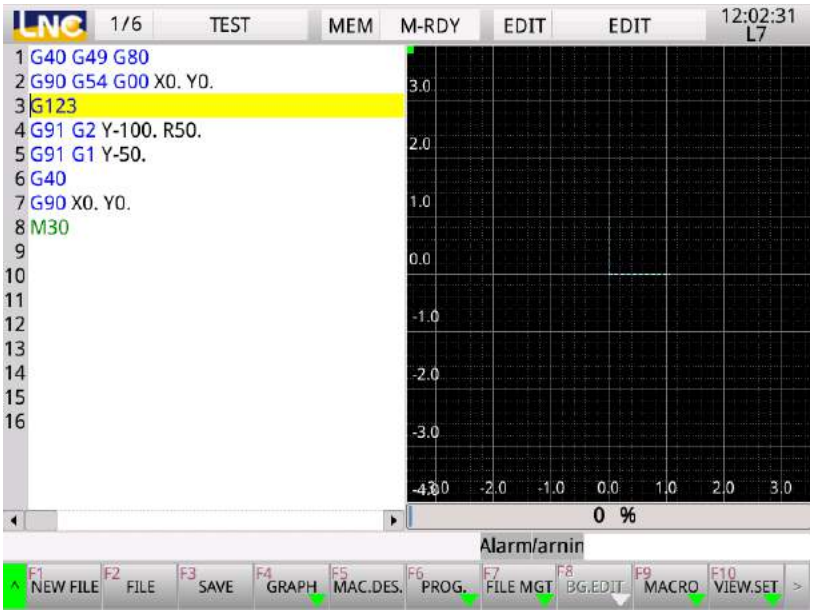
Type of filename	Examples of filename types	Description
maker_macro_g□□	maker_macro_g35	G code calls a manufacturer macro
maker_macro_m□□	maker_macro_m35	M code calls a manufacturer macro

maker_macro_t0	maker_macro_t0	All T codes calling this maker macro

- The editing method for here is identical to the program editing method for machining programs (please refer to **Opening and editing a file**). Please pay attention to the special regulations for macro programming, where "M99" must be added to the end of the program as shown in the figure below.

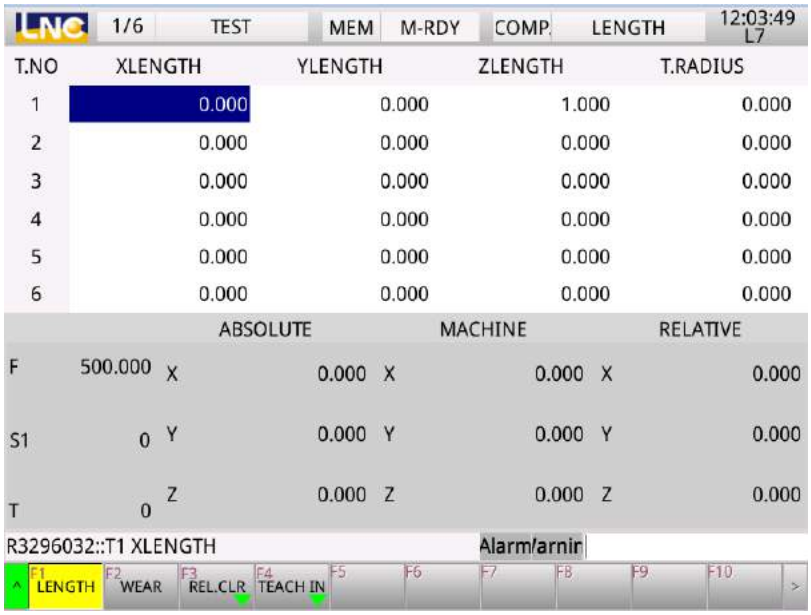


- If the name of a new macro file is MAKER_MACRO_G123, it can be called by a program command containing the command G123. When the program is executed to G123, contents of the MAKER_MACRO_G123 program will be executed.

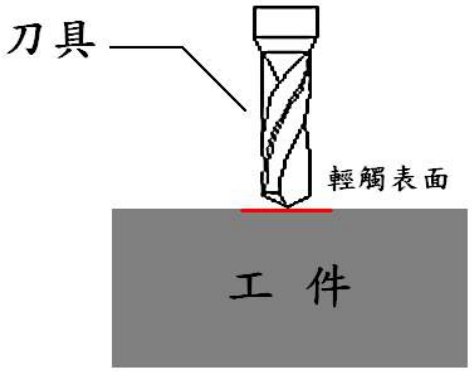


1.8.12 Tool offset settings

- Press the **<OFFSET>** key and the【**Tool management**】key on the human machine page, then press the【**Length**】key to access this function.

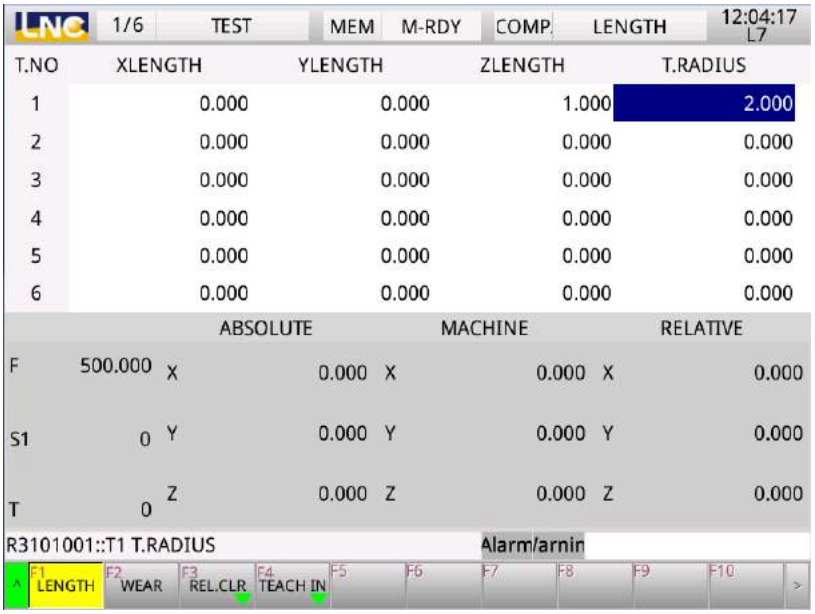


- Assume the tool is moving along the Z axis. The tool should be manually (by hand wheel or jog) positioned at the surface of the workpiece (as shown in the figure below).

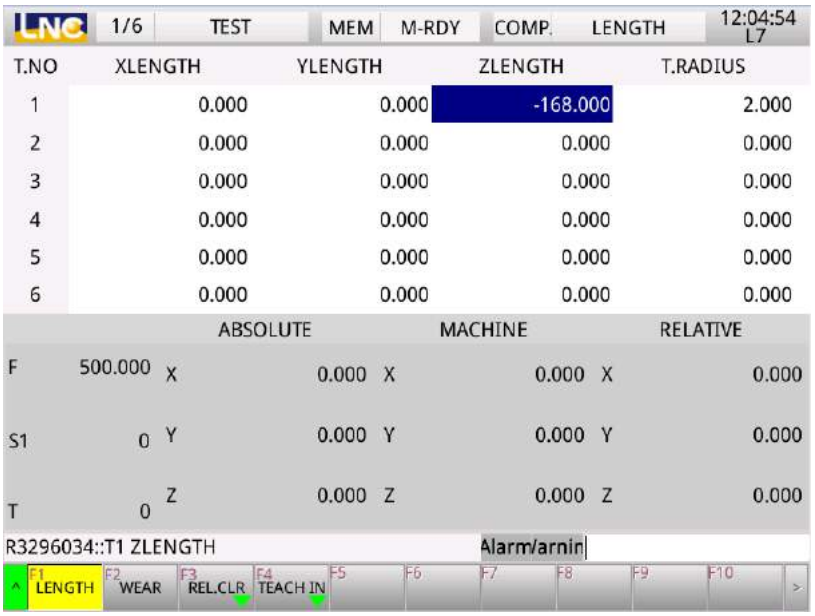


- Use arrow keys on the MDI panel to select the tool number requiring an offset, and enter the tool radius in the input area followed by pressing the **<Input>** key to enter the compensation value to the radius field of that tool number.

The input value's unit can be determined by its decimal point. For example: Input 2**.**=2(MM); input 2=2(LU)=0.002(MM).

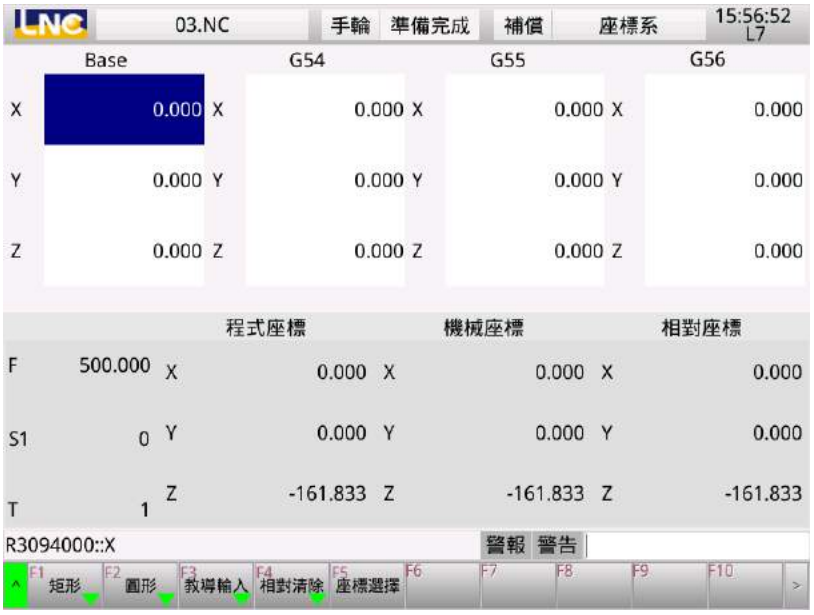


- Use arrow keys on the MDI panel to move the cursor to the tool number requiring an offset, press the **【Guided input】** key and then press the **【Guided Z】** to set the machine coordinate of the Z axis into Z's length field of that tool number to complete the tool offset setting. (If the Z axis field of the coordinate system has been set, the value must be subtracted during input to avoid repeating the offset)

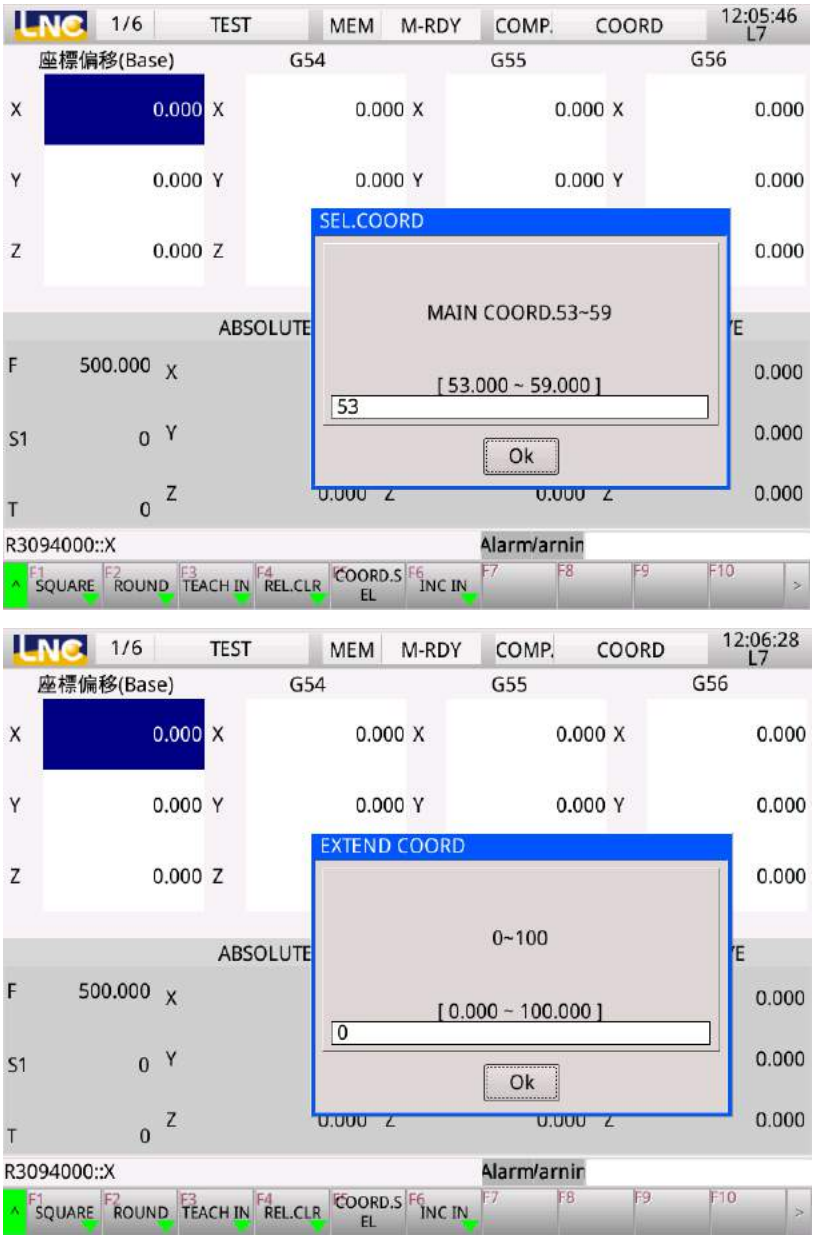


1.8.13 Coordinate system settings

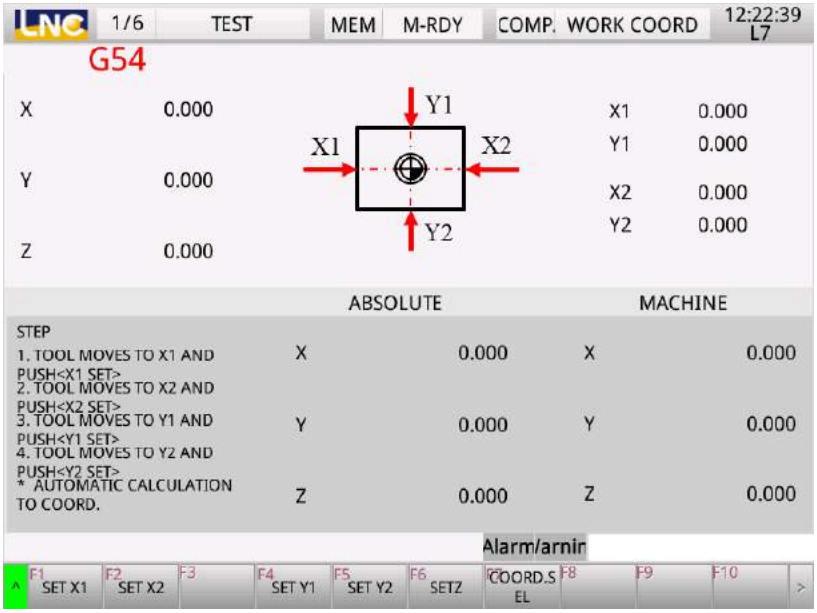
- Press the **<OFFSET>** key on the controller's MDI panel, followed by pressing the 【**Coordinate system**】 key on the human machine page to access this page (as shown in the figure below).



