FINGER CNC

Open Designs Customize The Future

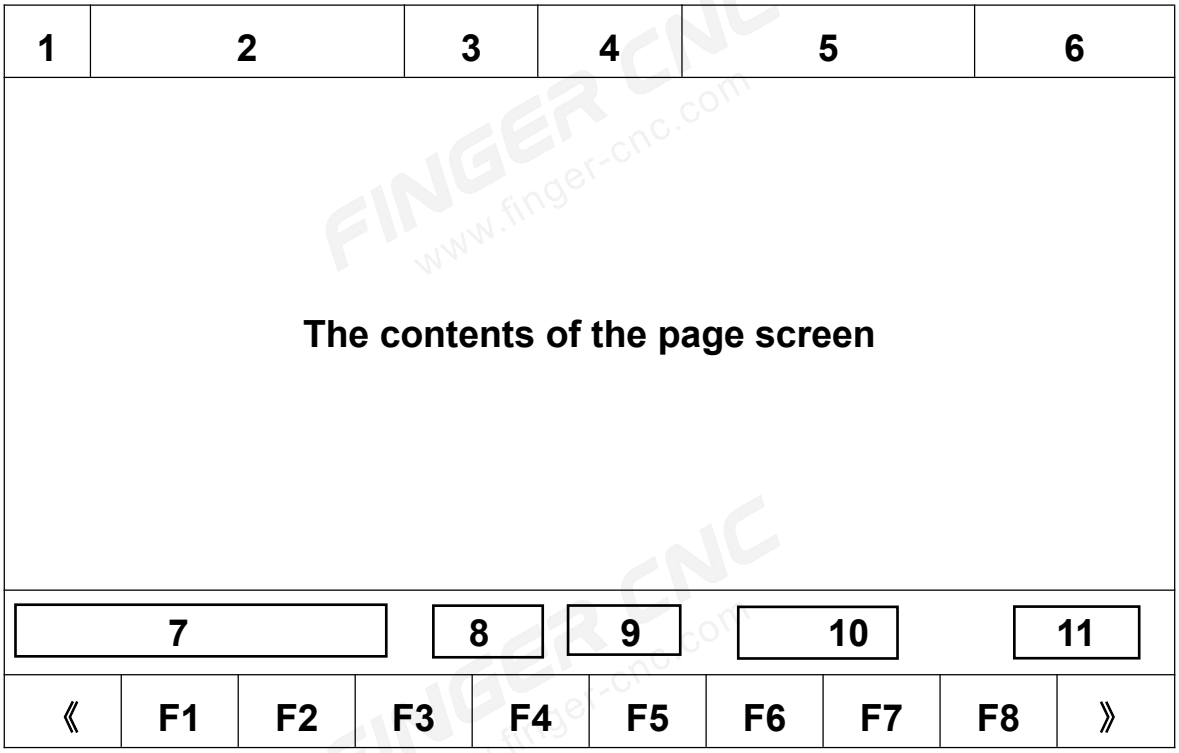
B Series Milling Machine Operation Manual

Version No：F202412MO-EN

Guangzhou Finger Technology Co., Ltd

CONTENTS

Part 1. Function Keys and System Overview



1.1 Main Screen Introduction

- **Screen Component Description**

1. The current working coordinate system
2. The name of the program currently being executed and the name of the subprogram file to which you are redirected
3. The line number of the currently executed program and the line number of the subprogram that you are redirecting
4. Screen Title
5. Date and Time
6. Operation Permission

7. Date Input
8. Current Status
9. Mode
10. Auto Machining Status
11. Alarms
12. F Function Keys

1.2 Machine Position

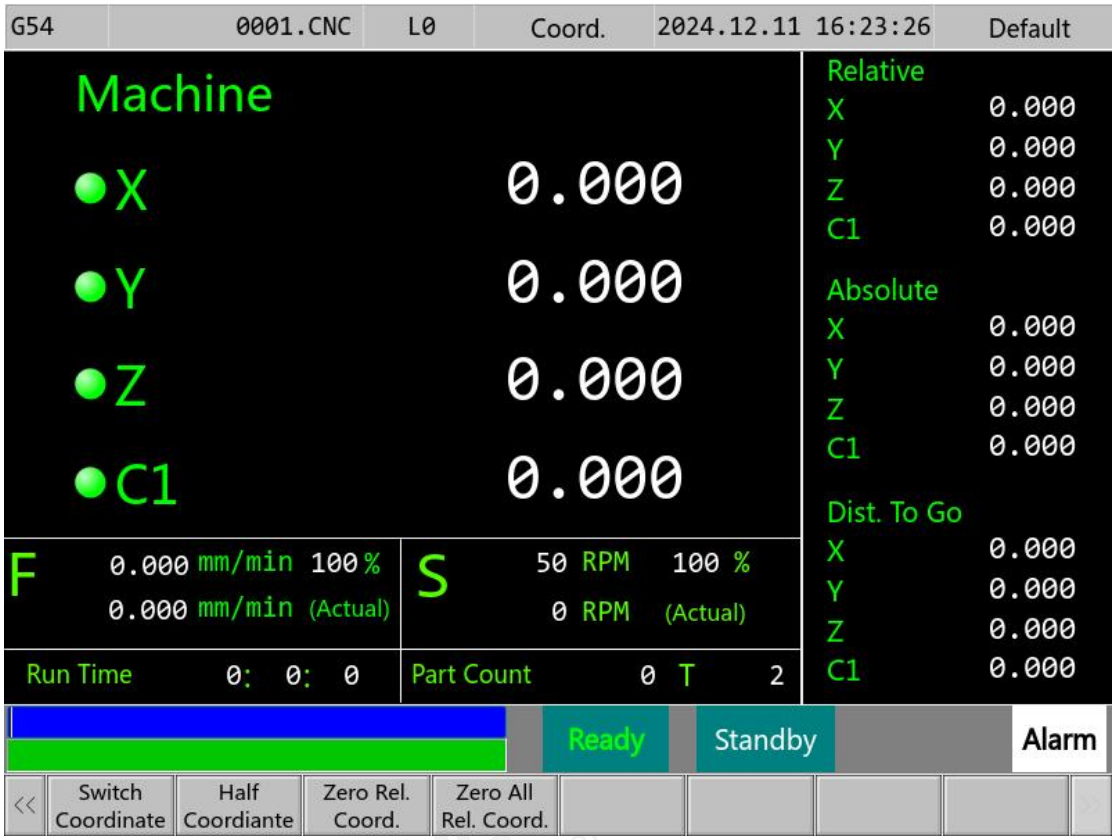
- **Operating Path**

On any page, press the system shortcut key [POS] to enter the "Machine Coordinates" page.

- **Explanation**

1. Operate the coordinates of the current machine;
2. Display commonly used processing information.

1.2.1 Screen Introduction



1.2.1.1 Coordinate Display

1. This screen displays 4 coordinate types at the same time;
2. You can use the [Switch Coordinate] button to switch the coordinate type (Mechanical coordinates, Relative coordinates, Absolute coordinates, Dist. To Go coordinates) displayed by the current primary coordinates.

1.2.1.2 F(Feedrate)

1. Displays the user-set feedrate.
2. Displays the actual feedrate.
3. Displays the user-set feedrate percentage (F value multiplier).

1.2.1.3 S (Spindle Speed)

1. Displays the user-set spindle speed.
2. Displays the actual feedback speed of the spindle.
3. Displays the user-set spindle speed percentage (spindle multiplier).

1.2.1.4 Run Time

Displays the time when the current program has been executed.

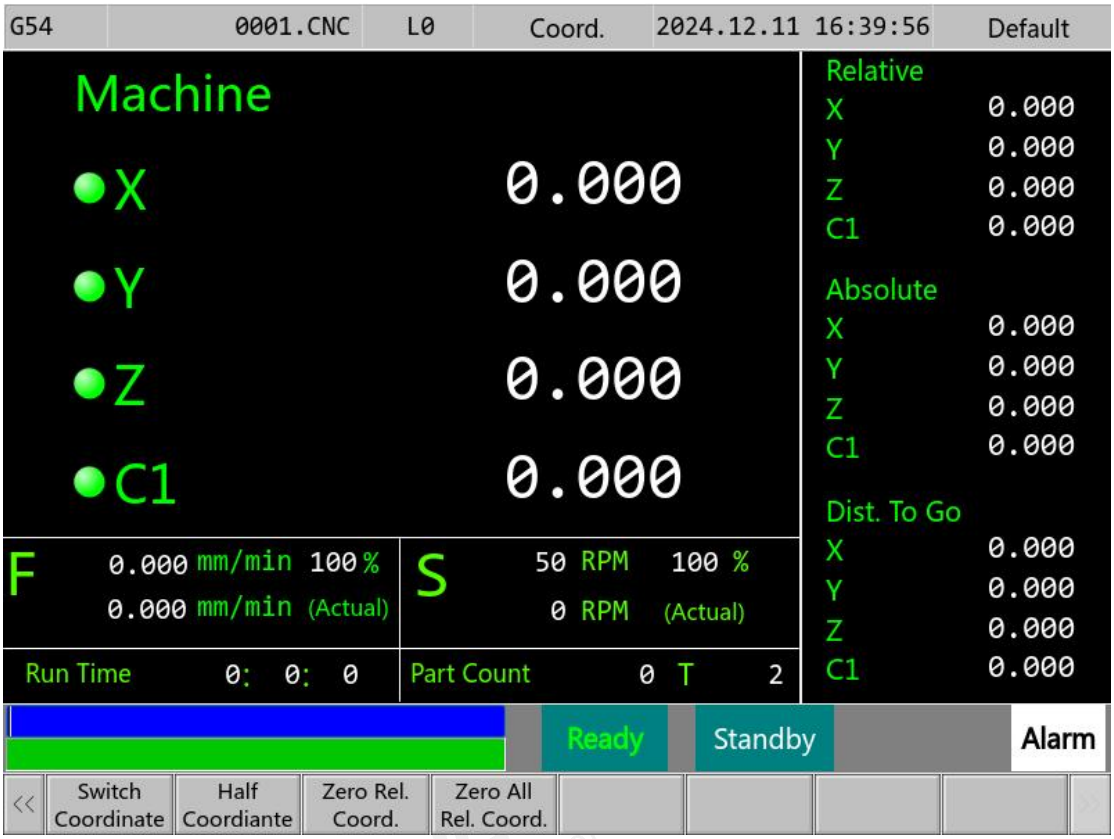
1.2.1.5 Part Count

Displays the number of times the process is added.

1.2.1.6 T(Tool number)

Displays the tool number of the currently called magazine.

1.2.2 Switch coordinates



- **Operating Path**

On the "Machine Coordinates" page→ F1[Coord.] → F1[Switching Coordinate].

- **Explanation**

This button can switch the display order of the machine's coordinate screen and coordinate type.

1.2.3 Half Coordiante

- **Operating Path**

On the "Machine Coordinates" page→ F1[Coord.] → F1[Half Coordinate].

- **Explanation**

1. Divide the corresponding axial relative coordinates by 2;
2. With the F3[Zero Relative Coordinate] function, you can quickly obtain the midpoint coordinates of any two points.

- **Mode of operation**

On "Machine Coordinates" page→ F1[Coord.]→Enter the axis to be set in the data input box →F2[Half coordinate]

- **Examples of operation**

1. The current relative coordinates of the X axis are 10.000;
2. Enter "X" in the data input box (no need to press the input button after entering);
3. Press the F1 [Half Coordinates] button;
4. The X-axis relative coordinates display value changes to 5.000.

1.2.4 Zero Relative coordinates

- **Operating Path**

On "Machine Coordinates" page→F1[Coord.] → F3[Zero Relative Coordinate]

- **Explanation**

You need to fill in the set relative coordinate axis in the input box, and if you do not enter the set value, the relative coordinate will be changed to 0.

- **Mode of operation**

Enter the axial symbol and coordinate value to be set in the input box, and then press F3 [Zero Relative Coordinate]

- **Examples of operation**

1. The current relative coordinates of the X axis are 10.000;
2. Enter "X0" in the screen input box;
3. Press the F3 [Zero Relative Coordinate] button;
4. The X-axis relative coordinate display value will be changed to 0.000.

1.2.5 Zero All Relative coordinates

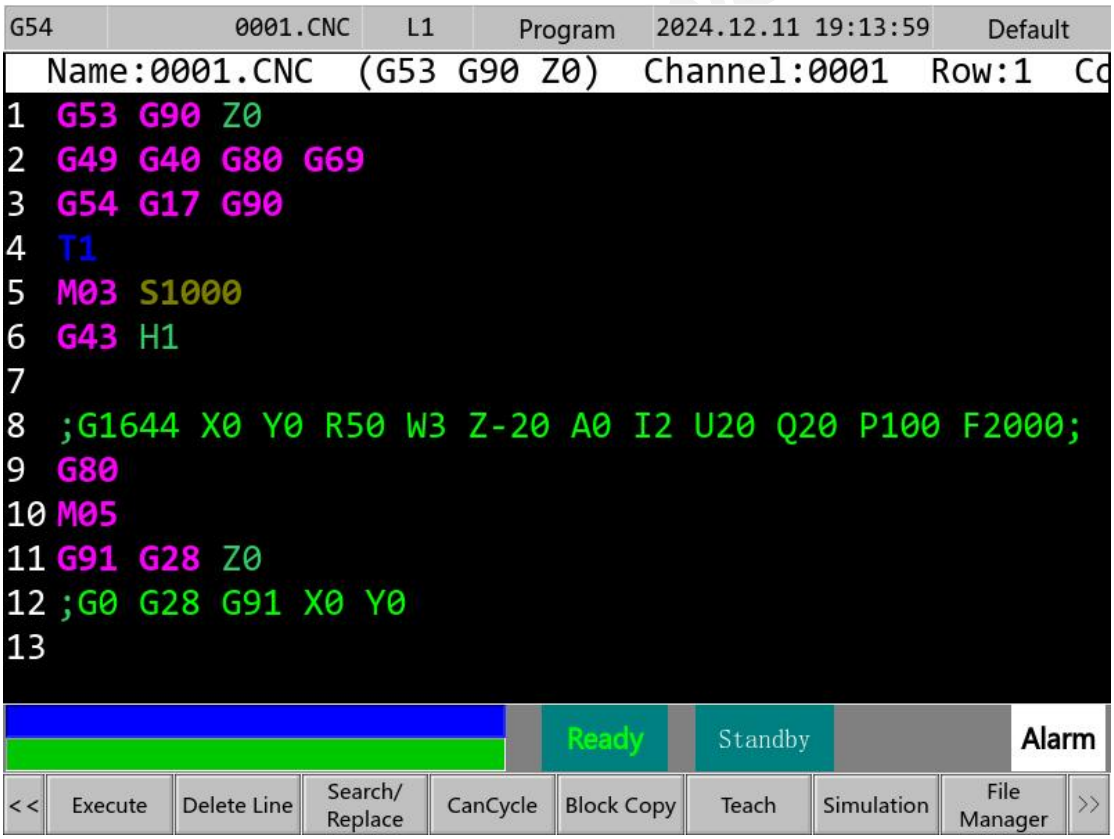
- **Operating Path**

On the "Machine Coordinates" page→F1[Coord.]→F4[Zero All Relative Coordinates].

- **Explanation**

Zeros out the relative coordinates of all axes

1.3 Eidt



- **Operating Path**

1. Path 1: Use the shortcut button [PROG/FILE] on the system panel to switch the page to the "Program Editing" page;
2. Path 2: "Machine Coordinates" page→ F2 [Edit] → "Program Editing" page;
3. Path 3: "File Management" page→ F8 [Edit] → "Program Editing" page.

- **Explanation**

It is used for editing operations on additive projects.

- **Operating instructions**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
2. Use [《》] to turn pages up and down;
3. Use [HOME/END] to quickly switch the cursor to the beginning and end of the row;

4. Use the shortcut button [PROG/FILE] on the system panel to quickly switch between the "Program Editing" and "File Management" pages.

1.3.1 Execute

- **Operating Path**

On the "Programming Editing" page→ F1 [Execute].

- **Explanation**

Use this button to designate the program in editing as an additive and switch the page to the "Monitor" page.

- **Note**

If the program is being executed, this button will not work.

1.3.2 Delete Line

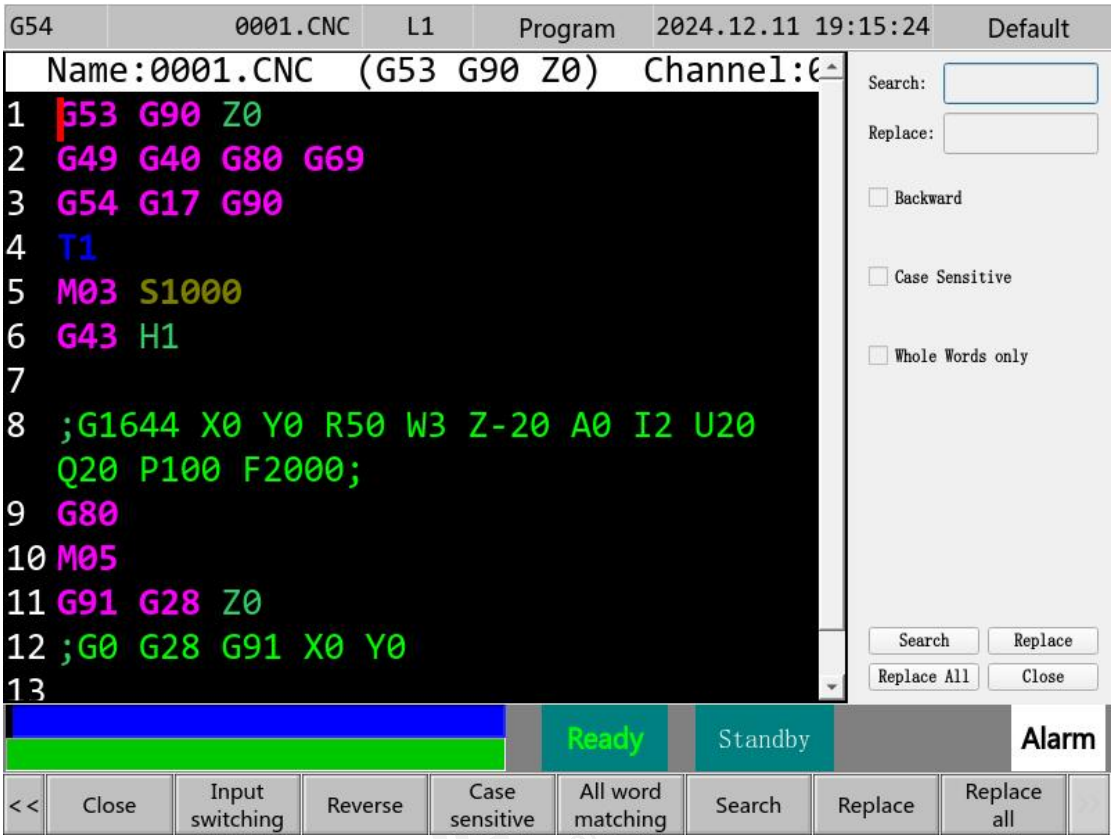
- **Operating Path**

On the "Program Editing" page→ F2 [Delete Line].

- **Explanation**

Deletes the cursor's row.

1.3.3 Search/Replace



- **Operating Path**

On the “Program Editing” page→ F3 [Search/Replace].

- **Explanation**

1. This button can search for characters or replace characters on the currently edited program content;
2. After pressing the [Search/Replace] button, the right sidebar and toggle button group will pop up, you can enter the characters you want to search in the " Search " input box, and enter the replaced characters in the "Replace" input box.

- **Operating instructions**

Use the [Input Switching] button on the current page to make the cursor switch between the "Search" and "Replace" input boxes.

1.3.3.1 Close

- **Operating Path**

On the “Program Editing” page→ F3 [Search/Replace] → F1 [Close].

- **Explanation**

When the “Search/Replace” function is no longer needed, press this key to close the right sidebar and return to the button group.

1.3.3.2 Input Search

- **Operating Path**

On the "Program Editing" page→ F3 [Search/Replace] → F2 [Input Switching].

- **Explanation**

You can make the cursor switch between the "Search" and "Replace" input boxes.

1.3.3.3 Reverse

- **Operating Path**

On the “Programming Edit” page→ F3 [Search/Replace] → F3 [Reverse].

- **Explanation**

When Searchs/replace, this button selects the direction of the search.

- **Operating instructions**

Press this button, the "Reverse Search" option in the right sidebar will be selected, at this time, press the "Search" or "Replace" button, and the direction of the search character will be found or replaced above the current cursor position. If it is not selected, it is used to search or replace it.

1.3.3.4 Case sensitive

- **Operating Path**

On the “Program Editing” page→ F3 [Search/Replace] → F4 [Case sensitive].

- **Explanation**

When searching/replace, press this button, and when searching for characters, the case of the letters is searched according to "Search" to enter the case of the letters.

- **Operating instructions**

Press this button, and the "Case Sensitive" option in the right sidebar will be displayed as selected, and the function will take effect.

1.3.3.5 All word matching

- **Operating Path**

On the "Program Editing" page→ F3 [Search/Replace] → F5 [All word matching].

- **Explanation**

When searching/replaceing, this key can precisely find characters with the same content as the "Search" input.

- **Operating instructions**

Press this button, and the "All Word Matching" option in the right sidebar will show that it is selected, and the function will take effect.

1.3.3.6 Search

- **Operating Path**

On the “Program Editing” page→ F3 [Search/Replace] → F6 [Search].

- **Explanation**

Search for the characters entered in the "Search" input box in the code editor.

1.3.3.7 Replace

- **Operating Path**

On the “Program Editing” page→ F3 [Search/Replace] → F7 [Replace].

- **Explanation**

Search for the characters entered in the "Search" input box in the code editor and replace them with the characters in the "Replace" input box. (Press this button once to replace one).

1.3.3.8 Replace all

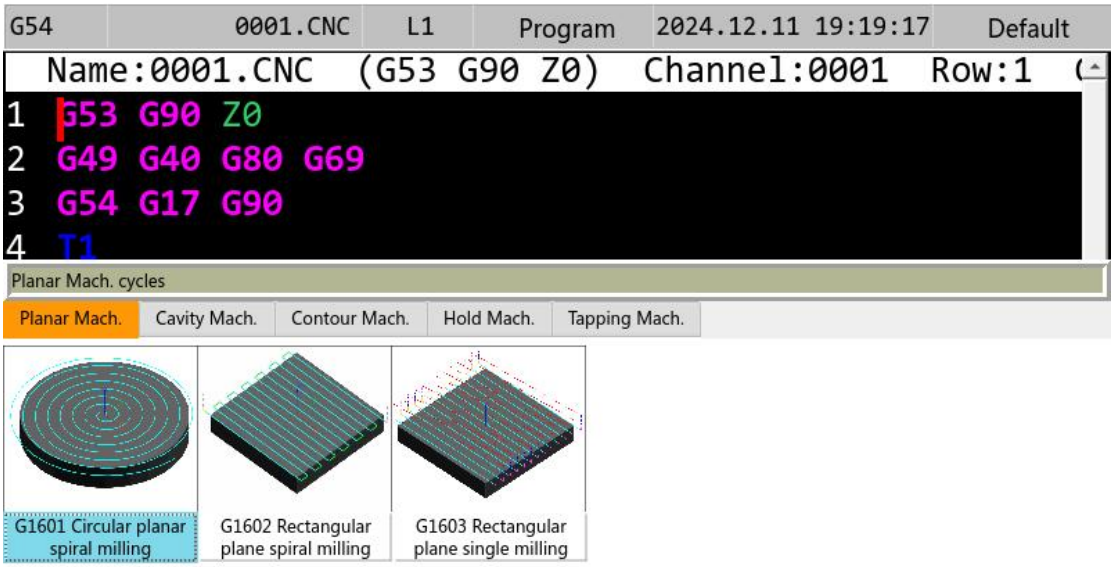
- **Operating Path**

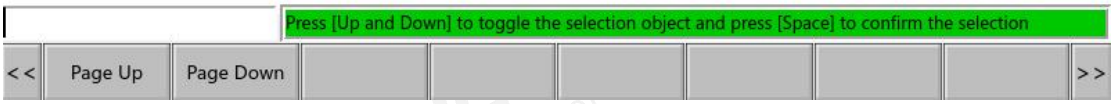
On the “Program Editing” page→ F3 [Search/Replace] → F8 [Replace All].

- **Explanation**

Search for the characters entered in the "Search" input box in the code editor and replace all the characters in the "Replace" input box.

1.3.4 CanCycle





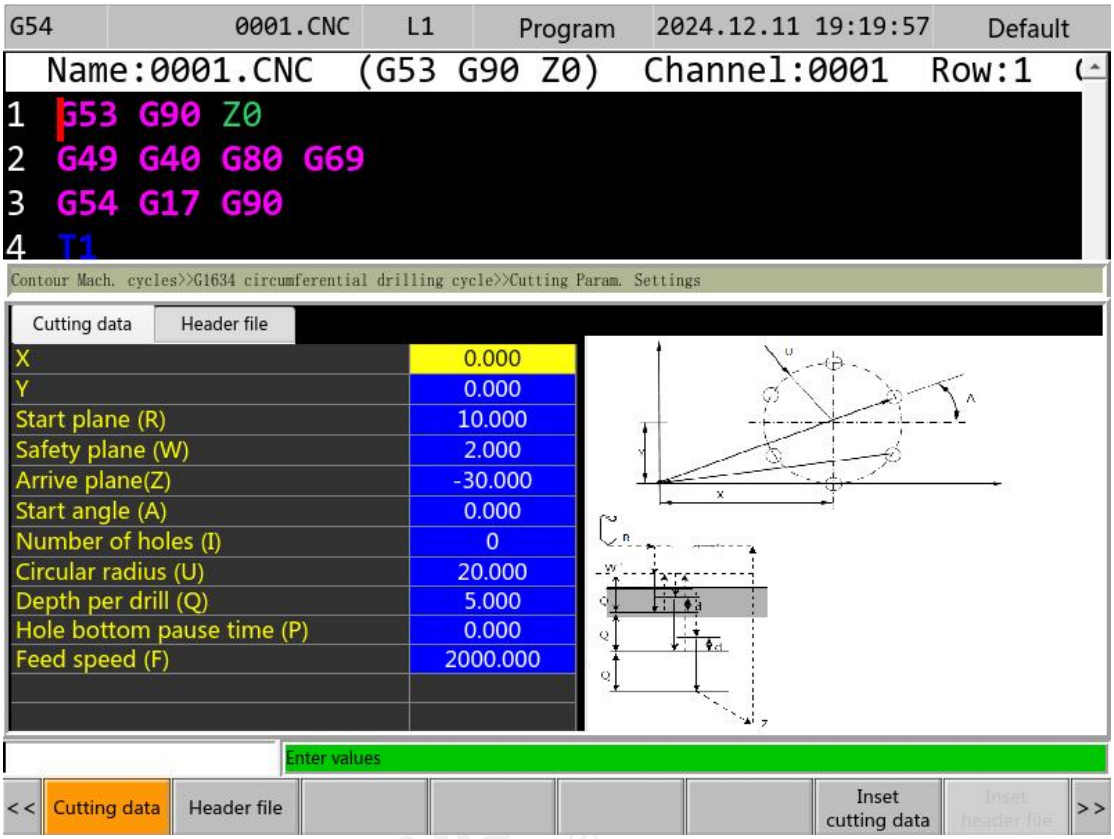
- **Operating Path**

On the “Program Editing” page→ F4 [CanCycle] → F1 [Insert Cycle].

- **Explanation**

1. Conversational editing and insertion of some cyclic milling commands;
2. At present, it provides dialogue editing and insertion of five cycle processing processes, including plane machining cycle, cavity machining cycle, contour machining cycle, hole machining cycle, and tapping processing cycle.

1.3.4.1 Inset Cycle



- **Operating Path**

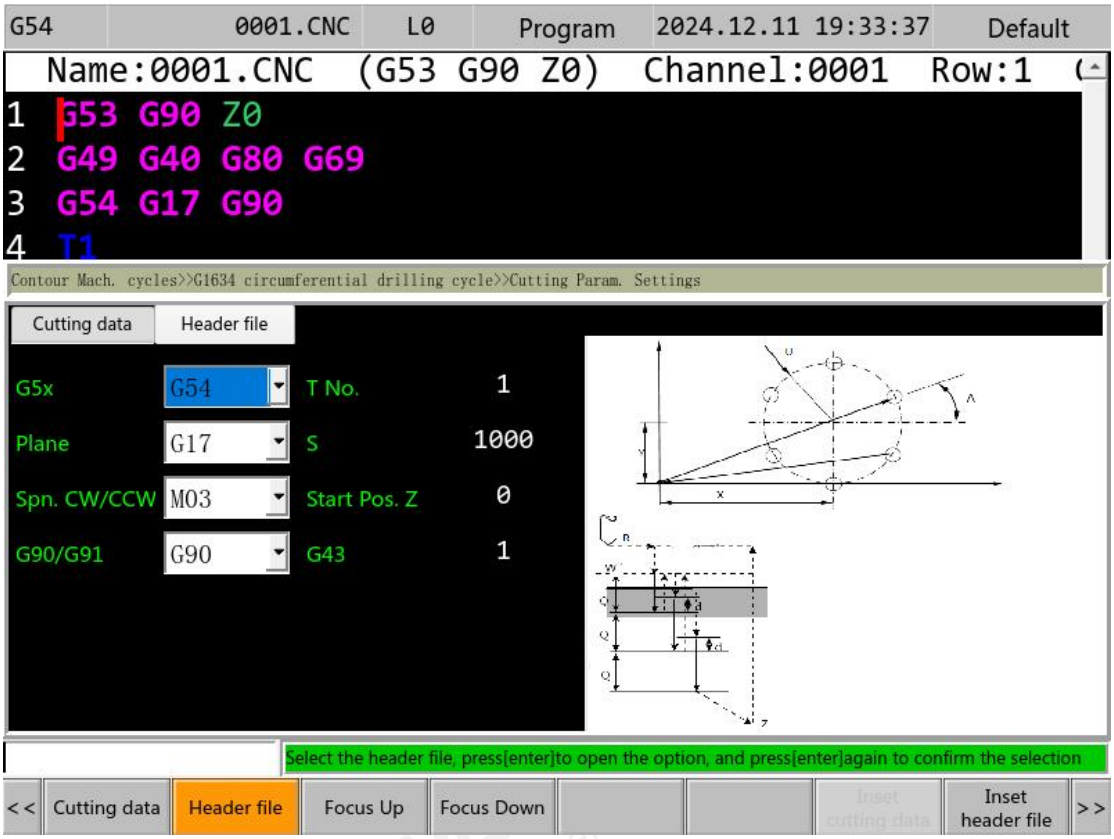
On the "Program Editing" page→ F4 [CanCycle] → F1 [Insert cycle] → move the cursor to the options and press the [Spacebar] key to enter.

- **Explanation**

1. Fill in the necessary parameters for processing according to the annotation of the drawing;

2. After the parameters are set, F7 [Insert cutting date] will insert the looper into the file of the current code editor.

1.3.4.2 Inset Header file



- **Operating Path**

On the “CanCycle” dialog page→ F2 [Header File].

- **Explanation**

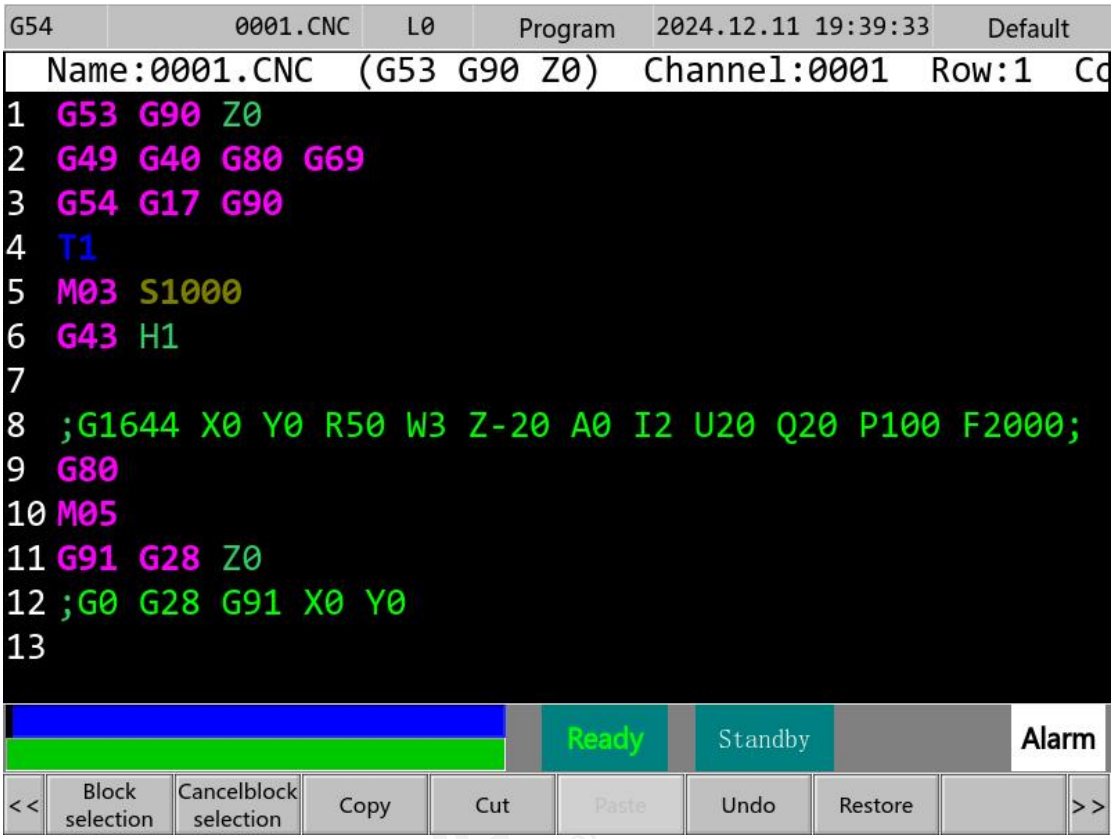
It includes workpiece coordinate system selection, machining plane, spindle rotation direction, incremental/absolute programming, spindle tool number, spindle speed, initial position, and tool complement number.

- **Operating instructions**

1. Move the cursor to the item to be modified through F3 [Focus Up] and F4 [Focus Down] to be modified;

2. After the parameters are set, F8 [Insert Header File] is to insert the header file into the file of the current code editor.

1.3.5 Block Copy



- **Operating Path**

On the "Program Editing" page→ F5 [Block Copy].

- **Explanation**

1. Press the [Block selection] button to select the block you want to select through [↑] [↓] [←] [→].

2. The selected block can be copied, cut, pasted, etc.

1.3.5.1 Cancel Block Selection

- **Operating Path**

On the "Program Editing" page→ [>>] → F2 [Cancelblock selection].

- **Explanation**

Used for deselected blocks.

1.3.5.2 Block Copy

- **Operating Path**

On the “Program Editing” page→ F5 [Block Copy] → F3 [Copy].

- **Explanation**

1. Copy the selected block in the current program;
2. Replication requires block selection to take effect.

1.3.5.3 Block Cut

- **Operating Path**

On the "Program Editing" page→ F5 [Block Copy] → F4 [Cut].

- **Explanation**

1. Cut the selected block in the program.
2. Clipping can only take effect when there is a block selection.

1.3.5.4 Block Paste

- **Operating Path**

On the “Program Editing” page→ F5 [Block Copy] → F5 [Paste].

- **Explanation**

Paste the content of the block executed by Cut or Copy into the row where the cursor is currently located.

1.3.5.5 Undo

- **Operating Path**

On the “Program Editing” page→ F5 [Block Copy] → F6 [Undo].

- **Explanation**

Undo the content of the previous edit.

1.3.5.6 Restore

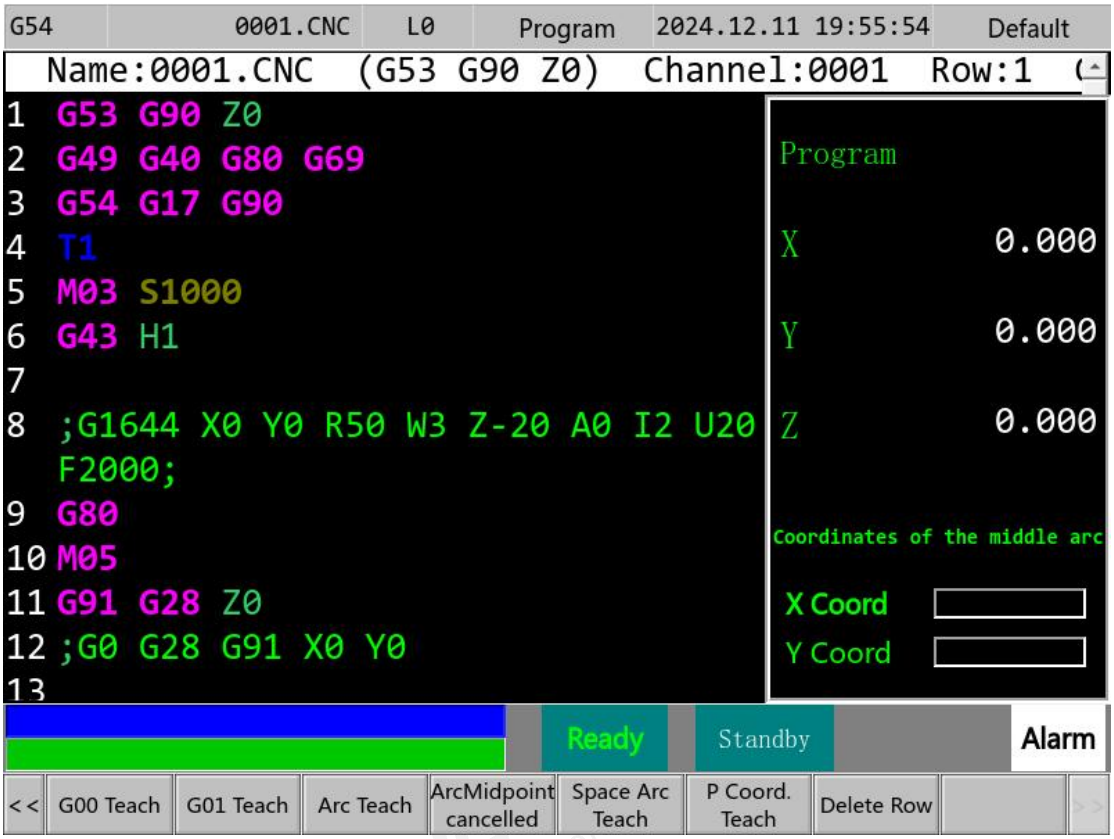
- **Operating Path**

On the “Program Editing” page→ F5 [Block Copy] → F7 [Restore].

- **Explanation**

Restore undone content.

1.3.6 Teach



- **Operating Path**

「程式编辑」页面→F6【教导】。On the “Program Editing” page → F6 [Teach].

- **Explanation**

You can use [Handwheel]/[Manual]/[Inch] to move the machine to the specified coordinates, and then use [Teach] to teach the coordinate values of the current system to the CNC program, saving the trouble of manual programming.

1.3.6.1 G00 Teach

- **Operating Path**

On the "Program Editing" page→ F6 [Teach] → F1 [G00 Teach].

- **Explanation**

Insert the coordinate value of the current machine into the line where the cursor of the current editing cursor is located in the form of G00.

1.3.6.2 G01 Teach

- **Operating Path**

On the "Program Editing" page→ F6 [Teach] → F2 [G01 Teach].

- **Explanation**

Insert the coordinate value of the current machine into the line where the cursor of the current editing plus project is located in the form of G01.

1.3.6.3 Arc Teach

- **Operating Path**

On the "Program Editing" page→ F6 [Teach] → F3 [Arc Teach].

- **Explanation**

After continuously moving the middle point and end point of the arc teaching of the machine according to the [G00 Teach], [G01 Teach], and [P Coord. Teach] carried out in the previous part as the starting point of the arc cutting instruction, the [G02 Clockwise Circular Cutting] or [G03 Clockwise Circular Cutting] will be automatically added to the line where the cursor of the current editing and engineering formula is added.

- **Operating instructions**

1. Carry out [G00 teach], [G01 teach], [P Coord. teach] arbitrary teaching, and locate the starting point of arc cutting teaching;
2. Move the machine to the middle point of arc cutting;
3. Click [Arc Teach];
4. The current absolute coordinate will be set to the value of the middle point of the arc cutting;
5. Move the machine to the end point of arc cutting;
6. Click [Arc Teach] again;

7. The controller will automatically calculate the relationship between the middle point and the end point, and judge it as [G02 clockwise circular cutting] or [G03 counterclockwise circular cutting];
8. The result of the calculation will be inserted into the cursor line of the program you are currently editing.

1.3.6.4 ArcMidpoint cancelled

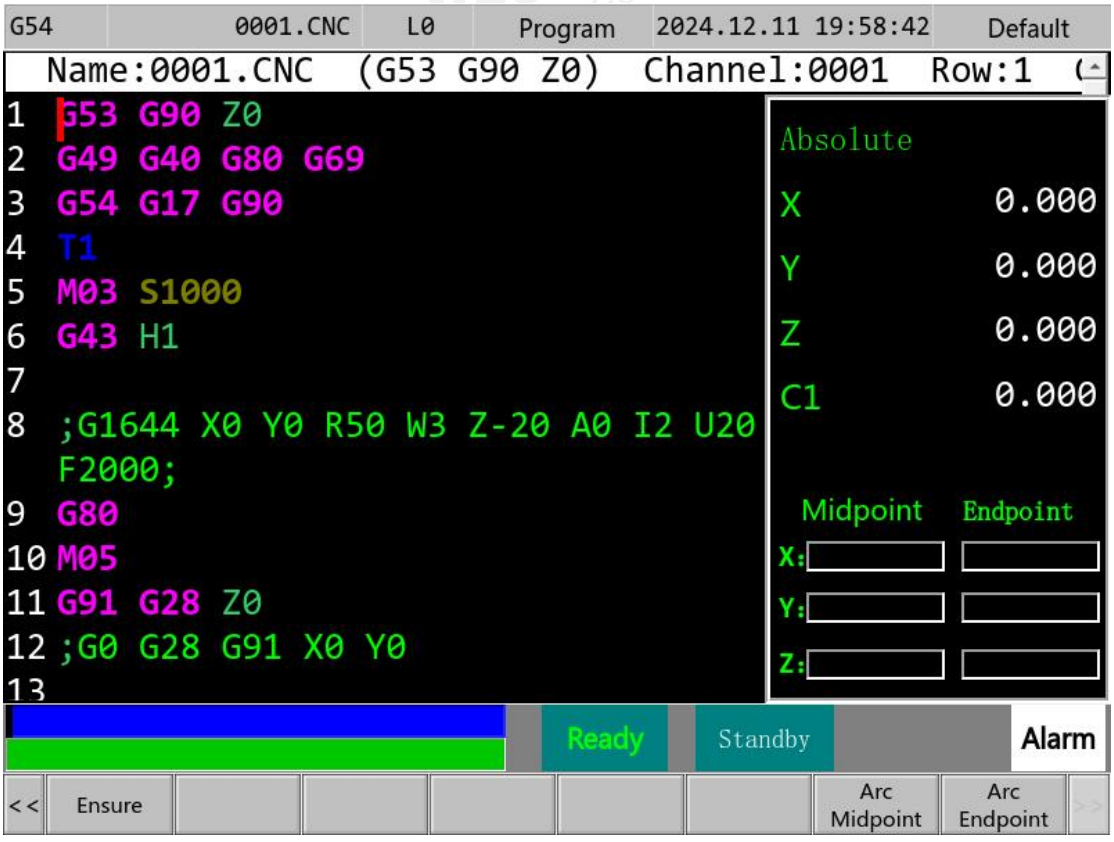
- **Operating Path**

On the "Program Editing" page→ F6 [Teach] → F4 [ArcMidpiont Cancelled]

- **Explanation**

1. Be clear about the value of the middle point of the arc that has been set;
2. If the middle point of the arc has not been set, this function button is invalid.