- After entering the coordinate system settings page, the 【**Coordinate selection**】 key can be pressed to select the coordinate system to be set up. After pressing the 【**Coordinate selection**】 key, a "Select coordinate system" input window will appear. After the code of the main coordinate system (53 to 59) is entered, select **Confirm** to move the cursor to the selected coordinate system; since coordinate system G54 is equipped with an extended coordinate system, if 54 is entered into the "Select coordinate system" field and **Confirm** is selected, an "Extended coordinate system" input window will also appear. If users simply want to select coordinate system G54, the value of 0 should be entered into the "Extended coordinate system" field before selecting **Confirm**. If another extended coordinate system is to be selected, its corresponding number should be entered into the "Extended coordinate system" field before selecting **Confirm**.

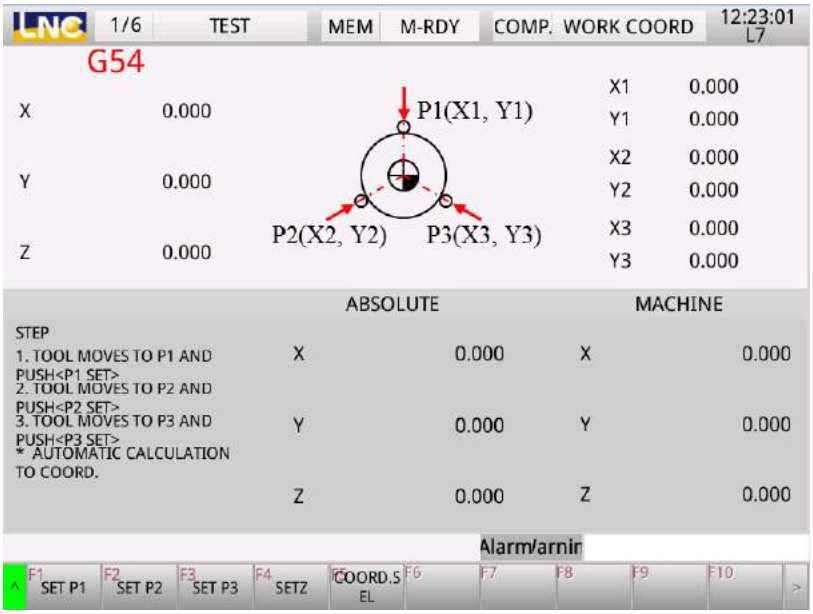


- Take G54 as an example.
- If the workpiece is rectangular in shape and the program's reference point is the center point of the workpiece, the 【**Rectangle**】 key on the human machine page can be pressed to enter the rectangle center settings page.

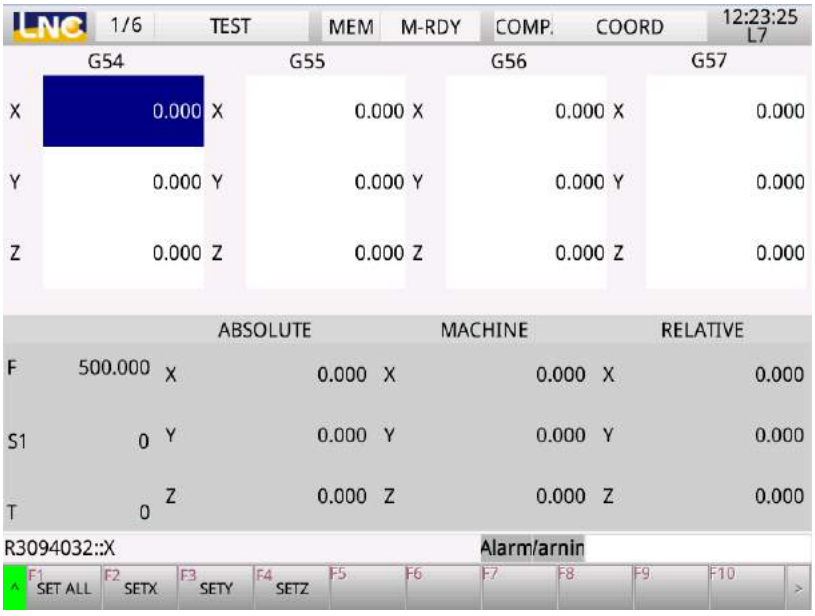


- machine coordinates (corresponding to X1 on the human machine page) on the edge of the workpiece along the X axis should be positioned manually before pressing the 【**Set X1**】 key for setting the X1 coordinates.

- Similarly, machine coordinates (corresponding to X2 on the human machine page) of the workpiece's other edge along the X axis should be positioned manually before pressing the 【**Set X2**】 key for setting X2 coordinates.
- At this moment, the controller will automatically calculate the workpiece's center coordinate of the X axis.
- The next step is to set the center coordinate of the Y axis (with the same setting method as the X axis).
- If the workpiece is circular in shape, the 【**Circle**】 key should be pressed to enter the circular center settings page. The setting method is also based on the manual positioning of machine coordinates (corresponding to P1, P2, and P3 on the human machine page) of the workpiece's edge, and the 【**Set P1**】 key, 【**Set P2**】 key, and 【**Set P3**】 key on the human machine page should be used to set the center coordinates of X and Y axis.



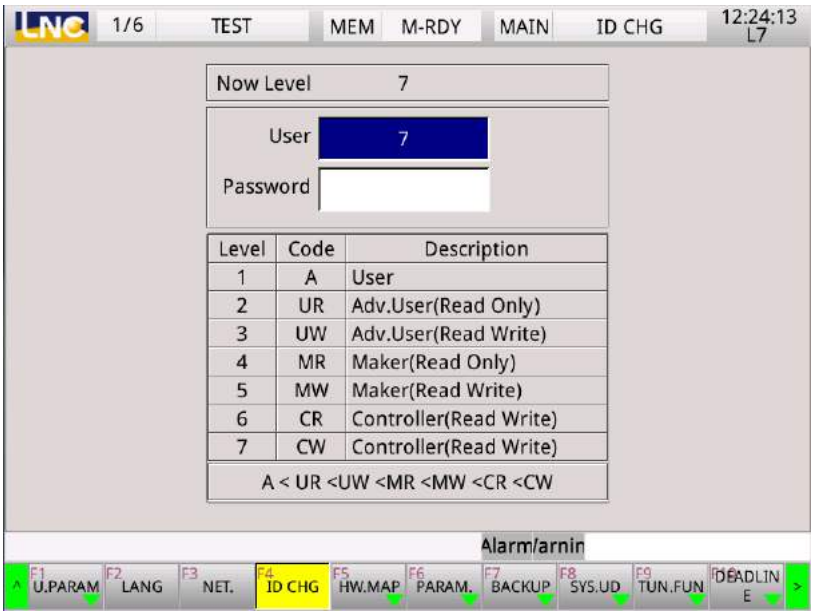
- In addition to the two methods described above, the 【**Guided input**】 key can also be pressed to manually position the reference point on the X axis of the workpiece before pressing the 【**Set X**】 key to directly set the X axis coordinate of coordinate system G54. The setting method for the X axis can be applied to other axes as well. The reference point of each axis of the workpiece can also be manually positioned before pressing the 【**Set all**】 key to directly set the coordinates of all axes in coordinate system G54.

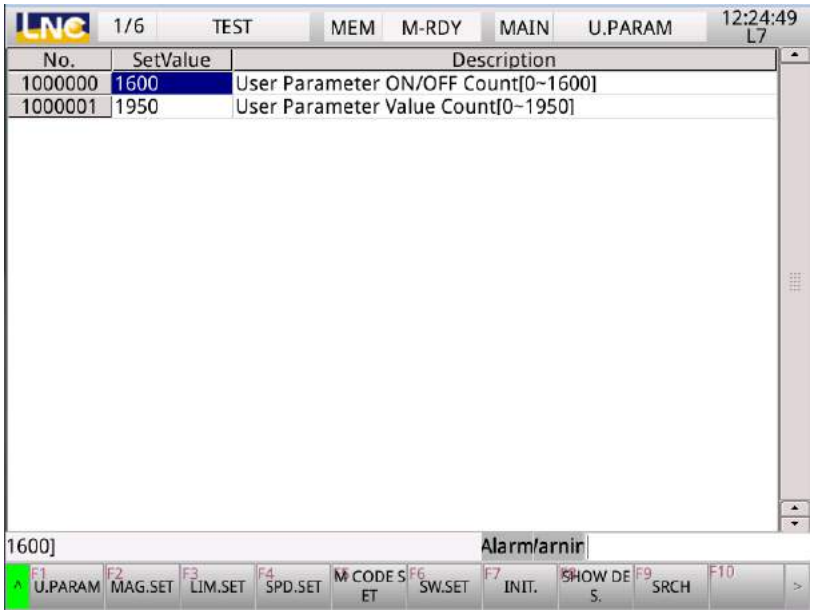


1.8.14 Parameter settings

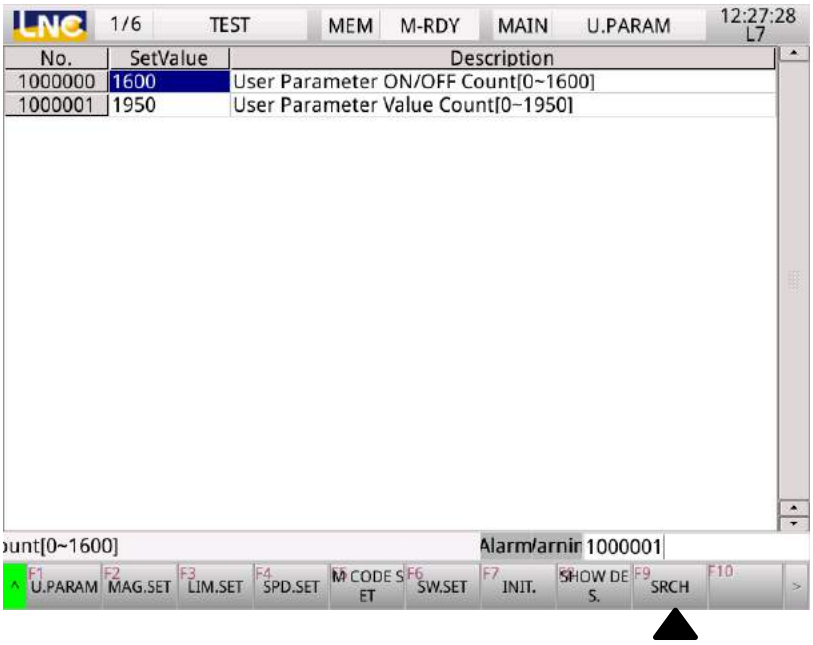
- Different parameters need to be set up in accordance with different user levels.
- By pressing the **<MAINTE>** key on the MDI panel, the controller provides two parameter setting functions to set "user parameters" and "system parameters".

- As for user parameters, the 【**User parameters**】 key on the human machine page can be pressed to enter user parameter setup page, and arrow keys on the MDI panel can be used to select parameters you wish to modify, enter numerical values in the input area followed by pressing the **<Input>** key to complete the setup as shown in the figure below.

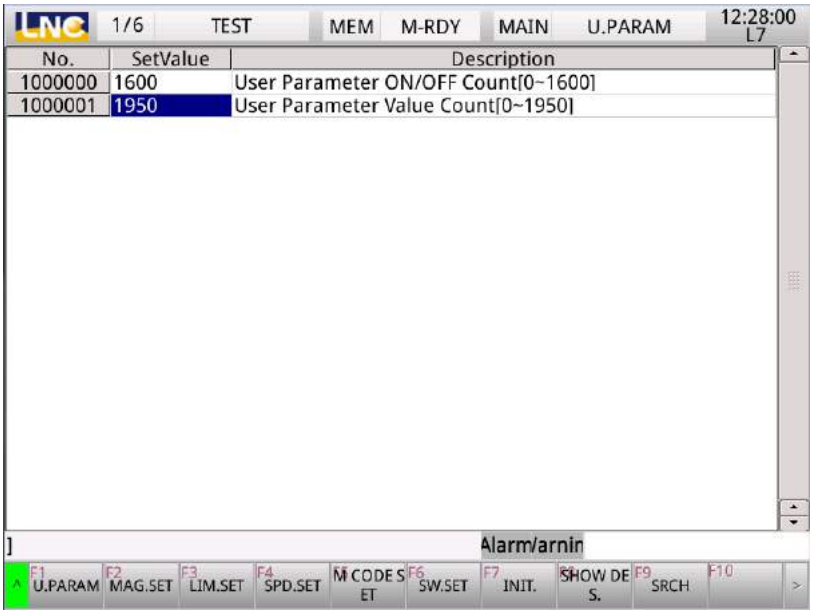


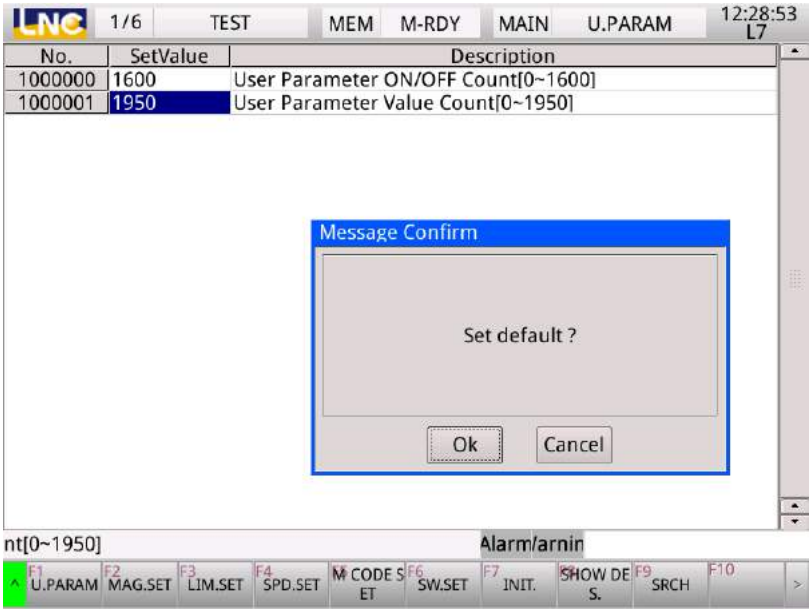


- If a known parameter needs to be found, its parameter number can be entered into the input area before pressing the 【**Find**】 key on the human machine page, the cursor will move directly to the parameter corresponding to this number as shown in the figure below.

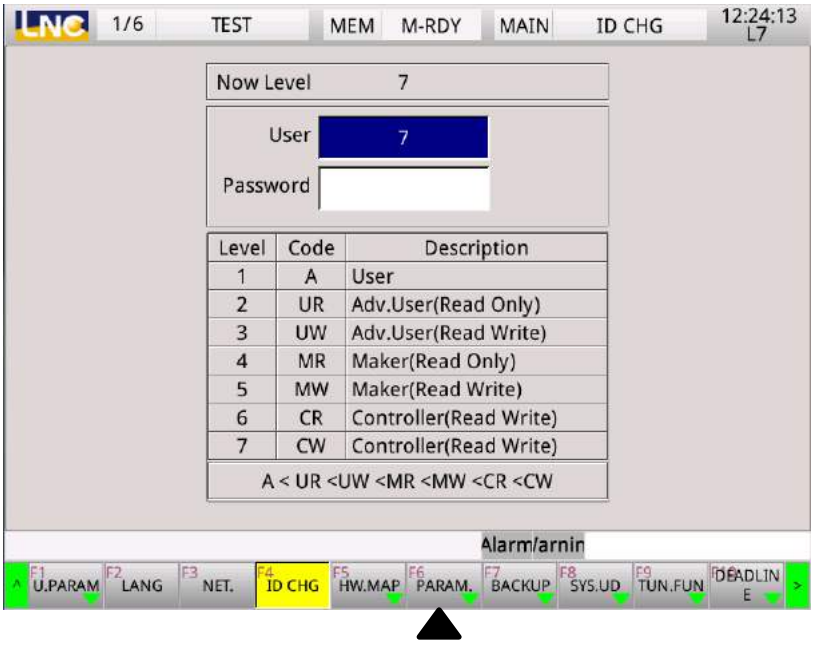


- If a parameter is to be set as a default value, the【**Initialization**】key can be pressed for a confirmation message window to appear, select **Confirm** to set the parameter as a default value.

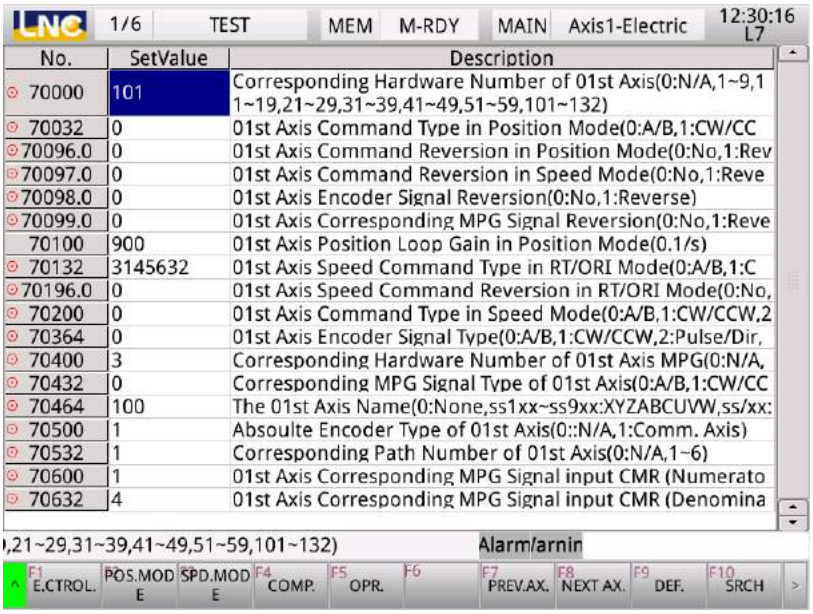




- As for system parameters, the 【**Parameters**】 key on the human machine page should be pressed to enter the system parameter setup page, and its setting method is identical to user parameters.

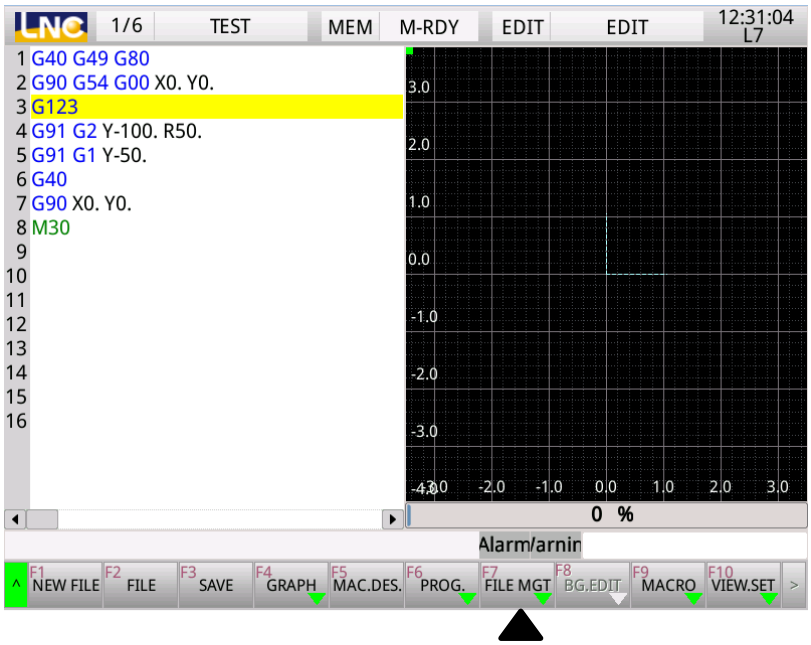


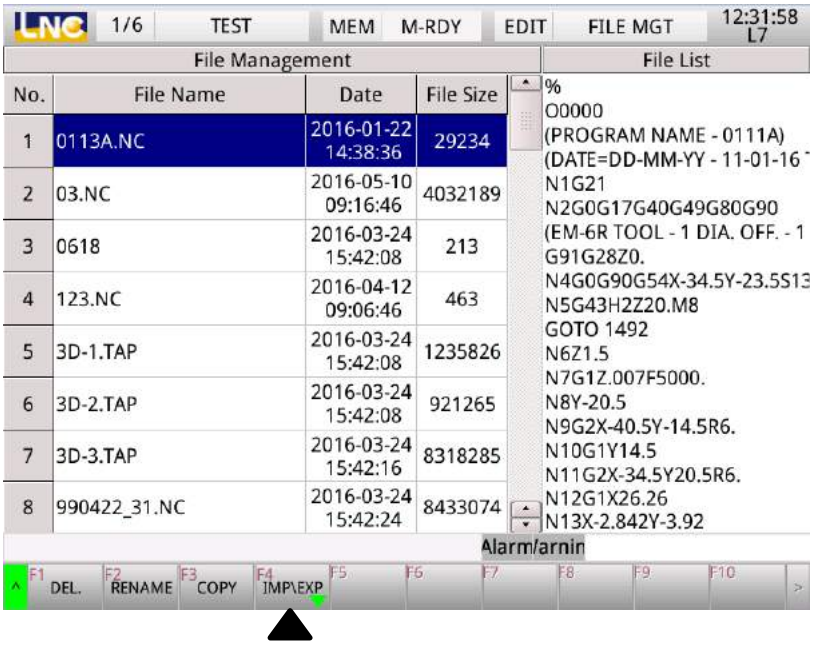
- If the position in front of a parameter number is **blank**, it means that the parameter will take effect immediately after setting is complete; if there is an **R** in front of the parameter number, it means that the parameter will take effect by pressing the <RESET> key after setting is complete; if there is a ⊙ in front of the parameter number, it means that the parameter will take effect after system reboot.


1/6
TEST
MEM
M-RDY
MAIN
Axis1-Electric
12:30:16
L7
No.
SetValue
Description
70000
101
Corresponding Hardware Number of 01st Axis(0:N/A,1~9,11~19,21~29,31~39,41~49,51~59,101~132)
70032
0
01st Axis Command Type in Position Mode(0:A/B,1:CW/CC
70096.0
0
01st Axis Command Reversion in Position Mode(0:No,1:Rev
70097.0
0
01st Axis Command Reversion in Speed Mode(0:No,1:Reve
70098.0
0
01st Axis Encoder Signal Reversion(0:No,1:Reverse)
70099.0
0
01st Axis Corresponding MPG Signal Reversion(0:No,1:Reve
70100
900
01st Axis Position Loop Gain in Position Mode(0.1/s)
70132
3145632
01st Axis Speed Command Type in RT/ORI Mode(0:A/B,1:C
70196.0
0
01st Axis Speed Command Reversion in RT/ORI Mode(0:No,
70200
0
01st Axis Command Type in Speed Mode(0:A/B,1:CW/CCW,2
70364
0
01st Axis Encoder Signal Type(0:A/B,1:CW/CCW,2:Pulse/Dir,
70400
3
Corresponding Hardware Number of 01st Axis MPG(0:N/A,
70432
0
Corresponding MPG Signal Type of 01st Axis(0:A/B,1:CW/CC
70464
100
The 01st Axis Name(0:None,ss1xx~ss9xx:XYZABCUVW,ss/xx:
70500
1
Absoulte Encoder Type of 01st Axis(0::N/A,1:Comm. Axis)
70532
1
Corresponding Path Number of 01st Axis(0:N/A,1~6)
70600
1
01st Axis Corresponding MPG Signal input CMR (Numerato
70632
4
01st Axis Corresponding MPG Signal input CMR (Denomina
,21~29,31~39,41~49,51~59,101~132)
Alarm/arnin
F1 E.CTROL.
POS.MODE
SPD.MODE
F4 COMP.
F5 OPR.
F6
F7 PREV.AX.
F8 NEXT AX.
F9 DEF.
F10 SRCH

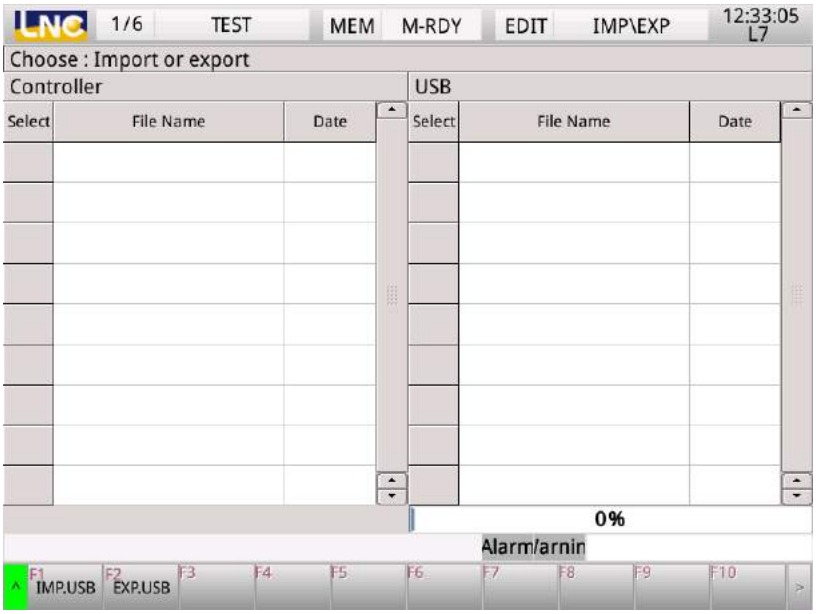
1.8.15 File backup - import

- To import or export a file, the **<PROG>** key on the controller's MDI panel can be pressed, then press the 【**File management**】 key to enter the import/export page via the 【**Import\export**】 key.

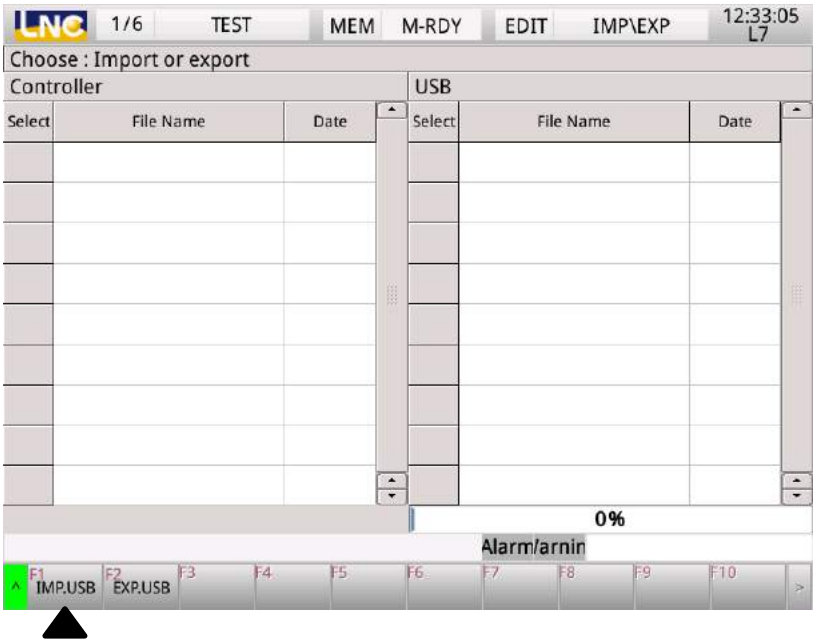




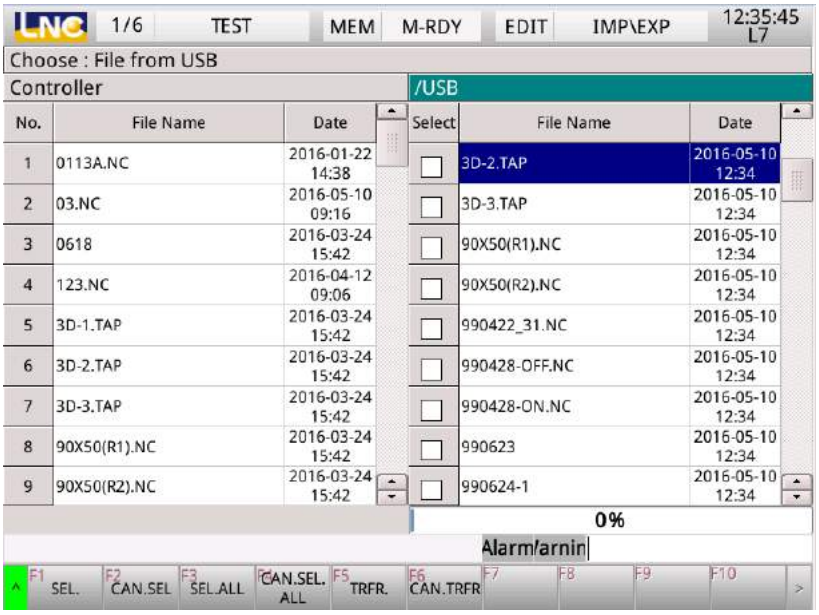
- The 【**Import from USB**】 key on the human machine page can be pressed to import files from a USB device as shown in the figure below.