1.3.6.5 Space Arc Teach



- **Operating Path**

On the "Program Editing" page→ F6 [Teach] → F5 [Space Arc Teach].

- **Explanation**

1. Move the XYZ axis to the middle point of the arc, and press F7 [Arc Midpoint] to get the middle point data;

2. Move the XYZ axis to the end of the arc and press F8 [Arc Endpoint] to get the end point data;

3. The result of F1 [OK] calculation will be inserted into the cursor line of the currently edited program

1.3.6.6 Point Coordinate Teach

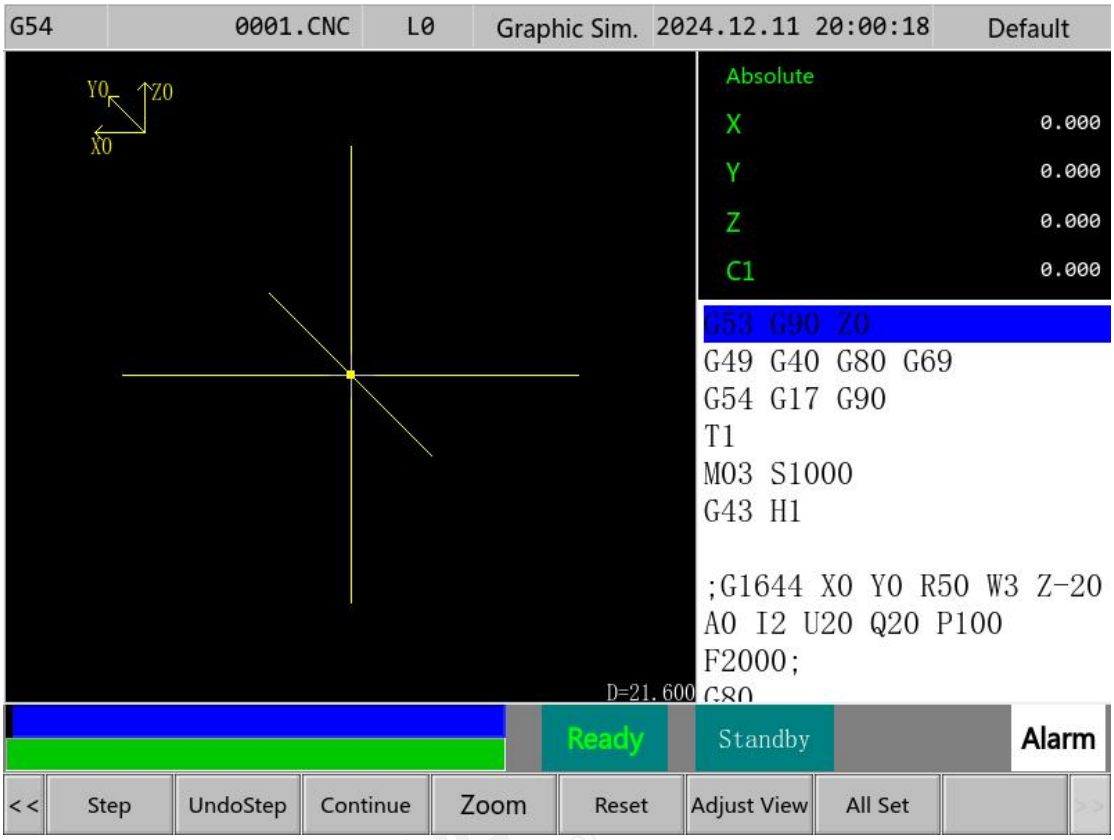
- **Operating Path**

「程式编辑」页面→F6【教导】→F6【点坐标教导】。On the "Program Editing" page→ F6 [Teach] → F6 [P Coord. Teach].

- **Explanation**

Inserts the current absolute coordinates as point coordinates into the line of the cursor in the currently edited additive project.

1.3.7 Simulation



- **Operating Path**

On the "Programming Editing" page→ F7 [Simulation].

- **Explanation**

1. You can view the machining path in advance for the currently edited program;
2. It has the function of checking whether the program is wrong.

1.3.7.1 Step

- **Operating Path**

On the “Graphics Simulation” page→ F1 [Step].

- **Explanation**

A single-section executor depicts the trajectory of a graphical simulation.

1.3.7.2 UndoStep

- **Operating Path**

On the “Graphics Simulation” page→ F2 [UndoStep].

- **Explanation**

Use with [Step] to rewind a single section.

1.3.7.3 Continue

- **Operating Path**

On the “Graphics Simulation” page→ F3 [Continue].

- **Explanation**

Execute all programs to trace the trajectory of the graphical simulation.

1.3.7.4 Zoom

- **Operating Path**

On the “Graphics Simulation” page→ F4 [Continue].

- **Explanation**

You can zoom in on the trajectory of the graphical simulation.

- **Operating instructions**

1. Press the F4 [Zoom] button, pop up the checkbox to select magnification, and move the checkbox to the local position to be enlarged through [↑] [↓] [←] [→].
2. Press the [Enter] key to zoom in on the position in the circle.

1.3.7.5 Reset

- **Operating Path**

On the “Graphics Simulation” page→ F5 [Reset].

- **Explanation**

Use with the [Zoom] function to restore the magnified graphic.

1.3.7.6 Adjust View

- **Operating Path**

On the “Graphics Simulation” page→ F6 [Adjust View].

- **Explanation**

You can move, zoom in, and zoom out of the entire viewport.

1.3.7.7 All Set

- **Operating Path**

On the “Graphics Simulation” page→ F7 [All Set].

- **Explanation**

This function sets settings such as the color of the drawing, the color of the layer, and the color of the background.

1.3.8 File Manager

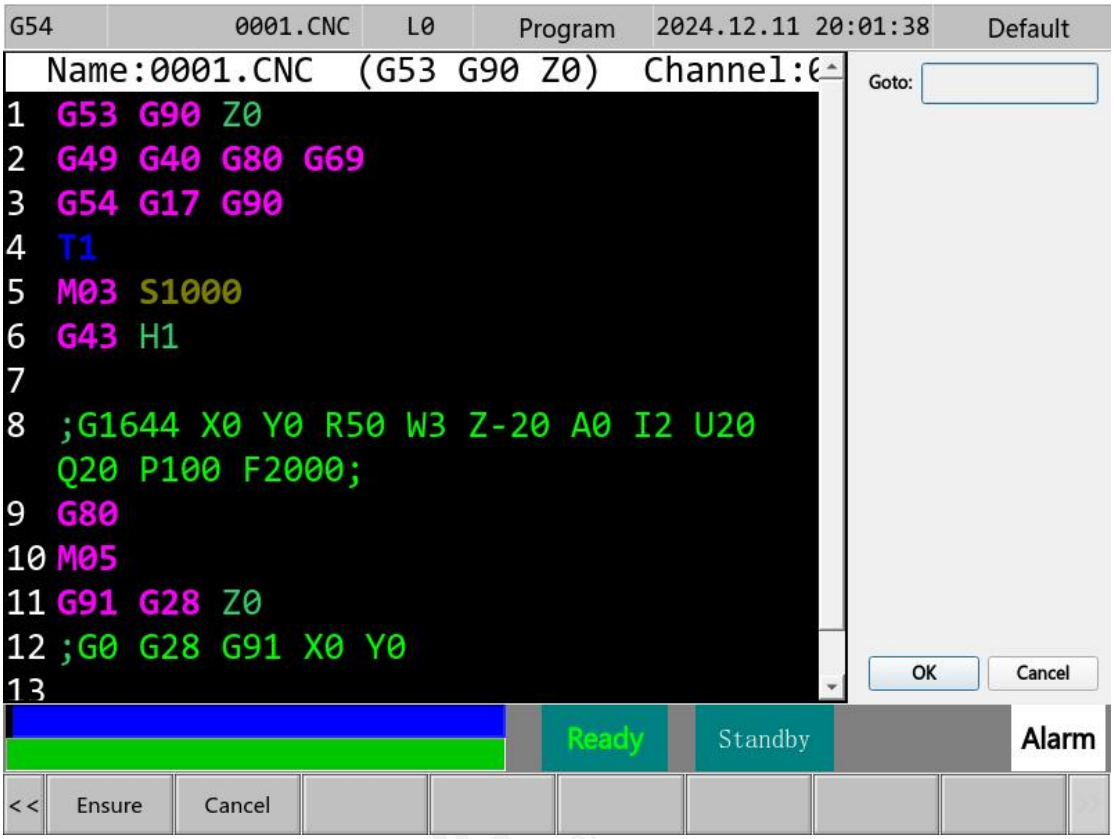
- **Operating Path**

On the "Program Editing" page→ F5 [File Management].

- **Explanation**

For the management of processing documents, please refer to 1.4 "File Management" Function Introduction.

1.3.9 Search Row



- **Operating Path**

On the "Program Editing" page→ [>>] → F1 [Search Row].

- **Explanation**

When the program is long, this function can directly position the cursor to the desired line number.

- **Operating instructions**

1. Press F1 [Search Row] to pop up the input dialog box, and enter the line number to be jumped to in the "Jump to" input area;
2. 2. Press F1 [Ensure], and the cursor will automatically position the specified line number.
3. 3. Press F2 [Cancel] to exit the jump function and close the input dialog box.

1.3.10 From Current Line Start

- **Operating Path**

On the "Program Editing" page→ [>>] → F2 [From Cur. Line Start].

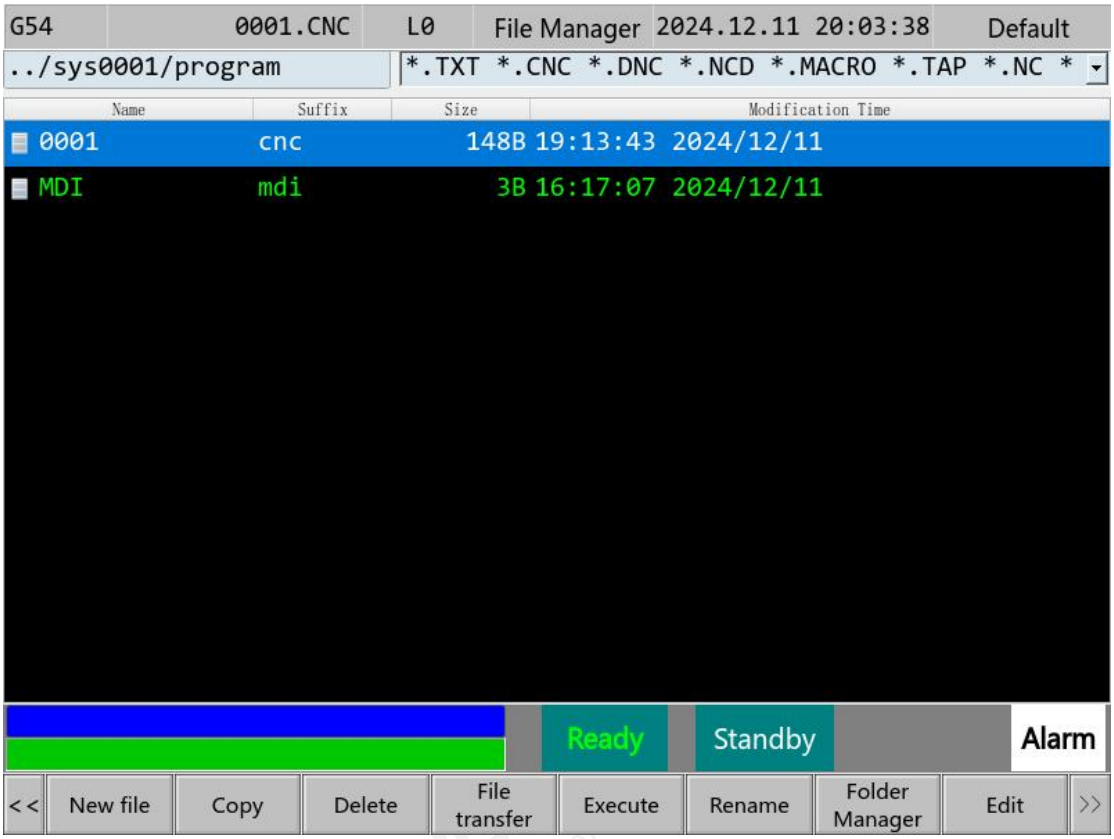
- **Explanation**

1. This button can select the starting position when restarting;
2. This function can only be used in Auto mode;
3. When this function is enabled, it will automatically jump to the processing monitoring interface.

- **Operating instructions**

1. In Auto mode, move the cursor to the line number that needs to be restarted;
2. Press the F2 [From Cur. Line Start] button to automatically switch to the processing monitoring page, and press the Start button to start the program from the line where the cursor is located.

1.4 File Manager



- **Operating Path**

1. Path 1: Use the shortcut button [PROG/File] on the system panel to switch the page to the "File Management" page;
2. Path 2: "Program Editing" page→ F8 [File Management].

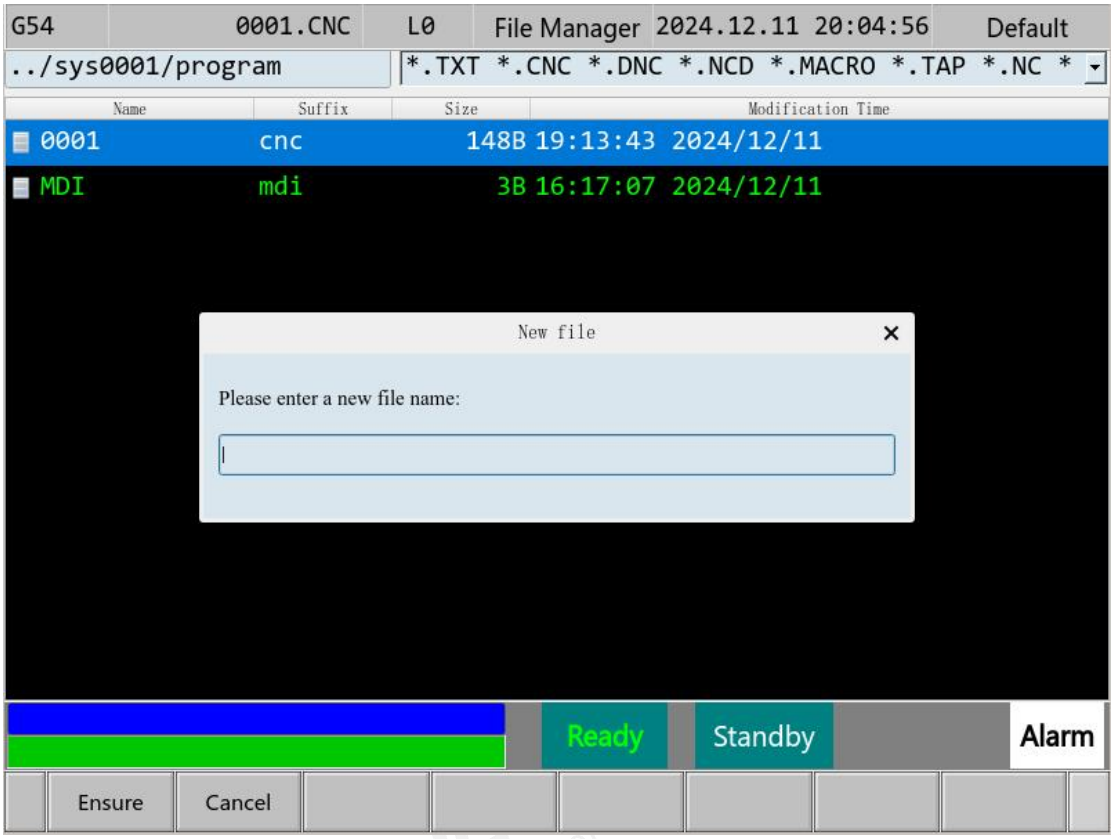
- **Explanation**

It is used to manage documents in the form of additional engineering.

- **Operating instructions**

1. Use the arrow keys [↑] [↓] to move the cursor to select the program document;
2. Use [⩓][⩔] to turn pages up and down;
3. Use the F8[Edit] button or the [Enter] button on the panel to specify the program document pointed by the current cursor as an edit file, and open it for editing.

1.4.1 New file



- **Operating Path**

On the "Program Editing" page→ F8 [File Management] → F1 [New File].

- **Explanation**

Used to create new program files.

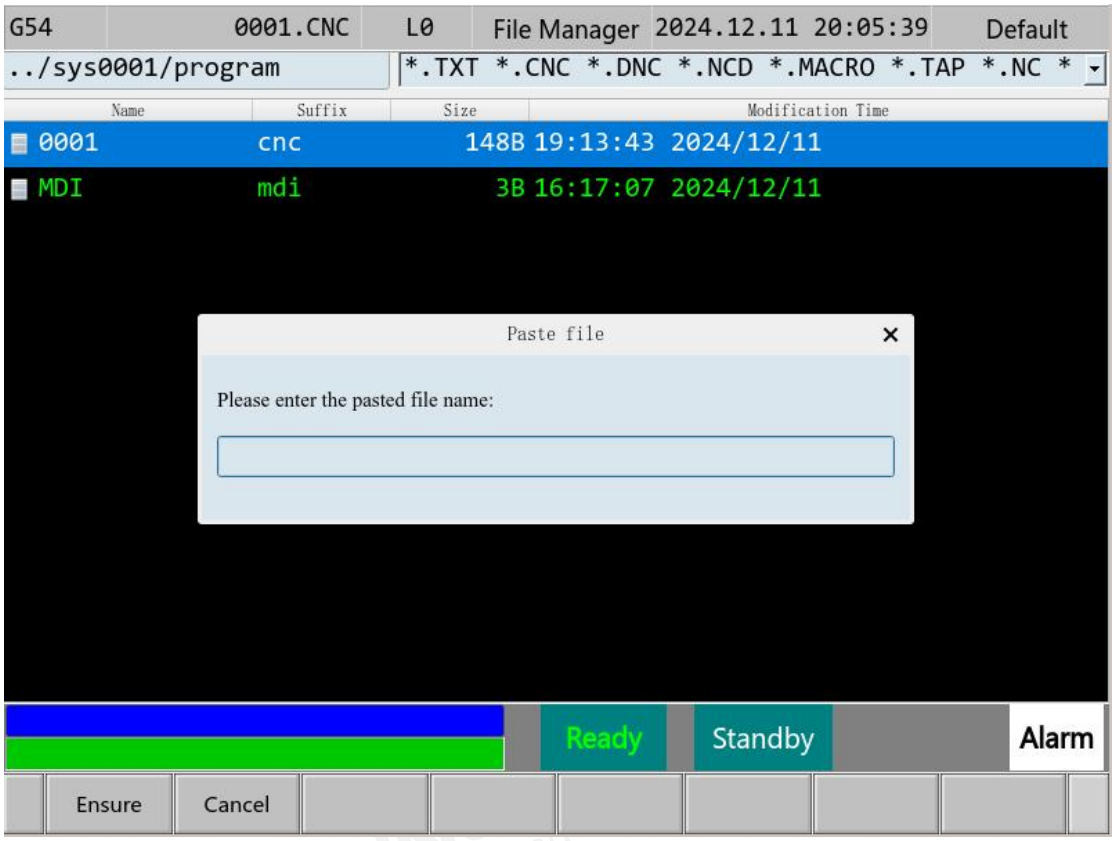
- **Operating instructions**

1. Press the [New File] button to pop up a dialog box, enter the file name of the new file;
2. Press the F1 [Ensure] button to complete the new file.

- **Note**

When the entered file name has no suffix, the suffix defaults to ". CNC”。

1.4.2 Copy file



- **Operating Path**

On the "Program Editing" page→ F8 [File Management] → F2 [Copy].

- **Explanation**

This button can be used to copy the file currently selected by the cursor.

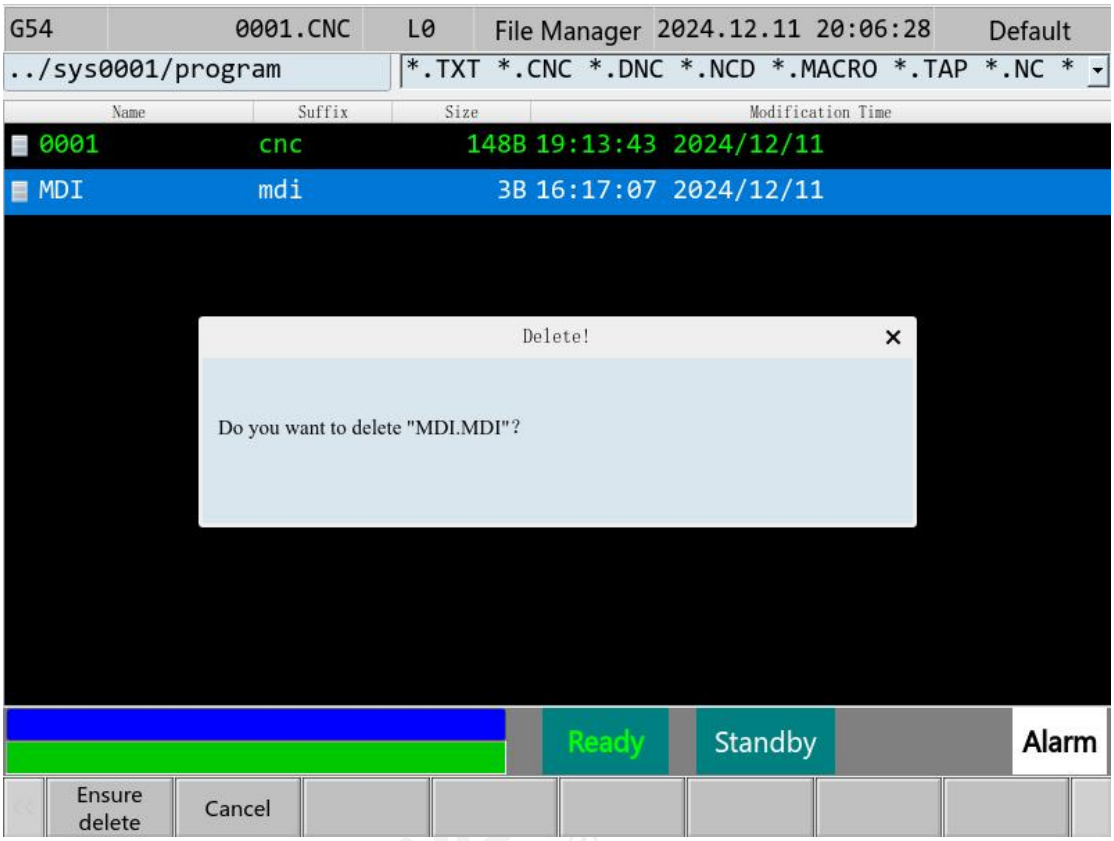
- **Operating instructions**

1. Use the arrow keys [↑] [↓] to move the cursor to select the file you want to copy;
2. Select and press [Copy].
3. In the dialog box that pops up, enter the file name of the new file.

- **Note**

By default, the file is in an unsuffixed format, if you want to open a file with a suffix, such as *. NC, enter *. NC is sufficient.

1.4.3 Delete file



- **Operating Path**

On the “File Management” page→ F3 [Delete].

- **Explanation**

This button is used to delete program documents.

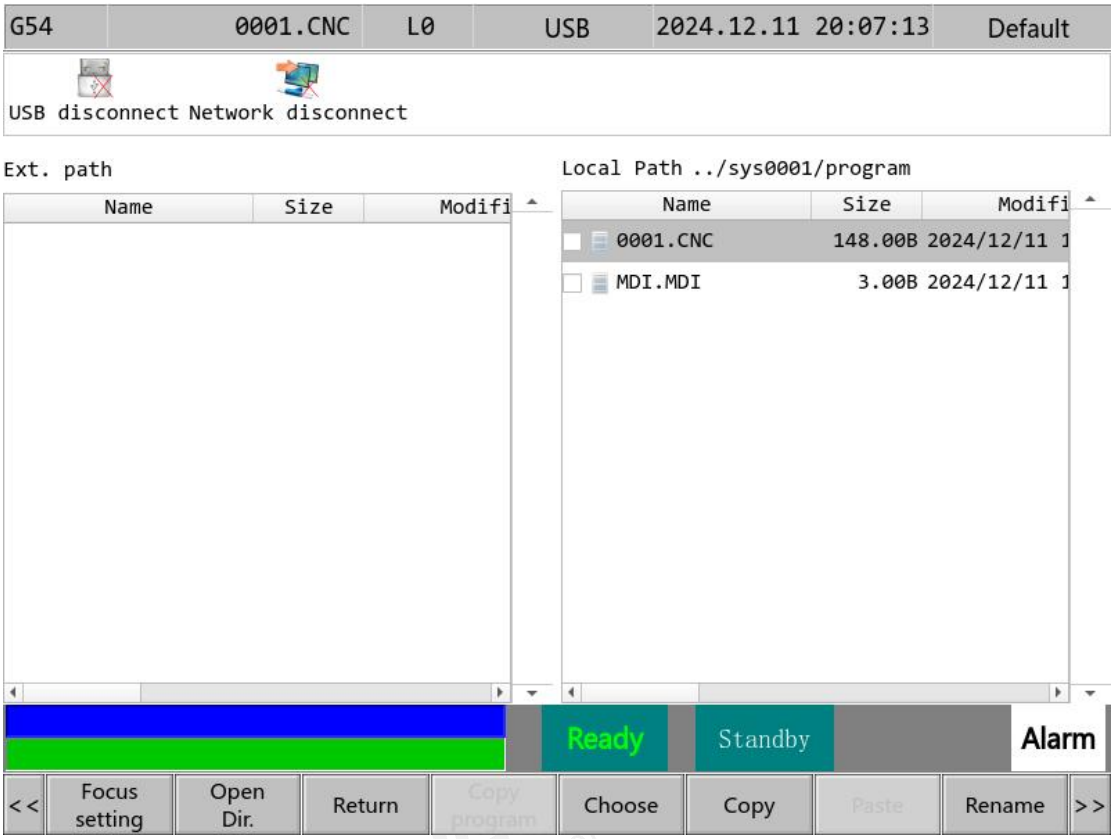
- **Operating instructions**

1. Use the arrow keys [↑] [↓] to move the cursor to select the program document to be deleted, and press the F6 [Delete] button to pop up the second confirmation box;
2. Press F1 [Ensure delete], the deletion is successful, and the second confirmation box will be automatically closed.
3. Press F2 [Cancel] to discard the deletion.

- **Note**

When the cursor selected program document is being edited or loaded for processing, the document cannot be deleted and the deletion fails.

1.4.4 File transfer



- **Operating Path**

On the "Program Editing" page→ F8 [File Management] → F4 [File transfer].

- **Explanation**

This function allows you to transfer and add engineering documents as well as system files via a USB stick.

- **Note**

This function is used for file sharing between the controller and the outside, file input/output, and external devices include USB flash drives, computers, etc.

1.4.4.1 Focus setting

- **Operating Path**

On the “USB” page→ F1 [Focus setting].

- **Explanation**

The focus of the controller file selector and the external device file selector is switched between each other, and the cursor can only be moved when the focus is on the corresponding file selector.

1.4.4.2 Open Directory

- **Operating Path**

On the “USB” page→ F2 [Open Dir.].

- **Explanation**

This button opens the folder in the current file directory.

1.4.4.3 Return

- **Operating Path**

On the “USB” page→ F3 [Return].

- **Explanation**

This button returns to the previous folder.

1.4.4.4 Copy program

- **Operating Path**

On the “USB” page→ F4 [Copy program].

- **Explanation**

Copies the currently selected file.

1.4.4.5 Choose

- **Operating Path**

On the “USB” page→ F5 [Choose].

- **Explanation**

Select the file selected by the current cursor, and you can "copy program" and "paste" the file only when the file is selected.

1.4.4.6 Copy

- **Operating Path**

On the “USB” page→ F6 [Copy].

- **Explanation**

Copies the currently selected file.

1.4.4.7 Paste

- **Operating Path**

On the “USB” page→ F7 [Paste].

- **Explanation**

Paste the file copied to the pasteboard to the directory where the current cursor is located.

1.4.4.8 Rename

- **Operating Path**

On the “USB” page→ F8 [Rename].

- **Explanation**

The duplicate name is the file selected by the current cursor.

1.4.4.9 New Directory

- **Operating Path**

On the “USB” page→[>>]→F1[New Dir.]。

- **Explanation**

Create a new folder in the current file directory.

1.4.4.10 External equipment set

- **Operating Path**

On the “USB” page→[>>]→F2[Ext. Equ. set]。

- **Explanation**

Used to switch external devices, including USB flash drives and computers.

1.4.4.11 Equipment selection

- **Operating Path**

On the “USB” page→[>>]→F3[Equipment selection]。

- **Explanation**

Used to select external equipment.

1.4.4.12 All selection

- **Operating Path**

On the “USB” page→[>>]→F4[All selection]

- **Explanation**

This button selects all files in the current directory.

1.4.4.13 All cancle

- **Operating Path**

On the “USB” page→[>>]→F5[All cancle]。

- **Explanation**

This button unchecks all files.

1.4.4.14 Delete

- **Operating Path**

On the “USB” page→[>>]→F5[Delete]。

- **Explanation**

This button deletes the file selected by the current cursor.

1.4.5 Execute

- **Operating Path**

On the “File Management” page→ F5 [Execute].

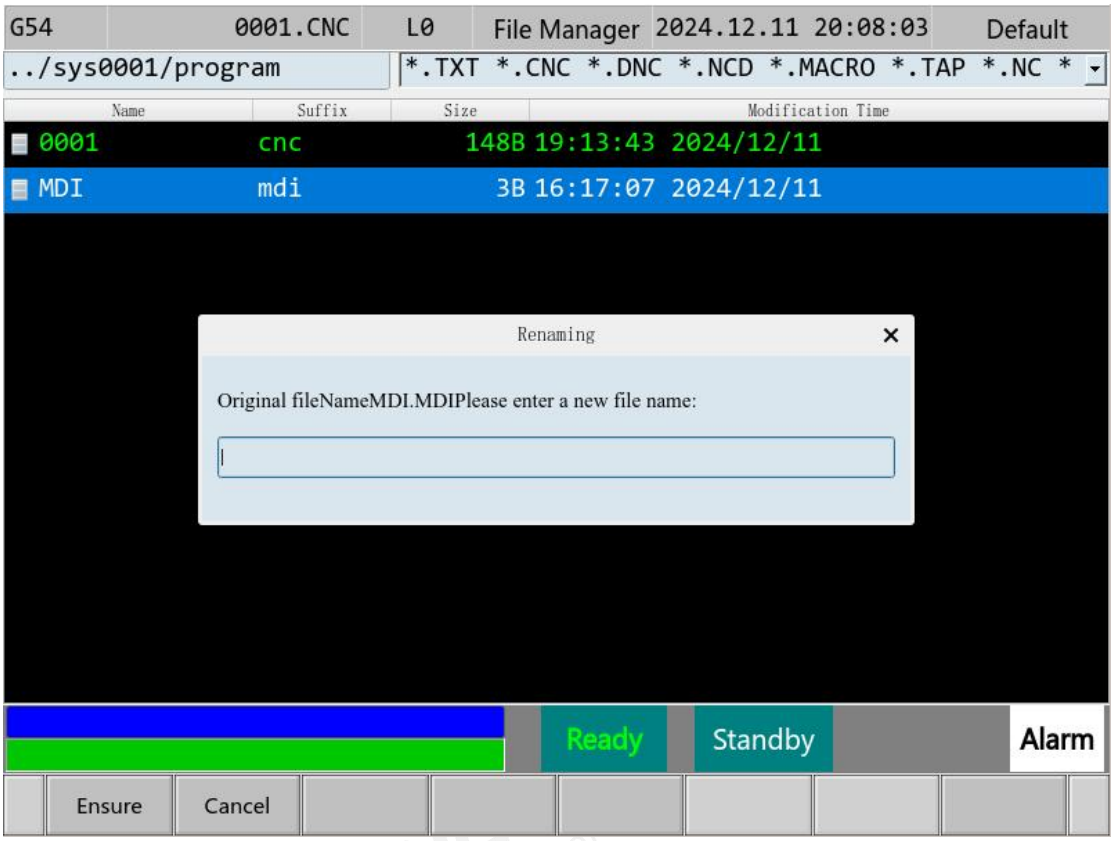
- **Explanation**

Use this button to specify the program document selected by the current cursor as an additive and switch the page to the "Process Monitoring" page.

- **Note**

If the program is being executed, this button will not work.

1.4.6 Rename



- **Operating Path**

On the “File Management” page→ F6 [Rename].

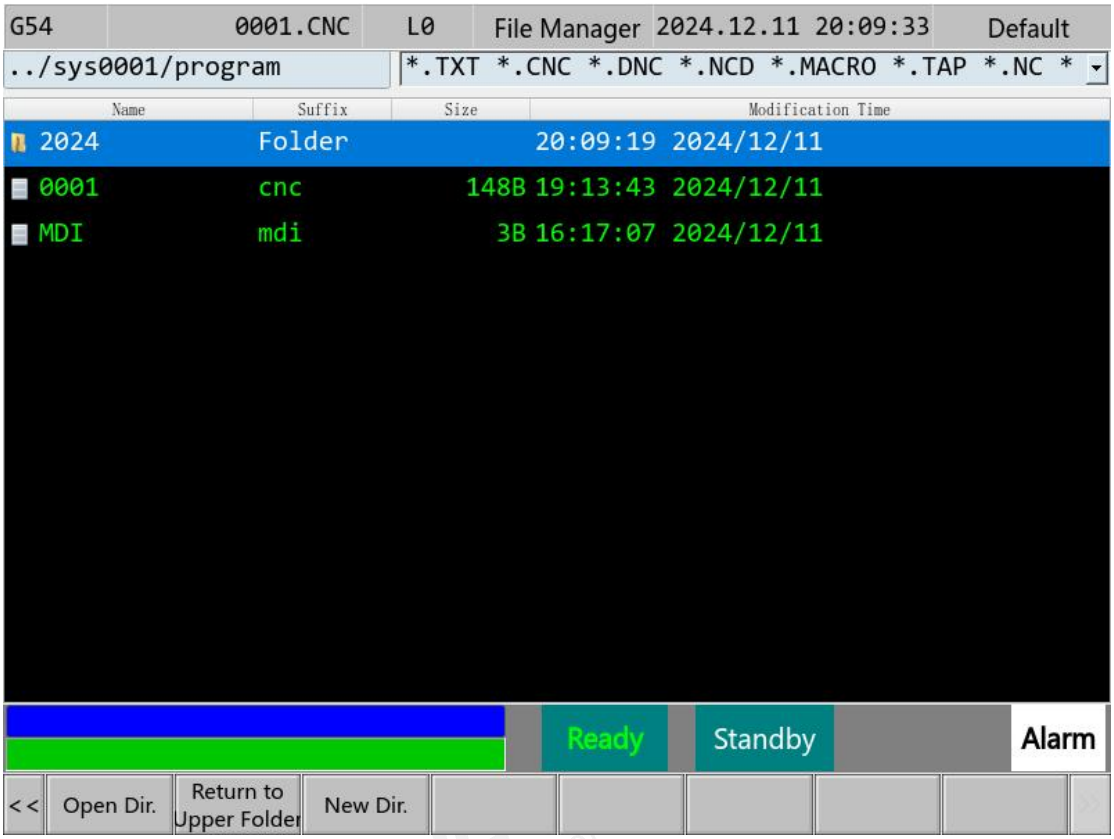
- **Explanation**

Lets you rename a plus engineering file.

- **Operating instructions**

1. Use the arrow keys [↑] [↓] to move the cursor to select the program file to be renamed, press the F6 [Rename] button, the input dialog box will pop up, and enter a new file name;
2. Press the F1 [OK] button to complete the renaming and close the input dialog box.
3. Press the F2 [Cancel] button to exit the rename and close the input dialog box.

1.4.7 Folder Manager



- **Operating Path**

On the "File Management" page→ F7 [Folder Management]

- **Explanation**

Enter the folder management button group, and you can manage folders and files by pressing the button [Open Dir.] [Return to Upper Folder] [New Dir.].

1.4.7.1 Open directory

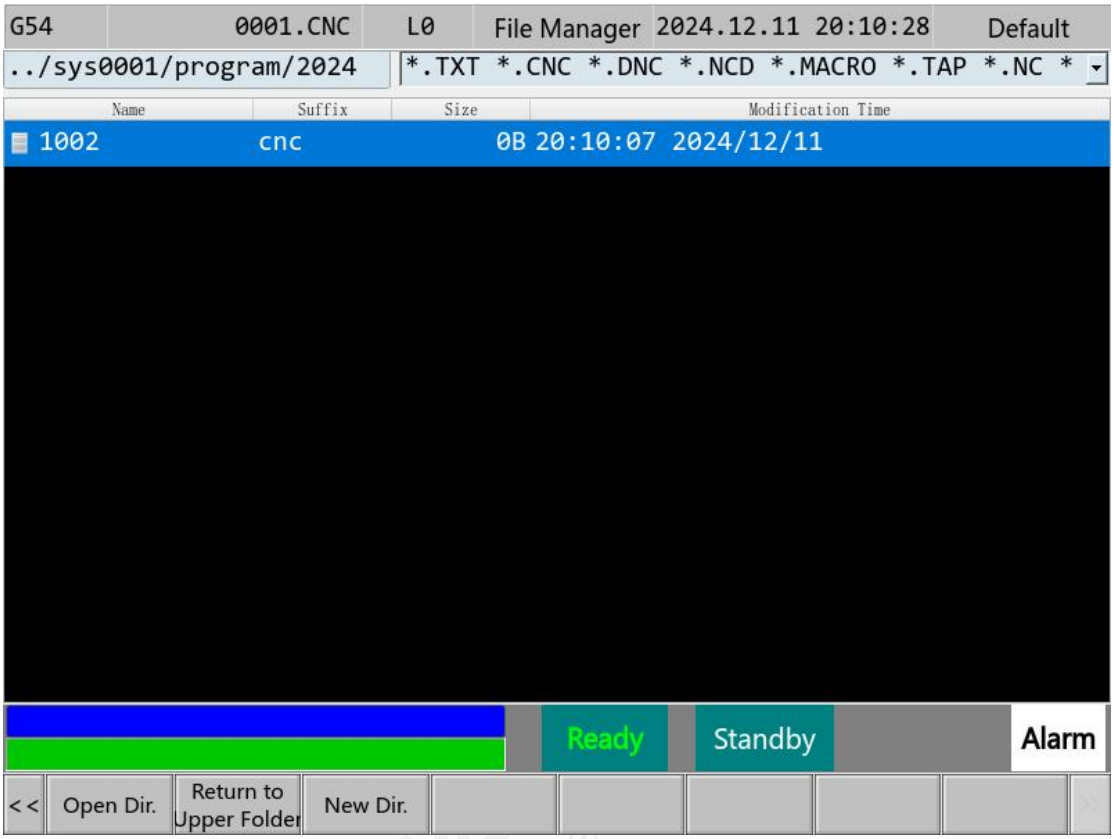
- **Operating Path**

On the "File Management" page→ F7 [Folder Manager] → F1 [Open Dir.].

- **Explanation**

This button opens the folder where the cursor is located in the current file directory.

1.4.7.2 Return to Upper Folder



- **Operating Path**

On the "File Management" page→ F7 [Folder Manager] → F2 [Return to Upper Folder].

- **Explanation**

This button returns to the previous folder.

1.4.7.3 New directory

- **Operating Path**

On the "File Management" page→ F7 [Folder Manager] → F3 [New directory].

- **Explanation**

Create a new folder in the current file directory.

1.4.8 Edit

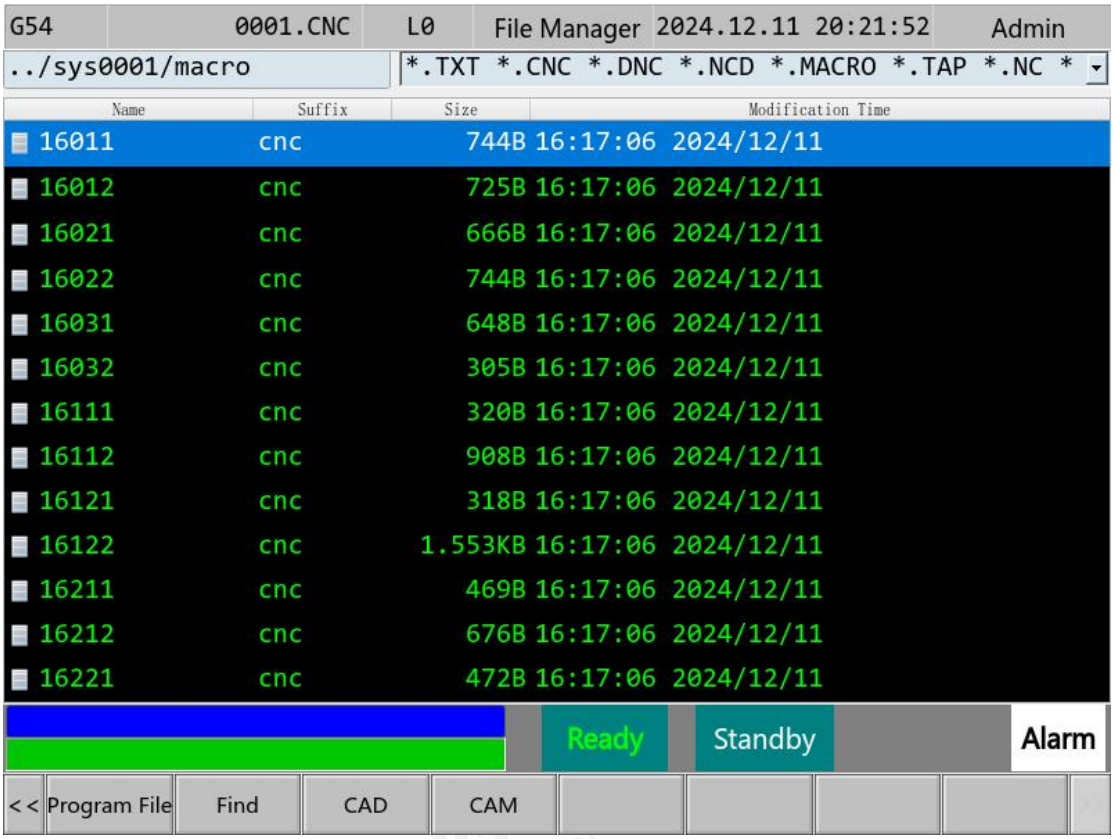
- **Operating Path**

On the "File Management" page→F8[Edit]。

- **Explanation**

For editing of additional projects, please refer to 1.3 "Program Editing" Function Introduction.