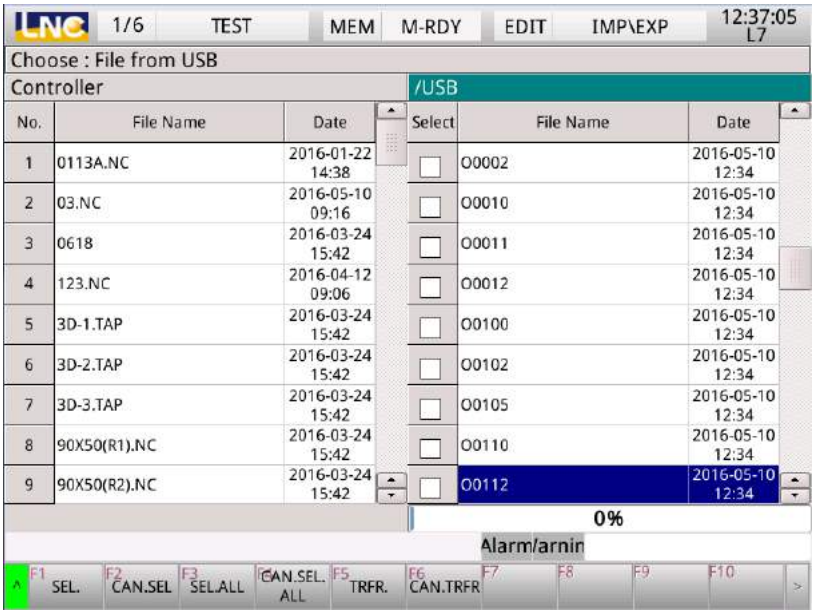
- At this time, a directory selection window will appear as shown in the figure below.



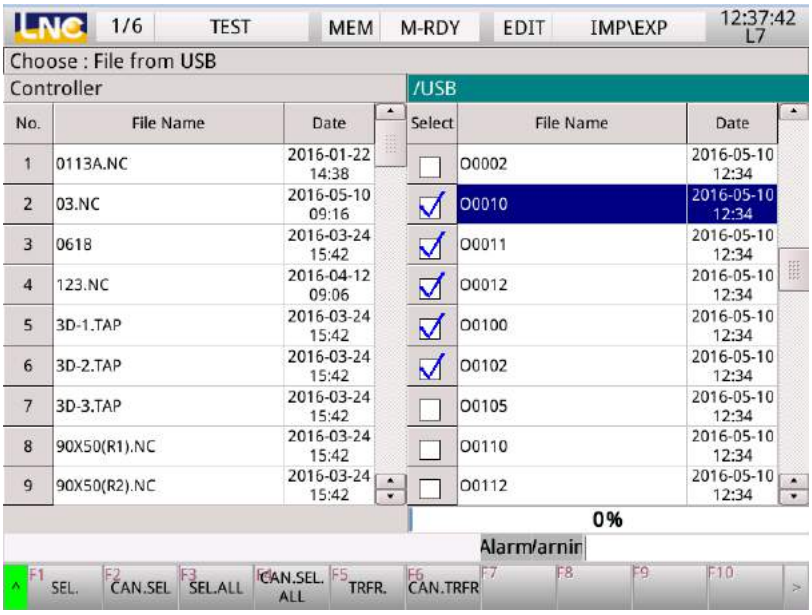
- Press the ▶ key on the MDI panel to open the subdirectory as shown in the figure below.



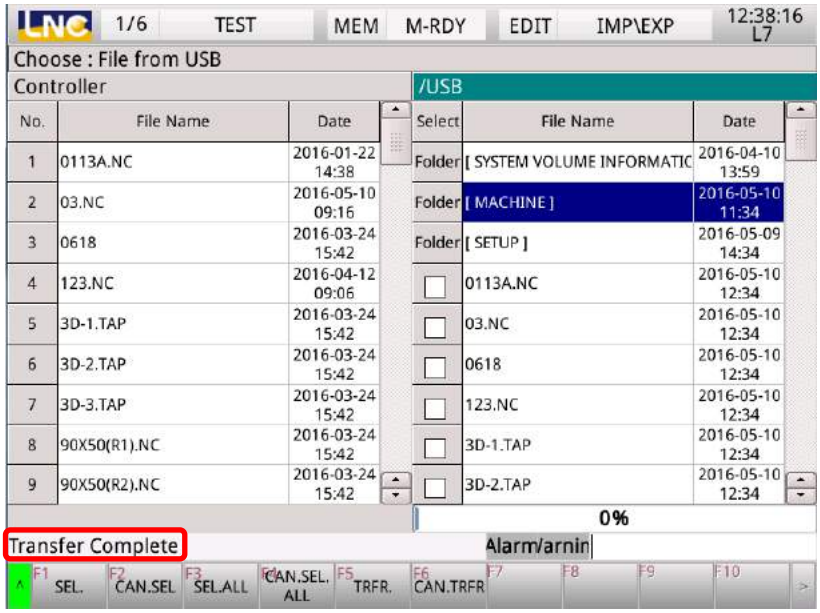
- Arrow keys on the MDI panel should be used to select the folder of file to be imported, then select **Confirm** to enter the file selection page.



- Arrow keys on the MDI panel should be used to move the cursor to select the file to be imported, press the 【**Select**】 key and the selected file will be checked (as shown in the figure below). Or the 【**Cancel select**】 key can be pressed to cancel the selection. In addition, the 【**Select all**】 key can be pressed to select all files.



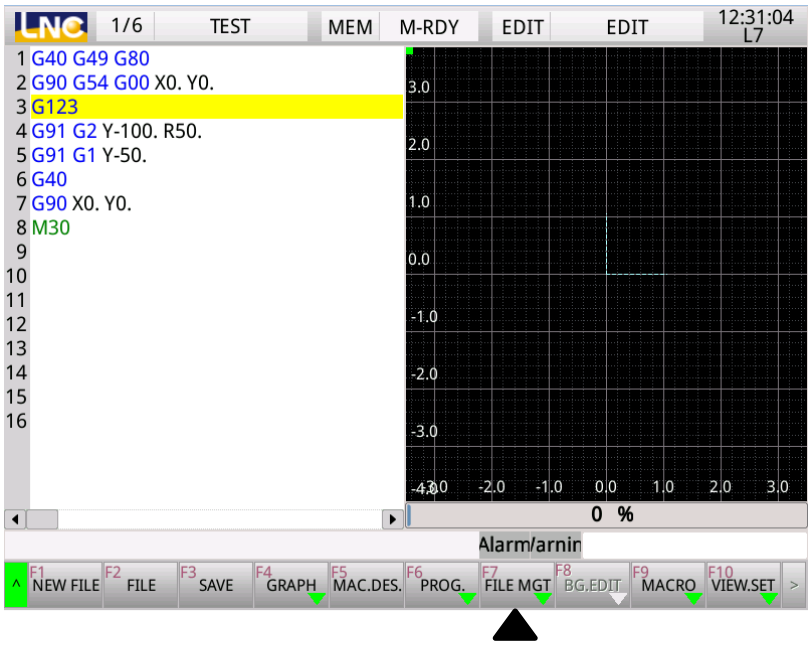
- Transfer will start after pressing the 【**Transfer**】 key. The "Transfer completed" message will be displayed in the message prompt area after the process is completed as shown in the figure below.

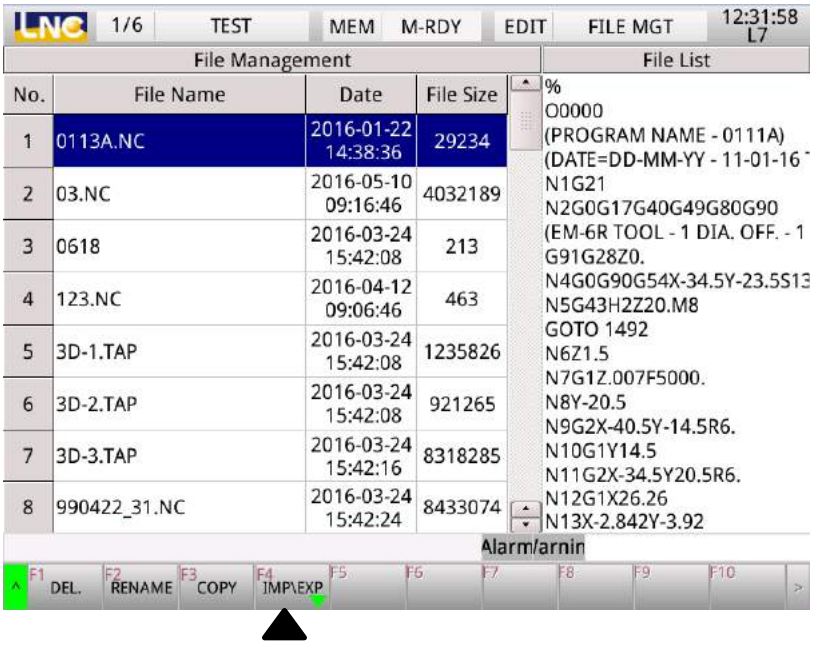


- The importing method of manufacturer macros is the same as the method described above.

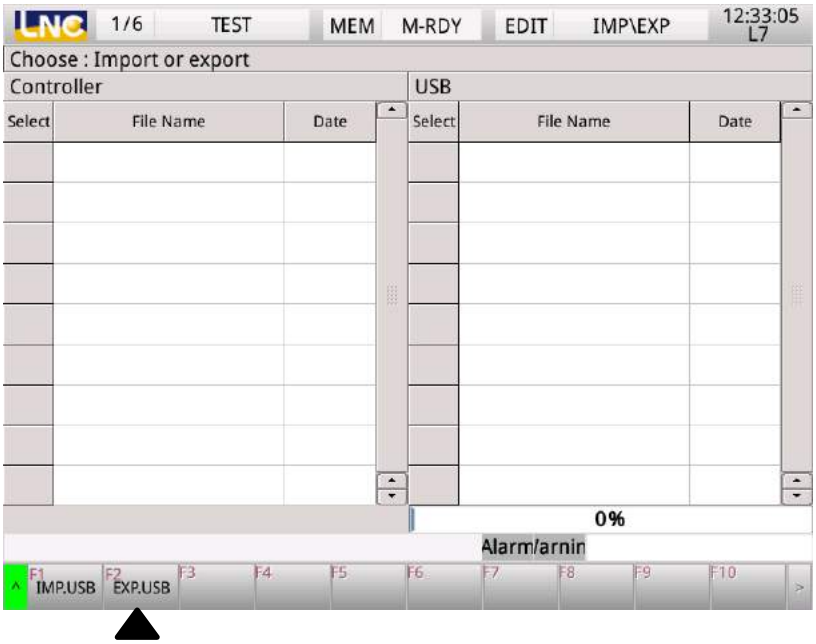
1.8.16 File backup - export

- To import or export a file, the **<PROG>** key on the controller's MDI panel can be pressed, then press the 【**File management**】 key to enter the import/export page via the 【**Import\export**】 key.

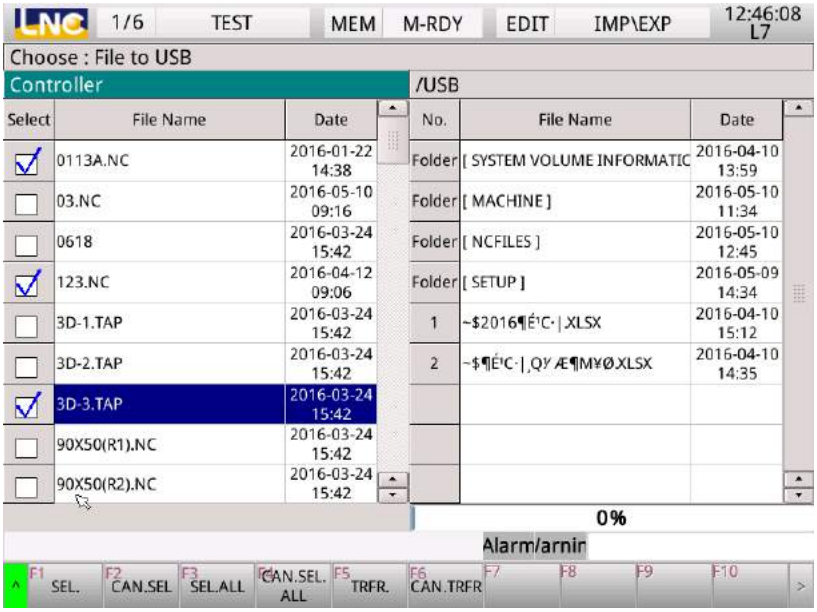




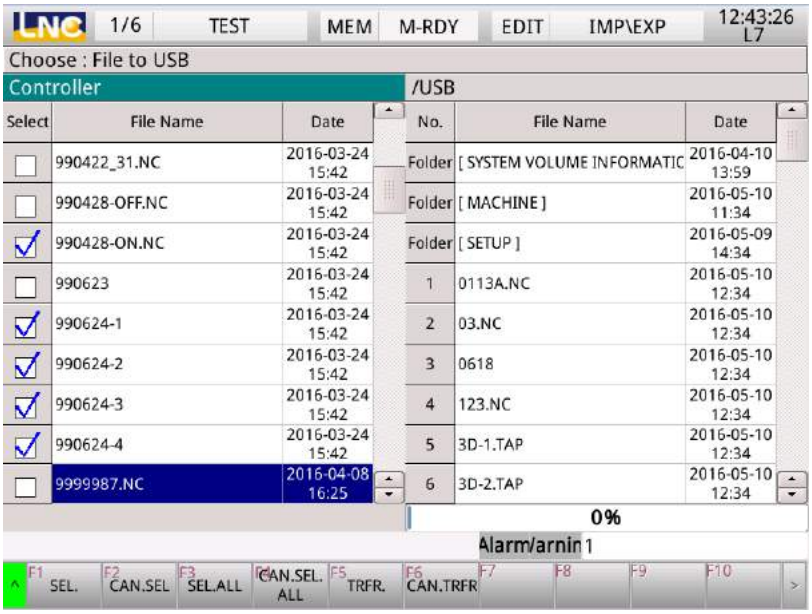
- The 【**Export to USB**】 key on the human machine page can be pressed to export files to a USB device as shown in the figure below.



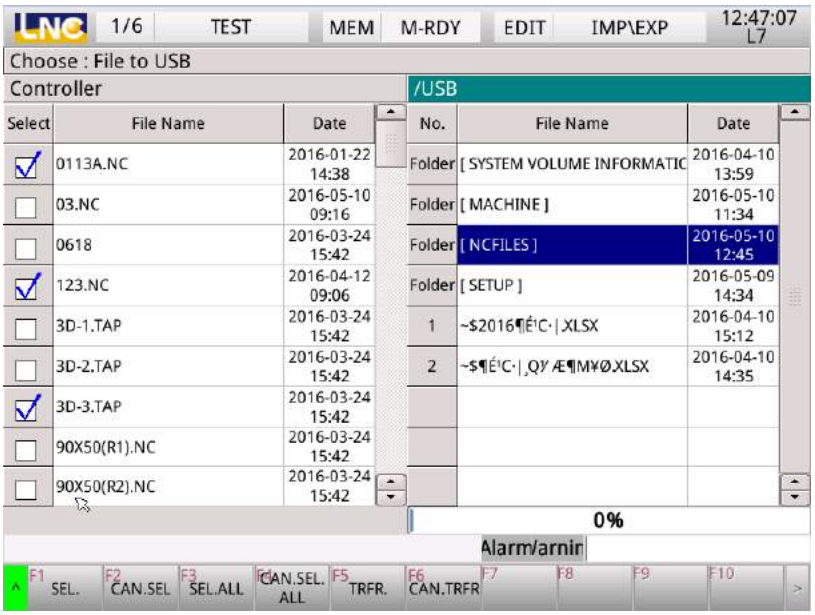
- Arrow keys on the MDI panel should be used to move the cursor to select the file to be exported, pressing the 【**Select**】 key and the selected file will be checked (as shown in the figure below). Or the 【**Cancel select**】 key can be pressed to cancel the selection. In addition, the 【**Select all**】 key can be pressed to directly export all files.



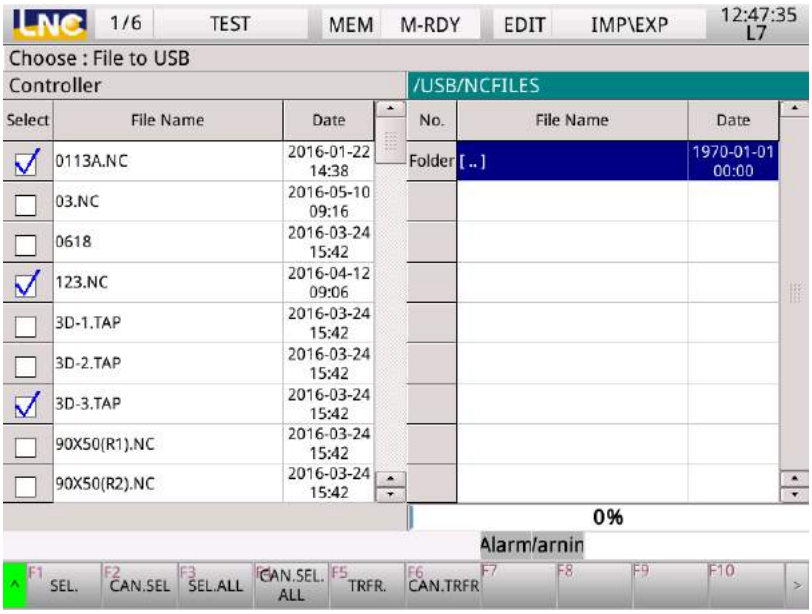
- After pressing the 【**Transfer**】 key on the human machine page, a window for directory selection will appear.



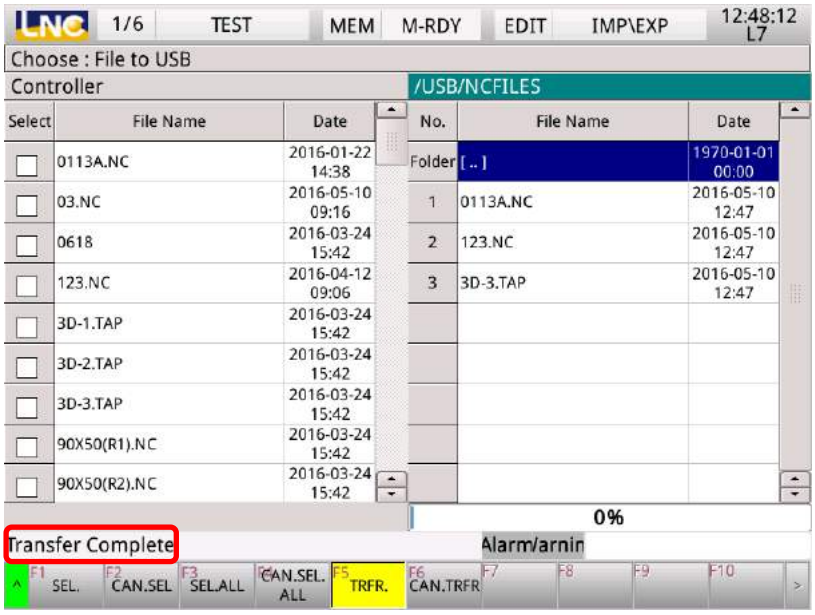
- Press the ▶ key to open the subdirectory as shown in the figure below.



- Arrow keys on the MDI panel should be used to select the folder to be exported, and then **Confirm** should be selected to start the transfer.



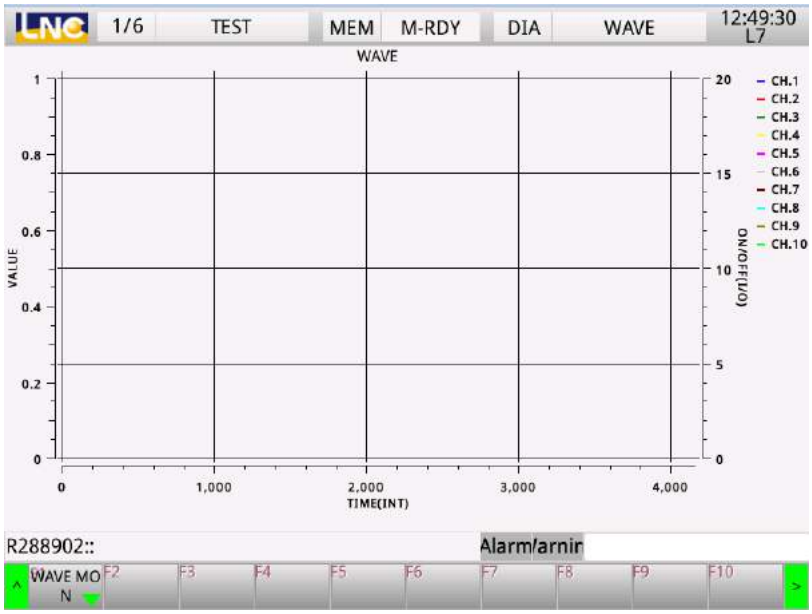
- After the transfer is completed, the "Transfer completed" message will be displayed in the message prompt area.

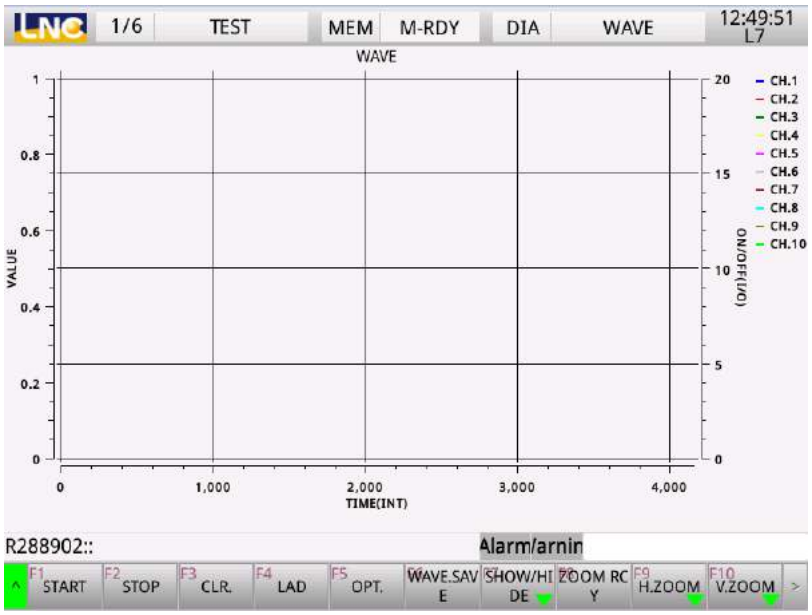


- The exporting method of manufacturer macros are the same as the method described above.

1.8.17 Waveform monitoring function

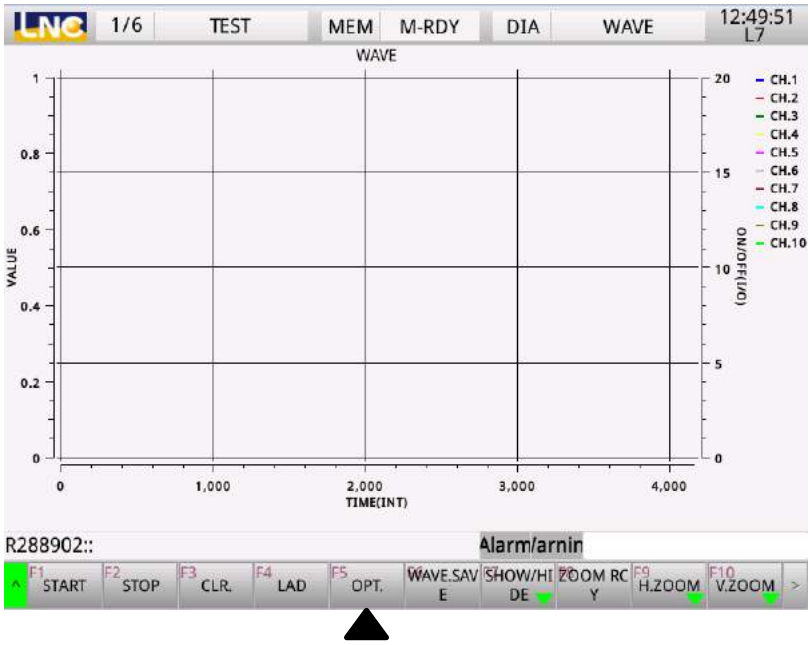
- Press the **<DGNOS>** key on the controller's MDI panel to enter diagnosis group, press the 【 > 】 key followed by 【**Waveform monitoring**】 to enter the waveform monitoring page from the human machine page as shown in the figure below.

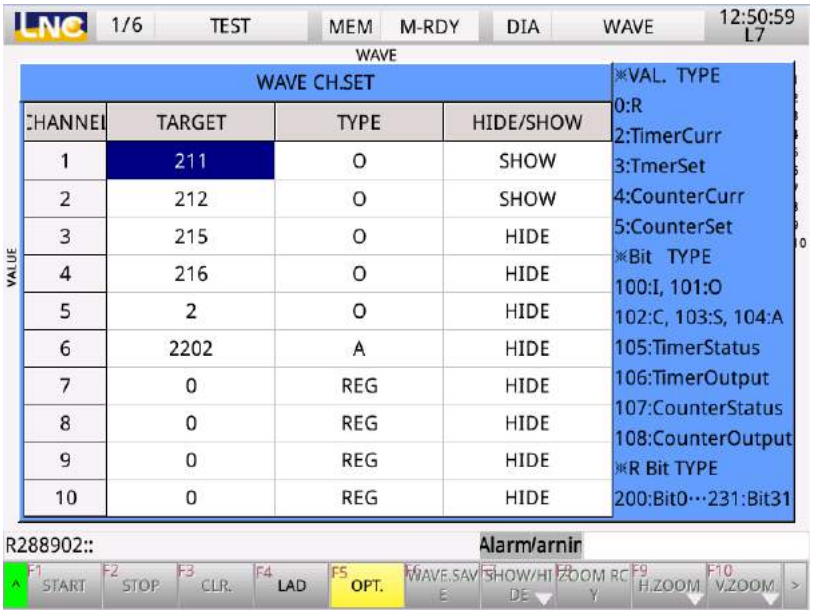




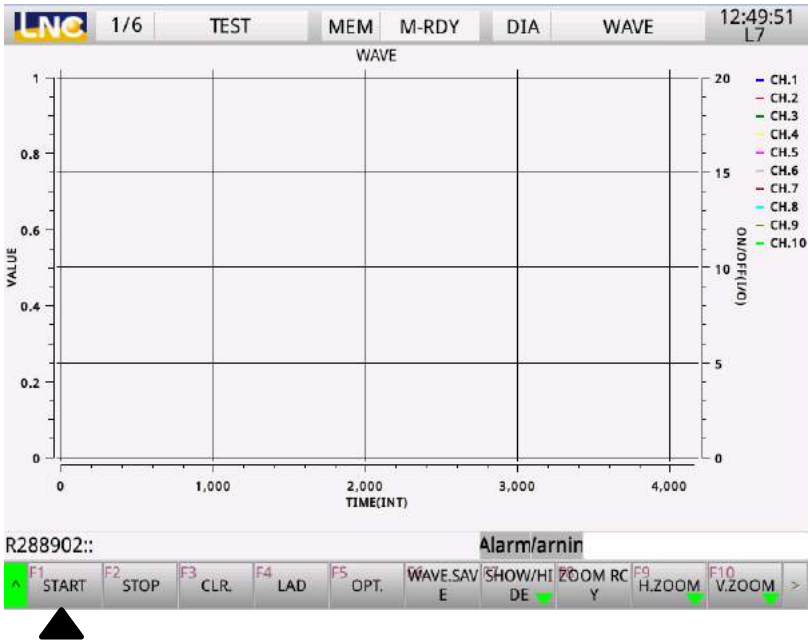
- A settings window will appear after pressing the 【**Option**】 key. The number can be set in the "Target" field, the type can be set in the "Type" field, and the "Hide" field can be used to hide the channel. When setting changes are complete, the 【**Option**】 key can be pressed again to close the setting window.