1.4.9 Macro File



- **Operating Path**

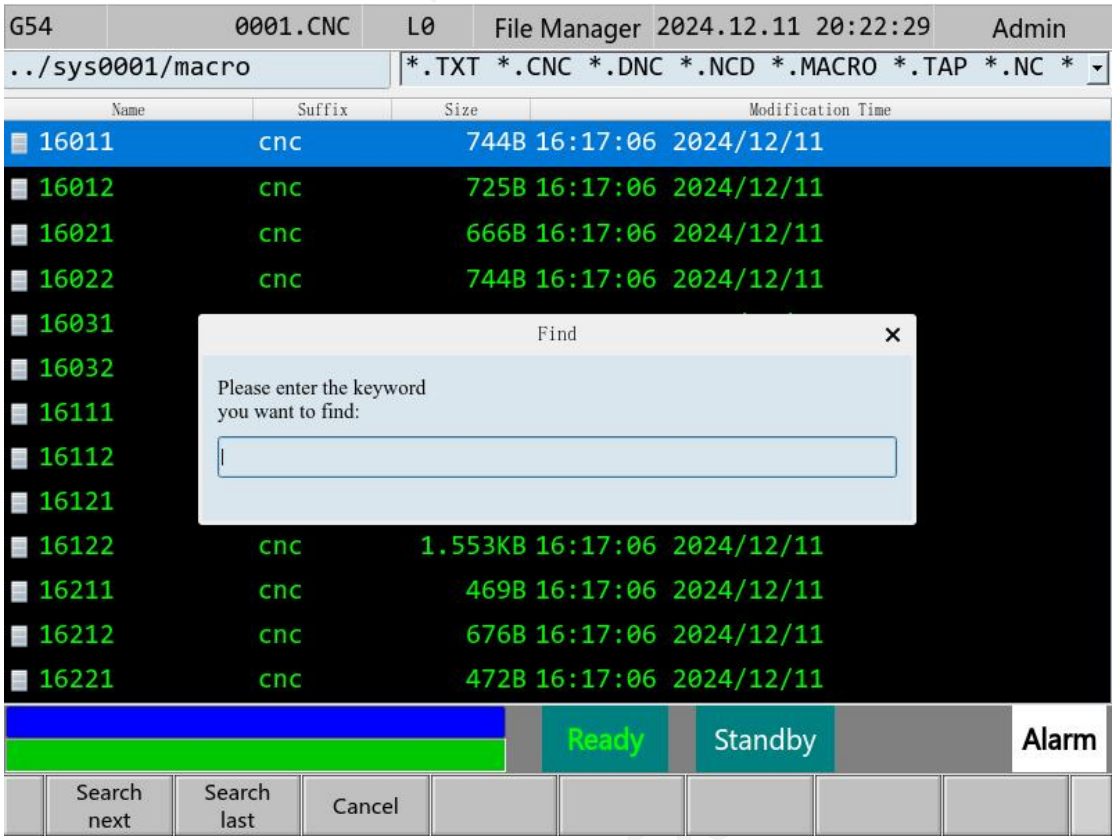
On the “File Management” page→ >> → F1[Macro File].

- **Explanation**

1. Click F1 [Macro File] to enter the macro file, and the F1 button will change to [Program File].

2. Click the [Macro File] button to view the macro file (suffix is MACRO) in the system storage space, and the new file format under this page is still a CNC file (suffix is . CNC）；
3. Click the [Program File] button to view the Add Project File (suffix is . CNC), and the F1 button changes to [Macro File].

1.4.10 Find



- **Operating Path**

On the “File Management” page→ [>>] → F2[Find].

- **Explanation**

Used to find file archives by file name.

- **Operating instructions**

1. Press the [Find] button to pop up a dialog box, enter the file name of the file you want to find or the keywords contained in the file name;

2. Press the F1 [Search next] and F2 [Search last] buttons, and in the file manager, find the file file file name that matches the entered file name;
3. Press the F3 [Cancel] button to exit the search and close the input dialog box.

1.4.11 CAD(Selected)

- **Operating Path**

On the “File Management” page→ [>>] → F3 [CAD]

- **Explanation**

CAD graphics can be drawn, and various machining tasks can be completed by docking with CAM modules, please refer to 1.15 "CAD" function introduction.

1.4.12 CAM(Selected)

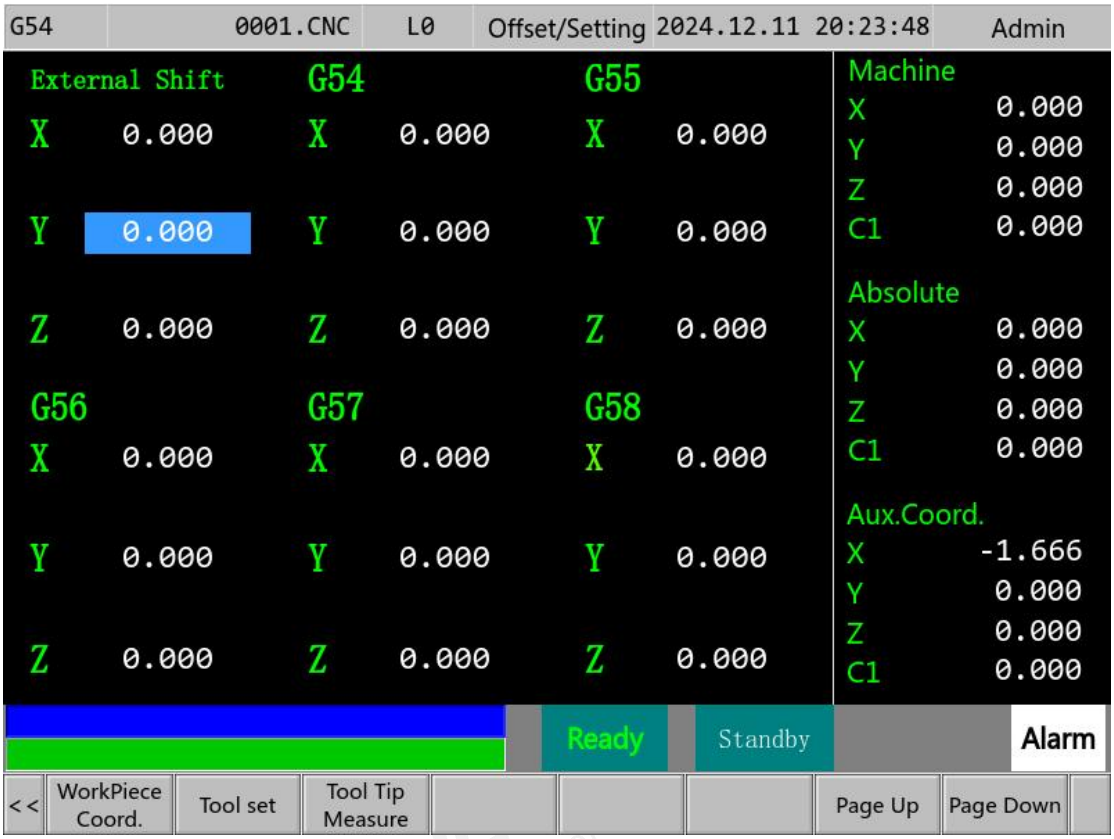
- **Operating Path**

On the “File Management” page→ [>>] → F4 [CAM]

- **Explanation**

It can be interconnected with the machine tool system through CAD graphics, and edit the graphics in a conversational manner, and generate the corresponding G-code after certain editing, so as to achieve milling, slotting, drilling, tapping and other processing, please refer to 1.16 "CAM" function introduction.

1.5 Offset/Setting



- **Operating Path**

1. Path 1: Use the System Panel shortcut button [Offset/Setting] to switch the page to the "Work Coordinate System" page;
2. Path 2: "Machine coordinates" page →F3 [Offset/Setting] to switch the page to the "Work Coordinate System" page.

- **Explanation**

1. This button switches the page to the Work Coordinate System page for workpiece coordinate system settings.
2. If there is no set G54.1P1-G54.1P48/G54-G59 in the user program, the default is the G54 coordinate system;
3. External workpiece coordinate system: Works on all G54.1P1-G54.1P48/G54-G59 coordinate systems.

- **Operating instructions**

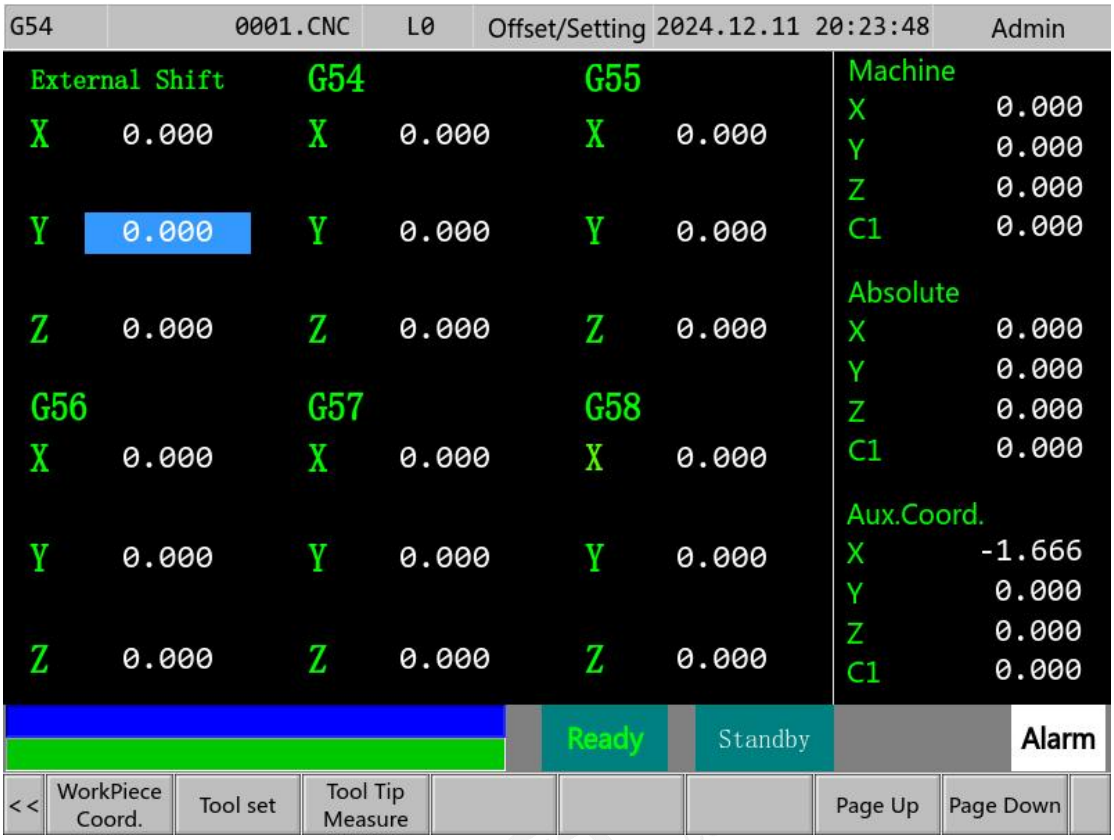
1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;

2. On the "Coordinate System" page → [Page Up] [Page Down] to go up and down the page;
3. You can enter a numeric value directly, or you can enter an axis name plus a numeric value for working coordinate system-to-tool operations.

- **Note**

If it is not for the overall offset, setting the completion coordinate system needs to be reoriented.

1.5.1 Workpiece Coordinate system



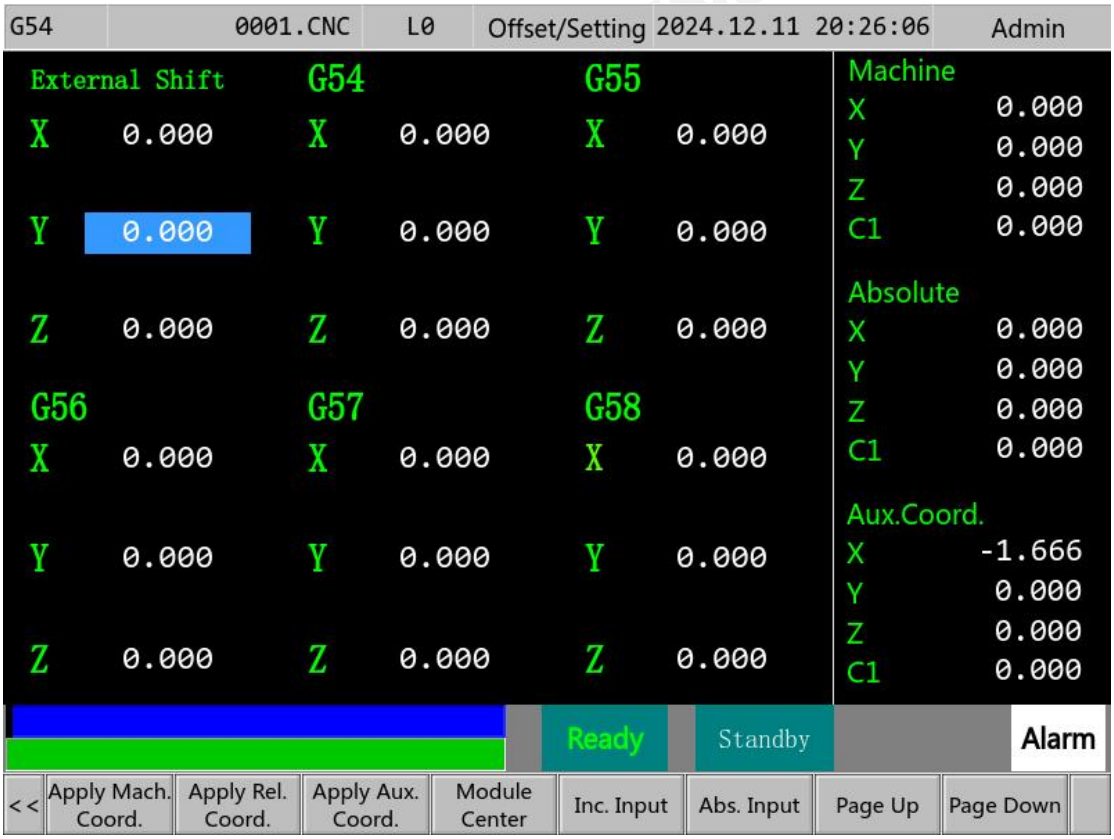
- **Operating Path**

1. Path 1: Use the System Panel shortcut button [Offset/Setting] to switch the page to the "Coordinate System" page →F1 [WorkPiece Coord.].
2. Path 2: "Machine coordinates " page →F3 [Offset/Settings] switch the page to " Coordinate System" page →F1 [WorkPiece Coord.].

- **Explanation**

This function is used for workpiece coordinate system pairs.

1.5.1.1 Workpiece Coordinate（Apply Mach. Coord.）



- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F1[Apply Mach. Coord.]

- **Explanation**

Enter the value of the current mechanical coordinates in the field where the cursor is located.

1.5.1.2 Workpiece Coordinate（Apply Rel. Coord.）

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F2[Apply Rel. Coord.]

- **Explanation**

Enter the current relative coordinate value into the field in which the cursor is located.

1.5.1.3 Workpiece Coordinate（Apply Aux. Coord.）

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F3[Apply Aux. Coord.]

- **Explanation**

Enter the value of the current auxiliary coordinates into the field in which the cursor resides

1.5.1.4 Module Center

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F4[Module Center]

- **Explanation**

To open the Medium Feature page, please see the 1.5.3 "Module Center" description.

1.5.1.5 Workpiece Coordinate（Inc. Input）

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F5[Inc. Input]

- **Explanation**

This key sets the input method, representing the input incremental value, adding the input value on the original basis.

1.5.1.6 Workpiece Coordinate（Abs. Input）

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F6[Abs. Input]

- **Explanation**

This key sets the way you enter, indicating an absolute value.

1.5.1.6 Page Up/Page Down

- **Operating Path**

On the "Workpiece Coordinate System" Page →F1 [WorkPiece Coord.] →F7-8[Page Up/Page Down]

- **Explanation**

Turn the pages up and down.

1.5.2 Tool set(Offset/Setting)

- **Operating Path**

1. Path 1: Use the system panel shortcut keys [Offset/Setting];
2. Path 2: "Machine Coordinates" page →F3 [Offset/Settings] →F2 [Tool Set]

- **Explanation**

1. Set tool length wear correction value (H), tool radius correction value (D);
2. (Tool radius compensation + tool radius wear compensation) is actual G41/G42 compensation amount;
3. (Tool Length Compensation + Tool Length Wear Compensation) is the actual G43/G44 compensation amount.

1.5.2.1 Clear Z Relative coordinates

- **Operating Path**

On the “Offset/Setting” page →F1[Clear Z Coord.]

- **Explanation**

Lets you clear the Z axis relative coordinates.

1.5.2.2 Set Tool Mechanical coordinates

- **Operating Path**

On the “Offset/Setting” page →F2[Set Tool Mach. Coord.]

- **Explanation**

Enter the value of the current Z-axis mechanical coordinates into the tool length data for the row in which the cursor is located.

1.5.2.3 Set Tool Relative coordinates

- **Operating Path**

On the “Offset/Setting” page →F3[Set Tool Rel. Coord.]

- **Explanation**

Enter the value of the current Z-axis relative coordinates into the tool-length data for the row in which the cursor is located.

1.5.2.4 Tool Information

- **Operating Path**

On the “Offset/Setting” page →F7[Tool Information]

- **Explanation**

Open the tool magazine parameter interface, see 1.5.6 tool number description for details.

1.5.2.5 Tool management

- **Operating Path**

On the “Offset/Setting” page →F8[Tool Man.]

- **Explanation**

Open the Tool Life Management interface, see 1.5.7 Tool Life Management Instructions.

1.5.3 Module Center

- **Operating Path**

Use the System Panel shortcut button "Offset/Setting" to switch the page to the "Workpiece Coordinate System" page →F1[Workpiece Coord.] →F4 [Module Center].

- **Explanation**

To find the center of the workpiece as the starting point of the process, so you need to touch both sides of the workpiece, get the coordinates on both sides of the workpiece by the system automatically find the center coordinates, the user selects another set of artifact coordinates (The external coordinate system ~ G54.1P44) is set in as the starting point for the process.

1.5.3.1 Manual Four Points（XY Center）

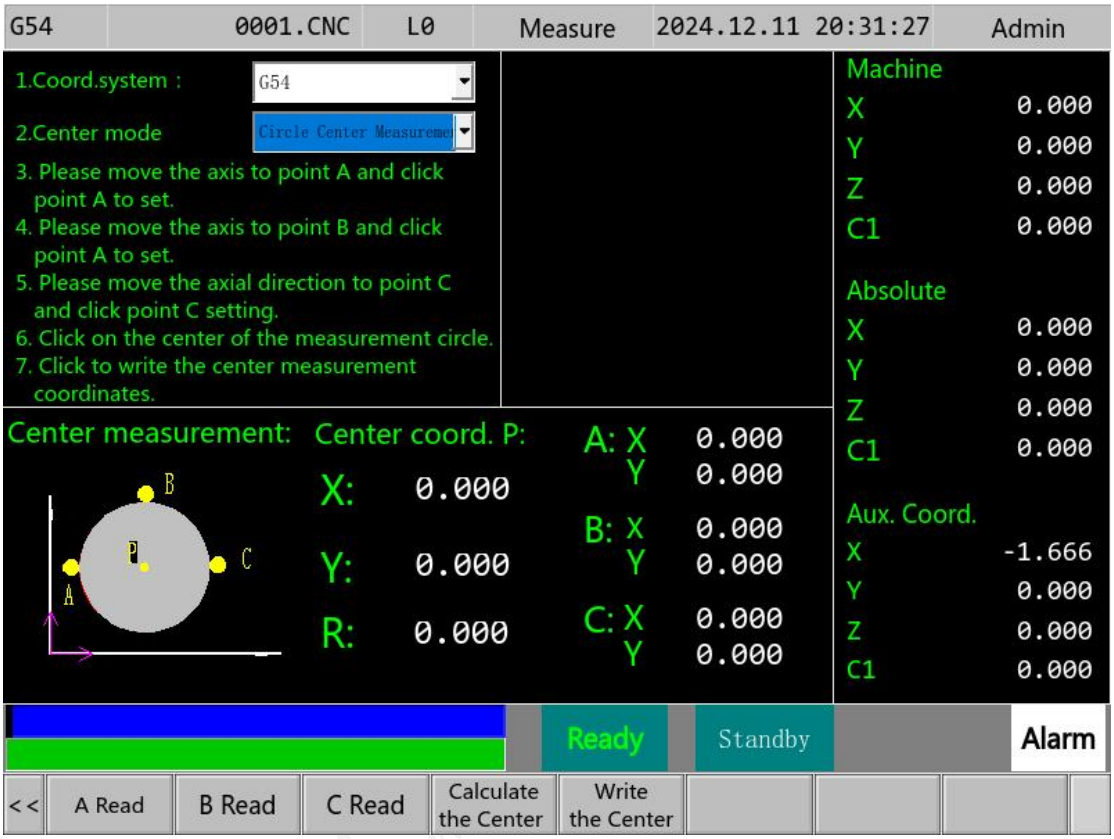
- **Operating Path**

1. Set the cutter coordinate system on the screen.
2. Switch the middle mode to XY Center, the system will switch to the middle screen.
3. Use the handwheel to go to the Px1 point in the touch chart, press F1[Px1 Set] setting, and then mark the Px1-point X mechanical seat on the screen. With Px2, the mechanical coordinates of the X-axis midpoint are calculated, placed in the Pxm field and the auxiliary point coordinate X-axis field.
4. Click the F2[Px2 Set] setting on the handwheel to go to the tool in the touch diagram Px2 and the Px2-point X mechanical seat will be marked on the screen. With Px1, the mechanical coordinates of the X-axis midpoint are calculated. Place in the Pxm field and the auxiliary point coordinate X-axis field.
5. Click the F3[Py1 Set] setting in the handwheel to remove the tool from the touch diagram Py1 and the Py1-point Y mechanical seat is marked on the screen. and

calculate the Y-axis intermediate point mechanical coordinates with Py2, Place in the Pym field and the auxiliary point coordinate Y-axis field.

6. Use the handwheel to go to the Py2 point in the touch diagram, press F4[Py2 Set] setting, and the Py2-point Y mechanical seat will be marked on the screen. and calculate the Y-axis intermediate point mechanical coordinates with Py1, Place in the Pym field and the auxiliary point coordinate Y-axis field.
7. Press F5 to write the value of Pxm, Pym to the tool coordinate system specified on the screen
8. Switch to the cutter 1.5.1 screen for the workpiece coordinate system. Move the cursor to the position of the artifact coordinate system to be set. Press F4[Apply Aux. Coord.] and the system will follow the axial direction of the cursor. Enter the auxiliary point coordinate value of the axis into the field in which the cursor resides.

1.5.3.2 Manual Three-Point Centerpiece（Circle Center Measurement）

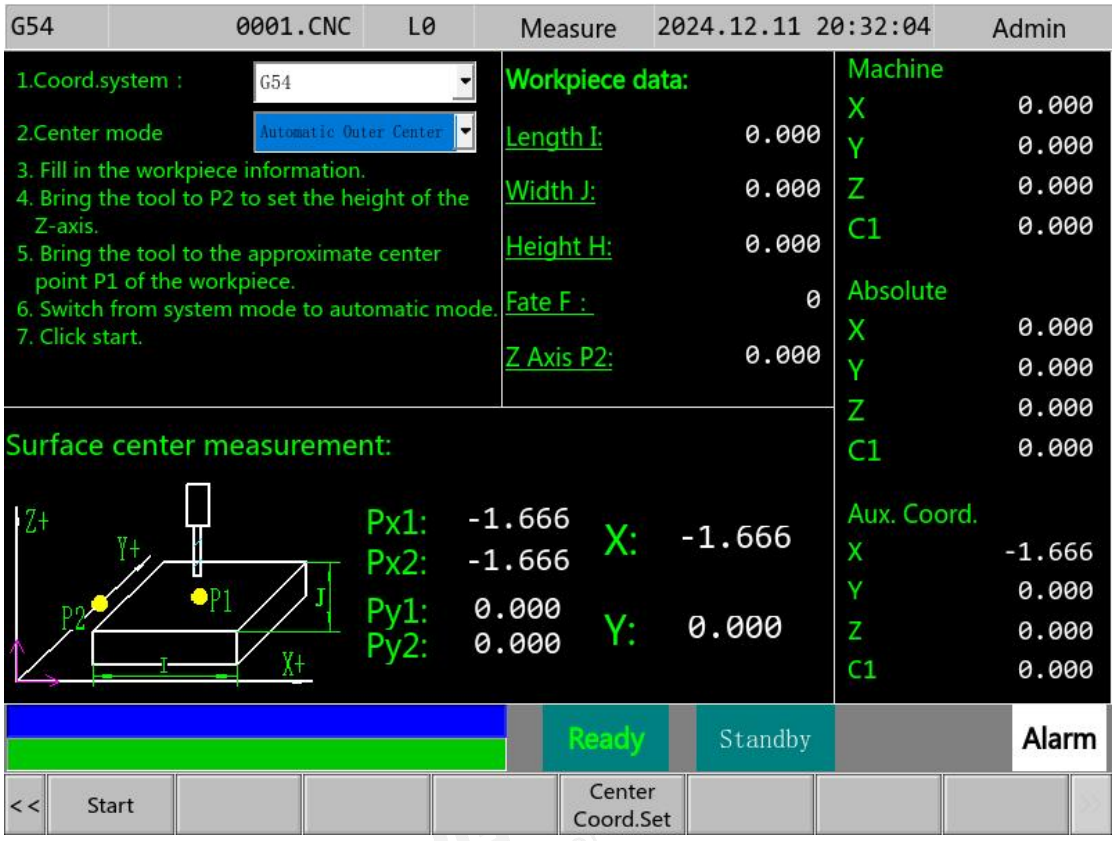


- **Explanation**

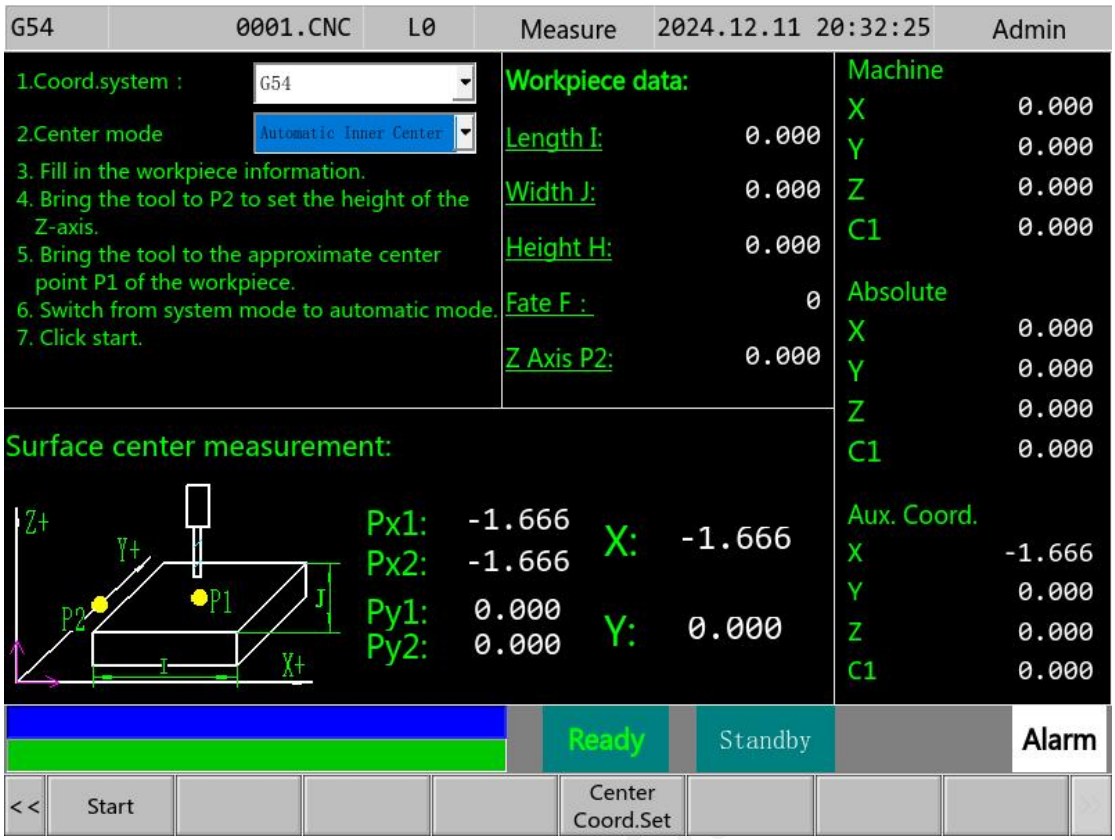
1. Set the cutter coordinate system on the screen.
2. Switch the middle mode to the Circle Center Measurement, the system will switch to the middle screen.
3. Move the probe or tool to point A, click Read point A coordinates, and the system automatically records point A coordinates.
4. Move the probe or tool to point B, click Read point B coordinates, the system will automatically record point B coordinates.
5. Move the probe or tool to point C, click Read C point coordinates, the system will automatically record C point coordinates.
6. Select the three-point centroid coordinates and press F4 to Measure the Centroid, which displays the centroid P on the screen.

7. Press F5 to Write Center Coordinate System, which writes the value of Center Coordinate P to the pair coordinate system specified on the screen.
8. Or teach the calculated numerical values to the row coordinate system in which the cursor resides through auxiliary point coordinates: Switch to the cutter 1.5.1 screen for the workpiece coordinate system. Move the cursor to the position of the artifact coordinate system to be set. Press F4[Apply Aux. Coord.] and the system will follow the axial direction of the cursor. Enter the auxiliary point coordinate value of the axis into the field in which the cursor resides.

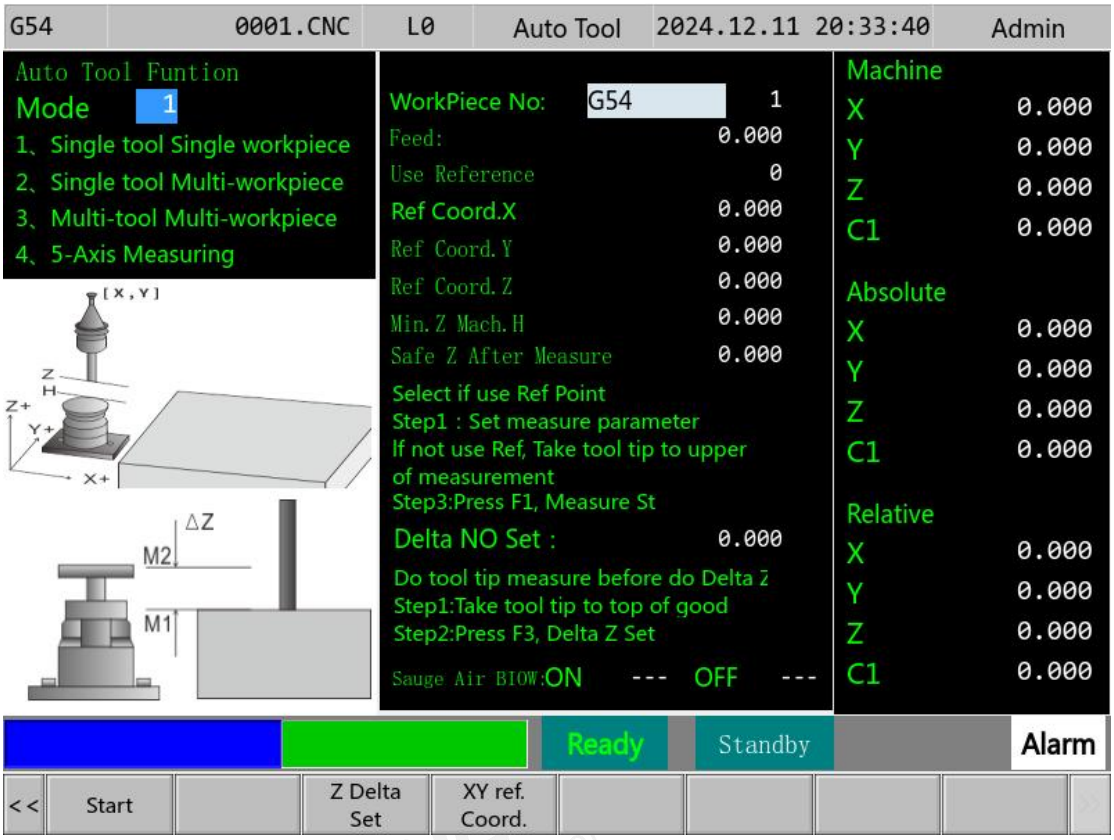
1.5.3.3 Automatic Outer Center



1.5.3.4 Automatic Inner Center



1.5.4 Auto Tool



- **Purpose of Operation**

1. The automatic tool alignment function is to measure the Z-axis coordinates of each tool's tip through the alignment tool on the machine. Because the difference between the tool surface and the workpiece datum plane is fixed, So once the auto-pair is complete, the controller can automatically calculate the Z-axis program origin of the tool against the corresponding workpiece.

   NOTE: Use this feature to ensure that the machine is equipped with a pair of tool

2. There are currently three pairs of tool modes, as described below:

   ❖ Single tool and single workpiece: select the workpiece coordinate number, automatically after the tool, save the tool length in the workpiece coordinate; Then move the tool to the workpiece datum plane, press the F3[Z Delta Set], save the drop to the external coordinate offset; The Z-axis program coordinate origin can be obtained after calculation;

   ❖ Single tool and multiple workpieces: after automatic tool, Save the Tool length to

an external coordinate offset. Then select the artifact coordinate number and press F3[Z Delta Set]. Save the drop value to the artifact coordinates. The Z-axis program coordinate origin can be obtained after calculation;

❖ Multi-tool and Multi-workpiece: Select tool number, after automatic tool, the length of the tool into the tool length compensation table, Then select the corresponding artifact coordinate number. Press F3[Z Delta Set] to save the drop value to the artifact coordinates. When computed, the Z-axis program coordinates the origin.

- **Operating Path**

On the "Workpiece Coordinate System" page →F3 [Tool Tip Measure]

- **Screen Description**

1. Mode：1：Single tool and single workpiece 2：Single tool and multiple workpieces 3：Multi-tool and Multi-workpiece 4：Five-axis measurement
2. Workpiece coordinate number P: Set the coordinate system to which the tool setting measurement value should be filled. (Mode 1 is available);
3. Select the tool number T: set the tool number to be measured at present, and fill in the measured value to the number of tool length compensation;
4. Measurement speed F: set the speed of the first downward probe and each rebound of the tool setting action;
5. Use reference point coordinates: set whether to move to the reference point first and then perform the tool setting action, if the tool setter has a fixed position, it is recommended to enable this function;

   ❖ Set 0: Do not use the coordinates of the reference point, and directly probe down at the current position to carry out the tool setting action

   ❖ Set 1: Use the reference point coordinates, the Z axis returns to the origin first, then the XY axis moves to the reference point, and finally the Z axis descends to the Z reference point, and then starts to set the tool.
6. X-axis reference point: set the X-axis reference point, which is usually the X coordinate of the center position of the tool setter, which can be filled in, or [XY

reference point setting] to set;

7. Y-axis reference point: set the Y-axis reference point, which is usually the Y coordinate of the center position of the tool setter, which can be filled in, or [XY ref. Coord.] to set;
8. Z-axis starting point: set the starting point of Z-axis tool setting, in the automatic process, after the Z-axis drops to this point, the measurement speed reduction will be used, and the input can be done directly;
9. Z lowest coordinate H: In the process of automatic tool setting, the lowest point where the Z axis can be lowered, usually the main shaft head will not hit the Z-axis coordinate of the tool setter, and the coordinate value can be directly entered.
10. Drop value: the distance between the surface of the tool and the surface of the workpiece;

    ❖ Mode 1: After the automatic tool setting is completed, move the tool tip to the workpiece datum plane, and set the drop value to the external coordinate offset through the [Z Delta Set] button

    ❖ Mode 2 or 3: Set the coordinate number of the workpiece, after the automatic tool setting is completed, move the tool tip to the datum plane of the workpiece, and set the drop value to the corresponding workpiece coordinate through the [Z Delta Set] button

    ❖ Tool setter setting (blowing, holding air): Before the automatic tool setting starts, sometimes it is necessary to blow to clean the surface of the tool setting instrument, this function provides the M code for setting blowing and holding.

- **Action Description**

1. The Z-axis moves to the mechanical origin at G00 speed;
2. The XY axis moves to the reference point at G00 speed;
3. The Z-axis moves to the start of the Z-axis at a speed of G00;
4. The Z-axis moves to [the lowest coordinate H of the Z-axis] for the first time at the measured speed, and stops immediately when it touches the tool setter during the descent; Note: If the lowest mechanical coordinate of the Z-axis is set incorrectly, the

Z-axis may be set upward, for example, the lowest coordinate of the Z-axis is set larger than the coordinate of the reference point;

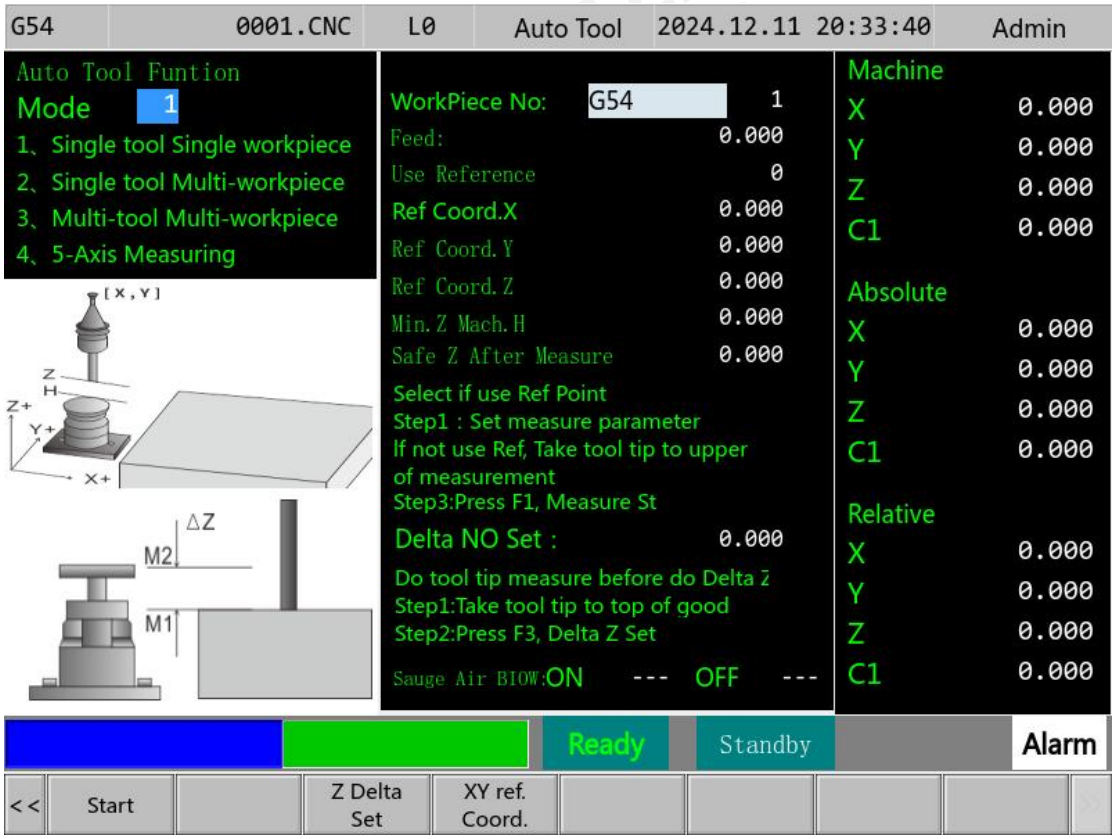
5. The Z-axis rises 3mm at the measurement speed;
6. The second Z-axis descends, moving down 5mm at a speed of 30mm/min, if it touches the tool setter during the descent, it will stop immediately and record the Z-axis coordinates;
7. The Z-axis returns to the mechanical origin at a speed of G00.

- **Alarm and Protection**

1. When the Z-axis descends for the first time, if the coordinates of the lowest point of the Z-axis are reached, but the tool setter has not yet been touched, the system will report that the Z-axis has reached the lowest point;
2. 2. When the Z-axis is lowered for the second time, if it does not touch the tool setter after dropping 5mm, the controller will report that the tool has not detected the tool setting signal.