  Specifications of the type field:

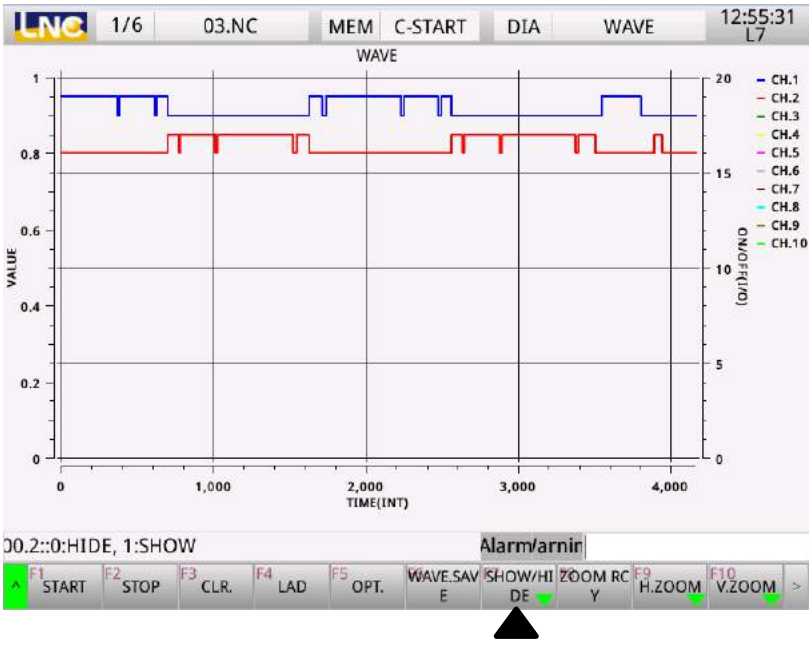


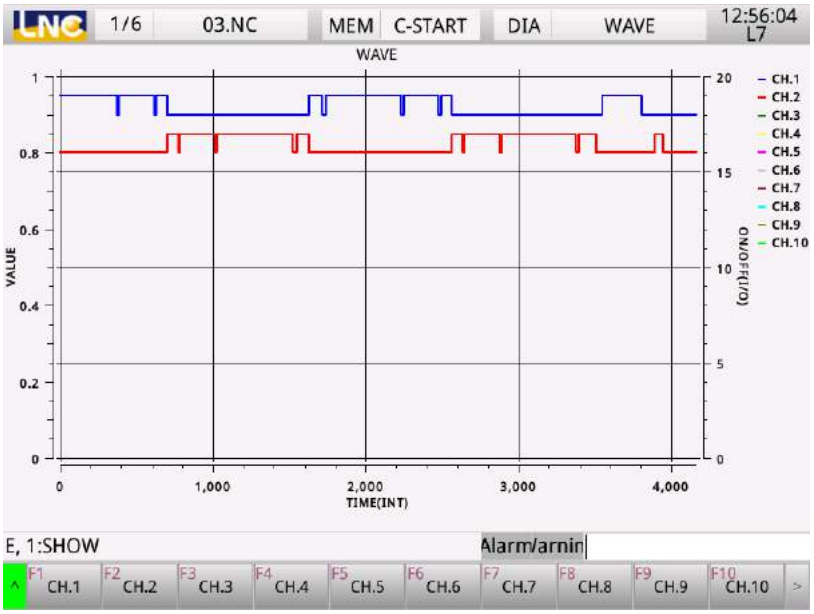


- Press the 【**Start**】 key to monitor the waveform; press the 【**Stop**】 key to stop at the current screen; press the 【**Clear**】 key to clear the waveform.

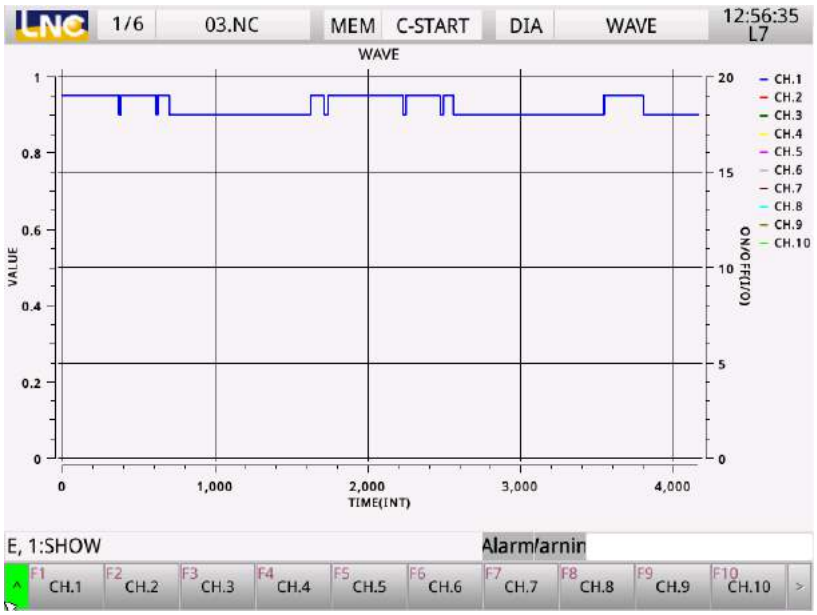


- To hide the waveform of channel 2, the 【**Display/hide**】 key can be pressed to access channel selection.

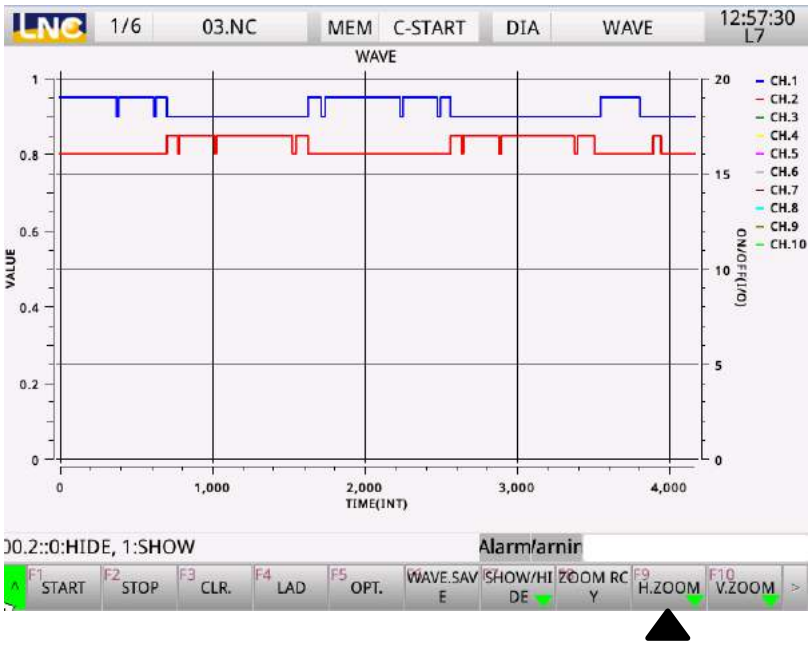


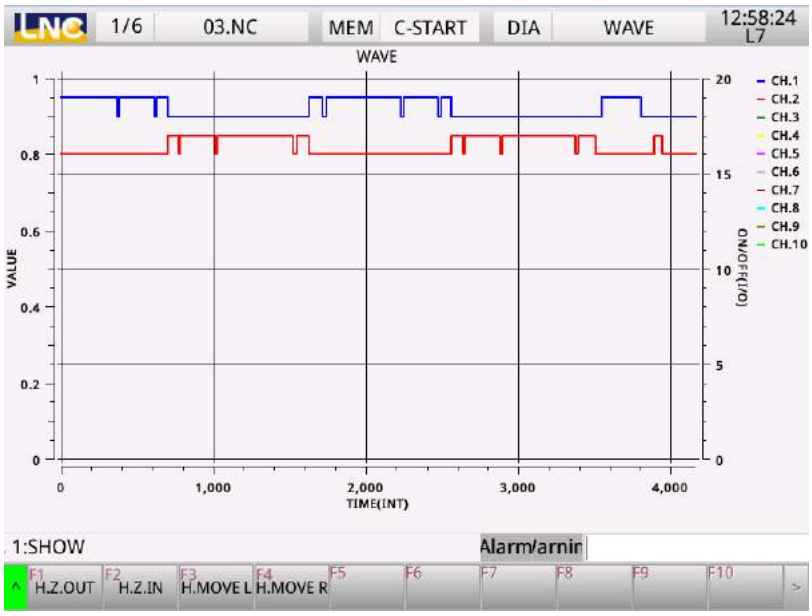


- Press the 【**Channel 2**】 key on the page to hide the waveform of channel 2.

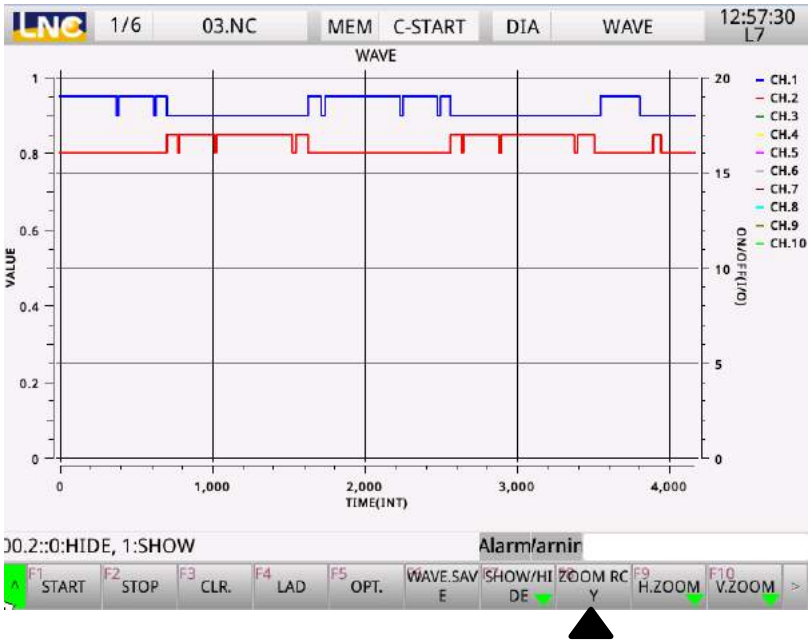


- Press the【**Channel 2**】key again to display the waveform of channel 2. The【 ^ 】key on the human machine page can be pressed to go back to the previous page.
- To zoom in/out horizontally on the waveform, the 【**Horizontal zoom**】 key can be pressed to access the horizontal function keys.

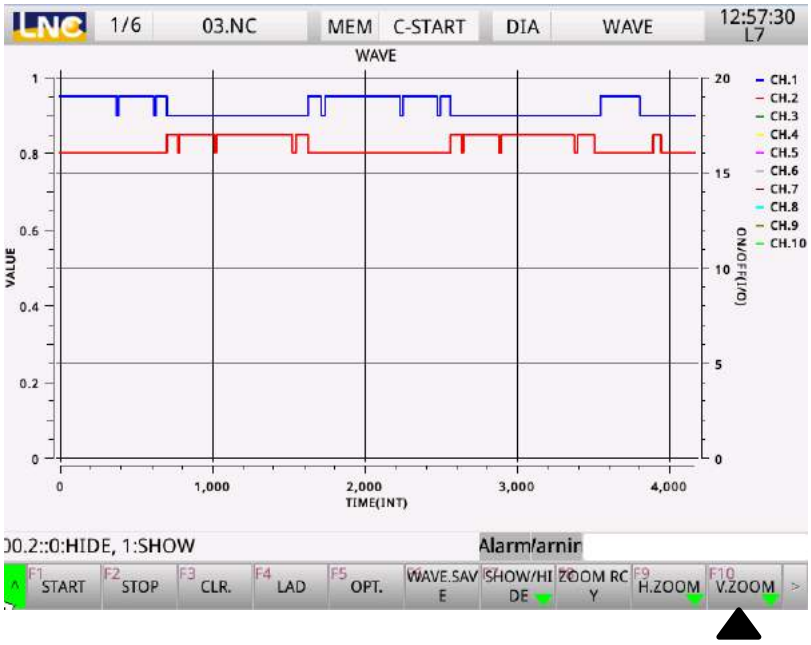


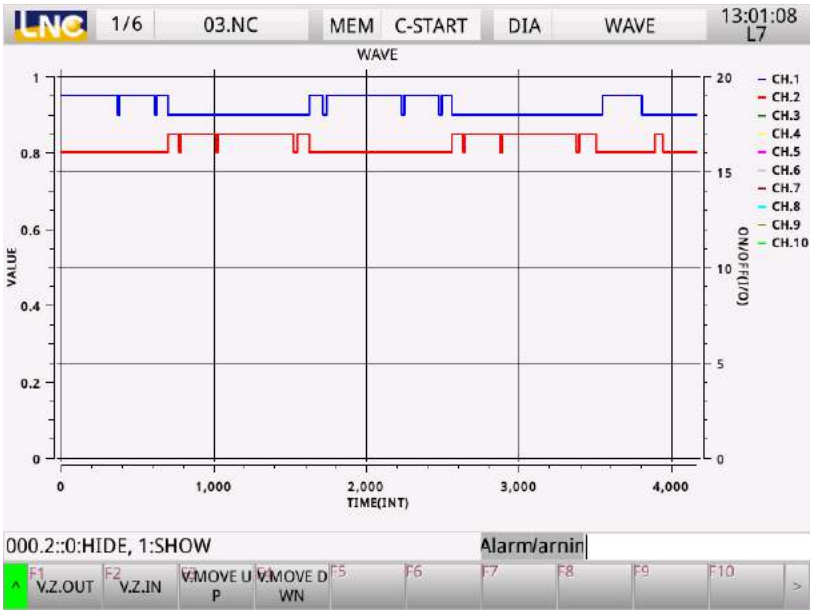


- Press the 【**Zoom-in horizontally**】 key to magnify the display of waveform along the horizontal direction.
- Press the 【Zoom-out **horizontally**】 key to shrink the display of waveform along the horizontal direction.
- Press the 【**Move left horizontally**】 key to move all waveforms to the left.
- Press the 【**Move right horizontally**】 key to move all waveforms to the right.
- To undo the zooming action, press the 【 ^ 】 key on the page followed by pressing the 【**Undo zoom**】 key on the human machine page.



- To zoom in/out vertically on the waveform, the 【**Vertical zoom**】 key on the human machine page can be pressed to access the vertical function keys.





- Press the 【**Zoom-in Vertically**】 key to magnify the display of waveform along the vertical direction; press the 【**Zoom-out Vertically**】 key to shrink the display of waveform along the vertical direction.
- Press the 【Move up **vertically**】 key to move the entire waveform upward.
- Press the 【Move down **vertically**】 key to move the entire waveform downward.
- To undo the zooming action, press the 【 **^** 】 key on the page followed by pressing the 【**Undo zoom**】 key on the human machine page.

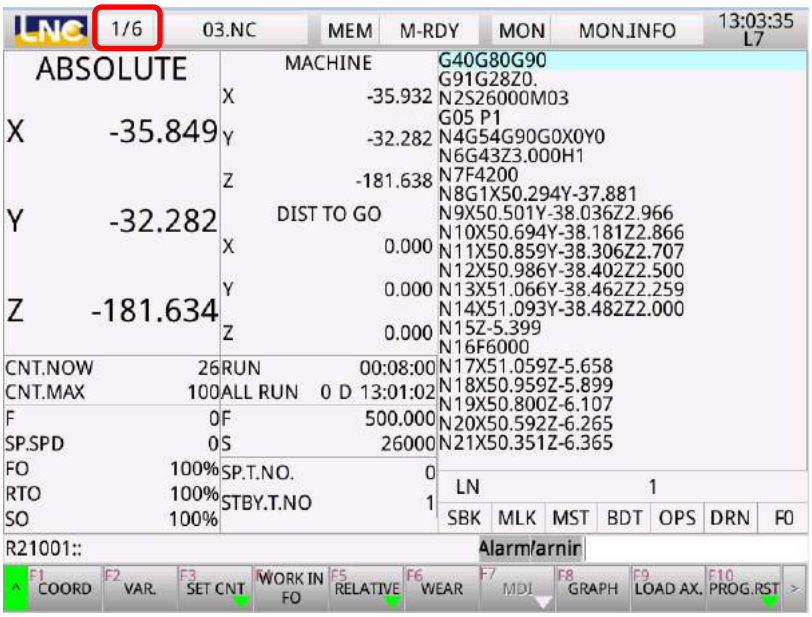
1.8.18 OnLine Help

- It will be displayed by pressing the **<Help>** key on the controller's MDI panel.
- Take this monitoring page as an example; the **<Help>** key on the MDI panel can be pressed to enter the Online Help screen as shown in the figure below.
- It can be used in coordination with ▲ ▼ keys and **<PgUp>** and **<PgDn>** keys on the MDI panel for browsing.
- Pressing the **<Input>** key on the human machine page to close Online Help.

1.8.19 Switching multiple paths

(Note: This function key is not available in the M5800 series)

- When the multiple path function is activated, an additional path field will be shown on the status bar; press the **<PATH>** key on the MDI panel to switch machining paths.


1/6
03.NC
MEM
M-RDY
MON
MON.INFO
13:03:35
L7
ABSOLUTE
X -35.849
Y -32.282
Z -181.634
MACHINE
X -35.932
Y -32.282
Z -181.638
DIST TO GO
X 0.000
Y 0.000
Z 0.000
CNT.NOW 26
CNT.MAX 100
F 0
SP.SPD 0
FO 100%
RTO 100%
SO 100%
RUN 00:08:00
ALL RUN 0 D 13:01:02
F 500.000
S 26000
SP.T.NO. 0
STBY.T.NO 1
G40G80G90
G91G28Z0.
N2S26000M03
G05 P1
N4G54G90G0X0Y0
N6G43Z3.000H1
N7F4200
N8G1X50.294Y-37.881
N9X50.501Y-38.036Z2.966
N10X50.694Y-38.181Z2.866
N11X50.859Y-38.306Z2.707
N12X50.986Y-38.402Z2.500
N13X51.066Y-38.462Z2.259
N14X51.093Y-38.482Z2.000
N15Z-5.399
N16F6000
N17X51.059Z-5.658
N18X50.959Z-5.899
N19X50.800Z-6.107
N20X50.592Z-6.265
N21X50.351Z-6.365
LN 1
SBK MLK MST BDT OPS DRN F0
R21001::
Alarm/arnir
F1 COORD
F2 VAR.
F3 SET CNT
F4 WORK IN FO
F5 RELATIVE
F6 WEAR
F7 MDI
F8 GRAPH
F9 LOAD AX.
F10 PROG.RST

2 Control panel operations

The control panel is designed by the machine maker based on different requirements. Only keys involving the most frequently used operations and functions are described here.

2.1 Operating panel

2.1.1 M5800 series

2.1.2 M6800 series

Based on function, it can be divided into the first operating panel and the second operating panel:

The first operating panel

The second operating panel

2.2 LED signal light (LED SIGNAL)

M6800 series

1: All axes have returned to the reference point.

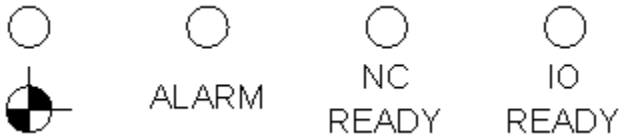
2: The light will come on automatically after the power is turned on.

3: The light will come on automatically when an alert is issued by the controller.

4: The light will automatically turn on when the level of grease is too low.

5: Insufficient air pressure.

6: Tool magazine malfunction.



M5800 series

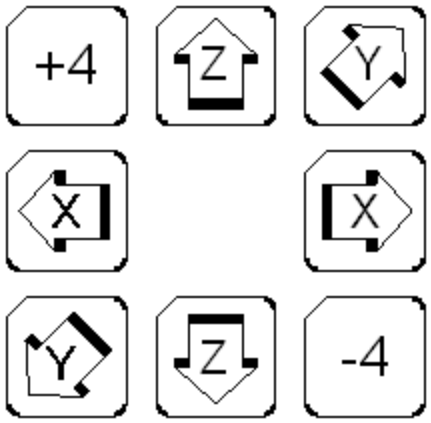
1: All axes have returned to the reference point.

2: The light will come on automatically when an alert is issued by the controller.

3: The light will come on automatically after the power is turned on.

4: The light will come on automatically the I/O axis card has successfully performed a self-hardware circuit test.

2.3 Axis selection (AXIS SELECTION)



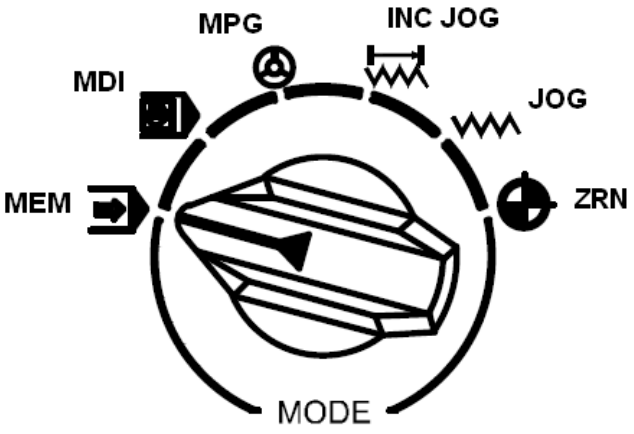
These keys can be used for assigning the direction of axial movement under manual continuous feeding (JOG) mode and reference point return mode (ZRN).

For example:

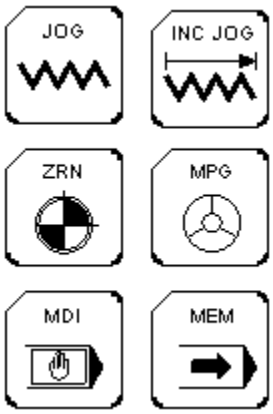
By pressing and holding +X under manual continuous feeding (JOG) mode, the X axis will move towards the positive direction, and it will stop after the key is released. The same operating method can be applied to movement along other axes.

By pressing +X once under the reference point return (ZRN) mode, the X axis will automatically return to the reference point along the positive direction. The same operating method can be applied to movement along other axes.

2.4 Mode selection (MODE SELECTION)



Knob style



Key style

There are a total of 6 modes on operating panel, including automatic machining mode (MEM), manual input mode (MDI), hand wheel mode (MPG), reference point return mode (ZRN), continuous jog mode (JOG), and incremental jog mode (INC JOG).

(1). **Memory mode (MEM)**

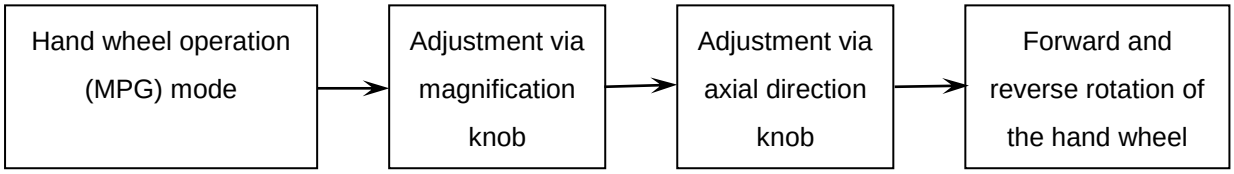
This mode is for executing the program automatically.

(2). **Manual input mode (MDI)**

This mode is mainly for executing program blocks, modifying parameters, and setting data.

(3). **Hand wheel mode (MPG)**

In this mode, users can use the hand wheel to control the feed of servo axis. The hand wheel control panel is equipped with magnification selection switches of 1X, 10X, and 100X with the minimum command unit (0.001 mm or 0.0001 in.) and axial direction selection knob.



(4). **Reference point return mode (ZRN)**

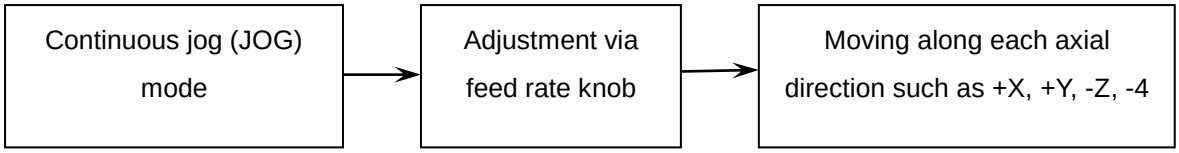
It is for performing the reference point return of each axis. After being switched to this mode, arrow keys (same keys as manual continuous feeding) used for returning to reference point for each axis can be pressed for the feed along that axis with a reference point return speed equal to what is set in the parameter until reaching the DOG. The servo axis will start searching for the reference point position until it reaches the reference point. At this moment, the + direction indicator light of that axis will come on; it will also come on every time users switch to HOME in order to remind users that the HOME operations have been completed by the machine. For each machine restart, reference point return should be implemented before executing other machining programs in order to ensure the correctness

of every axis coordinate.

If the servo mechanism is used in coordination with an absolute encoder, reference point return can be completed by directly returning to the reference point position without touching the DOG.

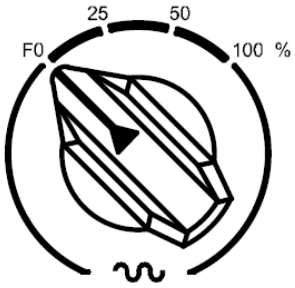
(5). **Continuous jog mode (JOG)**

Under this mode, operators can select the direction for axial movement, and the movement speed will be determined by the feed rate.



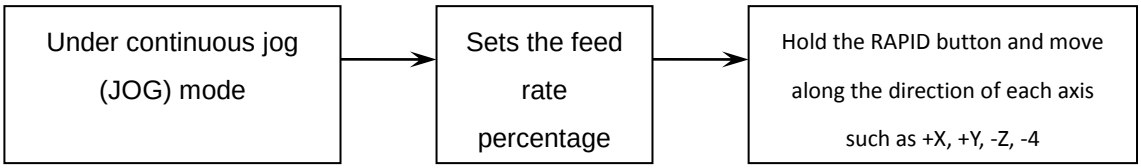
(6). **Rapid feeding (RAPID)**

Under continuous jog (JOG) mode, operators can use axial movement direction keys to move the axis while hold the **RAPID** key to start rapid feeding, and the movement speed will be determined by the rapid feeding % button. Rapid feeding is valid under three conditions: execution of G00 in the program, manual rapid feeding mode (RAPID), and velocity of the front section during reference point return. There are 4 stages for the feeding rate percentage knob, including F0 (LOW), 25%, 50%, and 100%. The speed of F0 is set by user parameter number 8051.



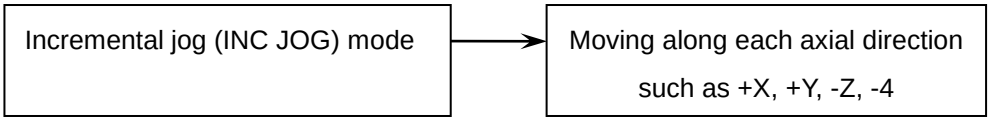
Knob style

Key style

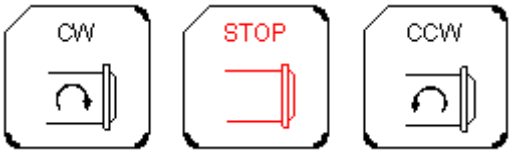


(7). **Incremental jog mode (INC JOG)**

Under this mode, operators can select the direction for axial movement, and the amount of each movement is set via user parameter numbers 8130 to 8134.