1.5.5 Automatic tool setting operation mode

1.5.5.1 Single-tool single-workpiece automatic tool setting operation mode



1. Set the [Measurement Mode] in the upper left corner of the tool setting screen to 1: single tool and single workpiece mode;

2. Set the [Workpiece Coordinate Number P] in the upper right corner of the screen as the working coordinate system to which the workpiece belongs;

3. Workpiece coordinate number P: 0: external coordinate offset; 1：G54； 2：G55； 3：G56.....；

4. Set the speed of the first descent and rise when the automatic tool setting is set in [Measurement Speed F];

5. If the tool setter has a fixed position on the machine, skip to step 6 and set the XY reference point coordinates. If the current position is the position of the tool setter, please

set [Use Reference Point Coordinates] to 0;

6. If step 5 has already been performed, skip this step. Please set [Use Reference Point Coordinates] to 1, then move the XY axis, let the tool align with the center of the tool setter, and press F4[XY Ref. Coord.], and the mechanical coordinates will be automatically filled in [X-axis reference point] and [Y-axis reference point];

7. Set the starting position of the Z-axis downward tool setting at the [Z-axis reference point];

8. When the lowest coordinate of the Z axis H is set for automatic tool setting, the lowest point that the Z axis can drop;

9. After determining the above settings, press F1[Start] to start automatic tool setting;

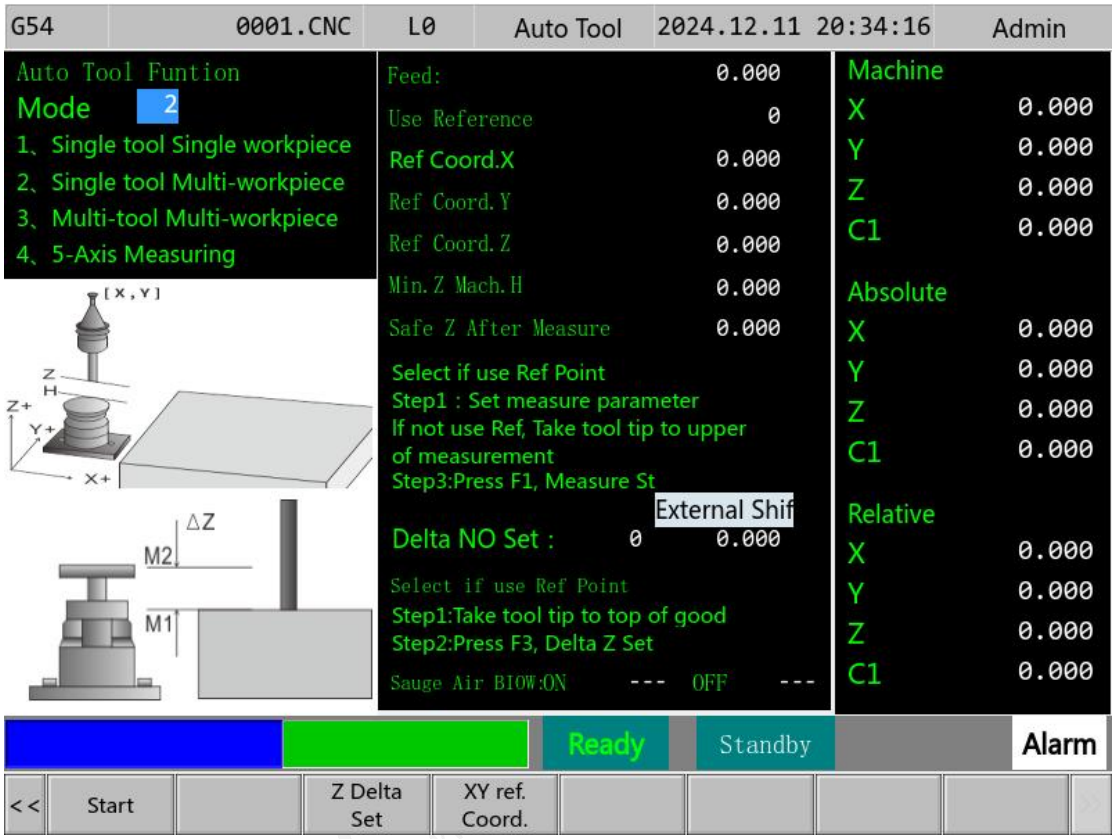
10. If any situation occurs during the process, immediately press the emergency stop or reset button to stop the automatic tool setting;

11. After the automatic tool setting is completed, the Z-axis coordinates of the tool tip touching the tool setting device will automatically save the tool length value and the set workpiece coordinates;

12. Move the speed of the tool tip to the surface of the workpiece, press F3[Z Delta Set], and the drop value between the tool counter and the surface of the workpiece will be filled in with the [drop value] and the external coordinate offset;

13. Single-tool single-workpiece automatic tool setting is completed.

1.5.5.2 Single-tool multi-workpiece automatic tool setting operation mode

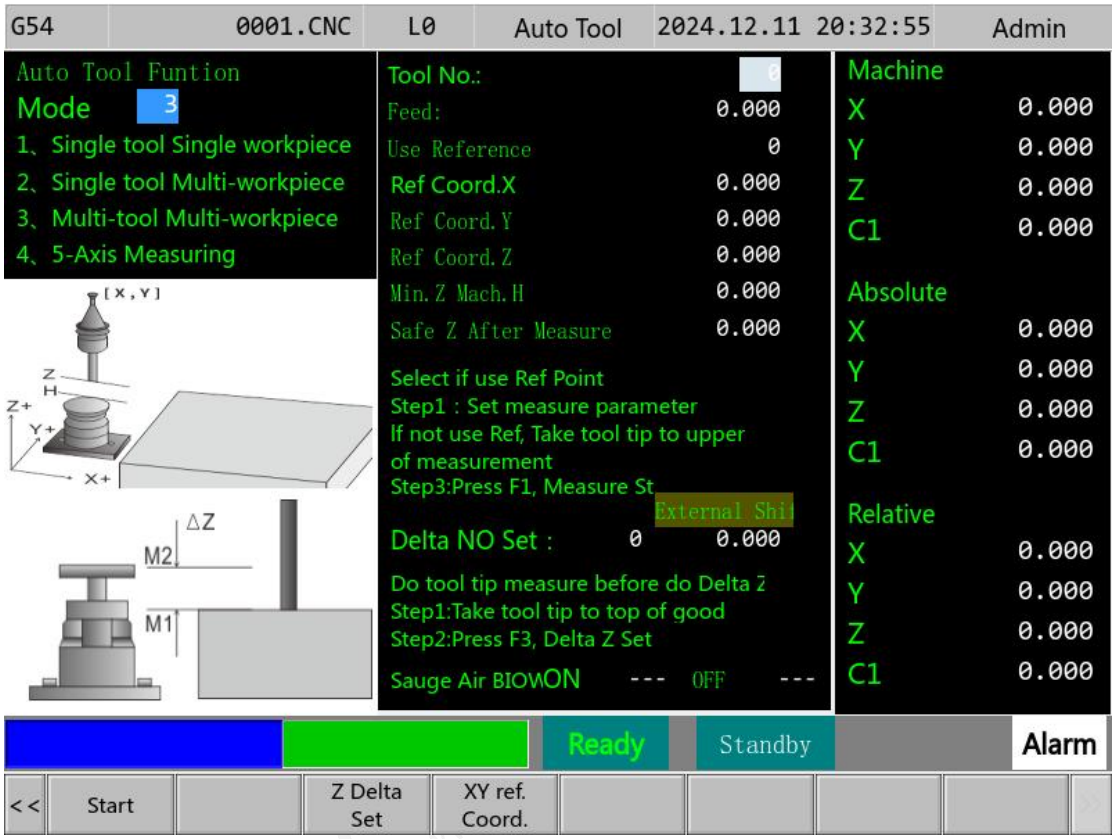


1. Set the [Measurement Mode] in the upper left corner of the tool setting screen to 2: Single tool multi-workpiece mode;
2. Set the speed of the first descent and rise when the automatic tool setting is set in [Measurement Speed F];
3. If the tool setter has a fixed position on the machine, please skip to step 4 and set the XY reference point coordinates, if the current position is the tool setter position, please set [Use reference point coordinates] to 0;
4. If you have already performed step 3, skip this step. Please set [Use Reference Point Coordinates] to 1, then move the XY axis, let the tool align with the center of the tool setter, and press F4[XY Ref. Coord.], the current mechanical coordinates will be automatically filled in the [X-axis reference point] and [Y-axis reference point];
5. Set the starting position of the Z-axis downward tool setting at the [Z-axis reference

point].

6. When the lowest coordinate of the Z axis H is set to the automatic tool setting, the lowest point that the Z axis can drop;
7. After confirming the above settings, press F1[Start] to start automatic tool setting;
8. If any situation occurs during the process, immediately press the emergency stop or reset button to stop the automatic tool setting;
9. After the automatic tool setting is completed, the Z-axis coordinate of the tool tip touching the tool setting device will automatically save the tool length value and the external coordinate offset.
10. Move the speed of the tool tip to the surface of the workpiece, and then set the [Drop Setting Coordinate System] Press F3[Z Delta Set], the drop value between the cutter and the surface of the workpiece will be filled in the Z-axis coordinates of the specified workpiece coordinate system;
11. Drop setting coordinate system: 0: external coordinate offset; 1：G54； 2：G55； 3：G56.....；
12. If there are other workpieces to be set, repeat step 10 to complete the automatic tool setting of single tool and multiple workpieces.

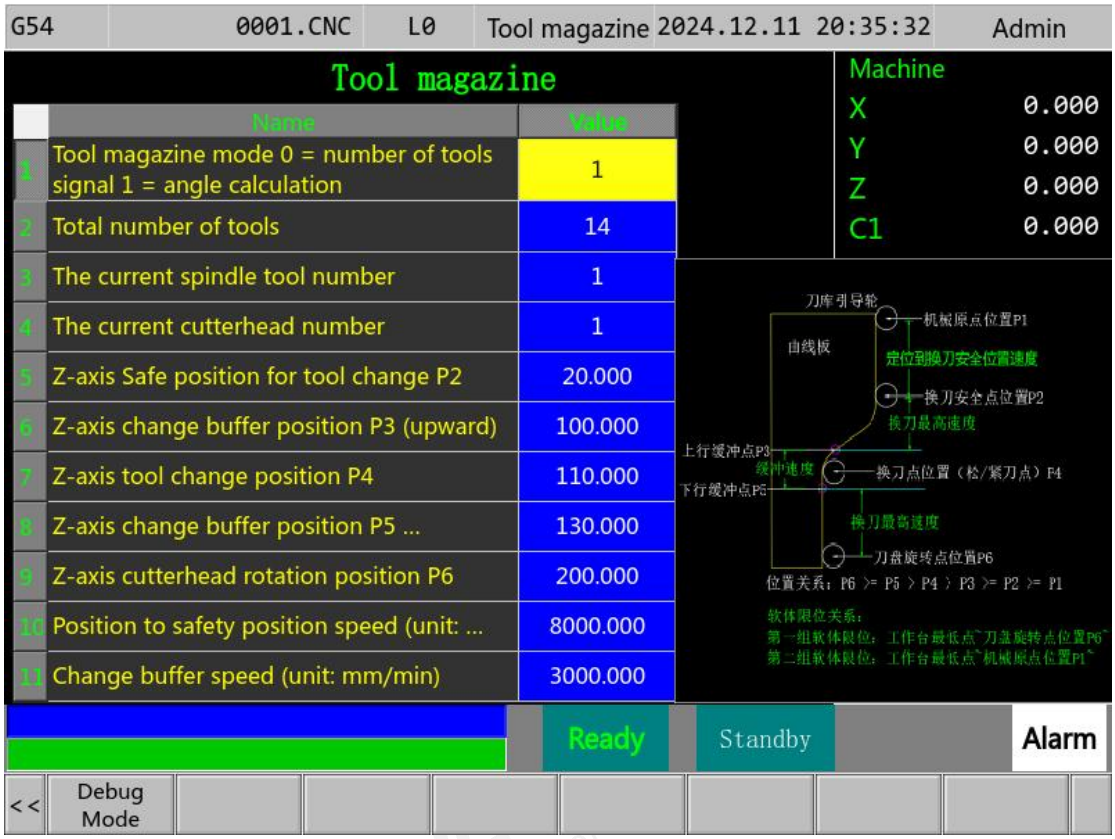
1.5.5.3 Multi-tool and multi-workpiece automatic tool setting operation mode



1. Set the [Measurement Mode] in the upper left corner of the tool setting screen to 3: Multi-tool and multi-workpiece mode;
2. Enter the target tool number into [Tool Number Selection T];
3. Set the speed of the first descent and rise when the automatic tool setting is set in [Measurement Speed F];
4. If the tool setter has a fixed position on the machine, skip to step 5 and set the XY reference point coordinates. If the current position is the position of the tool setter, please set [Use Reference Point Coordinates] to 0;
5. If you have already performed Step 4, skip this step. Please set [Use Reference Point Coordinates] to 1, then move the XY axis, let the tool align with the center of the tool setter, and press F4[XY Ref. Coord.], and the mechanical coordinates will be automatically filled in [X-axis reference point] and [Y-axis reference point];

6. Set the starting position of the Z-axis downward tool setting at the [Z-axis reference point];
7. When setting the lowest coordinate of the Z axis H, the lowest point that the Z axis can descend when setting the automatic tool setting;
8. After confirming the above settings, press F1[Start] to start automatic tool setting;
9. If any situation occurs during the process, immediately press the emergency stop or reset button to stop the automatic tool setting;
10. After the automatic tool setting is completed, the Z-axis coordinate of the tool tip touching the tool setting device will be automatically stored in the tool length value;
11. Move the speed of the tool tip to the surface of the workpiece, and then set the [Drop Setting Coordinate System] Press F3[Z Delta Set], the drop value between the tool and the surface of the workpiece will be filled in the Z-axis coordinates of the specified workpiece coordinate system;
12. Drop setting coordinate system: 0: external coordinate offset; 1：G54； 2：G55； 3：G56.....；
13. If there are other tool numbers or workpiece tool setting requirements, please repeat the above steps 2~12 to complete the automatic tool setting of multiple tools and multiple workpieces.

1.5.6 Tool Information



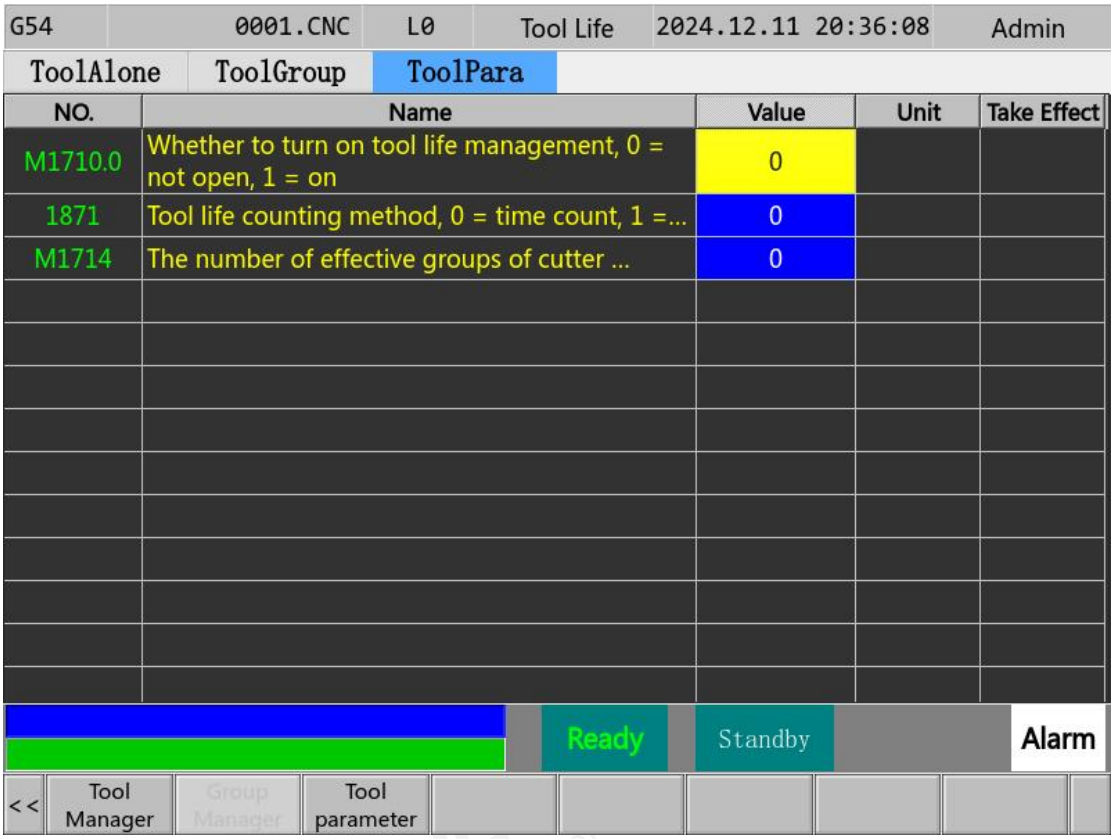
- **Operating Path**

Use the shortcut button [Offset/Settings] on the system panel to switch the page to the "Offset/Settings" page →F7 [Tool Information]

- **Explanation**

1. Set the relevant parameters of the tool magazine;

1.5.7 Tool management



- **Operating Path**

Use the shortcut button [Offset/Setting] on the system panel to switch the page to the "Offset/Setting" page→F8 [Tool Mag.]→F1 [Tool Manager].

- **Operating instructions**

1. It is used for the management of the tool, including the current use time, the current number of uses, the life management status, the arrival time setting, the arrival times setting, the wear compensation number setting, the length compensation number setting, the tool number setting, etc.;
2. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
3. Use [⩓] [⩔]to turn pages up and down;
4. Use [HOME/END] to quickly switch the cursor to the beginning and end of the row.

1.5.7.3 Group management

- **Operating Path**

On the Offset/Settings page→ F8 [Tool Management] → F2 [Group management].

- **Explanation**

1. Manage the group of tools, including the starting tool number, the end tool number, the number of effective tools, the life time limit of the tool set, the life management status of the tool set, etc.;
2. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
3. Use [⩤] [⩥]to turn pages up and down;
4. Use [HOME/END] to quickly switch the cursor to the beginning and end of the row.

1.5.7.4 Tool parameters

- **Operating Path**

On the Offset/Settings page → F8 [Tool Management] → F3 [Tool Parameters].

- **Explanation**

1. This function is used to set the tool parameters;
2. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
3. Use [⩤] [⩥]to turn pages up and down;
4. Use [HOME/END] to quickly switch the cursor to the beginning and end of the row.

1.6 Monitor

1.6.1 Screen description

- **Operating Path**

1. Path 1: Use the shortcut button [MON] on the system panel to switch the page to the "Monitor" page;
2. Path 2: "Machine coordinates" page→ F4 [Monitor] → "Monitor" page.

- **Explanation**

This page provides the necessary information for monitoring during processing.

1.6.1.1 Machine control area

- **This area displays the current machine information**

1. Absolute coordinates

2. Remaining distance
3. Feed rate
4. Spindle speed

1.6.1.2 Code monitoring area

1. This area displays the program content that is currently being processed.
2. The yellow cursor will indicate the single section that the current program is executing.

1.6.1.3 Processing information display area

- **Explanation**

1. This area overlaps with the "Processing Information Settings" area;
2. Use the F4 [Machining setting] button to switch the display.

- **Screen description**

1. G-Code Status
   - ❖ Displays the G-code in the current system execution.
2. Run Time
   - ❖ Displays the machining time of the currently machined workpiece;
   - ❖ When the program starts running again, the time will be recalculated.
3. Accumulated Time
   - ❖ The total number of machining time from the first execution of the program to the present;
   - ❖ Use the Clear Accumulated Time button to clear the cumulative processing time.
4. MFO
   - ❖ G00 MFO
   - ❖ G01 MFO
   - ❖ MPG MFO

- ❖ Spn.C MFO

5. Accumulated parts
   - ❖ The total number of workpieces that have been machined by the machine;
   - ❖ The system will not clear zero, you can set it manually, and the cumulative completion items set in the processing information setting are set with the input box.
6. Parts count
   - ❖ Set the number of workpieces of the current workpiece;
   - ❖ When it becomes more engineering, the number of workpieces will be cleared once;
   - ❖ If you need to cooperate with the M code, you will add 1 to trigger the pause when the number of artifacts reaches the required number of artifacts. (M15 is the count plus 1, and M16 is the count zero).
7. Machining tool data
   - ❖ T represents the current machining tool number;
   - ❖ H represents the current called tool length and tool length wear group number;
   - ❖ D represents the currently called tool diameter and tool diameter wear group number.
8. M Code
   - ❖ The M-code that is currently executed.
9. The starting program section number
   - ❖ Interrupt start: You can set a single section to start again, press the [Enter] button to trigger and then start a single section search.

1.6.2 Edit

- **Operating Path**

On the "Monitor" page→ F1 [Edit].

- **Explanation**

Load the currently executed file into the code editor, and switch to the "Program Editing" page.

- **Note**

If you press this button while the program is running, the screen will switch to the "Program Editing" page, but you cannot edit the processing file.

1.6.3 Simulation switch

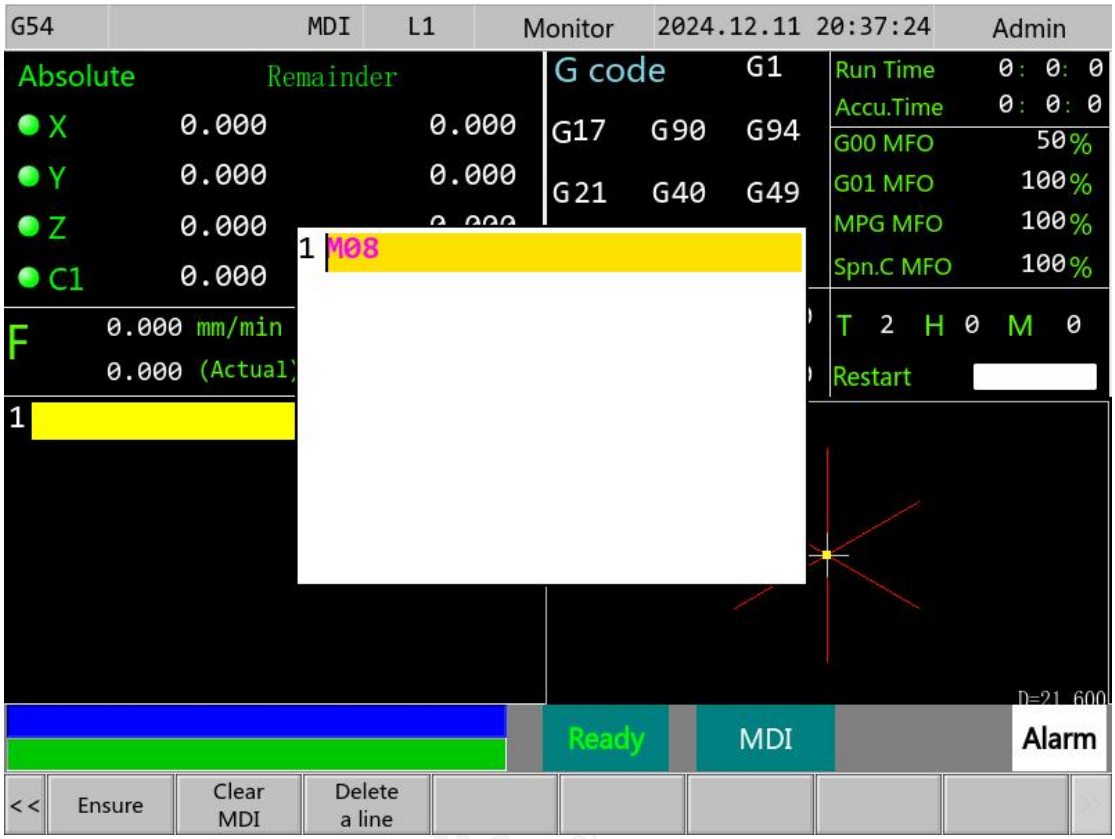
- **Operating Path**

On the "Monitor" page→ F2 [Simul. Switch].

- **Explanation**

1. Used to toggle whether the graphics simulator is displayed or not;
2. Graphic adjustment can only be made when the graphics simulator is displayed;
3. For details, please refer to 1.6.10 "Graphics Adjustments".

1.6.4 MDI Input



- **Operating Path**

1. Path 1: Switch to the "Monitor" page through the [MDI] button on the system panel and pop up the MDI input box
2. Path 2: "Monitor" page → F3 [MDI] in MDI mode.

- **Explanation**

Edit the program executed by MDI.

- **Operating instructions**

1. Switch the mode to "MDI" mode;
2. Press the [MDI] button, and an edit dialog box will pop up.
3. In the editing dialog box, after editing the program, press the [Ensure] button, and the system will automatically load the edited program into the processing storage area;
4. Press the [Start] button to perform MDI programming.

- **Note**

1. This button only works in "MDI" mode, the "MDI" button on the auxiliary panel can

switch the mode to MDI mode;

2. [Clear MDI] can clear the currently edited MDI content;

3. [Delete a line] deletes the currently edited row.

1.6.5 Machining setting

- **Operating Path**

On the "Monitor" page→ F4 [Machining setting].

- **Explanation**

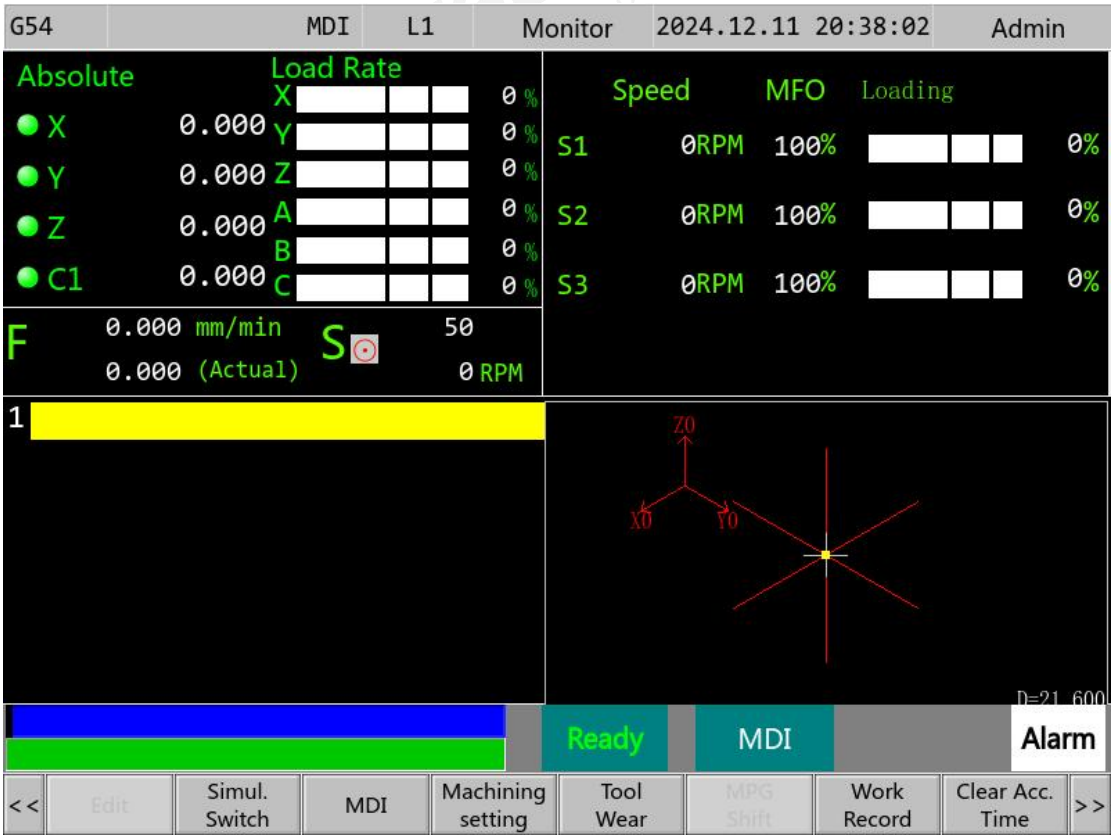
It is used to switch the display between "Processing Information" and "Processing Settings".

1.6.5.1 Processing information 1

- **Explanation**

1. It can monitor the status of G code, this working hour, cumulative working hours, cumulative completion, this completion, G00 magnification, G01 magnification, handwheel magnification, spindle magnification, T code, M code;
2. You can set to restart a single section.

1.6.5.2 Processing information 2



- **Explanation**

The load factor of each axial and the load factor of the spindle can be monitored.

1.6.5.3 Processing information settings

- **Explanation**

You can set the total number of parts, the number of parts, the number of required parts, the feed rate, the spindle speed, and view the program breakpoint line number and serial number information.

1.6.6 Tool Wear

- **Operating Path**

On the "Monitor" page→ F5 [Tool Wear].

- **Explanation**

It is used to adjust the tool compensation data.

- **Operating instructions**

Use the panel arrow keys [↑] [↓] [←] [→] to move the cursor to the corresponding tool number information and enter the data to be compensated.

1.6.7 MPG Shift

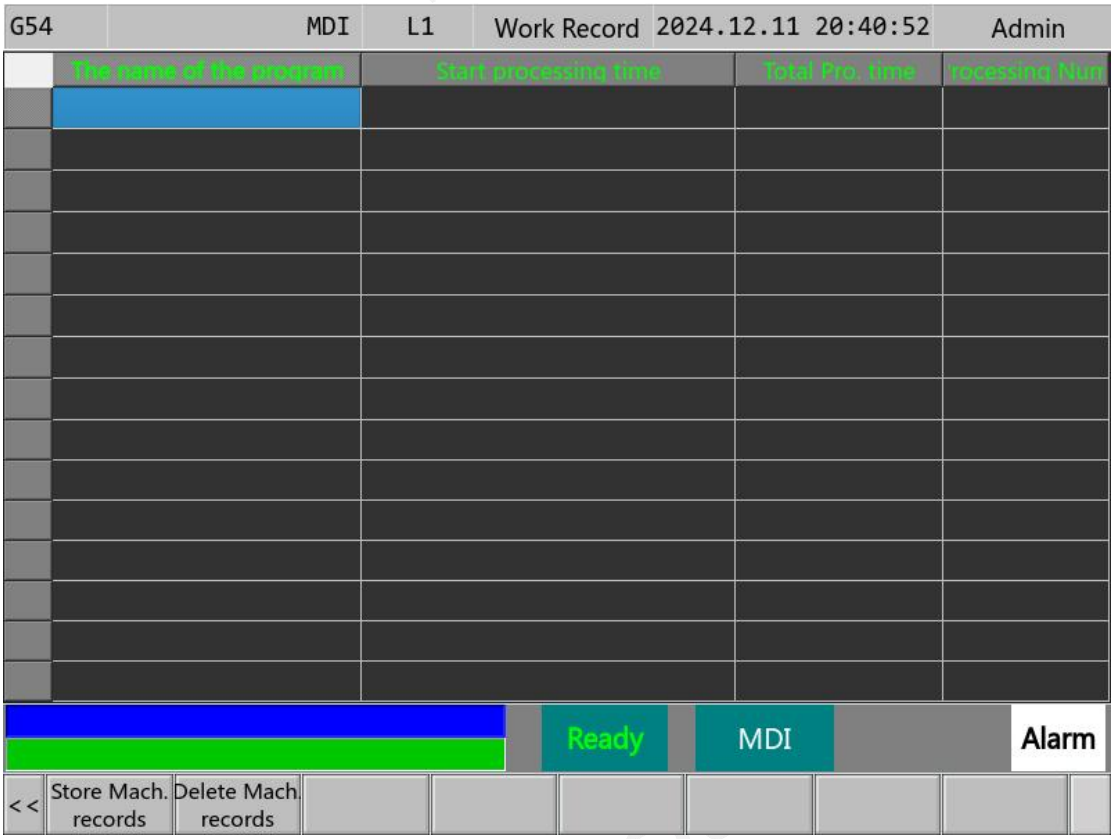
- **Operating Path**

On the "Monitor" page→ F6 [MPG Shift].

- **Explanation**

1. Eject the coordinates of the MPG offset eject.
2. When running the program in automatic or MDI mode, an incremental zero offset is generated by using a handwheel. When the handwheel moves, the workpiece coordinate system does not change, but the mechanical coordinate system changes according to the amount of handwheel movement. The offset is not displayed in the actual value display area of the axis.

1.6.8 Work Record



- **Operating Path**

On the "Monitor" page→ F7 [Work Record].

- **Explanation**

This function is used to record the information of program processing, including program name, starting processing time, total processing time, total processing time, etc.

1.6.9 Clear the accumulated time

- **Operating Path**

On the "Monitor" page→ F8 [Clear Acc. Time].

- **Explanation**

The cumulative machining time is reset to zero.

1.6.10 Graphical adjustments

- **Operating Path**

On the “Monitoring” page→ [>>] → F1 [Graphic Adjustments].

- **Explanation**

Appropriate adjustments can be made to the graphical simulation.

1.6.10.1 Plane selection

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F1[Plane Selection].

- **Explanation**

You can switch between the planes displayed in the graphical simulation.

1.6.10.2 Zoom in

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F2[Zoom in].

- **Explanation**

You can zoom in on the trajectory of the graphical simulation.

1.6.10.3 Zoom out

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F3[Zoom out].

- **Explanation**

You can zoom out on the trajectory of the graphical simulation.

1.6.10.4 Zoom Org

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F4[Zoom Org].

- **Explanation**

The trajectory of the graphic simulation can be moderately processed, and the position

can be automatically adjusted to see all the tracks.

1.6.10.5 Clear item

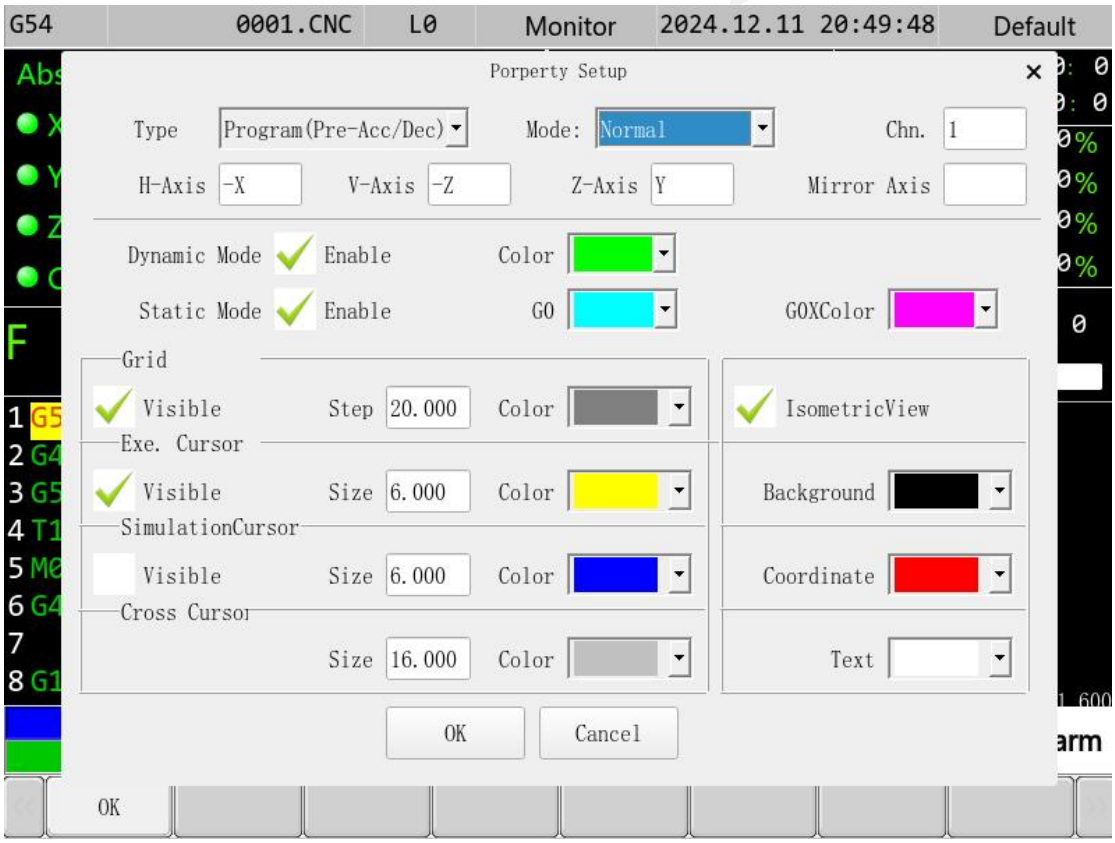
- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F5[Clear item].

- **Explanation**

You can clear the trajectory of the drawing simulation.

1.6.10.6 Set up



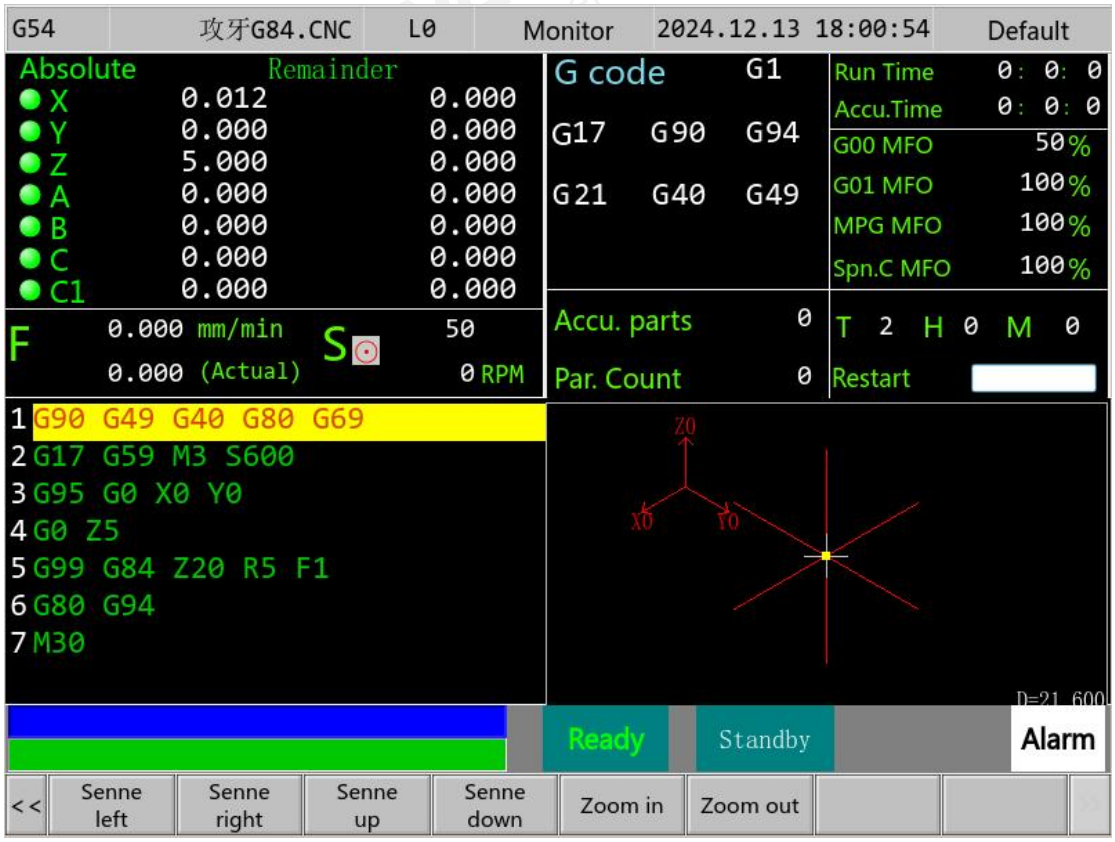
- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F6[Set up].

- **Explanation**

This function sets settings such as the color of the drawing, the color of the layer, and the color of the background.

1.6.10.7 Windows adjustments



- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→ F7[Win. Adj.].

- **Explanation**

This feature moves the viewport of the graphics simulator.

1.6.10.7.1 Senne Left

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F1 [Senne Left]

- **Explanation**

This feature allows you to move the viewport on the graphics simulator to the left.

1.6.10.7.2 Senne Right

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F2 [Senne Right]

- **Explanation**

This feature allows you to move the viewport on the graphics simulator to the right.

1.6.10.7.3 Senne Up

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F3 [Senne Up]

- **Explanation**

This feature allows you to move the viewport on the graphics simulator upwards.

1.6.10.7.4 Senne Down

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F4 [Senne Down]

- **Explanation**

This feature allows the viewport on the graphics simulator to be moved downward.

1.6.10.7.5 Zoom In

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F5 [Zoom In]

- **Explanation**

You can zoom in on the trajectory of the graphical simulation.

1.6.10.7.6 Zoom out

- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F7[Win. Adj.]→F6 [Zoom Out]

- **Explanation**

You can zoom out on the trajectory of the graphical simulation.

1.6.10.8 Static tracing

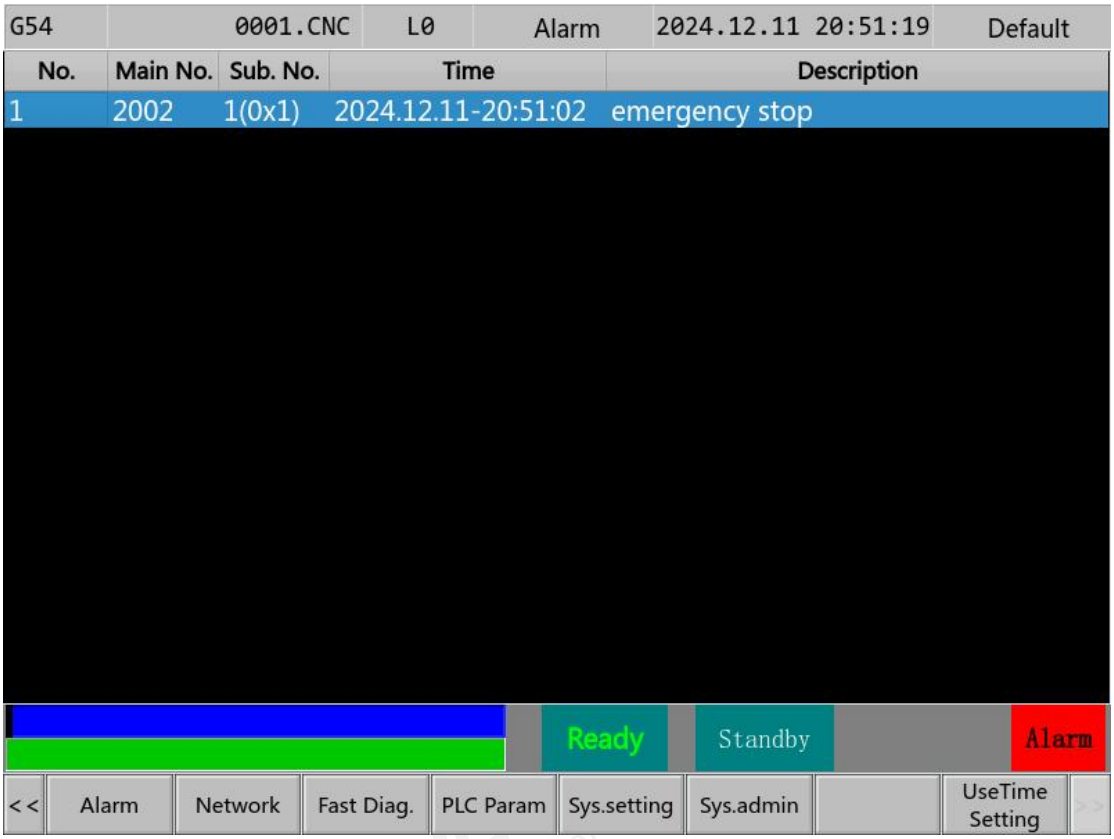
- **Operating Path**

On the “Monitor” page→[>>]→F1[Graphic Adjustments]→F8[Static tracing]

- **Explanation**

1. Scan the content of the program;
2. After the scanning is completed, the graphical simulation will be carried out according to the program content until the program is fully simulated.

1.7 Maintain(Help/Alert)



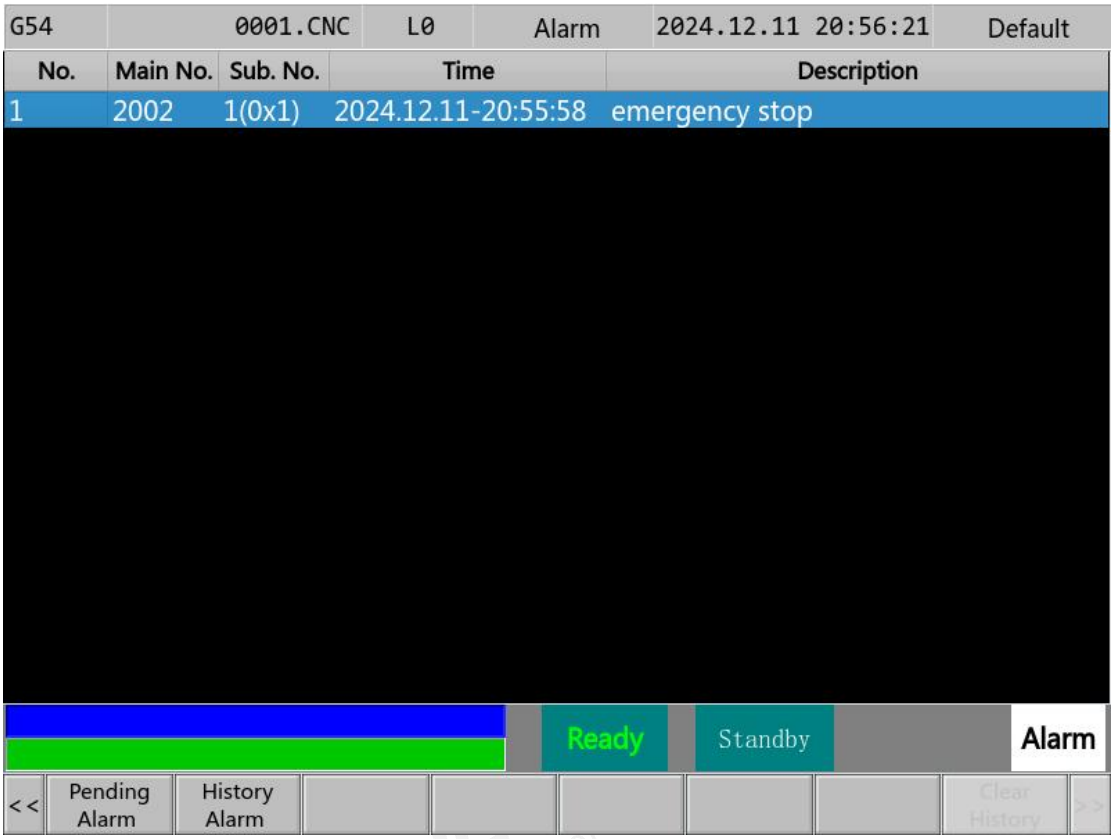
- **Operating Path**

1. Path 1: "Machine Coordinates" page→ F5 [Maintain].
2. Path 2: The [HELP/ALARM] button on the system panel can switch to the "Alarm" page

- **Explanation**

It is used to view auxiliary functions such as alarm signals, network settings, I/O diagnosis, and data backup

1.7.1 Alarm signs



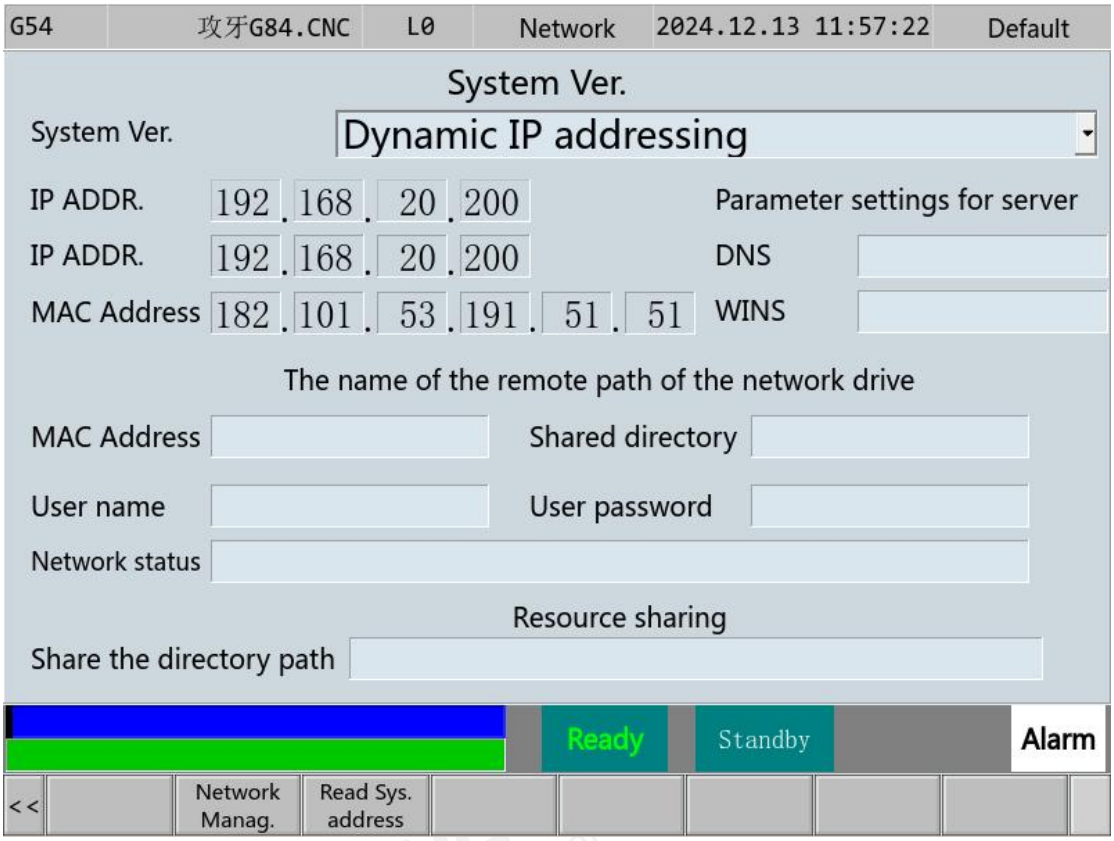
- **Operating Path**

On the "Machine Coordinates" page→ F5 [Maintain]→F1[Alarm]

- **Explanation**

It is used to view the current alarms, historical alerts, and clear historical alerts.

1.7.2 Network Setting



- **Operating Path**

On the "Machine Coordinates" page→ F5 [Maintain]→F2[Network]

- **Explanation**

Used to read and set the IP address of the controller and to view the physical address of the controller.

1.7.2.1 Network management

- **Operating Path**

On the "Machine Coordinates" page→ F5 [Maintain]→F2[Network]→F2[Network Manag.]

- **Explanation**

Used to read and set the IP address of the controller.

1.7.2.1.1 Read IP

- **Operating Path**

On the "Machine Coordinates" page→ F5 [Maintain]→F2[Network]→F2[Network Manag.]→F1[Read IP]

- **Explanation**

This feature is used to read the controller IP address.

1.7.2.1.2 Set IP

- **Operating Path**

On the "Machine Coordinates" page→ F5 [Maintain]→F2[Network]→F2[Network Manag.]→F2[Set IP]

- **Explanation**

This function is used to set the IP address of the controller.

- **Operating instructions**

1. If you want to set the IP address, enter the corresponding IP address in the IP address input box on the page;
2. Press [Set IP], then the IP address of the controller will be set to the value set by the input box.

1.7.2.2 Read the physical address of the system

- **Operating Path**

On the "Machine Coordinates" page→F5 [Maintain]→F2[Network]→F3[Read Sys. address]

- **Explanation**

Used to read and set the IP address of the controller.

1.7.3 Fast diagnosis

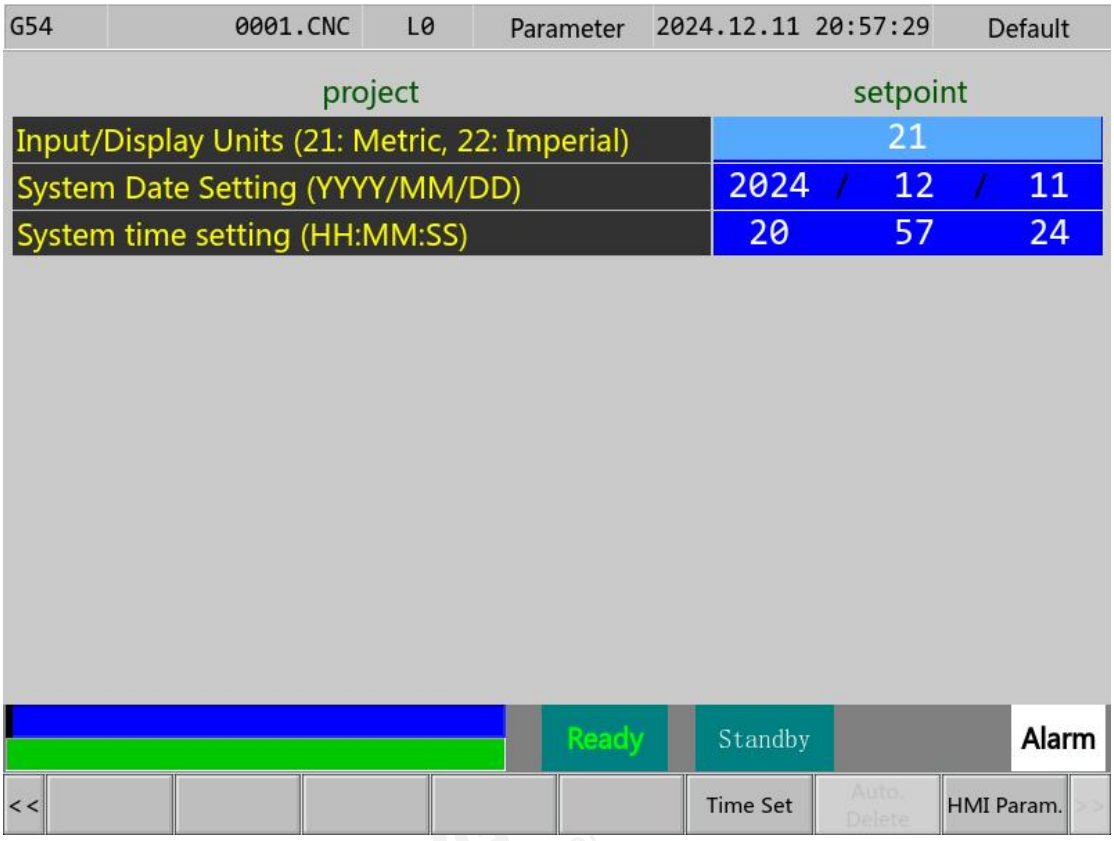
- **Operating Path**

1. Path 1：On the "Machine Coordinates" page→F5 [Maintain]→F3[Fast Diag.]
2. Path 2：On the "Machine Coordinates" page→F7[Fast Diag.]

- **Explanation**

For details, please refer to 1.8 Description.

1.7.4 System settings



- **Operating Path**

On the "Machine Coordinates" page→F5 [Maintain]→F5[Sys. setting]

- **Explanation**

1. Time used to calibrate the controller;
2. It is used to set UI customization functions.

1.7.4.1 Time Set

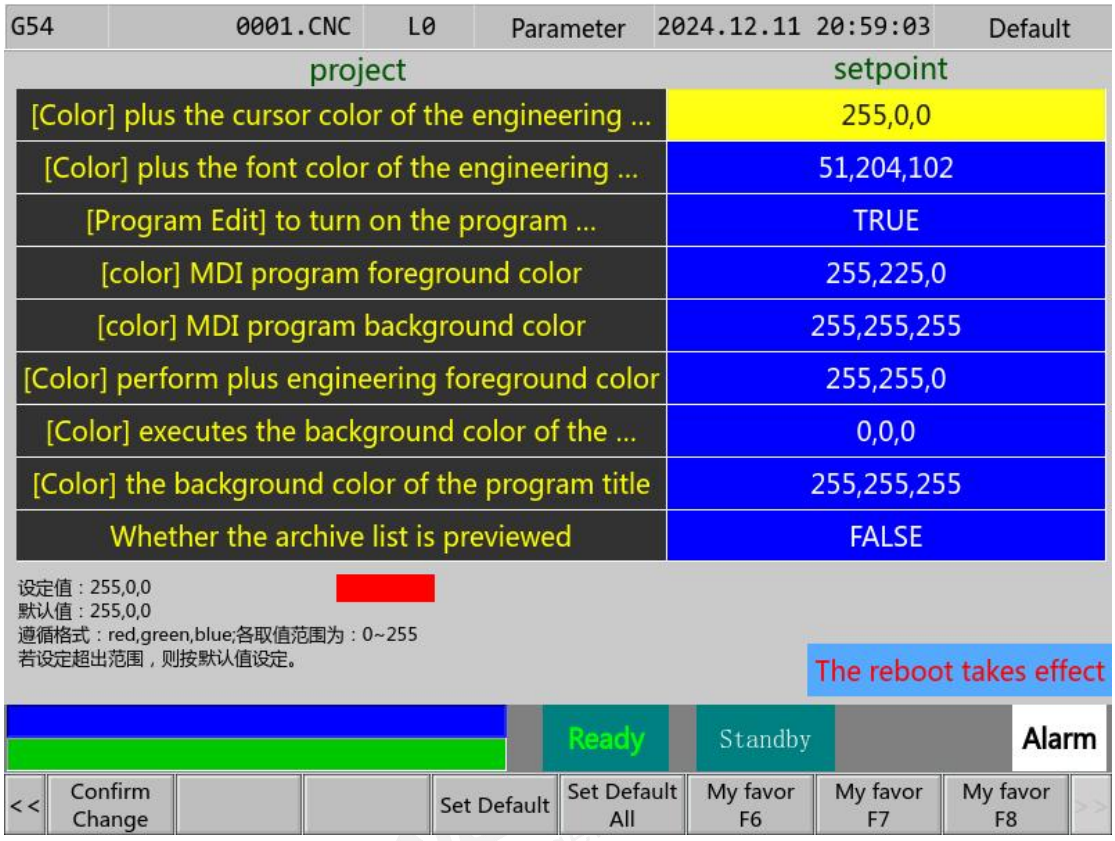
- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F5 [Sys. Settings]→ F6 [Time Set].

- **Explanation**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor to the corresponding position to enter and modify data.
2. After the modification is completed, press the F6[Set Time] button to save the settings.

1.7.4.2 Set MHI parameters



- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F5 [Sys. Settings]→ F8 [HMI Param.].

1.7.4.2.1 Set the plug-in color

- **Explanation**

1. Use the arrow keys [↑] [↓] to move the cursor to the corresponding item to enter and modify the data.
2. 2. After the modification is completed, press the F1[Confirm Change] button to save the settings.
3. 3. After the modification is complete, it needs to be restarted to take effect.

1.7.4.2.2 Single-item are recovered to defaults

- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F5 [Sys. Settings]→ F8 [HMI Param.]→F4[Set Default]

- **Explanation**

Restores the default setting for the cursor row in the setting item.

1.7.4.2.3 All items are restored to default

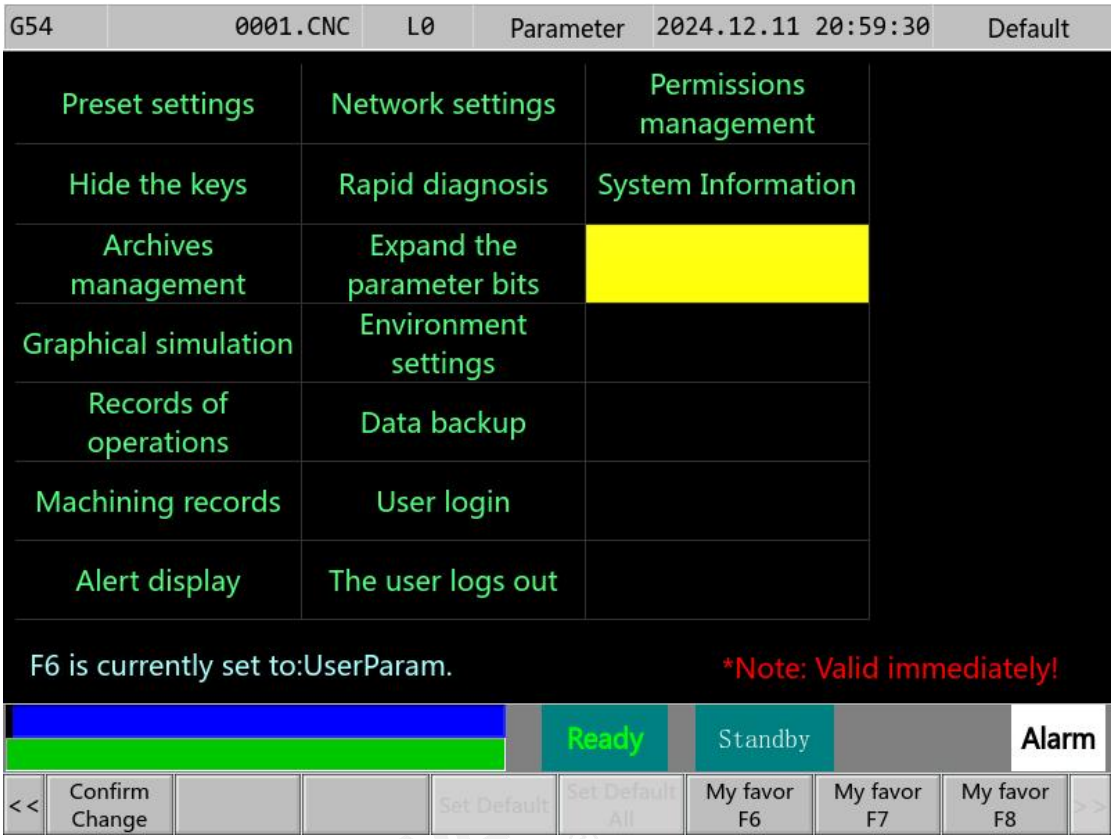
- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F5 [Sys. Settings]→ F8 [HMI Param.]→F5[Set Default All]

- **Explanation**

All items in the settings are restored to the default settings.

1.7.4.2.4 My favor F6



- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F5 [Sys. Settings]→ F8 [HMI Param.]→F6[My favor F6]

- **Explanation**

Set the function of F6 in the "Machine Coordinates" page.

- **Operating instructions**

1. Use the system arrow keys [↑] [↓] [←] [→] to move the cursor to the desired option.
2. Press F1 [Confirm Change] to take effect.

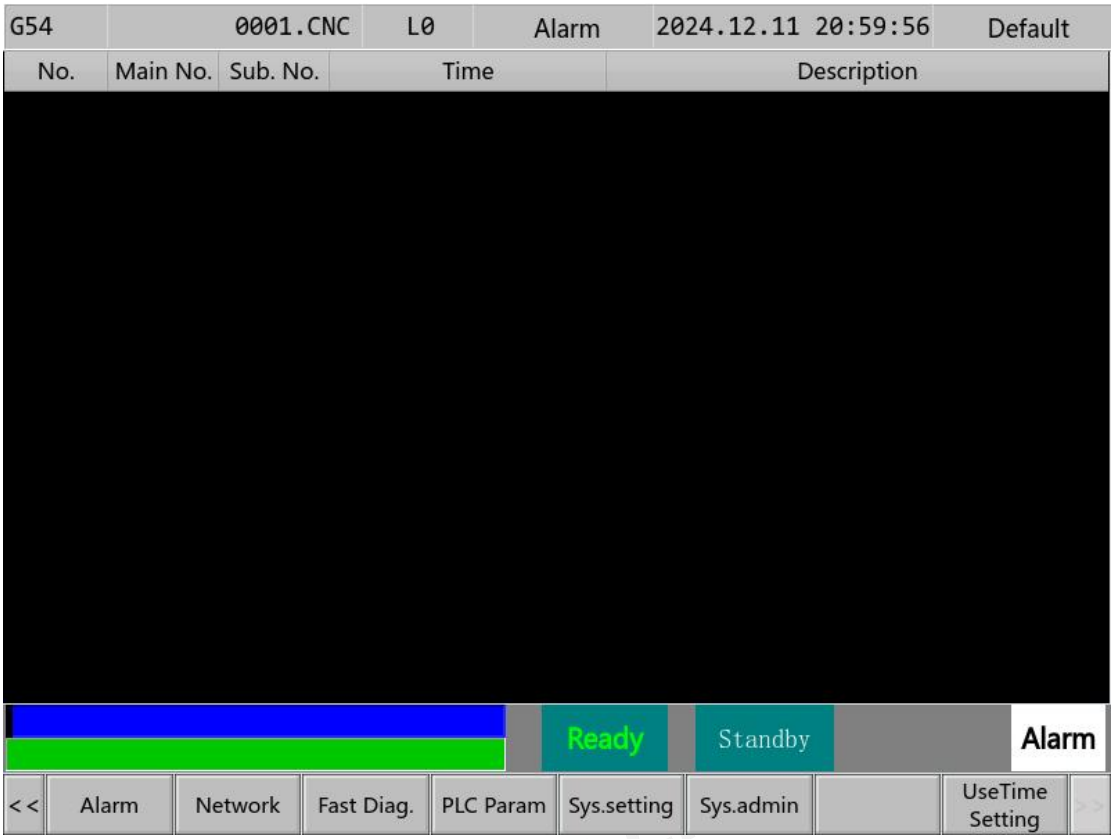
1.7.4.2.5 My favor F7

The same is true for F6 in 1.7.6.2.5.

1.7.4.2.6 My favor F8

The same is true for F6 in 1.7.6.2.5.

1.7.5 The remaining usage time of the CNC(Controller unlocking/locking)



- **Operating Path**

On the "Machine Location" page→ F5 [Maintain]→ F8 [UseTime Settings]

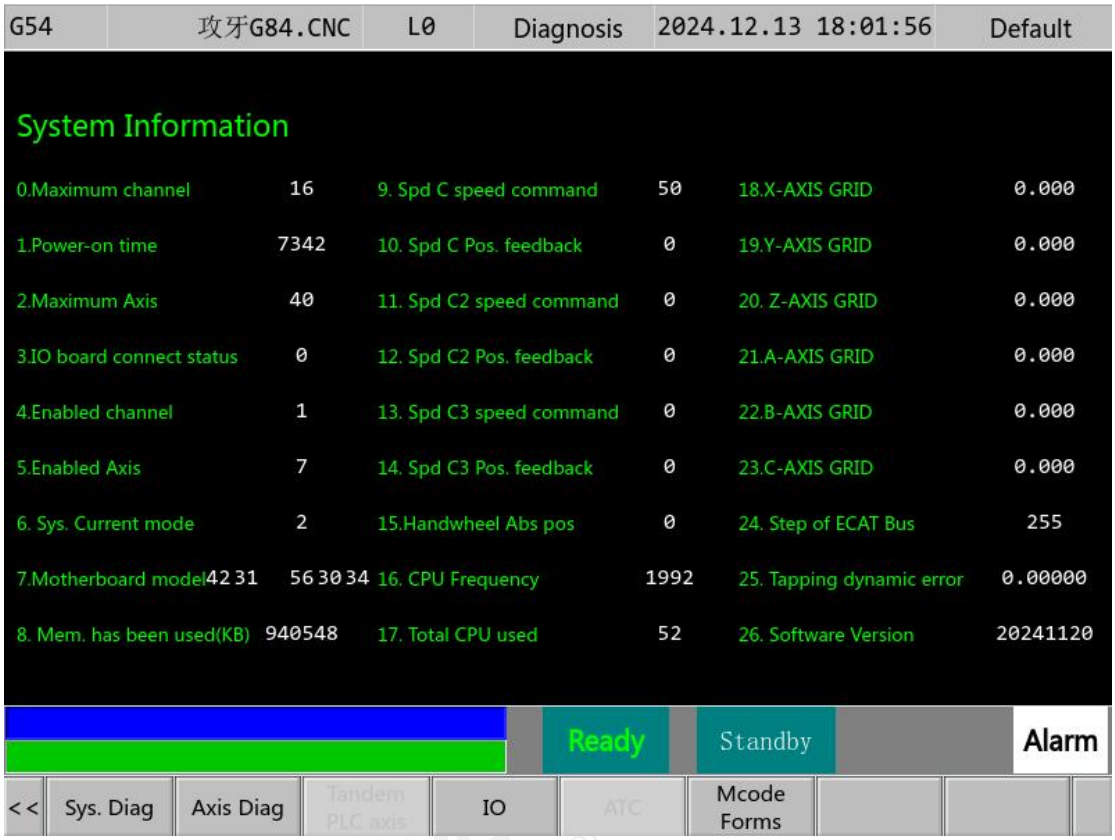
- **Explanation**

1. This function is used to lock and unlock the controller;
2. This function requires administrators and above permissions to operate.

- **Operating instructions**

Please refer to the "Lock-and-Lock Operation" documentation.

1.8 Fast diagnosis



- **Operating Path**

1. Path 1：On the "Machine Coordinates" page→F5 [Maintain]→F3[Fast Diag.]

3. Path 2：On the "Machine Coordinates" page→F7[Fast Diag.]

- **Explanation**

It is used to monitor system information, axial information, and IO diagnosis.

1.8.1 System diagnosis

- **Operating Path**

On the "Machine Coordinates" page→F7[Fast Diag.]

- **Explanation**

Displays some configuration information of the system, spindle command feedback, and axial GRID point distance.

1.8.2 Axial Information (Oscilloscope)

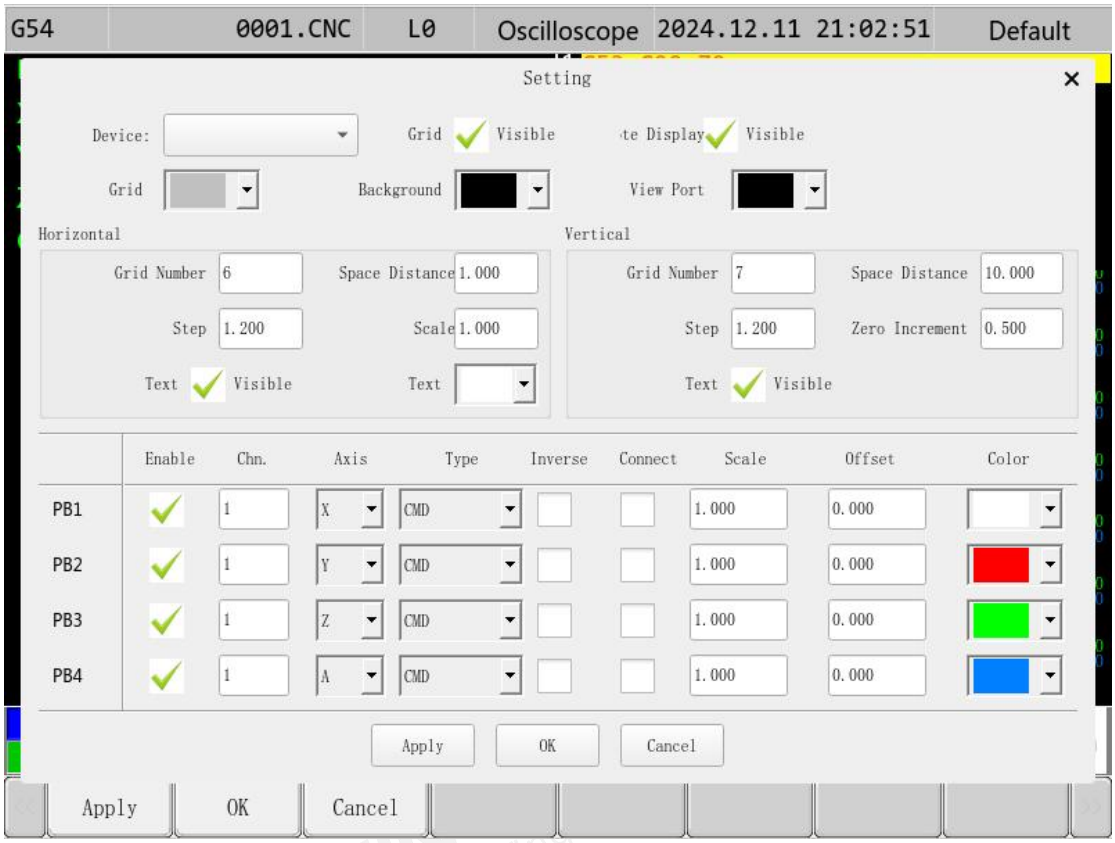
- **Operating Path**

1. Path 1：On the "Machine Coordinates" page→F5 [Maintain]→F3[Fast Diag.]→F2[Axis Diag.]
2. Path 2：On the "Machine Coordinates" page→F7[Fast Diag.]→F2[Axis Diag.]

- **Explanation**

It is used to monitor the waveform in the axial direction.

1.8.2.1 Setting



- **Operating Path**

On the "Machine Coordinates" page→F7[Fast Diag.]→F2[Axis Diag.]→F1[Setting]

- **Explanation**

This function is used to set the channel, axial, and pulse type of the controller corresponding to multiple channels of the oscilloscope. Multiple channels are set up in the same way, and an example of an oscilloscope channel 1 setting is shown below.

- **Operating instructions**

1. Press F1[Settings] button to automatically pop up the setting table;
2. On the PB1 line, enter 1 for the corresponding channel, X for the axis, and enter the command for style. (Axial options include: X, Y, Z, A, B, C, X1, Y1, Z1, A1, B1, C1…… X5, Y5, (type options include: command, feedback, voltage and other six ways);
3. Press the [Apply] button to set the oscilloscope channel;

4. In this way, the waveform of oscilloscope channel 1 is set to the command pulse of the X-axis in system channel 1;
5. You can also set whether the net is displayed, whether the probe is displayed, horizontal layout, vertical layout, waveform negation, waveform linear, waveform proportional zero, and color.

1.8.2.2 Clean

- **Operating Path**

On the “Oscilloscope” page→ F2 [Clean].

- **Explanation**

This key function is used to erase all waveforms from the oscilloscope window.

1.8.2.3 Pause the oscillograph

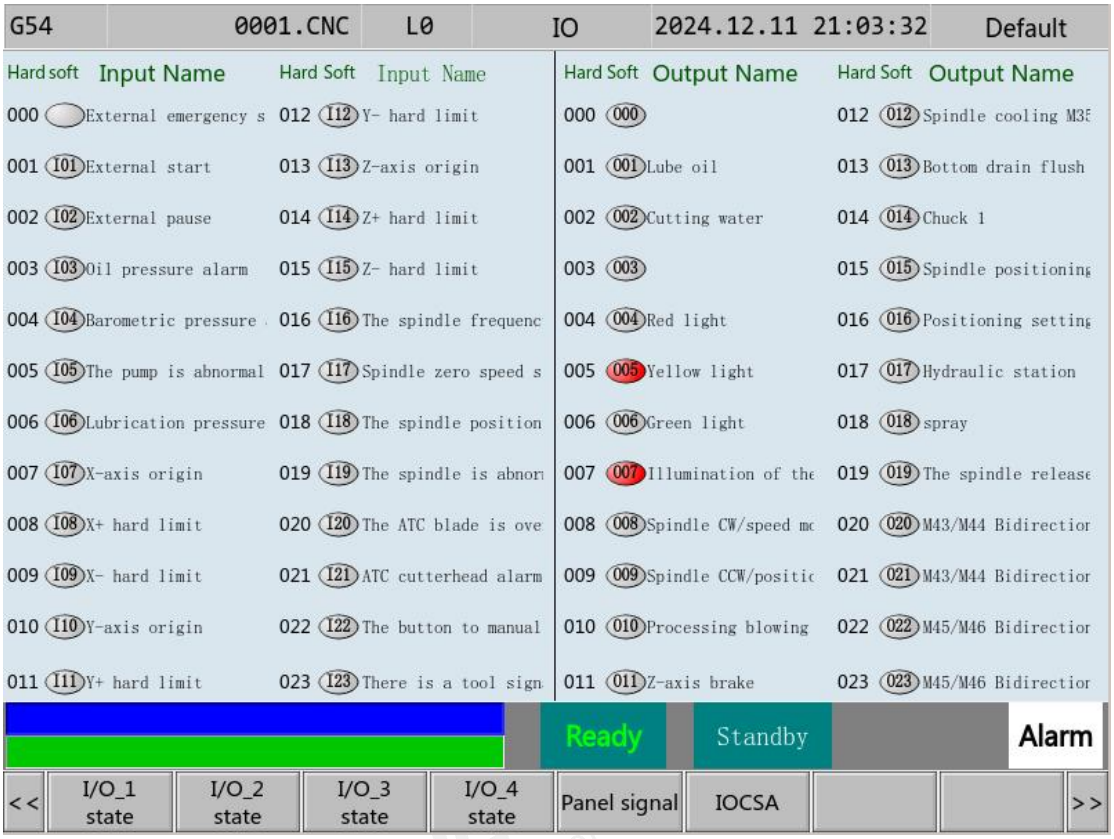
- **Operating Path**

On the “Oscilloscope” page→ F3 [Pause Osc.].

- **Explanation**

Press this button to stop the waveform from refreshing, and keep the current waveform for easy viewing.

1.8.3 IO diagnosis



- **Operating Path**

1. Path 1：On the "Machine Coordinates" page→F5 [Maintain]→F3[Fast Diag.]→F4[IO]
2. Path 2：On the "Machine Coordinates" page→F7[Fast Diag.]→F4[IO]

- **Explanation**

This function is used to monitor the input/output signals.

1.8.3.1 Secondary panel

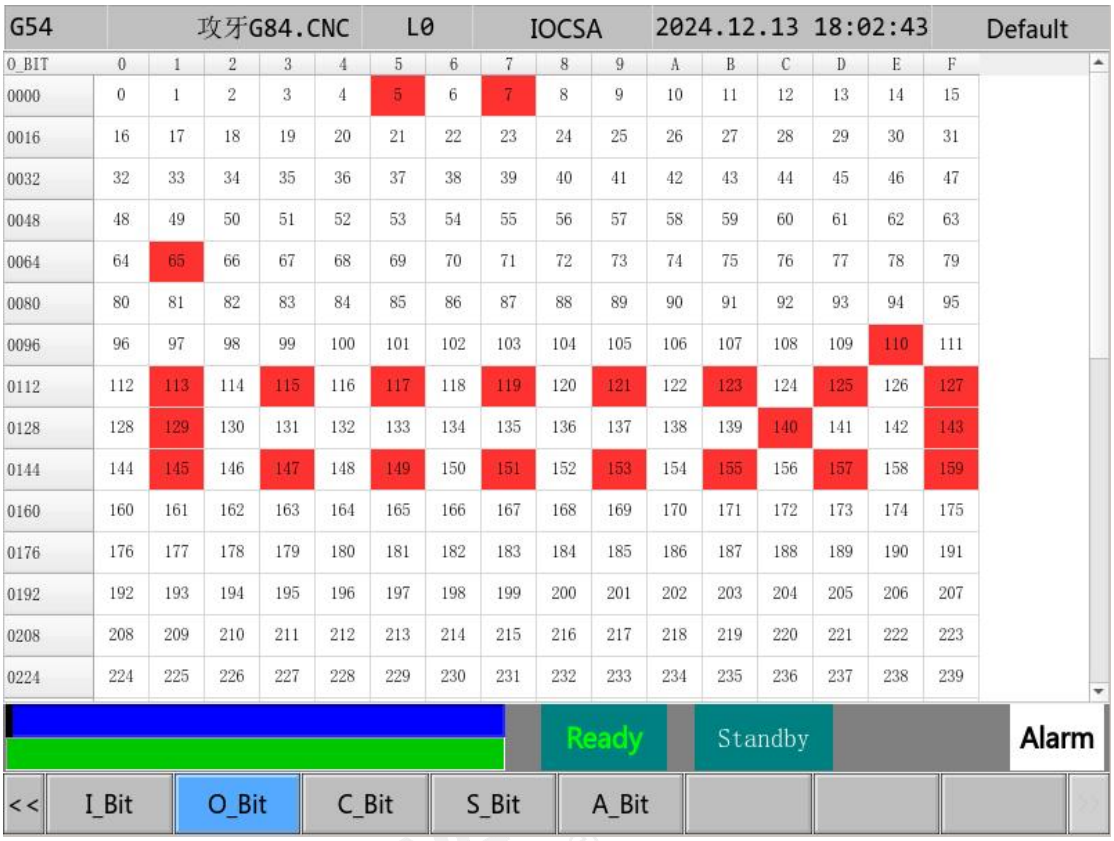
- **Operating Path**

On the "IO diagnosis" page→F5 [Panel]

- **Explanation**

This function is used to monitor the status of the auxiliary panel keys of the controller.

1.8.3.2 IOCSA



O_BIT	0	1	2	3	4	5	6	7	8	9	A	B	C	D	E	F
0000	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
0016	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31
0032	32	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47
0048	48	49	50	51	52	53	54	55	56	57	58	59	60	61	62	63
0064	64	65	66	67	68	69	70	71	72	73	74	75	76	77	78	79
0080	80	81	82	83	84	85	86	87	88	89	90	91	92	93	94	95
0096	96	97	98	99	100	101	102	103	104	105	106	107	108	109	110	111
0112	112	113	114	115	116	117	118	119	120	121	122	123	124	125	126	127
0128	128	129	130	131	132	133	134	135	136	137	138	139	140	141	142	143
0144	144	145	146	147	148	149	150	151	152	153	154	155	156	157	158	159
0160	160	161	162	163	164	165	166	167	168	169	170	171	172	173	174	175
0176	176	177	178	179	180	181	182	183	184	185	186	187	188	189	190	191
0192	192	193	194	195	196	197	198	199	200	201	202	203	204	205	206	207
0208	208	209	210	211	212	213	214	215	216	217	218	219	220	221	222	223
0224	224	225	226	227	228	229	230	231	232	233	234	235	236	237	238	239

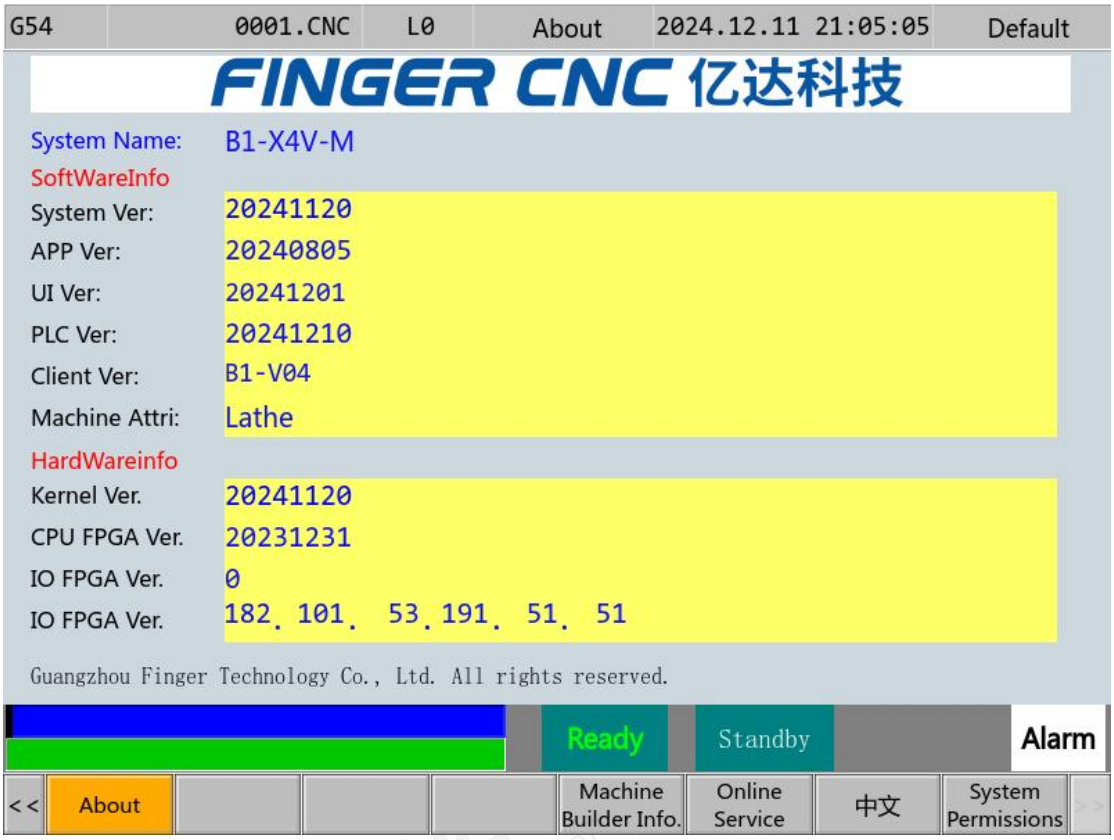
- **Operating Path**

On the "IO diagnosis" page→F6 [IOCSA]

- **Explanation**

1. This function is used to monitor the software IOCSA status of the controller;
2. If the background color is red, it means that the bit is On.