Incremental jog (INC JOG) mode
Moving along each axial direction such as +X, +Y, -Z, -4

2.5 Spindle operation & spindle speed adjustment



Spindle operation

Under manual mode (here it refers to JOG, RAPID, and MPG modes), the spindle operation can be controlled by these three keys at any time.

CW : Spindle in forward rotation.

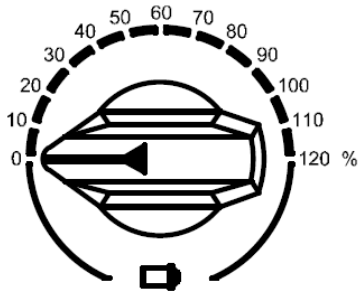
STOP : spindle operation is stopped.

CCW : spindle in reverse rotation.

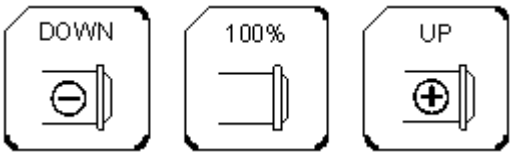
Under manual mode, the rotational speed command is adjusted between 0% to 120% regardless if the spindle is turning clockwise or counterclockwise. One important thing to note is that the switching of rotational speed must go through the process of pressing the STOP key or it will be invalid.

Adjustment of spindle speed

When the spindle is under memory mode or manual data input mode, after spindle rotation is started via the command M3 (or M4) Sxx....., the spindle's actual rotation speed can be adjusted by this knob (key) in between 0% to 120%. For example: with the issued command of M3 S1000 and the speed fixed at 120%, the actual rotational speed will be 1200RPM.



Knob style



Key style

2.6 Supplementary function keys

The controller can be used in coordination with frequently used supplementary function keys to facilitate operation, such as the over travel release or PLC's self-defined key function; functions of various keys are described below:

1. **MPG DRN (hand wheel mode)**

This is the switch for controlling MPG dry run. The MPG dry run will be activated by pressing this key, and it will be turned off by pressing this key again.

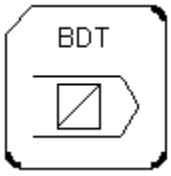
After the system enters its cycle start status, the program can be controlled by the hand wheel to execute its operation, and program coordinates as well as servo axis will change accordingly. The clockwise hand wheel program will be executed forward, and faster hand wheel rotation will lead to faster feeding; however, the hand wheel's maximum rotational speed will not exceed the value in the program's feed command. The program will be stopped when the hand wheel is stopped.

2. OP STOP (optional pause)

This is the switch for controlling optional pause. Optional pause will be activated by pressing this key, and it will be stopped by pressing it again.

Every time program execution has reached M01, it will be paused, and users must press the program start〔CYCLE START〕key in order to resume program execution.

3. BDT (optional skip)



This is the switch for controlling optional skip. Optional skip will be activated by pressing this key, and it will be stopped by pressing it again.

Program blocks start with the symbol " / " will be skipped during execution.

4. SBK (single block execution)

This is the switch for controlling single block execution. Single block execution will be started by pressing this key, and it will be terminated by pressing this key again.

When the single block execution switch is ON, program operation will be based on executing single blocks without continuous action, and the operation of each block must be executed by pressing the program start 〔CYCLE START〕 key.

1.

2.
3.
4.
5. **MST (MST ignore)**

This is the switch for controlling M, S, T commands to be ignored. The MST ignore will be activated by pressing this key, and it will be turned off by pressing this key again.

The M, S, T commands in the program will be ignored and not executed.

1.
2.
3.
4.
5.
6. **MLK (machine lock)**

This is the switch for controlling machine lock, which will be activated by pressing this key, and it will be turned off by pressing this key again.

During program execution, the controller will continue with program execution, but the servo axis will stop issuing movement commands; so the servo axis is actually stopped.

1.
2.
3.
4.
5.
6.
7. **ORI (spindle orientation)**

This key can be pressed to set the spindle's location. The **<**Reset**>** key can be pressed to cancel the setting.

2、

3、

4、

5、

6、

1.

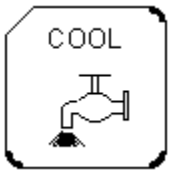
2.

3.

4.

5.

8. COOL (cutting fluid)



This is the switch for controlling cutting coolant. The cutting coolant can be activated by pressing this key, and it can be shut off by pressing this key again.

9. AIR BLOW (blowing air)

This is the switch for controlling the blow of air. Air blow can be started by pressing this key, and it can be turned off by pressing this key again.

1.

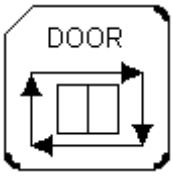
2.

3.

4.

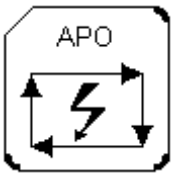
5.

6.
7.
8.
9.
10. **DOOR (safety door)**



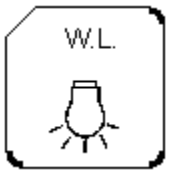
The safety door will be locked after pressing this key.

1.
2.
3.
4.
5.
6.
7.
8.
9.
10.
11. **APO (automatic power outage)**



After pressing this key, the power will be automatically disconnected after the program is executed to the M30 command.

11.
12. **W.L (work light)**



This is the switch for controlling the work light. The work light can be turned on by pressing this key, and it can be turned off by pressing this key again.

1.
2.
3.
4.
5.
6.
7.
8.
9.
10.
11.
12.
13. **MAG CW (magazine clockwise rotation)**

This is for manually rotating the magazine in the clockwise direction. Under manual mode, (in this instance it refers to JOG, RAPID, MPG), this key can be pressed (the MAG CW indicator light will be turned on), and the tool holder plate will rotate clockwise; it will stop at the next position after the finger is removed from this key. This key cannot enter a self-sustaining state, which means the condition is canceled when the key is released (and the indicator light will go off).

14. **MAG CCW (magazine counterclockwise rotation)**

This is for manually rotating the magazine in the counterclockwise direction. Operating procedure is the same as the MAG CW key.

15. **CHIP CW (chip former clockwise rotation)**

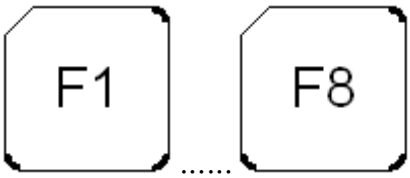
This is the switch for controlling the clockwise rotation of the chip former. The chip former will be activated by pressing this key, and it will be turned off by pressing this key again.

16. **CHIP CCW (chip former counterclockwise rotation)**

This is the switch for controlling the counterclockwise rotation of the chip former. The chip former will be activated in counterclockwise rotation by pressing this key, and it will be stopped after this key is released. This key cannot enter a self-sustaining state, which means the condition is canceled when the key is released (and the indicator light will go off).

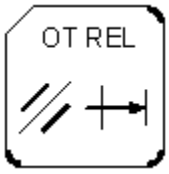
1.
2.
3.
4.
5.
6.
7.
8.
9.
10.
11.
12.
13.
14.
15.
16.

17. **F1 - F8 (additional function options)**



This function key is open for definition and used by the tool machine manufacturer.

18. **OT REL (over-travel limit release)**

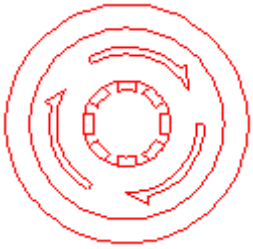


OT REL is the abbreviation of Over-Travel Release (to release over-travel). There is one limit switch on each of the two ends of the servo axis stroke to prevent damage due to servo mechanism collision. Over travel takes place every time the servo mechanism reaches the stroke limit. When over travel of this controller occurs, it is regarded as an emergency stop, and the message "EMERGENCY STOP OR OVER TRAVEL" will be displayed on the screen with indicator lights flashing. The servo mechanism must be examined for over travel.

If over travel is confirmed, it will be switched to hand wheel (MPG) operation mode or continuous jog mode (JOG) before pressing this key (with indicator lights turned on). The controller will temporarily ignore the emergency situation caused by over travel and allow the operator to move the servo axis back within the stroke via the hand wheel or axis arrow keys; the (OT REL) can be released for the system to resume stroke inspection. If everything has been restored to normal status, "Not ready" will be replaced by "Ready" to indicate that normal operation can be resumed. If other alert messages are also appearing, the **<Reset>** key must be pressed before restoring to normal operation. Please pay attention to the direction and speed of movement while moving back the servo mechanism in order to avoid collision.

(Note) when the "Not ready" status suddenly shows up, it could be caused by over travel, so please include over travel as one of the items to be inspected.

2.7 Emergency stop (EMG-STOP)

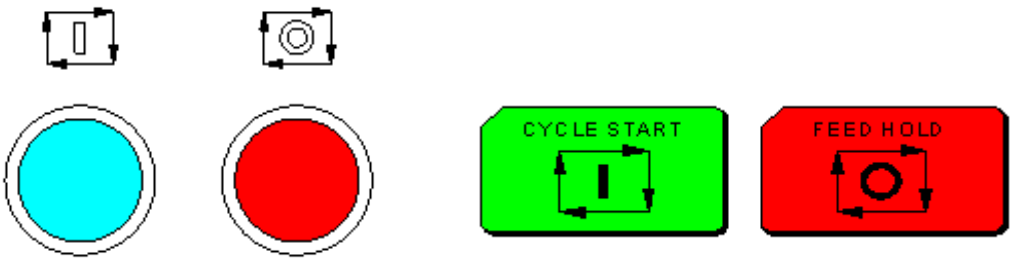


In case of a dangerous or emergency situation, this button can be pressed to stop all operation. The method for releasing this button is to rotate the button along the direction of the arrow on the button, then the button will automatically bounce up.

When this button is pressed, the system will be in a state of not ready (the status field will show "Not ready"). For absolute safety, the driving power in the electrical control box will be disconnected. It must be confirmed that the cause of malfunction has been resolved before clearing the emergency stop. After clearing the emergency stop, the reference point return operation should be executed again in order to ensure that the coordinate position is correct.

Please note: As soon as the emergency stop is pressed, reference point return must be performed after each clear, yet the absolute reference point is not affected by this limitation.

2.8 Program start (CYCLE START) & program pause (FEED HOLD)



Button type Key style

Program start (CYCLE START)

After program input, the operation mode should be switched to memory mode (MEM) or manual data input mode (MDI) before pressing the program start key to execute the program. The indicator light will be turned on while the program is executing. There are several uses for the program start (CYCLE START) key as shown below:

a. Automatic execution under memory mode (MEM)

When a program is selected, the program start key can be pressed in memory mode to execute the program. During program execution, the program indicator light will remain on until program execution has ended. All three axis must return to their reference point before program execution starts. If not, the reference point requirement on the user function settings screen can be switched to Off for the program to be executed without returning to the reference point.

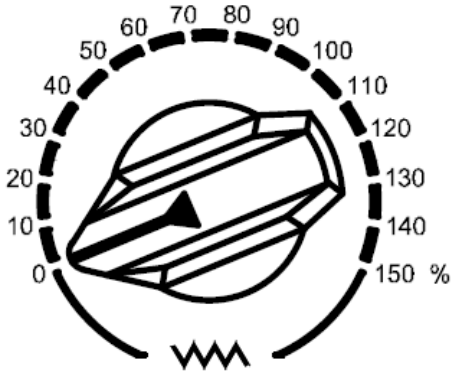
b. Automatic execution under manual data input mode (MDI)

Under manual data input mode, users can enter single block program commands such as G91 G01 X100. Z100., followed by pressing the program start (CYCLE START) key to execute the block command. The objective of this execution mode is different from memory mode; it is usually used for testing certain functions. During execution, the indicator will remain turned on until execution has ended.

Program pause (FEED HOLD)

Press this key to pause program execution. During the pause period, the FEED HOLD indicator will be turned on. Miscellaneous function (M), spindle function (S), and tool function (T) will maintain their current status. Press the program start key again to resume program execution.

2.9 Feed rate adjustment

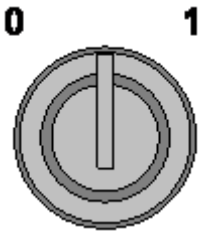


Knob style Key style

Feed rate (FEEDRATE):

When the servo axis is feeding via the command G01…F… under memory (MEM) mode or manual data input (MDI) mode, the actual feed rate can be adjusted in stages via this knob (key) within a range between 0% and 150%. For example, the designation of F100 indicates that the feed rate is 100 mm/min, but if the knob is adjusted to 50%, the actual feed rate will be only 50 mm/min. On many machines, this knob (key) can be used for adjusting the servo feed rate under manual continuous feeding (JOG) mode. When dry run is valid, feed rate can be adjusted via this knob (key).

2.10 Program protection lock

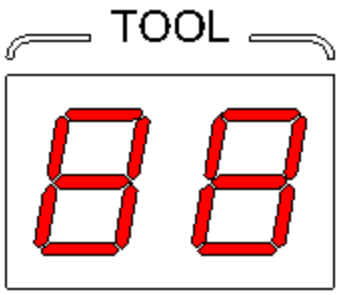


Users can lock program edit mode via the program protection lock. A locked program can only be opened by file explorer; it cannot be modified.

0: Release the program protection lock

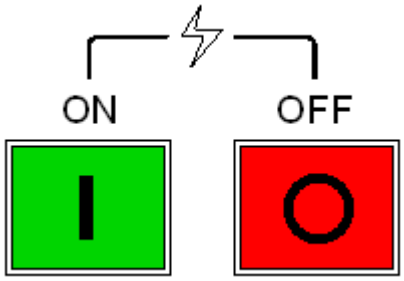
1: Activate the program protection lock

2.11 Seven-segment display of tool number



Users can find out which tool number is currently in use (spindle tool number) via the seven-segment display of tool number.

2.12 Power on/off



When users press the power on key, the CNC controller will be turned on. On the contrary, when the power off key is pressed, the CNC controller will delay its shut down after the servo is completely discharged.