1.9 About System Information



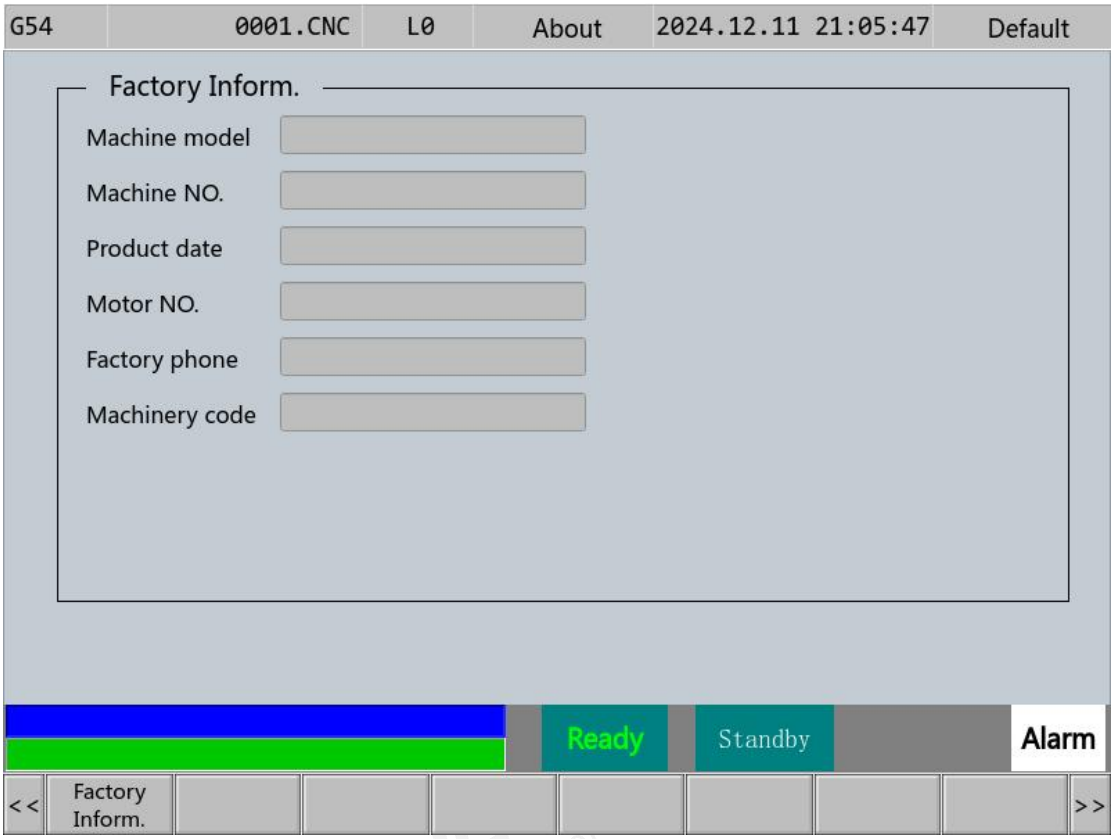
- **Operating Path**

On the "Machine Coordinates" page→ F8 [About].

- **Explanation**

This page is used to view the controller's software information, hardware information, equipment factory information, repair service, Chinese and English quick switching, permission management, etc.

1.9.1 Factory information



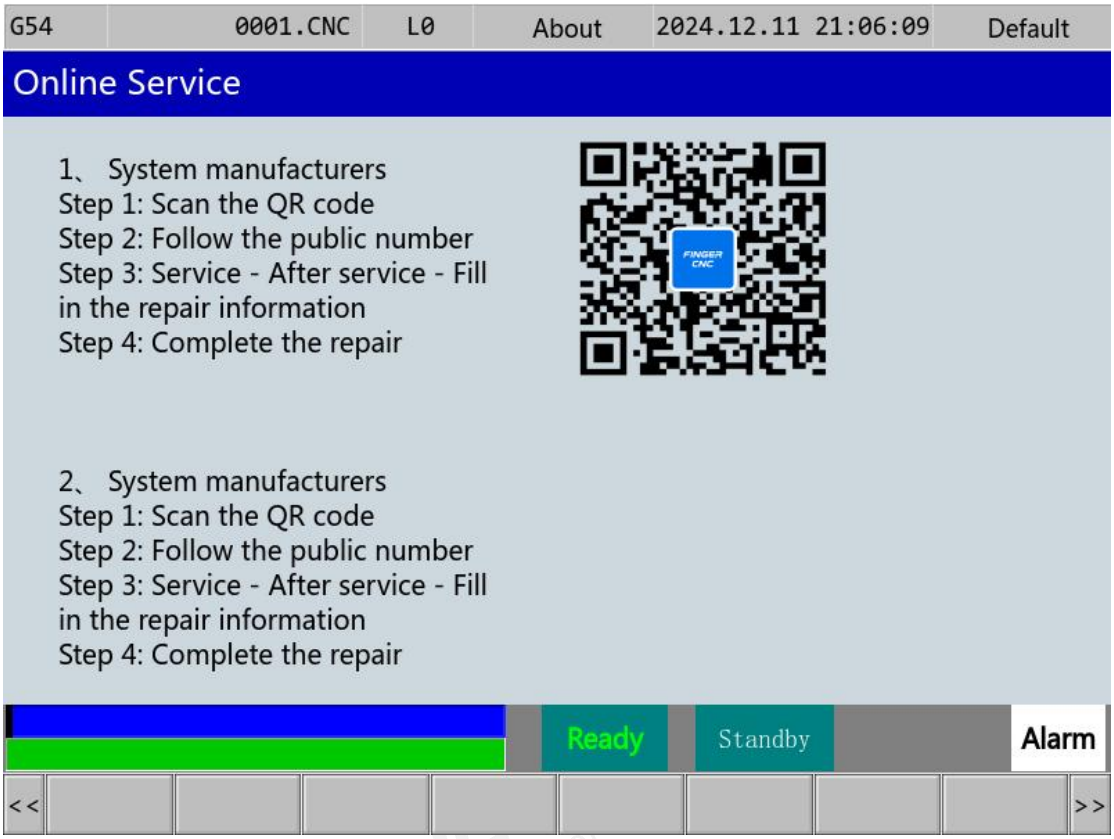
- **Operating Path**

On the "Machine Coordinates" page→ F8 [About] → F5 [Factory Information].

- **Explanation**

You can set the relevant information only after logging in to the device factory permission.

1.9.2 Online Service



- **Operating Path**

On the "Machine Coordinates" page→F8 [About] →F6 [Online Service].

- **Explanation**

Follow the instructions on the page.

1.9.3 Quick switch between Chinese and English

- **Operating Path**

On the "Machine Coordinates" page→F8 [About] →F7 [English/Chinese].

- **Explanation**

Quick switch button for Chinese and English languages.

1.9.4 System Permissions

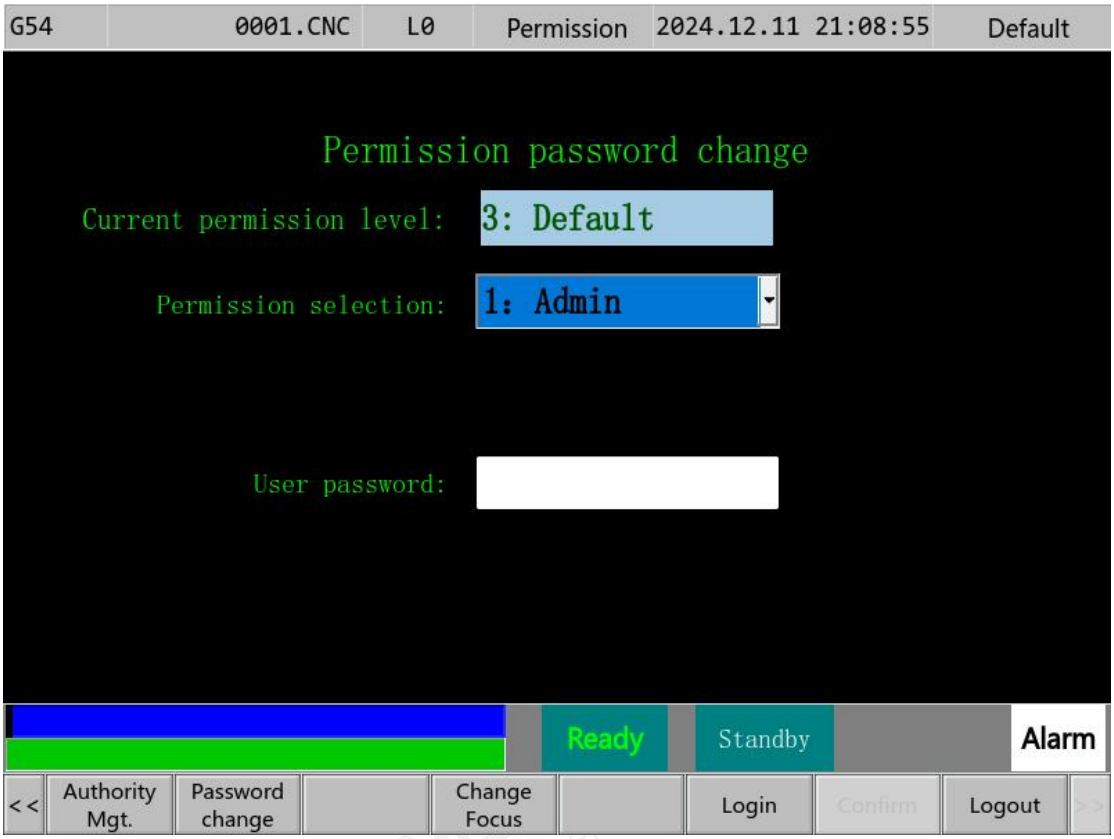
- **Operating Path**

On the "Machine Coordinates" page→F8 [About] →F8 [System Permissions].

- **Explanation**

1. You can log in to verify the current operation permission;
2. Currently, there are three types of permissions: not logged in, user, and manager.

1.9.4.1 Login permissions



- **Operating Path**

On the "Machine Coordinates" page→F8 [About] →F8 [System Permissions].

- **Explanation**

1. Permission acquisition: Login permission can be verified;
2. Password modification: You can switch to the permission password modification

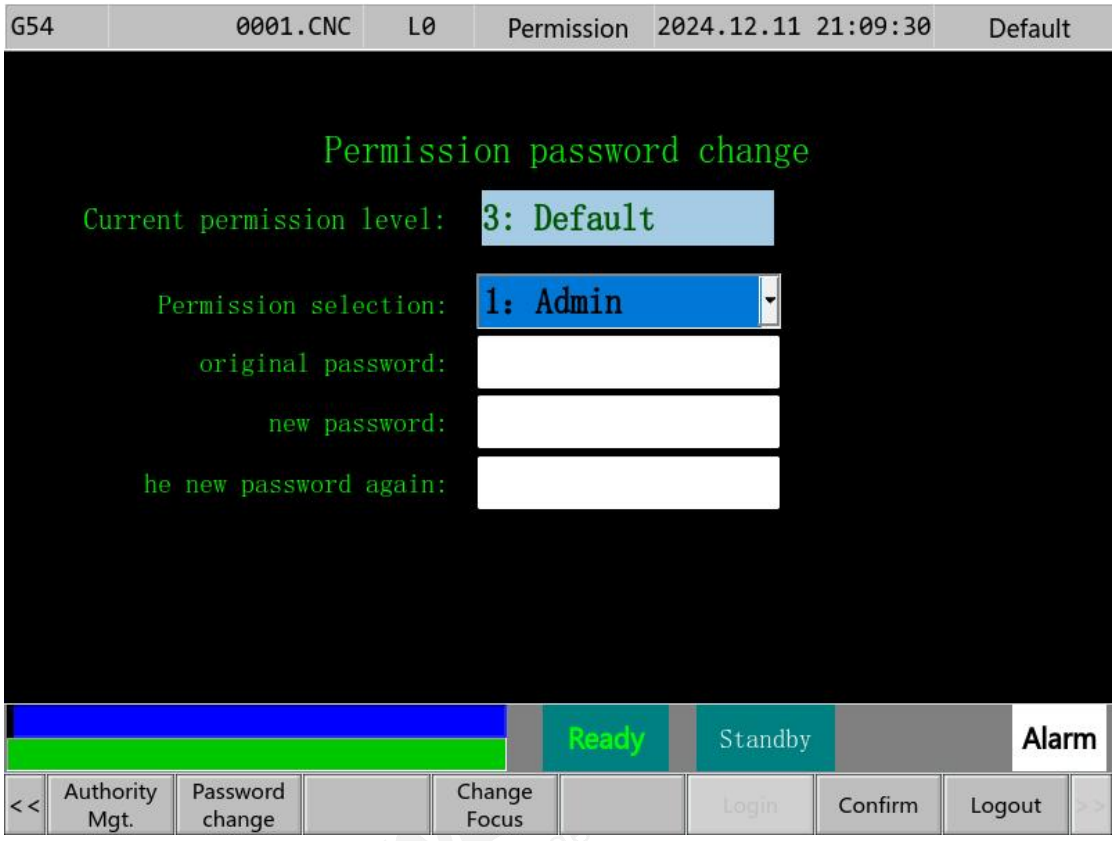
interface;

3. Switch input box: switch the input focus of the screen;
4. Login: Login permissions.

- **Operating instructions**

1. Select the corresponding permissions.
2. Switch the input box and enter the permission password.
3. Login Permissions.

1.9.4.2 Change the permission password



- **Operating Path**

On the "Machine Coordinates" page→F8 [About]→F8 [System Permissions]→F2 [Password change].

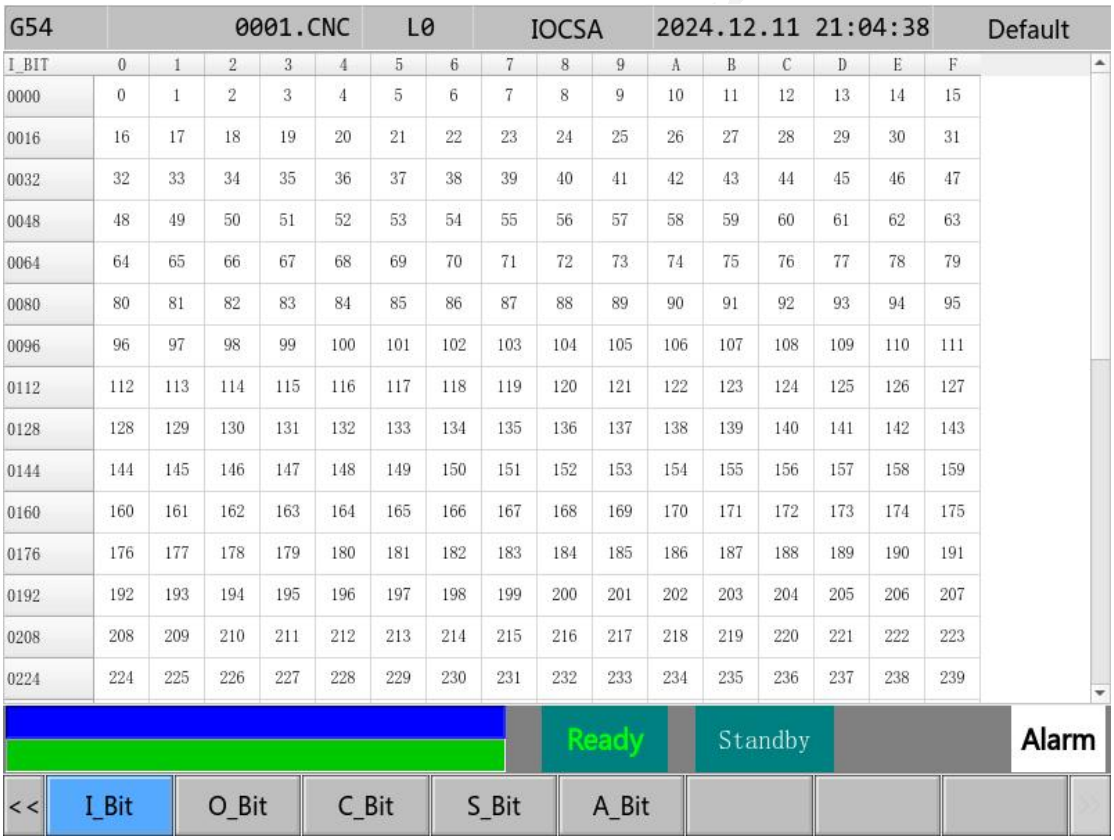
- **Explanation**

1. Permission acquisition: Login permission can be verified;
2. Password modification: You can switch to the permission password modification interface;
3. Switch input box: switch the input focus of the screen;
4. Confirm the password change: Confirm the password change.

- **Operating instructions**

1. Enter the original password.
2. Enter the new password twice;
3. F7[Confirm] to change password.

1.10 PLC status



- **Operating Path**

On the "Machine Coordinates" page→ [>>] → F1 [PLC Status].

- **Explanation**

1. This function is used to monitor the software IOCSA status of the controller;
2. 2. If the background color is red, it means that the bit is ON.

1.11 Diagnostic features

- **Operating Path**

On the "Machine Coordinates" page→ [>>] → F2 [Diag.].

- **Explanation**

This function is used to monitor the variable status of the controller.

1.11.1 System information

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F2 [Diag.]→F1 [Sys. Info.]

- **Explanation**

The address of the SYS variable associated with the monitoring system.

1.11.2 Comment variables

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F2 [Diag.]→F2[Comment Var.]

- **Explanation**

Monitor the address of the USR variable associated with the system.

1.11.3 Program variables

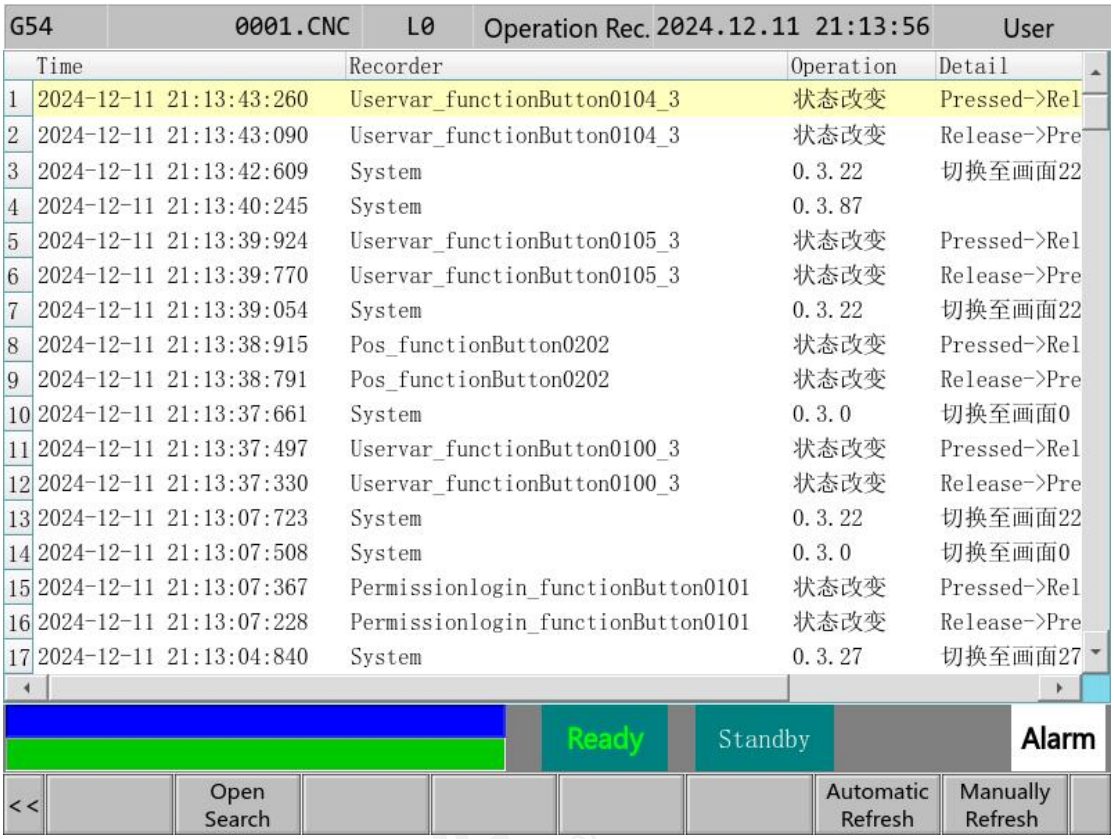
- **Operating Path**

On the "Machine Coordinates" page→[>>]→F2 [Diag.]→F3[Program Var.]

- **Explanation**

Monitor the address of the REG variable associated with the system.

1.11.4 Records of operations



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F2 [Diag.]→F4[Operation Record]

- **Explanation**

View the system's action history form.

1.11.4.1 Search

- **Operating Path**

On the "Operation Record" page→ F2 [Open Search]

- **Explanation**

1. A prompt box pops up
2. F1 Change focus
3. F2 Search All

4. F3 searches by time

1.11.4.2 Automatic refresh

- **Operating Path**

On the "Operation Record" page→ F7 [Automatic refresh]

- **Explanation**

Automatically refresh the operation record

1.11.4.3 Manually refresh

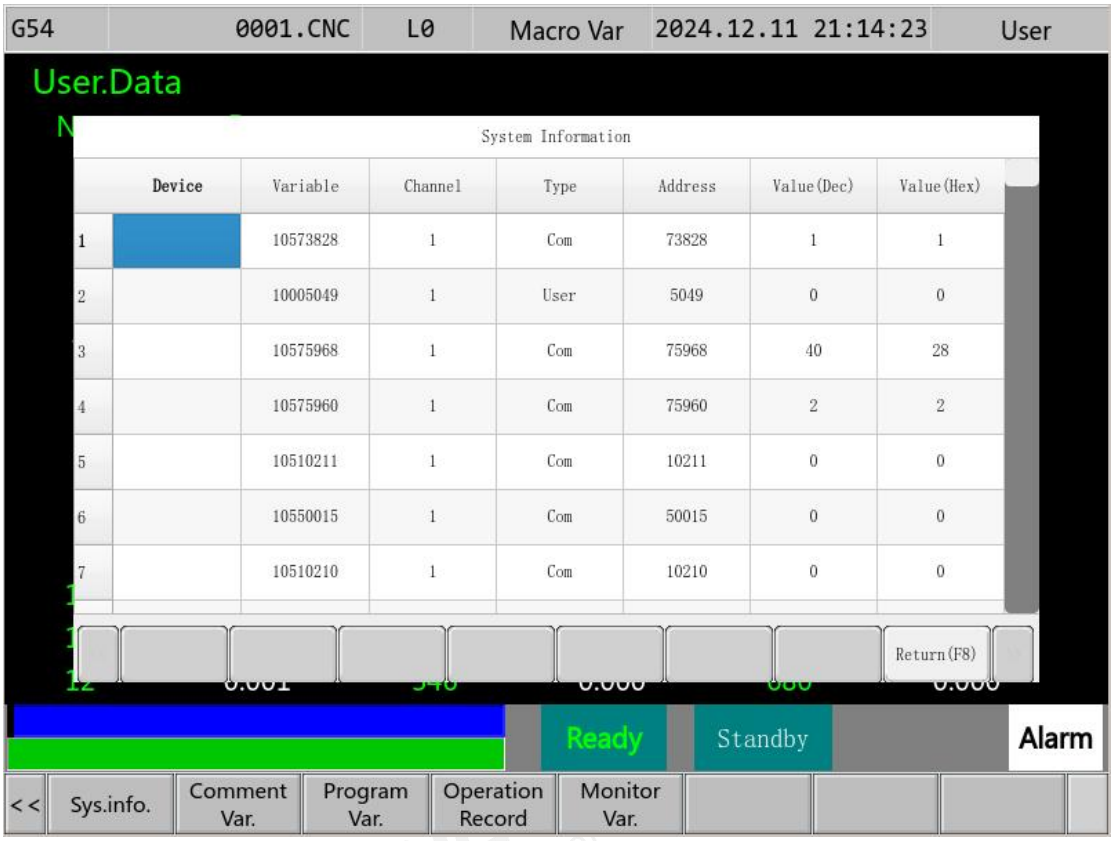
- **Operating Path**

On the "Operation Record" page→ F8 [Manually refresh]

- **Explanation**

Manually refresh the operation record

1.11.5 Monitor variables



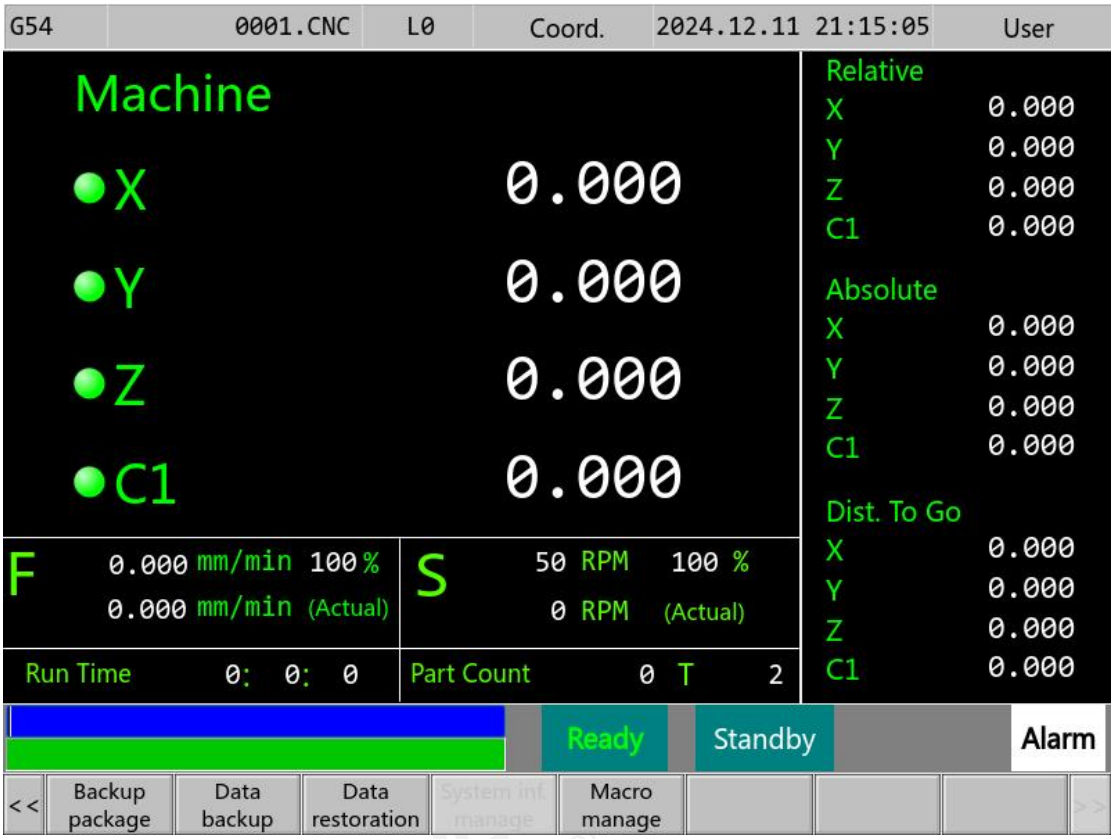
- **Operating Path**

On the "Machine Coordinates" page→[>>]→F2 [Diag.]→F5[Monitor Var.]

- **Explanation**

This function is used to view the variable values of the system.

1.12 System administration



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→Verify permissions.

- **Explanation**

1. This function is used to back up the controller-related data and convert it into a package.
2. After entering the data backup interface, press F8 "Return" to go to the backup package management interface.
3. After entering the backup package management interface, press F8 "Cancel" to go to the system data restoration interface.

1.12.1 Backup package

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→F1 [Backup package]

- **Explanation**

1. You can view all backup packages of the current controller;
2. It can be copied to a USB flash drive.

1.12.2 Date backup

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→F2[Date Backup]

- **Explanation**

1. It can back up the contents of the current controller;
2. You can choose to back up the contents.

- **Operating instructions**

1. On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→Verify permissions→F2[Date Backup];
2. The F1 and F2 buttons can be moved up and down, and F4 can be used to select files or not;
3. After selecting the file content that needs to be backed up, the F6 button will be used to back up;
4. Wait for the progress bar to be 100% ready to complete the backup.

1.12.3 Data restoration

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→Verify permissions→F3[Data restoration];

- **Explanation**

You can restore previously backed up files.

- **Operating instructions**

1. On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→Verify permissions→F3[Data restoration]
2. The F1 and F2 buttons can be moved up and down, and F4 can be used to update and restore the files.
3. Select the content of the file to be updated, and then click the F3 button to update.
4. Wait for the progress bar to reach 100%, prompting restart, restart.

1.12.4 System information management

- **Operating Path**

ignore

1.12.5 Macro manage

- **Operating Path**

On the "Machine Coordinates" page→[>>]→F4 [Sys. Admin]→Verify permissions→F5[Macro manage];

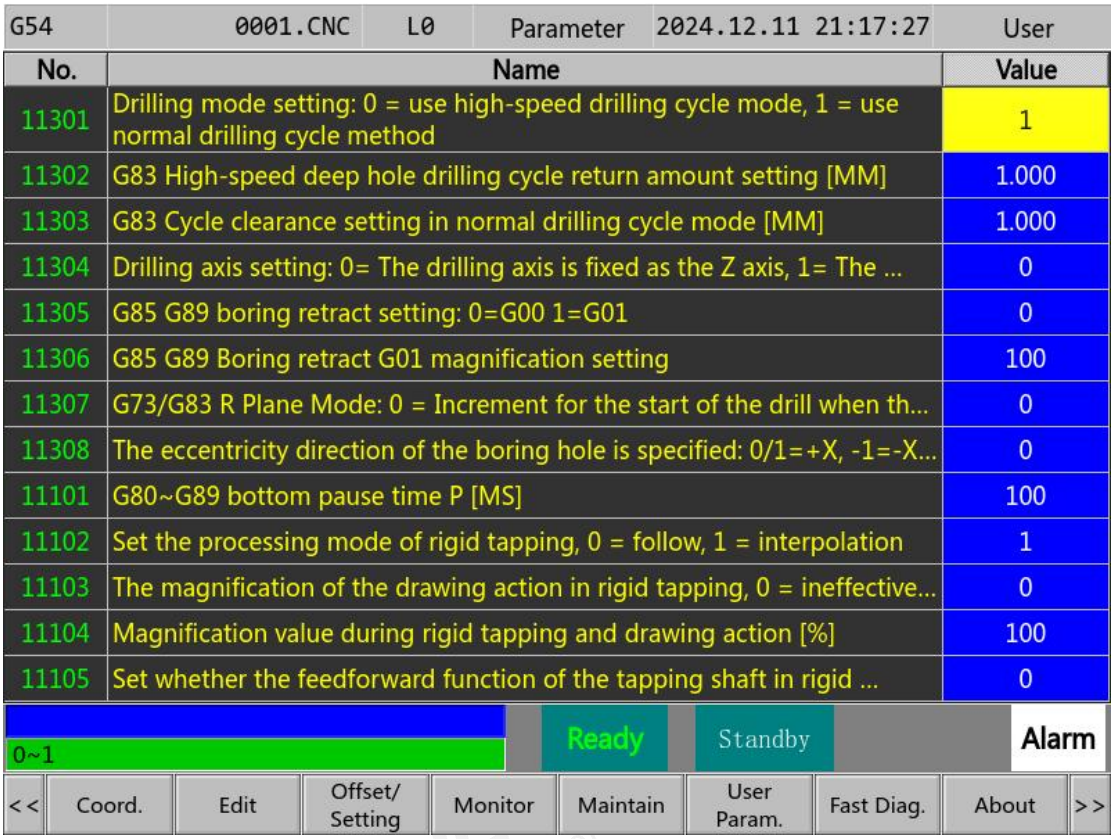
- **Explanation**

1. This function is used to set the configuration table of macro jumps for developers to use;
2. This function requires administrators and above permissions to operate.

- **Operating instructions**

1. Set the letter, value, type, file name, priority, modality, and whether to use the jump on the configuration table.
2. Press F2 [Save chain table] to automatically generate the configuration file.

1.13 User parameters



- **Operating Path**

On the "Machine Coordinates" page→ F6 [User Param.].

- **Explanation**

This function is used by the user to modify the parameters commonly used in machining.

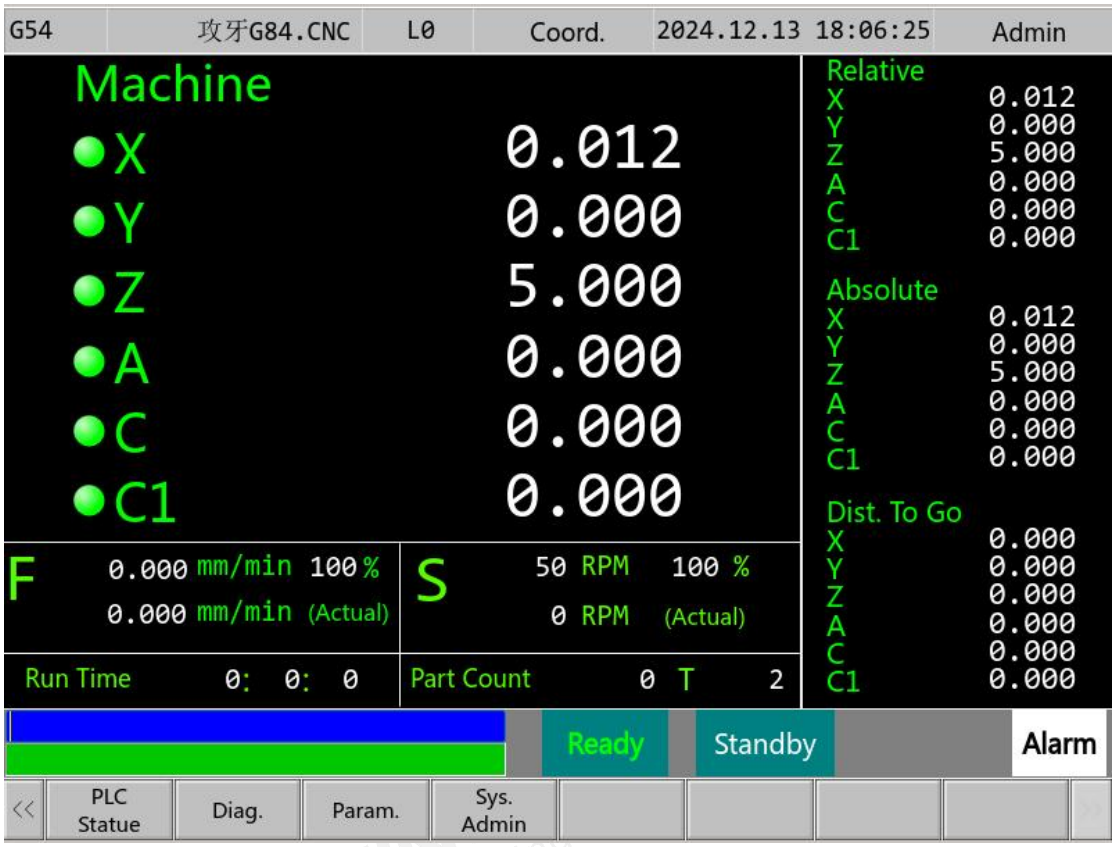
- **Operating instructions**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
2. Use [⩓] [⩔] to turn pages up and down;
3. Use the [Enter] button to enter.

- **Note**

After the parameter is modified, you need to press the [Reset] button to reset it before it can take effect.

1.14 Parameter setting



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F3 [Param.].

- **Explanation**

This function can set the control parameters of each axis of the controller, user parameters and other auxiliary parameters.

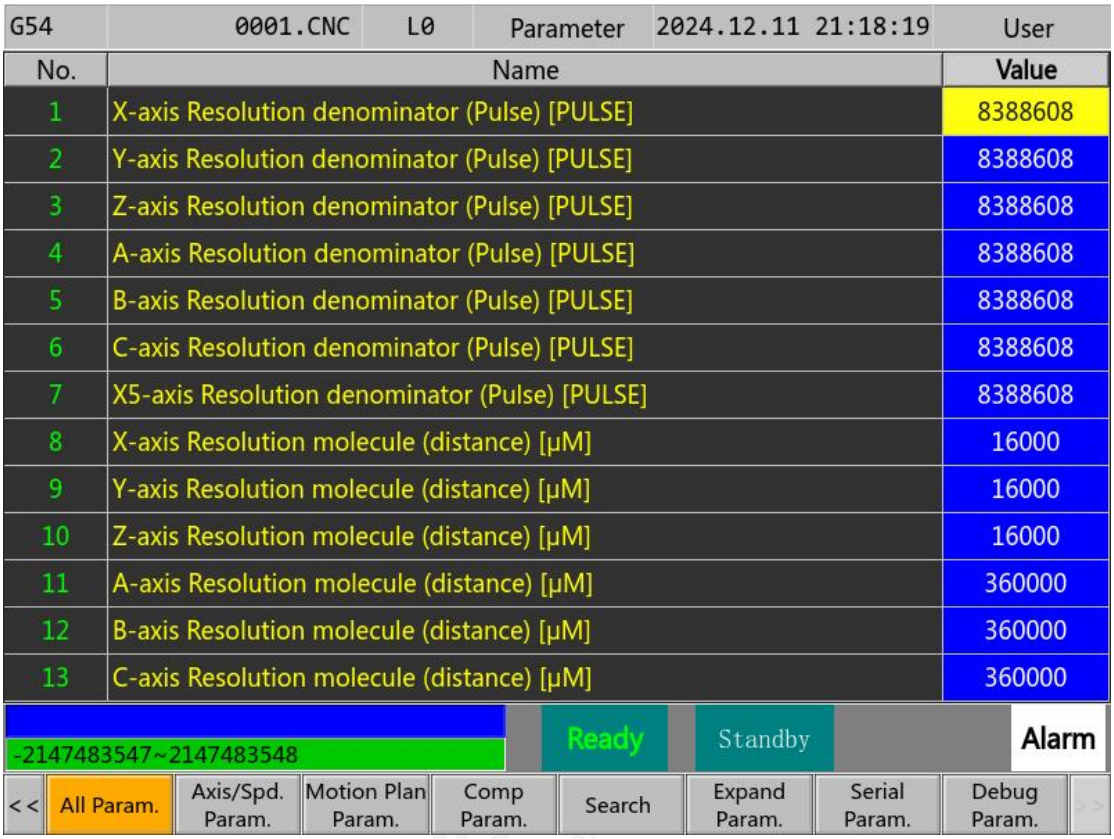
- **Operating instructions**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor;
2. Use [⩓] [⩔] to turn pages up and down;
3. Use the [Enter] button to enter.

- **Note**

After the parameter is modified, you need to press the [Reset] button to reset it before it can take effect.

1.14.1 All parameters



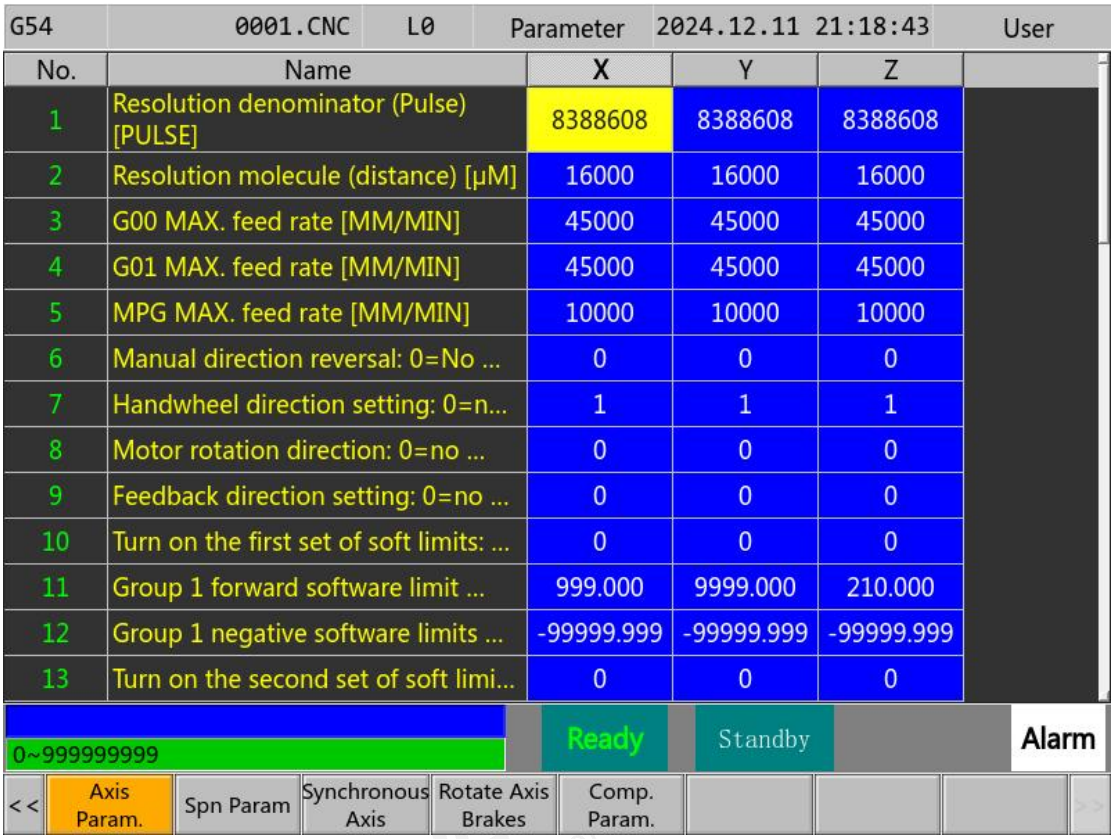
- **Operating Path**

On the "Machine Coordinates" page→[>>]→F3 [Param.].

- **Explanation**

You can search for parameters corresponding to the serial number

1.14.2 Axis/spindle parameters



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F3 [Param.]→F2 [Axis/Spd. Param.]

1.14.2.1 Axis/spindle parameters

For details, please refer to the corresponding function manual

1.14.2.2 Spindle parameters

For details, please refer to the corresponding function manual

1.14.2.3 Synchronized axes

For details, please refer to the corresponding function manual

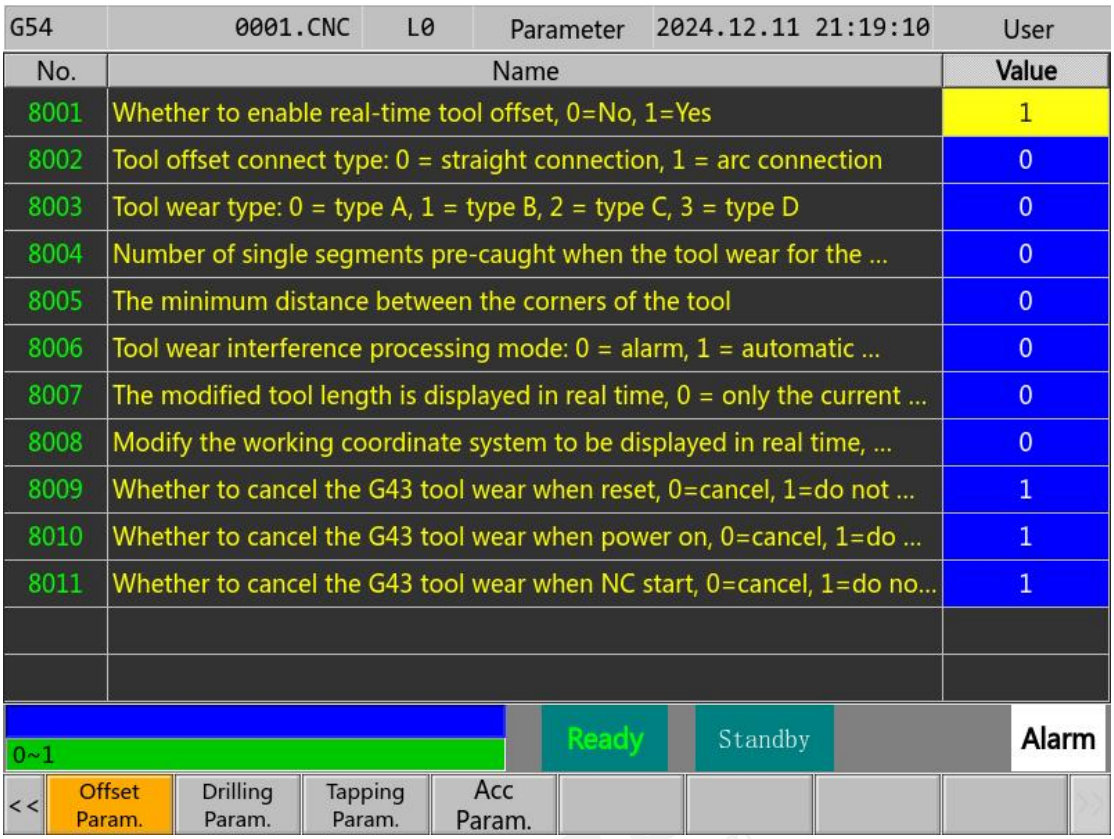
1.14.2.4 Rotate axis brakes

For details, please refer to the corresponding function manual

1.14.2.5 Comprehensive parameters

For details, please refer to the corresponding function manual

1.14.3 Motion plan parameters



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F3 [Param.]→F3 [Motion Plan Param.]

1.14.3.1 Offset parameters

For details, please refer to the corresponding function manual

1.14.3.2 Drilling parameters

For details, please refer to the corresponding function manual

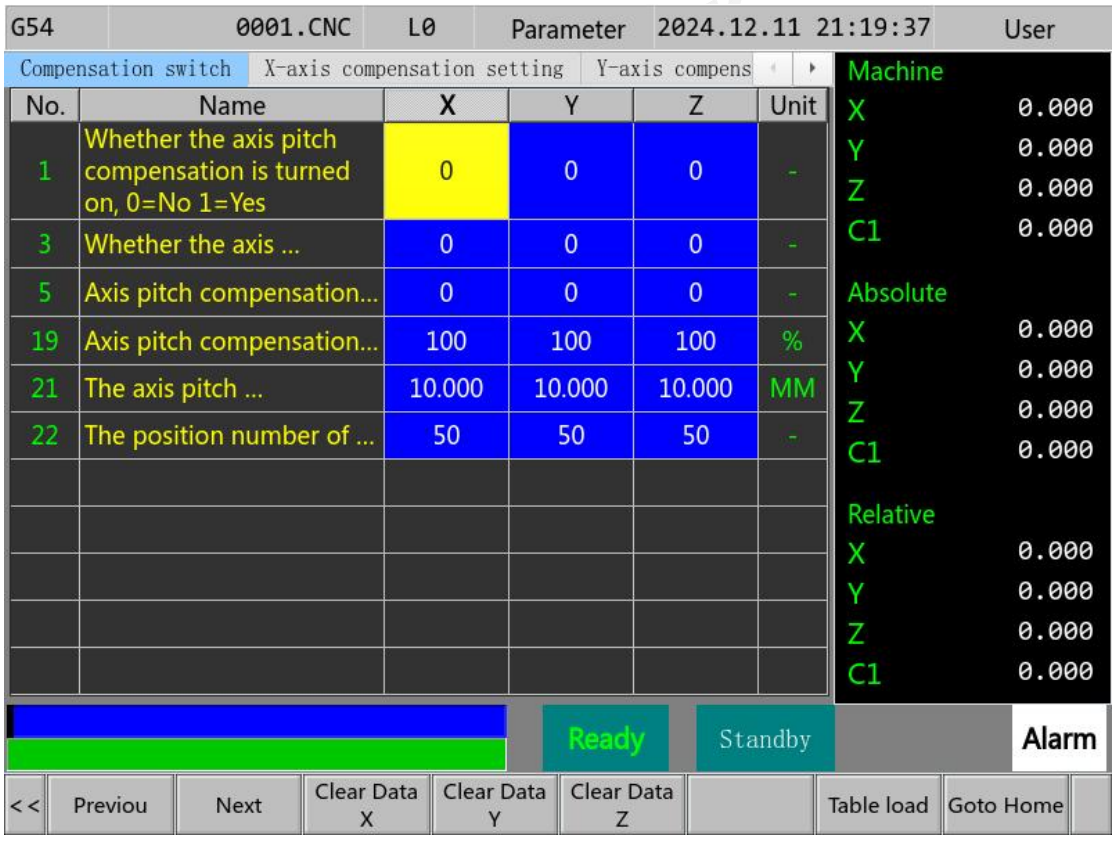
1.14.3.3 Tapping parameters

For details, please refer to the corresponding function manual

1.14.3.4 Acceleration and deceleration parameters

For details, please refer to the corresponding function manual

1.14.4 Compensation parameters



- **Operating Path**

On the "Machine Coordinates" page→[>>]→F3 [Param.]→F4 [Comp Param.]

- **Explanation**

Pitch compensation function to compensate for the error of the screw.

1.14.4.1 Previou

- **Operating Path**

On the “Pitch Compensation” page→ F1[Previous].

- **Explanation**

This button toggles the parameter item.

1.14.4.2 Next

- **Operating Path**

On the “Pitch Compensation” page→ F2[Next].

- **Explanation**

This button toggles the parameter item

1.14.4.3 Clear Date X

- **Operating Path**

On the “Pitch Compensation” page→ F3[Clear Date X].

- **Explanation**

This button clears the pitch compensation parameter in the X-axis direction

1.14.4.4 Clear Date Y

- **Operating Path**

On the “Pitch Compensation” page→ F4[Clear Date Y].

- **Explanation**

This button clears the pitch compensation parameter in the Y-axis direction

1.14.4.5 Clear Date Z

- **Operating Path**

On the “Pitch Compensation” page→ F5[Clear Date Z].

- **Explanation**

This button clears the pitch compensation parameter in the Z-axis direction

1.14.4.6 RTL Import

- **Operating Path**

On the “Pitch Compensation” page→ F7[Table load].

- **Explanation**

This button imports the generated RTL file

1.14.4.7 Goto Home

- **Operating Path**

On the “Pitch Compensation” page→ F8[Goto Home].

- **Explanation**

This key returns the location of the reference group

1.14.5 Jump to parameter number

- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F5 [Search].

- **Explanation**

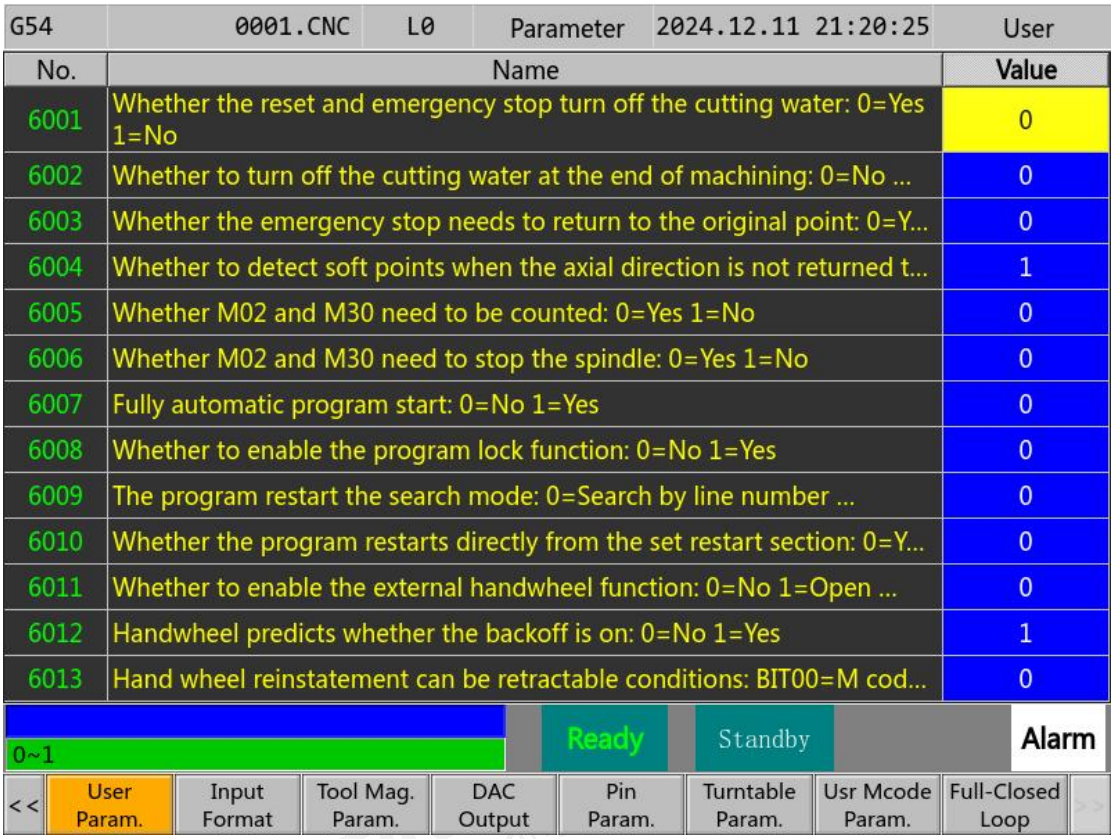
This function is used to search for parameters corresponding to the number.

1.14.6 Expand parameters

- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F6 [Expand Param.].

1.14.6.1 User parameters



- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F6 [Expand Param.]→F1 [User Param.]

- **Explanation**

1. This function is mainly aimed at the user parameters that the operator needs to set in actual operation;
2. User parameters need to be modified by the operator or above

1.14.6.2 Tool magazine parameters /Input format

For details, please refer to the corresponding function manual

1.14.6.3 DAC Output

For details, please refer to the corresponding function manual

1.14.6.4 Pin parameters

For details, please refer to the corresponding function manual

1.14.6.5 Turntable parameters

For details, please refer to the corresponding function manual

1.14.6.6 User Mcode parameters

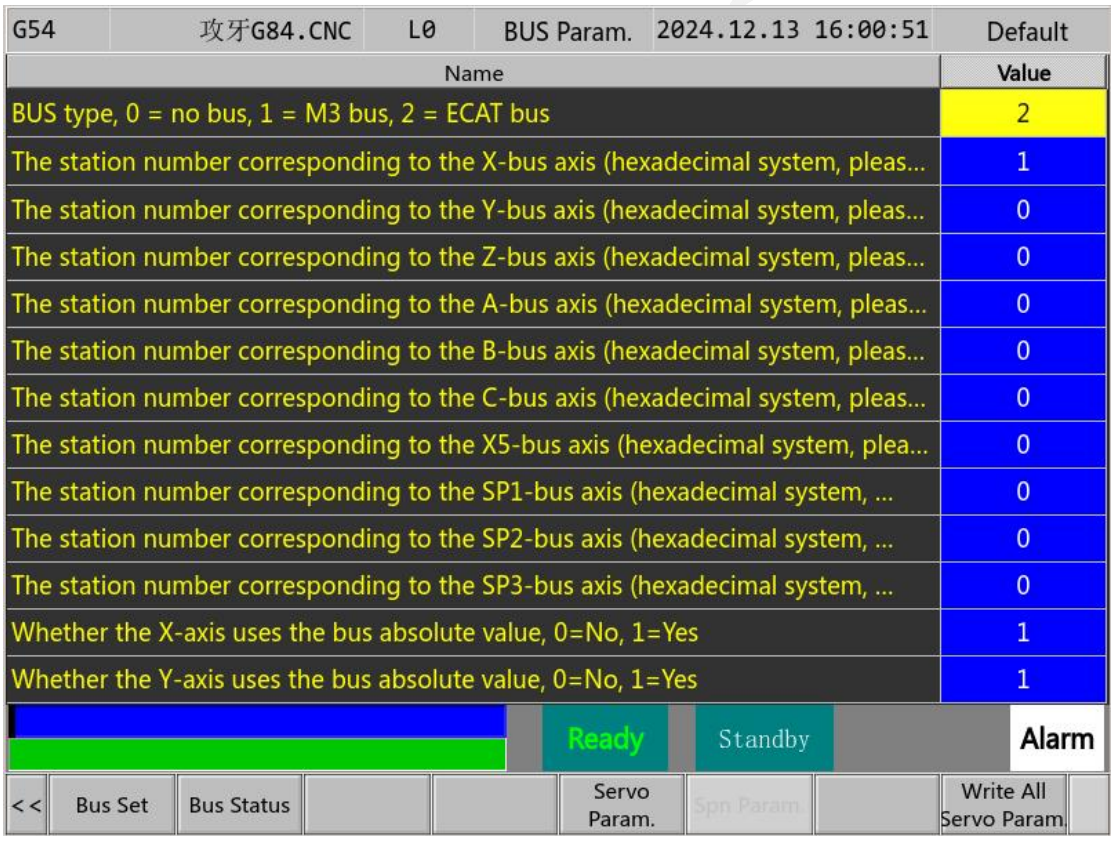
For details, please refer to the corresponding function manual

1.14.6.7 Full-closed Loop

For details, please refer to the corresponding function manual

1.14.7 Serial parameters

1.14.7.1 Bus Set



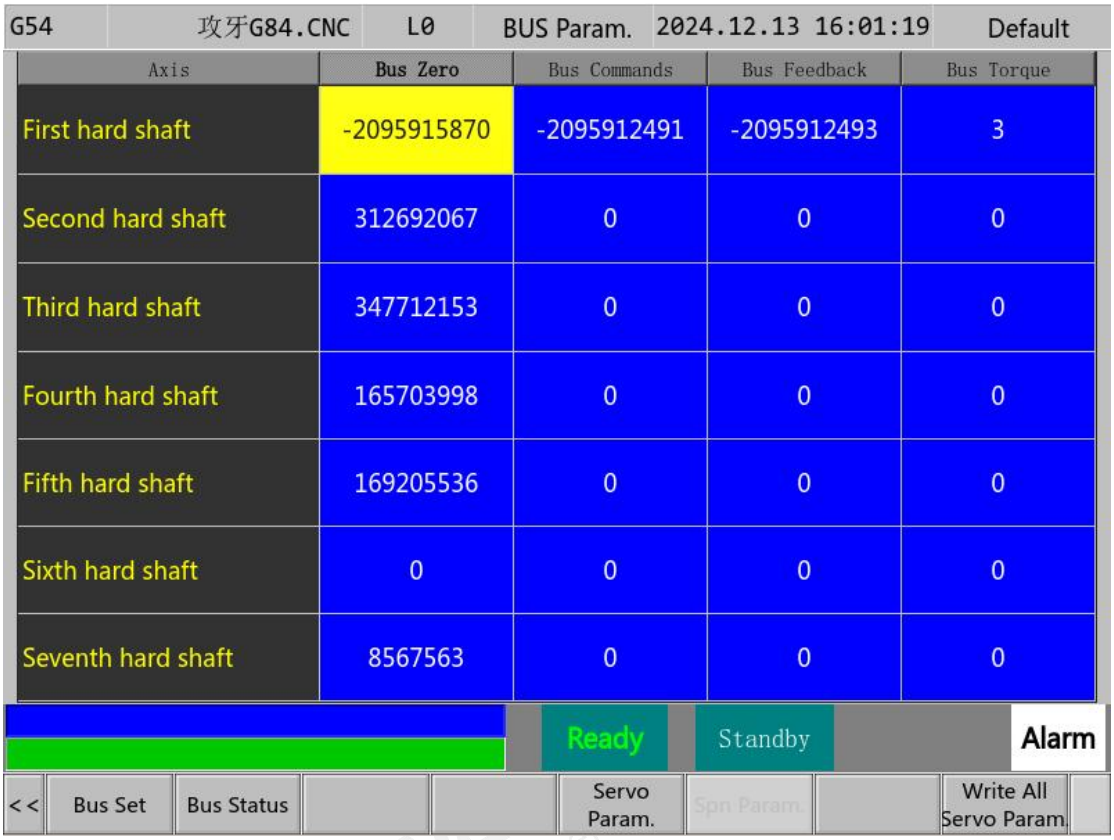
- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F7 [Serial Param.].

- **Explanation**

This function is used to set bus-related parameters, including bus type, bus axis corresponding station number, servo type selection, etc.

1.14.7.2 Bus Status



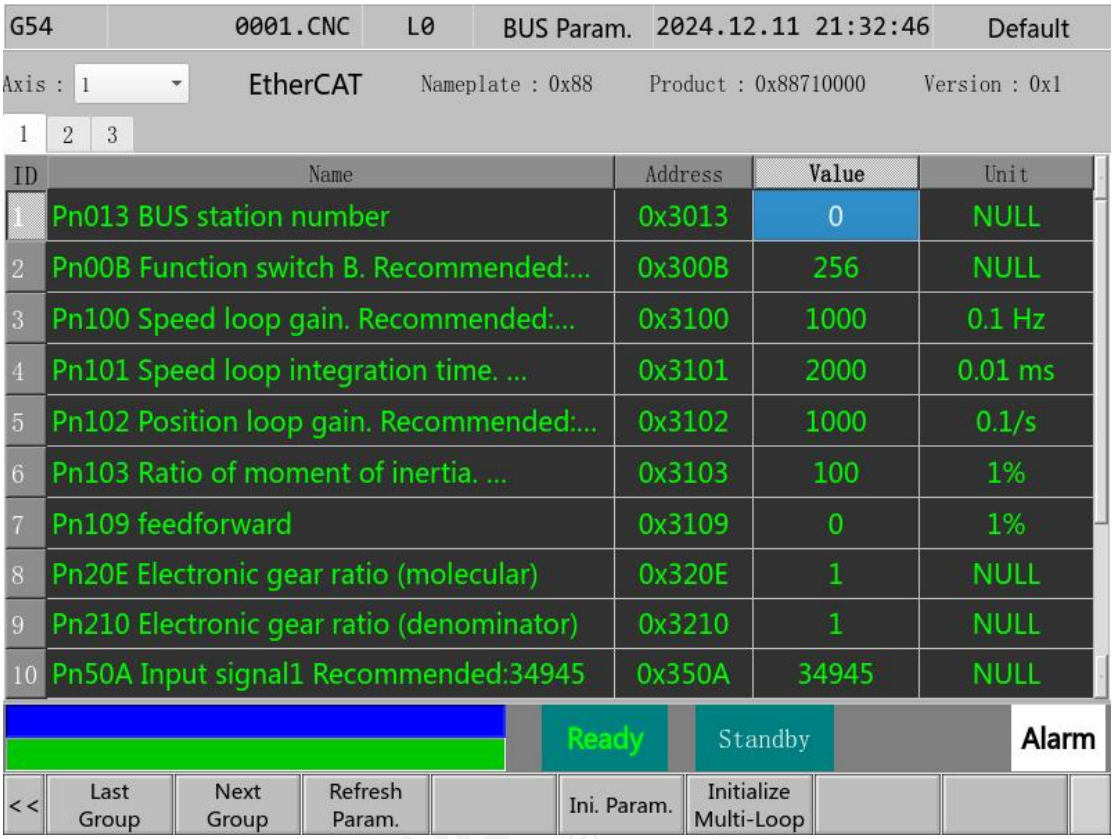
- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F7 [Serial Param.]→F2 [Bus Status].

- **Explanation**

This function is used to monitor the information of commands, feedbacks, and torques for each hardware axis of the bus.

1.14.7.3 Servo parameters



- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F7 [Serial Param.]→F5[Servo Param.].

- **Explanation**

This feature is used to set drive device parameters.

- **Note**

This button is only valid if the bus function is enabled and the bus hardware axis is set.

1.14.7.4 All drive parameters are loaded

- **Operating Path**

On the "Machine Coordinates" page[>>]→F3 [Param.]→F7 [Serial Param.]→F8[Write All Servo Param.].

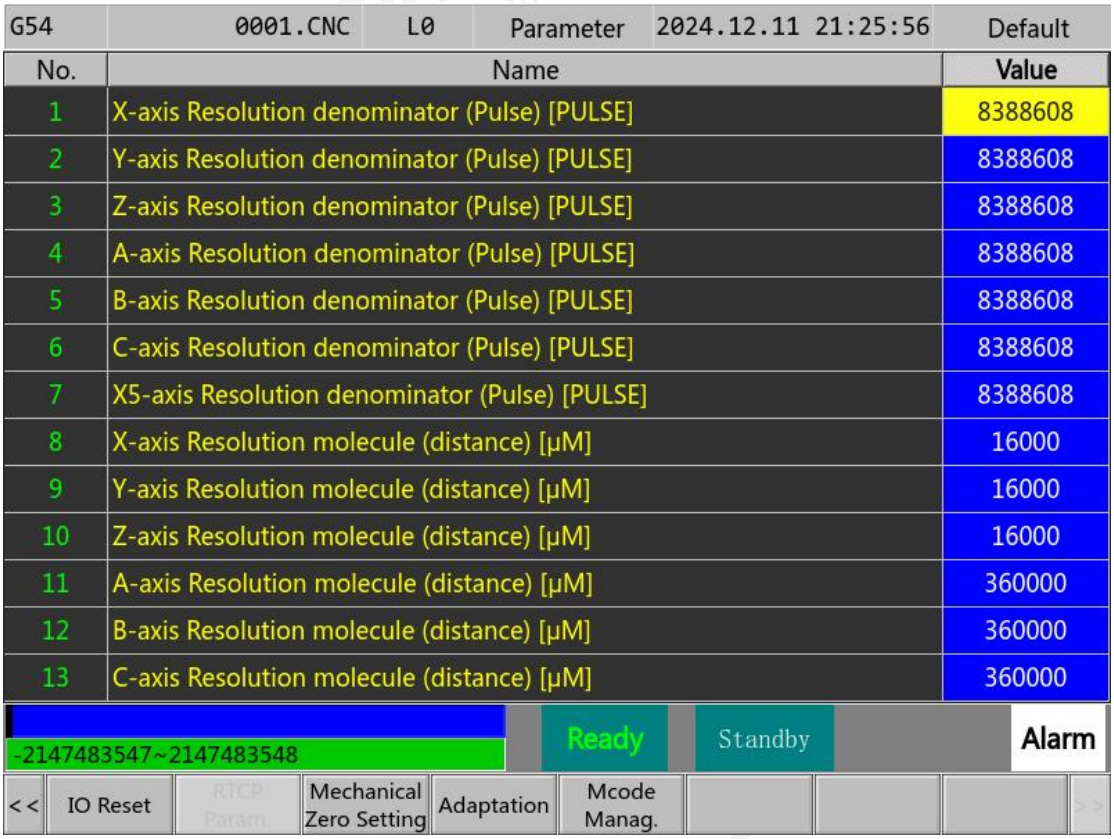
- **Explanation**

This feature is used to set drive device parameters.

- **Note**

This button is only valid if the bus function is enabled and the bus hardware axis is set.

1.14.8 Debugging parameters



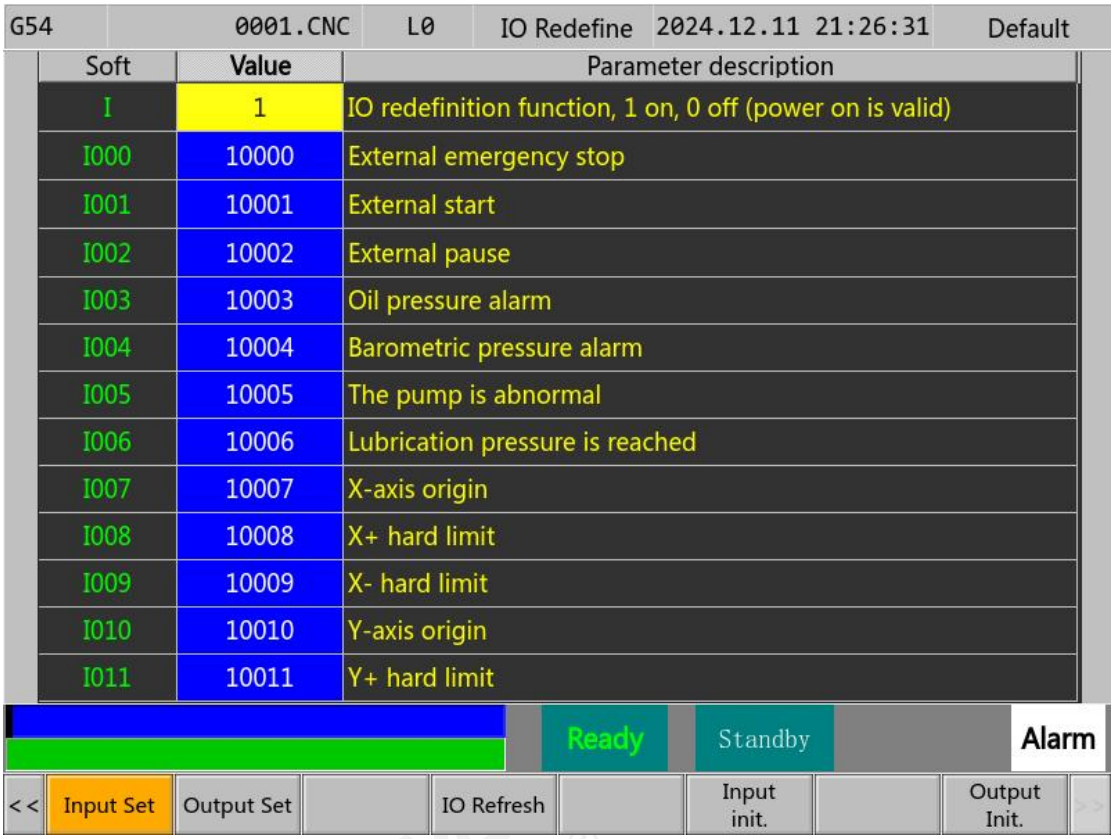
- **Operating Path**

On the “Parameter Settings” page→ F8 [Debug Param.].

- **Explanation**

This function is used for IO redefinition, 5-axis adjustment, mechanical zero setting, adjustment, M-code management and other related functions.

1.14.8.1 IO Reset



- **Operating Path**

On the “Parameter Settings” page→ F8 [Debug Param.] → F1 [IO Reset].

- **Explanation**

In this function, the hardware IO signal corresponding to the PLC software IO signal can be set to facilitate customer wiring.

- **Example**

Suppose that I03 on the software defines an oil pressure alarm; The actual oil pressure alarm on the scene is connected to I01, so the parameters redefined by I003 can be changed to 10001 at this time, which can be realized; There is no need to change the route.

- **Note**

1. Multiple software O points cannot correspond to the same hardware O point, otherwise, the system will alarm. A single software I point can correspond to one or

more hardware I points;

2. After the setting is completed, you need to press the [Reset] button to reset and restart the computer before it can take effect.

1.14.8.1.1 Input Set

- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F1[IO Reset]→F2 [Output Set].

- **Explanation**

The page switches to the I-point settings page, where you can redefine the I-point.

1.14.8.1.2 Output Set

- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F1[IO Reset]→F2 [Output Set].

- **Explanation**

The page switches to the O-point settings page to redefine the O-point.

1.14.8.1.3 Input initialize

- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F1[IO Reset]→F6 [Input Ini.].

- **Explanation**

Restore the hardware I-point to the initial settings.

1.14.8.1.4 Output initialize

- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F1[IO Reset]→F8 [Output Ini.].

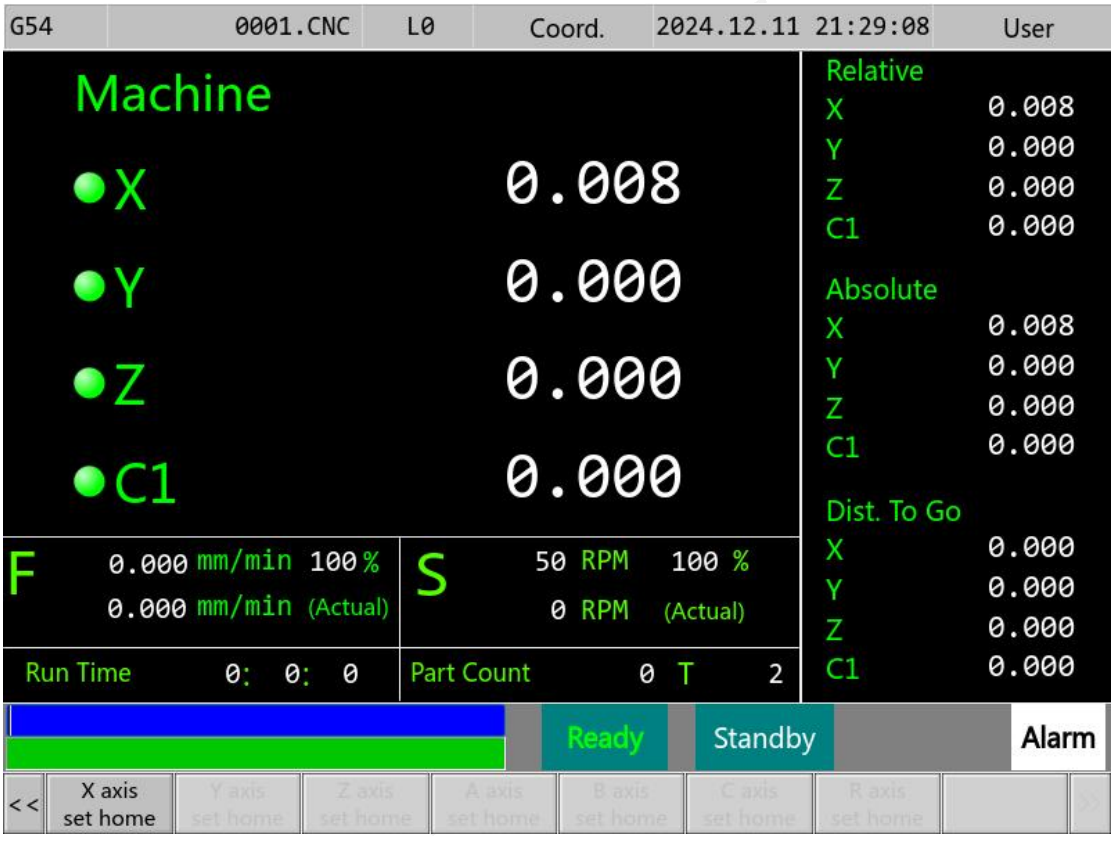
- **Explanation**

Restore the hardware O-point to the initial settings.

1.14.8.2 RTCP parameters

Please refer to the corresponding function manual

1.14.8.3 Mechanical zero setting



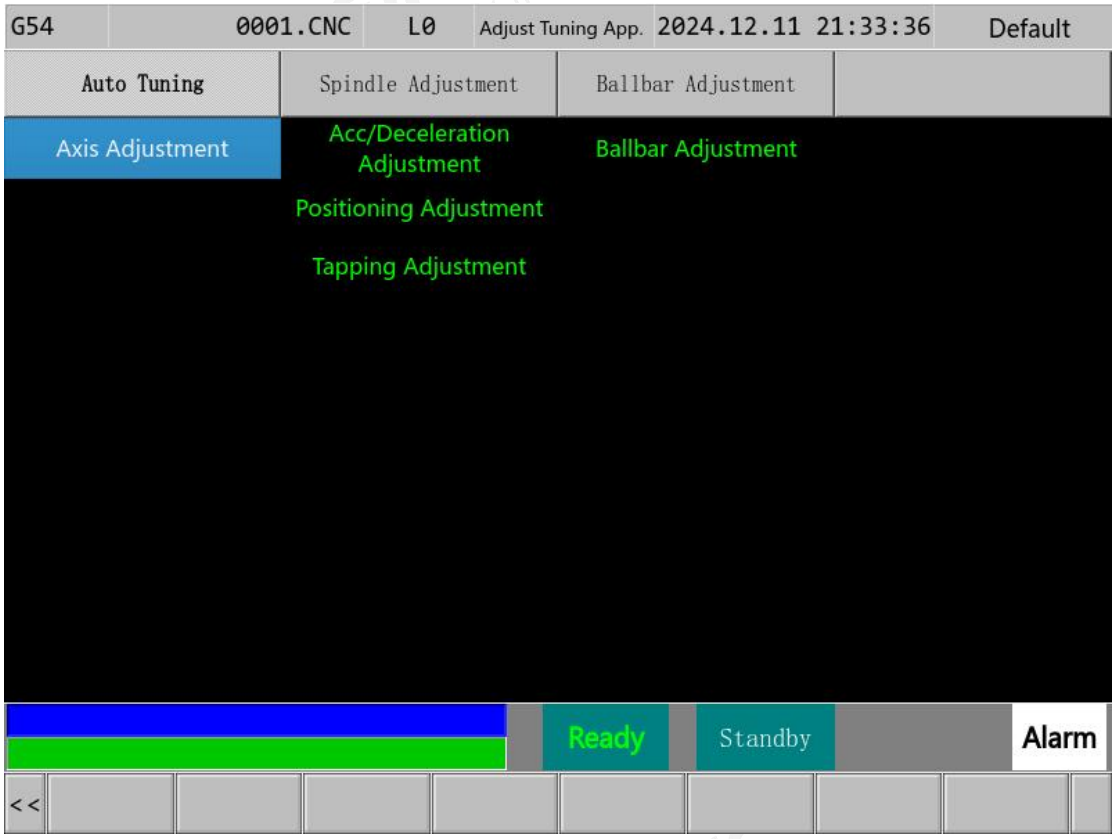
- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F3[Mechanical zero setting]→F1-7[X、Y、Z、A、B、C、R(X5) Mechanical zero setting].

- **Explanation**

1. This function is used to set the zero point of each axial machine;
2. Valid only when the bus spool is used; For non-bus spools, please use the form of change switch.

1.14.8.4 Adaptation features



- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F4[Adaptation]

- **Operating instructions**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor to the option you want to adjust.
2. Press the [Enter] button to jump to the corresponding adjustment page

1.14.8.4.1 Axis adaptation

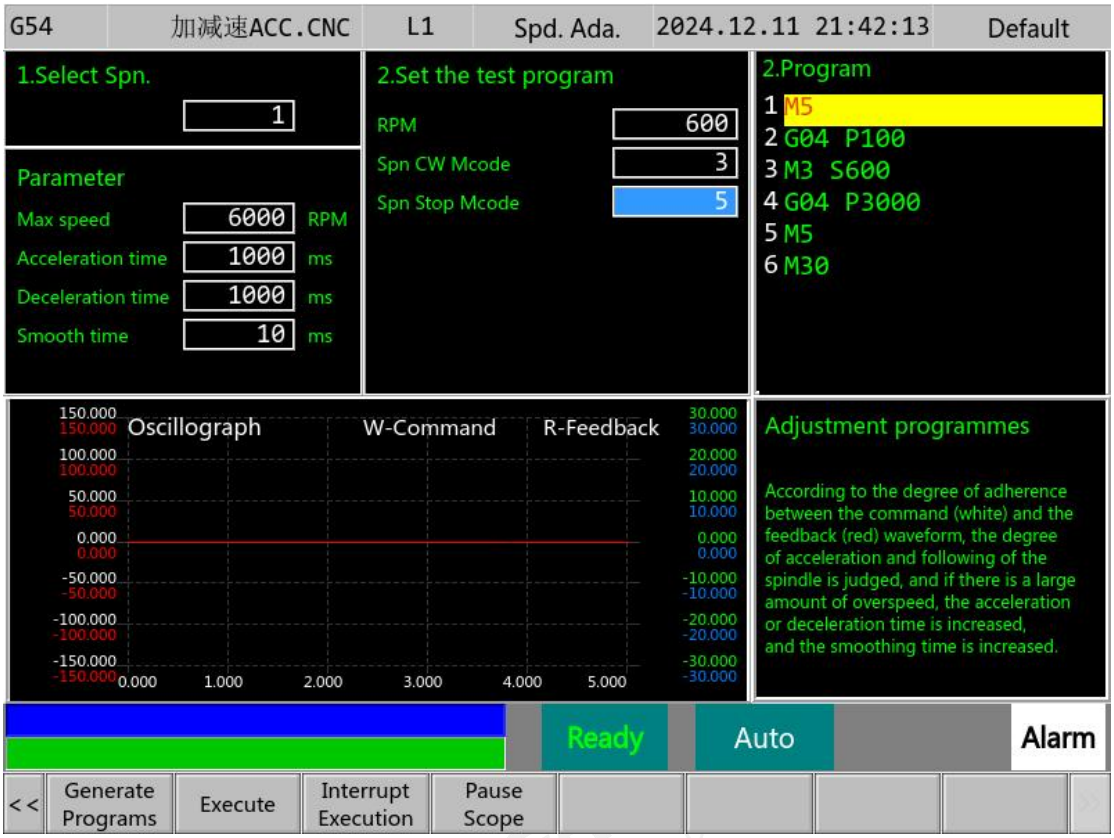
- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F4[Adaptation] → cursor to move to "Axis adaptation" and press [Enter]

- **Explanation**

Adaptive adjustment of servo parameters for bus-type axes.

1.14.8.4.2 Spindle acceleration and deceleration adaptation



- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F4[Adaptation] → cursor to move to " Acc./Deceleration adaptation" and press [Enter]

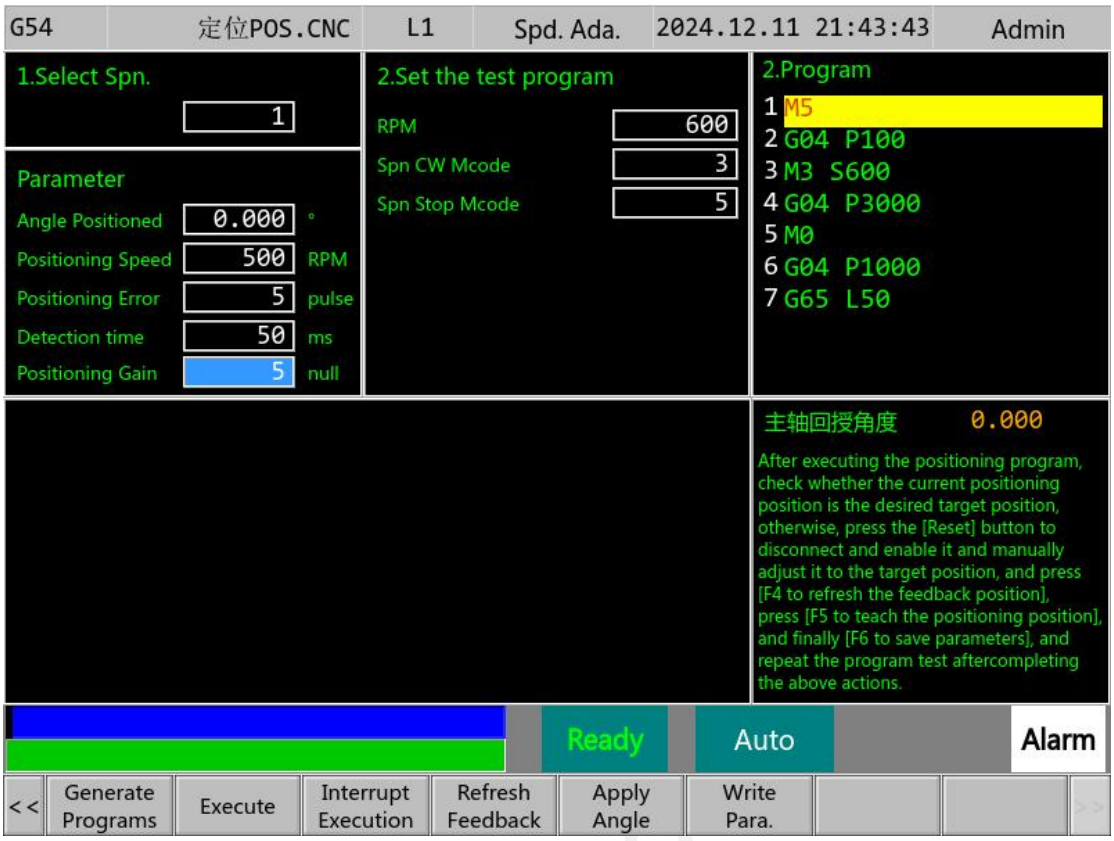
- **Explanation**

It is used to adjust the acceleration and deceleration of the spindle to a reasonable value.

- **Operating instructions**

1. Select the spindle;
2. After setting the relevant parameters of the generated program, the F1[Generate Program] button will generate the corresponding program in the code display.
3. F2 [Execute] Start the test program
4. Acceleration and deceleration can be adjusted by monitoring the waveform.

1.14.8.4.3 Spindle positioning adaptation



- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F4[Adaptation] → cursor to move to " positioning adaptation " and press [Enter]

- **Explanation**

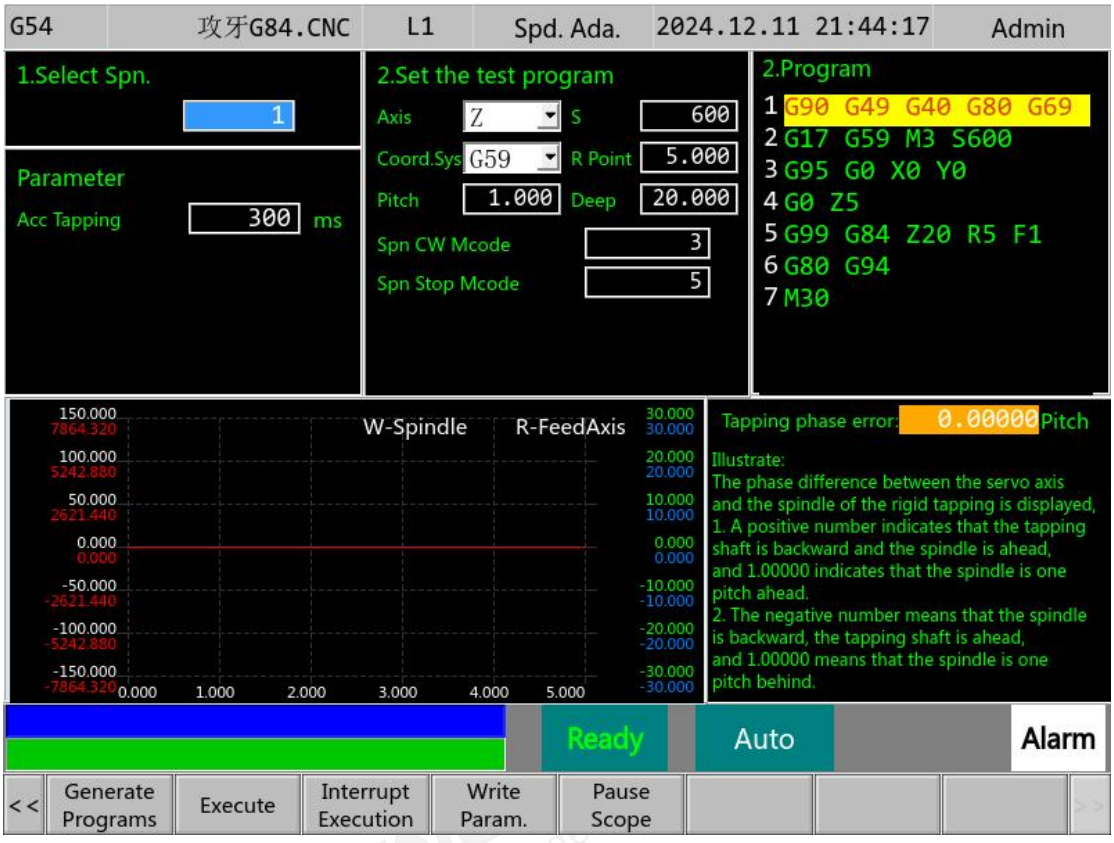
It is used to adjust the positioning angle of Finger Bus type spindle.

- **Operating instructions**

1. Select the spindle;

2. After setting the relevant parameters of the generated program, the F1[Generate Program] button will generate the corresponding program in the code display;
3. F2 [Execution] Start the test program;
4. After executing the positioning program, check whether the current positioning position is the desired target position, otherwise, press the [Reset] button to disconnect the enabled and manually adjust it to the target position, and press F4[Refresh Feedback], press F5 [Apply Angle], and finally F6[Write Param.], and repeat the program test after completing the above actions.

1.14.8.4.4 Tapping adaptation



- **Operating Path**

On the “Parameter Settings” page→F8[Debug Param.]→F4[Adaptation] → cursor to move to " Tapping adaptation" and press [Enter]

- **Explanation**

It is used for the adjustment of the tapping effect.

- **Operating instructions**

1. Select the spindle;
2. After setting the relevant parameters of the generated program, the F1[Generate Program] button will generate the corresponding program in the code display;
3. F2 [Execution] Start the test program;
4. According to the phase error value, the tapping acceleration and deceleration time and the gain-related parameters of the tapping feed axis and tapping spindle are adjusted according to the waveform.

Part 2. Operation

2.1 System Status

The system displays different status indications depending on its current state. The following describes the triggering conditions for each state.

2.1.1 Not Ready

The system imposes certain restrictions based on different alarms and conditions.

- **Triggering conditions**

1. The system has active alarms.
2. Axes have not returned to the home position.

2.1.2 Ready

The system is ready for various operations.

- **Triggering conditions**

1. Alarms have been cleared, and axes have returned to the home position.
2. While in the processing/pause state, pressing the "Reset" button will switch the system status to "Ready."

2.1.3 Processing

The system is executing program processing.

- **Triggering conditions**

The system is in the "Ready" state, and program processing is initiated, causing the

system status to switch to "Processing."

2.1.4 Pause

During program processing, the program can be paused.

- **Triggering conditions**

While the system is in the "Processing" state, triggering a pause will switch the system status from "Processing" to "Pause."

- **Note**

In the "Pause" state, the spindle can still operate normally.

2.2 Alarm Status

Indicates whether the system has any active alarms.

2.2.1 Alarm Flashing

- **Triggering conditions**

The system has active alarms.

2.2.2 Alarm Not Flashing

- **Triggering conditions**

The system has no active alarms.

2.3 Machine Preparation

2.3.1 Manual Functions

The controller provides four manual control functions for the axes: Manual Continuous Feed, Manual Rapid Feed, Jog Feed, and MPG Feed. The following sections will explain each of these functions.

2.3.1.1 Manual Continuous Feed

- **Description**

1. Drives the axis to move continuously in one direction.
2. Can drive multiple axes simultaneously.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. The operating mode is switched to "Manual Mode."

- **Operation**

1. Click on the axis keys, such as "X←, X→, Y↑, Y↓, Z↑, Z↓, 4↑, 4↓," to control the corresponding axis for continuous movement.
2. Feed rate and acceleration/deceleration time can be set through parameters.
3. The feed rate is limited by the maximum feed rate of G00.

2.3.1.2 Manual Rapid Feed

- **Description**

1. Drives the axis to move continuously in one direction at the feed rate of G00.
2. Can drive multiple axes simultaneously.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. All axes have returned to the home position.
3. The operating mode is switched to "Manual Mode."

- **Operation**

1. Press and hold the "Rapid" button until the indicator light turns on to activate the rapid feed mode.
2. Click on the axis keys, such as "X←, X→, Y↑, Y↓, Z↑, Z↓, 4↑, 4↓," to control the corresponding axis for continuous rapid movement.
3. The feed rate is adjusted using the maximum feed rate of G00.
4. The acceleration/deceleration time is set through parameters.

2.3.1.3 Jog Feed

- **Description**

1. Drives the axis to move in one direction for a fixed distance.
2. Can drive multiple axes simultaneously.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. The operating mode is switched to "Jog Mode."

- **Operation**

1. Click on the axis keys, such as "X←, X→, Y↑, Y↓, Z↑, Z↓, 4↑, 4↓," to control the corresponding axis movement.
2. Each click triggers a single movement.
3. The distance of each jog movement is set as follows:
   - ❖ X1: Move 0.001mm per jog.
   - ❖ X10: Move 0.010mm per jog.
   - ❖ X100: Move 0.100mm per jog.
   - ❖ This setting is shared with the handwheel multiplication factor selection.

2.3.1.4 MPG Feed

- **Description**

Drives the axis to move continuously in one direction.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. The operating mode is switched to "MPG Mode."

- **Operation**

1. Use the axis selection on the MPG to choose the axis to be driven by the MPG.
2. Rotate the MPG to drive the corresponding axis. The direction of rotation determines the direction of axis movement.
3. The distance of each MPG rotation is set as follows:
   - ❖ X1: Move 0.001mm per rotation.
   - ❖ X10: Move 0.010mm per rotation.
   - ❖ X100: Move 0.100mm per rotation.
   - ❖ This setting is shared with the jog feed distance selection.

2.3.2 Program Machining

2.3.2.1 Automatic Execution

- **Description**

Executes the machining program.

- **Operating Conditions**

1. The system is in a non-alarm state.
2. The operating mode is switched to "Auto Mode."

- **Operation**

1. Set the knob to the "AUTO" position(Or press the "AUTO" key).
2. Click the "Start" button to execute the current machining program.
3. The system status changes from "Ready" to "Machining."
4. Once the program machining is completed, the system status switches back to "Ready."

2.3.2.1 Single Block Execution (MDI)

- **Description**

Executes the MDI (Manual Data Input) machining.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. The operating mode is switched to "MDI Mode."

- **Operation**

1. Set the knob to the "MDI" position(Or press the "MDI" key).
2. The system will execute the content of the current single block program.
3. The system status changes from "Ready" to "MDI."
4. Once the program machining is completed, the system status switches back to "Ready."

2.3.3 Home Mode

Before machining, the coordinates required for machining are based on the machine's home position. Therefore, every time the controller is powered on, a homing action is required to establish the machine's home position.

- **Operating Conditions**

1. The system is not in a critical state, such as emergency stop or axis malfunction.
2. The operating mode is switched to "Home Mode."

- **Operation**

1. First, turn the knob to the "HOME" position(Or press the "HOMI" key).
2. Click on the axis buttons "X←, X→, Y↑, Y↓, Z↑, Z↓, 4↑, 4↓" to control the machine to move towards the origin point of the corresponding axis. Alternatively, you can click the "Start" button, and the system will move all axes toward their respective origin points.
3. After the axis completes the homing action and comes to a stop, the system will reset the mechanical coordinates of that axis to zero.
4. You can configure parameters to specify the homing method, direction, speed, and other settings.

- **Note**

The software limits are ineffective during the homing process.

2.4 Tool Preparation

When editing a machining program, it describes the relationship between the tool tip and the workpiece, without considering the actual position of the tool and the differences in tool lengths. Therefore, the offset settings for each tool need to be accurately measured and input into the controller before machining. The system provides four correction methods: tool length compensation, tool wear compensation, tool radius compensation, and workpiece coordinate offset.

- **Tool Length Compensation:**

Tool length correction is enabled through G43/G44 Hxx in the program, and the xx in Hxx corresponds to the tool length correction and tool length wear correction in the compensation group number.

- **Tool Radius Compensation:**

Enable tool diameter correction through G41/G42 Dxx in the program, and the xx in Dxx

corresponds to the tool diameter correction and tool diameter wear correction in the compensation group number.

- **Workpiece Coordinate Offset:**

G54.1P1-G54.1P48 and G54-G59 coordinate system settings.

2.5 Program Preparation and Machining Execution

This section will explain how to specify a machining program for machining and how to perform machining tests.

2.5.1 Specify Machining File

- **Operating Conditions**

Except for "MDI" mode.

- **Steps**

1. Specify the currently edited program file as the machining file.
2. Switch to the "Program Editing" page.
3. Click "Execute" to switch the screen to the "Monitor" page, and the program file will be designated as the machining program file.
4. Specify a program file from the file management as the machining program file.
5. Switch to the "File Management" page.
6. Move the cursor to the desired program to be loaded for machining.
7. Click "Execute" to switch the screen to the "Monitor" page, and the program will be designated as the machining program.

- **Verification**

1. The successful designation of the machining program can be confirmed in two places:
2. The current machining program name at the top of the screen.

3. The program content in the "Monitor" page.

2.5.2 Graphical Simulation

The system provides a convenient program content simulation feature. After finishing program editing, users can easily simulate the toolpath of the program. This feature also includes program checking, which helps users quickly identify syntax errors or unreasonable actions in the machining program. It is recommended that users use this feature to check the program content after completing program editing.

- **Steps**

1. On the "Program Selection" page, select the program to be edited.
2. After editing the program, go to the "Monitor" page and click on "Simul. Switch" -> "Graphical Adjustment" -> "Static tracing".
3. Scan the program content.
4. Once the scanning is complete, the system will begin the graphical simulation based on the program content until the entire program has been simulated.

- **Explanation**

1. In the simulation graphics:
   - ❖ Solid lines represent cutting paths.
   - ❖ Dashed lines represent travel paths.
2. During the scanning program process, if there are any syntax or content errors, the system will generate an alarm message and indicate the line number where the error occurred.

2.5.3 Test Machining

2.5.3.1 Program Prediction (MPG Simulation)

This section explains how to perform test machining using the MPG simulation.

- **Prerequisites**

"MDI" and "Auto" modes can be used.

- **Steps**

1. Execute the program for machining.
2. Press the "PROG-SIM" button on the panel.
3. Use the MPG to perform machining.
4. Rotate the MPG clockwise to move to the next program line for machining from the current line.
5. Rotate the MPG counterclockwise to move to the previous program line for machining. This function is also known as "MPG backfeed".

- **Verification**

You can confirm the successful activation of the MPG simulation through the following:

1. The "PROG-SIM" indicator light on the auxiliary panel is lit.
2. When the MPG simulation is activated during machining, the machine immediately decelerates to 0 until the MPG is moved or the MPG simulation is canceled.

2.5.3.2 Single Block Execution

This section explains how to perform program machining using the single block mode.

- **Prerequisites**

"MDI" and "Auto " modes can be used.

- **Steps**

1. Execute the program for machining.
2. Press the "S.BK" button on the panel.
3. The system will decelerate after executing the current block until it reaches 0 speed, entering the single block stop state.

4. Press the "Start" button to continue the machining. The system will enter the single block stop state after executing the next block.

2.5.4 Machining Monitoring

This section explains how to manage the quantity of workpieces during machining.

- **Prerequisites**

None

- **Explanation**

1. The total number of workpieces processed by the machine is cumulatively recorded.
2. Required Workpiece Count:
   - ❖ The specified number of workpieces required by a single machining program. When the program continues processing with M99, it will pause and send a notification when the workpiece count reaches the specified required workpiece count.
3. Workpiece Count:
   - ❖ The workpiece count continues to accumulate when the program is running with M99.
   - ❖ The workpiece count is reset (cleared to 0) under the following conditions:
     a. When the required workpiece count is reached.
     b. When changing the machining profile.
     c. When modifying the required workpiece count, and the new count is lower than the current workpiece count.

2.6 System Alarm Handling

In order to prevent safety hazards caused by erroneous operations, the system or PLC is equipped with various protections. When these protection conditions are triggered, the

system will issue warnings or alarms to alert the user. This section explains how to check and troubleshoot alarms when they occur.

2.6.1 Emergency Stop

When a machine malfunction or an unexpected action that may endanger personal or machine safety occurs, pressing the emergency stop button will immediately stop the machine. Once this button is pressed, it is locked, and depending on the manufacturer, the procedure to unlock it may vary, but typically rotating the button will unlock it. This button interrupts machine actions, and before unlocking it, the cause of the malfunction must be addressed.

2.6.2 Alarm Display

Alarms are divided into current alarms and historical alarms. Refer to section 1.7 "Alarms" for more information on their display.

2.6.2.1 Current Alarms

1. Displays the current alarm status of the system.
2. When an alarm occurs, the controller will display a pop-up window showing the current alarm details.
3. Clicking the back button "《" cancels the pop-up window.
4. If the alarm has not been cleared, clicking "Reset" will display the alarm window again.
5. Switching to the "Alarms" page will automatically display the current alarms.

2.6.2.2 Historical Alarms

1. Displays the alarms that have occurred in the past, allowing you to determine the possible causes of the alarms.
2. Switching to the "Alarms" page and clicking on "Historical Alarms" will display the historical alarms.
3. When multiple alarms occur, they are listed in chronological order from top to bottom based on the time of occurrence.

Part 3. Mechanical Operation Panel Description

B0x-8.4 Inch Milling Machine CNC System (Horizontal)

3.1 Panel Operation Buttons

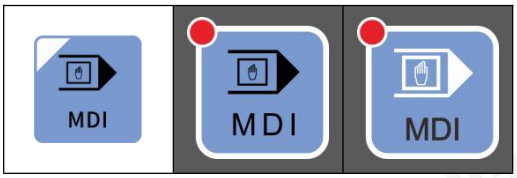
When there is a safety issue with personnel or a machine operation failure, pressing this button will immediately stop the machine movement, trigger an alarm on the controller, and the I/O section of the controller will be controlled according to the specific situation.

- **Auto Mode**

In this mode, pressing the Start button will automatically execute the machining program.

- **Homing Mode**

In this mode allows the axes to perform a homing operation, returning to the specified zero position or machine zero position



- **MDI Mode**

This mode is used to execute programs that are entered directly without the need for a machining file.

- **JOG Mode**

This mode is used to manually drive each axis of the machine.

- **MPG Mode**

This mode allows manual control of the axes using MPG.

- **Inch Mode**

This mode is used for inching or jog feeding the axes in small increments.

- **Program Prediction**

This function allows the use of MPG to simulate the execution of programs in Auto Mode or MDI Mode. It helps to verify if the machining paths are correct.

- **Single Block Execution**

This function allows step-by-step execution of the program during Auto Mode or MDI Mode machining

- **Skip Selection**

This function determines whether to skip program blocks that contain the "/" character during Auto Mode or MDI Mode machining

- **Optional Stop (M01)**

This function allows pausing the program execution at an optional stop (M01) during Auto Mode or MDI Mode.

- **Spindle Forward**

Start the spindle rotation in the forward direction.

- **Spindle Reverse**

Spindle Reverse: Start the spindle rotation in the reverse direction.

- **Spindle Stop**

Stop the spindle rotation.

- **Coolant On**

Manually activate the cutting coolant.

- **Lubrication Oil**

Manually trigger a lubrication oil cycle.

3.2 System Text Key Description

1. POS: Switch to the Machine Position page.
2. FILE: Switch to the Program Selection page.
3. PROG: Switch to the Program Editing page.
4. MON: Switch to the Machining Monitoring page.
5. OFFSET/SETTING: Toggle between the Workpiece Coordinate System page and the Tool Wear page.
6. HELP/ALARM: Used to switch between the Help page and the Alarm Monitoring page.
7. AZ keys: Letter keys, a total of 26 keys.
8. 0~9 keys: Numeric keys.
9. INPUT: Used for inputting letters, numbers, symbols, etc.
10. DELETE: Backspace delete key.
11. SPACE: Insert a space.
12. RESET: Generally used to stop a program, apply parameter changes, clear alarms after modifying parameters, etc.
13. HOME/END: Move the cursor to the beginning or end of a line.
14. Other commonly used keys: Not further explained.

Part 4. Wiring Instructions

4.1 System Installation Environmental Requirements

The B1 series controllers must be installed and used in the following environments. Using them outside these specified ranges may result in abnormal operation.

Ambient temperature	During operation	Rate of temperature change
	During storage or transportation	Others
Humidity	Under normal conditions	Rate of temperature change
	Within a short period of time	Others
Vibration limit	Under operating conditions	Rate of temperature change
Noise	Under operating conditions	Others
Rate of temperature change		Rate of temperature change
Other		Others

4.2 Considerations for Controller Box Design

- The controller and auxiliary panel box must be sealed to prevent dust ingress.
- The temperature rise inside the box should not exceed 10°C compared to the

surrounding environment.

- Cable entry and exit points must be sealed.
- To avoid noise interference, there should be a minimum distance of 100mm between cables, each unit, and AC power sources. If there is a magnetic field, the distance should be at least 500mm.
- For servo drive installation, please refer to the servo operation manual.

4.3 Design of Internal Temperature in the Box

The maximum temperature rise inside the box compared to the surrounding environment should not exceed 10°C. When designing the box, the main factors to consider are heat sources and heat dissipation area. For the controller, customers have limited control over heat sources but the heat dissipation area is an important consideration. The permissible temperature rise inside the box can be estimated using the following formulas:

1. With cooling fans, the permissible temperature rise is 1°C/6W/1m².
2. Without cooling fans, the permissible temperature rise is 1°C/4W/1m².

These formulas indicate that in the presence of cooling fans, if the heat dissipation area of the box is 1m² and there is a heat source of 6W (4W without cooling fans), the internal temperature will rise by 1°C. The heat dissipation area is calculated as the box's surface area minus the area in contact with the ground.

Example 1: (With Cooling Fans)

Heat Dissipation Area = 2m²

Maximum Permissible Temperature Rise Inside the Box = 10°C.

Therefore, the maximum permissible heat source inside the box = 6W x 2m² x 10 = 120W.

If the heat source inside the box exceeds 120W, additional heat dissipation devices such

as cooling fins must be added.

Example 2: (Without Cooling Fans)

Heat Dissipation Area = 2m²

Maximum Permissible Temperature Rise Inside the Box = 10°C.

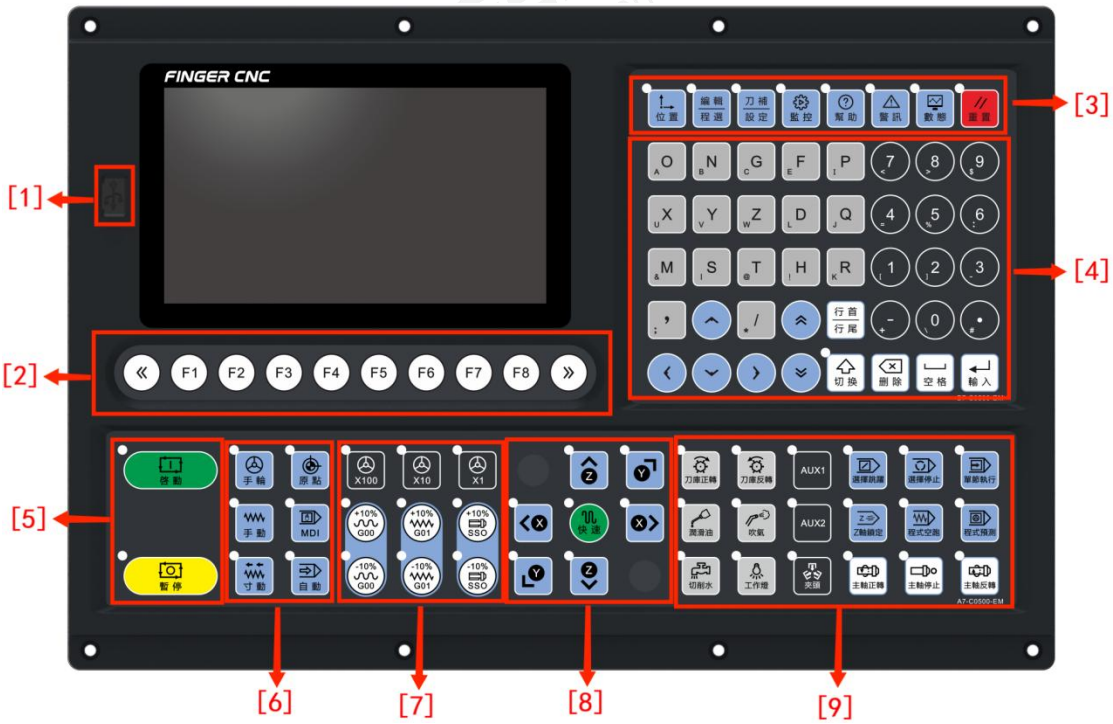
Therefore, the maximum permissible heat source inside the box = 4W x 2m² x 10 = 80W.

If the heat source inside the box exceeds 80W, additional heat dissipation devices such as fans or cooling fins must be added.

4.4 Interface Description and Installation Dimensions of Various Controller Models

4.4.1 B0x-7-inch Controller (Horizontal)

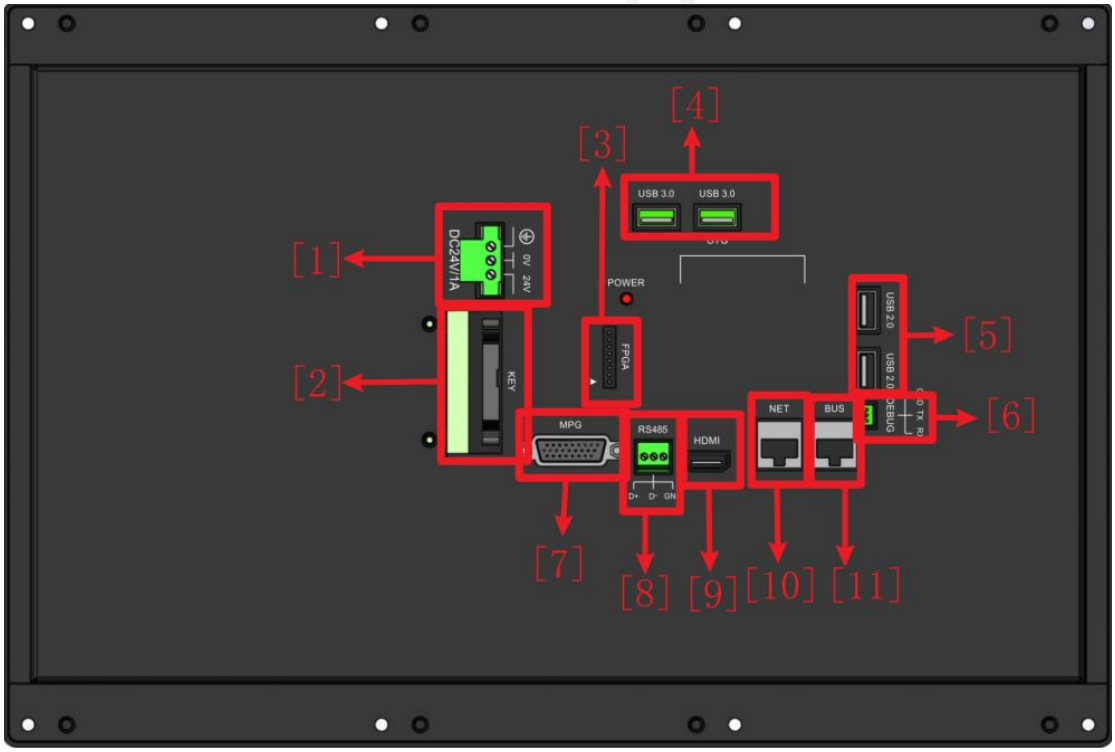
- **Interface Description**



Front View

No.	Feature And Function Introduction
1	USB communication port
2	F1-F8 function selection buttons
3	Quick menu key
4	Normal editing and operation aera
5	Cycle start and stop buttons
6	Auxiliary Panel
7	Speed ratio selection

8	JOG feedrate switch (slow, fast)
9	Function-key

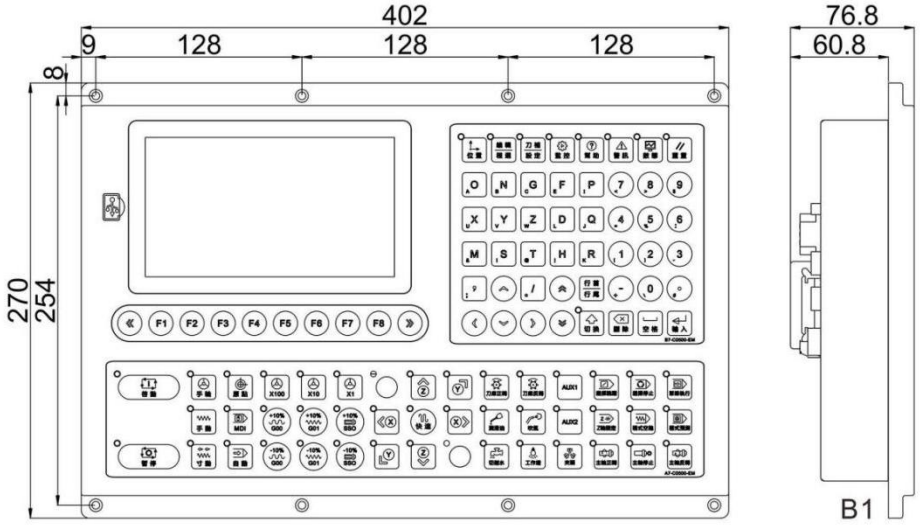


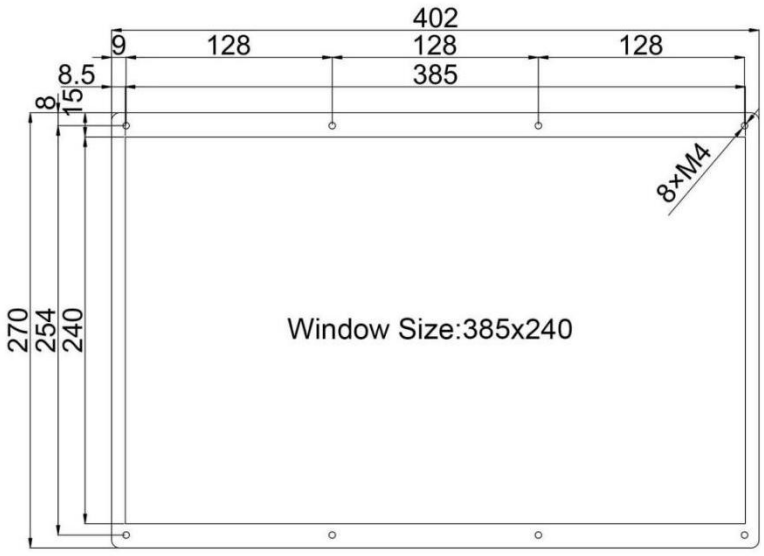
Rear View

No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port

9	HDMI interface for external displayer connection
10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

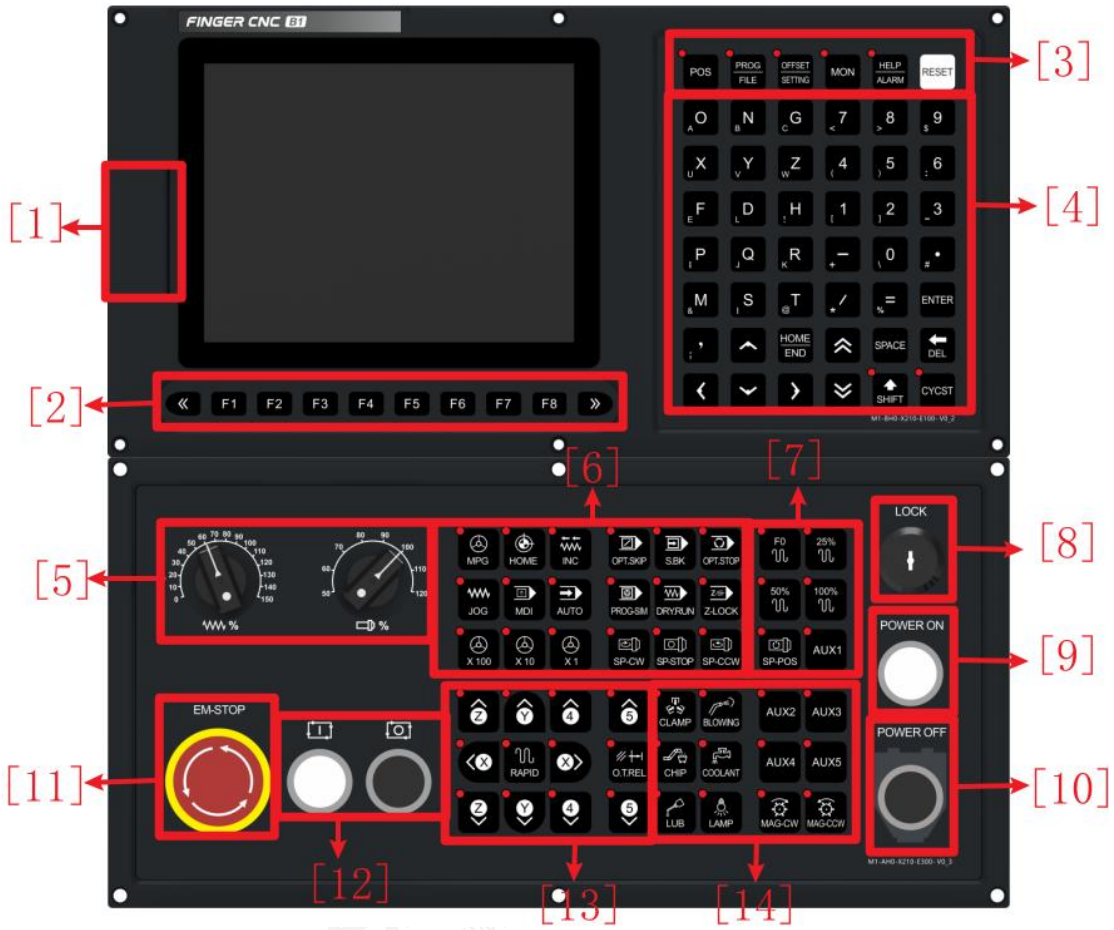
- **Installation dimensions and hole positions**





4.4.2 B0x-8 inch controller (horizontal)

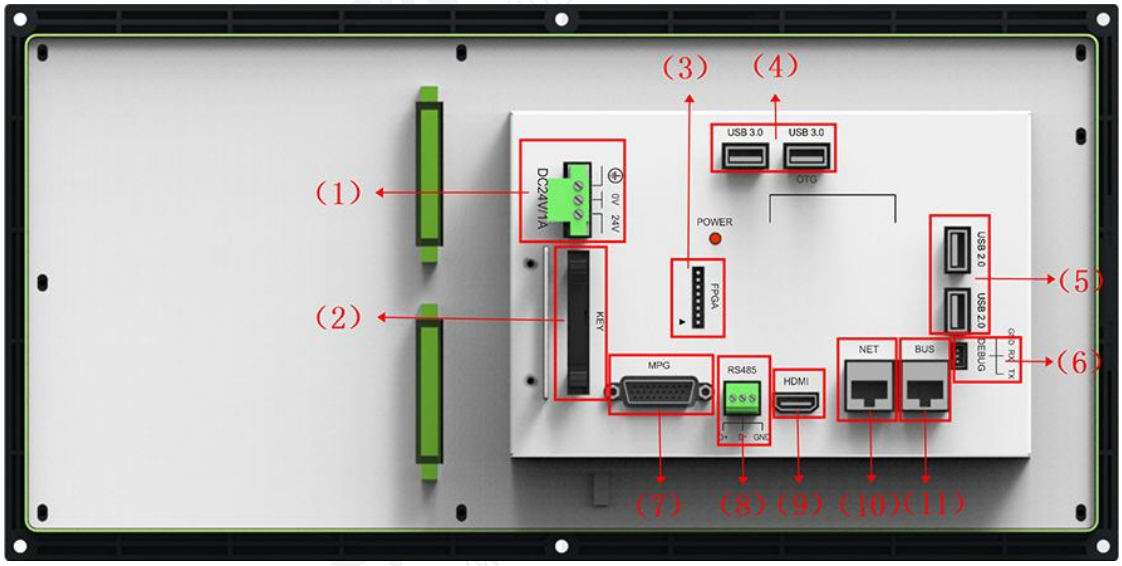
- **Interface description**



Front View

No.	Feature And Function Introduction
1	USB communication port
2	F1-F8 function selection buttons
3	Quick menu key
4	Normal editing and operation aera
5	Speed ratio selection
6	Auxiliary Panel
7	Speed ratio selection
8	Program lock
9	Power off switch

10	Power on switch
11	Emergency-stop switch
12	Cycle start and stop buttons
13	JOG feedrate switch (slow, fast)
14	Other operation aera

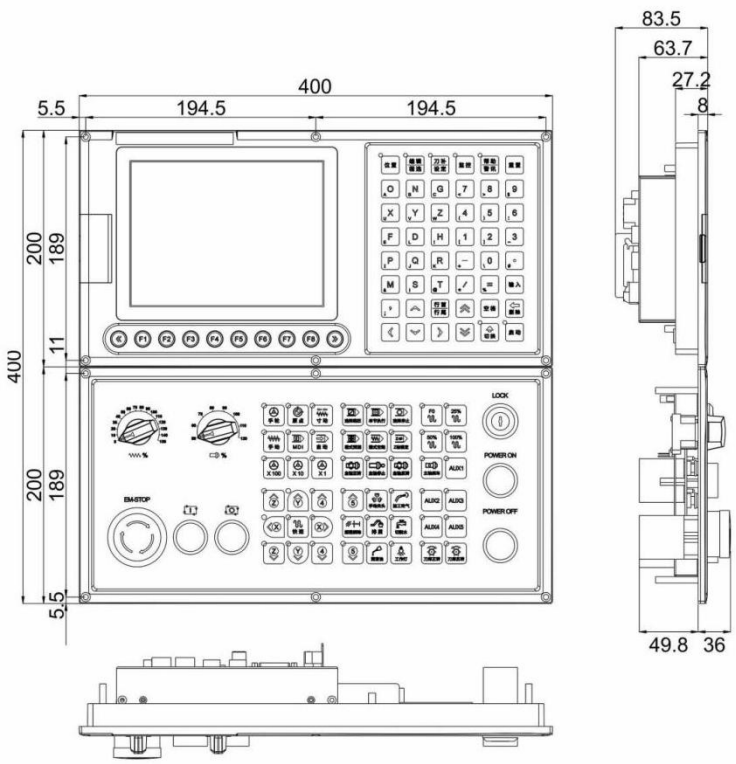


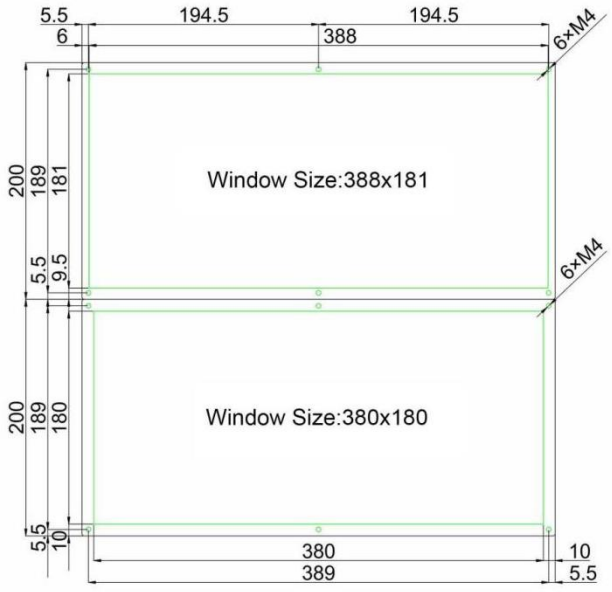
Rear View

No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port
9	HDMI interface for external displayer connection

10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

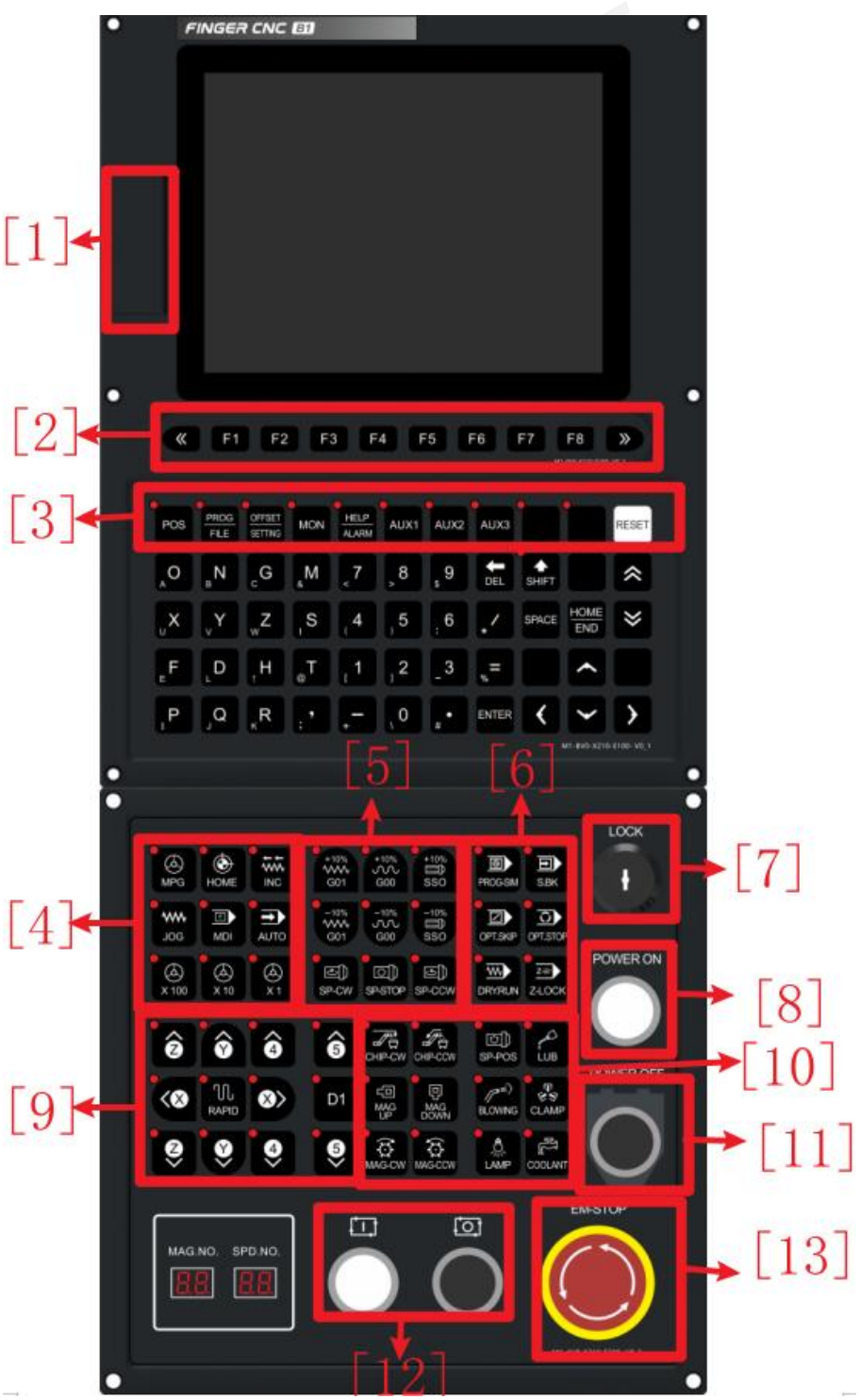
- **Installation dimensions and hole positions**





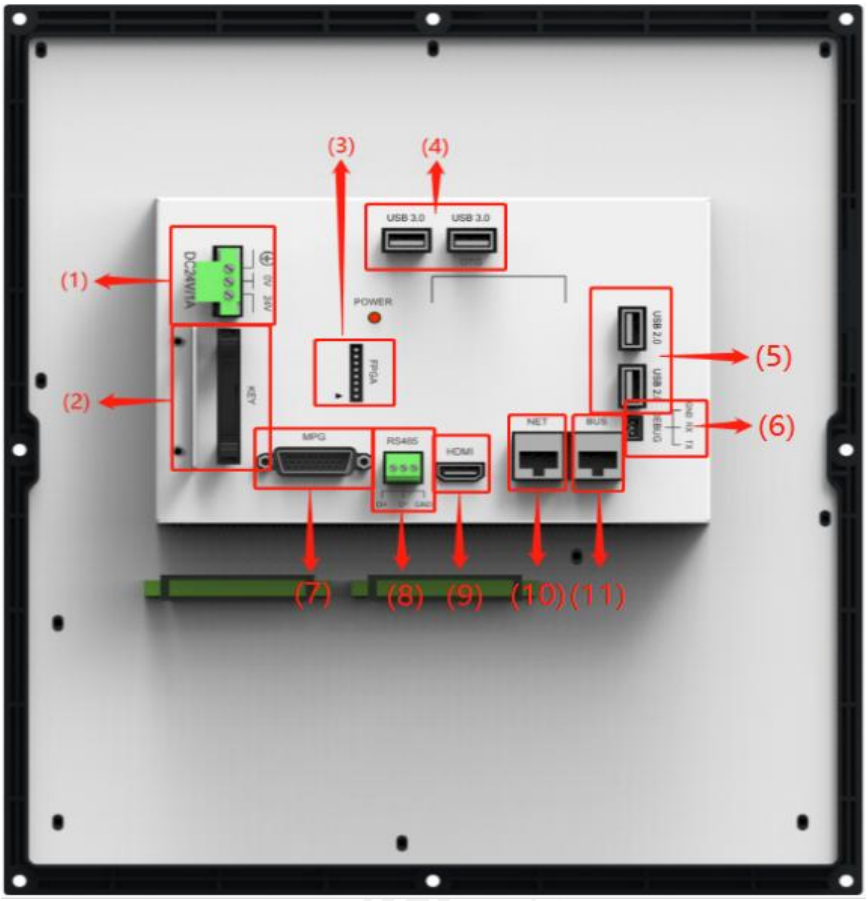
4.4.3 B0x-8 inch controller (vertical)

- **Interface description**



Front View

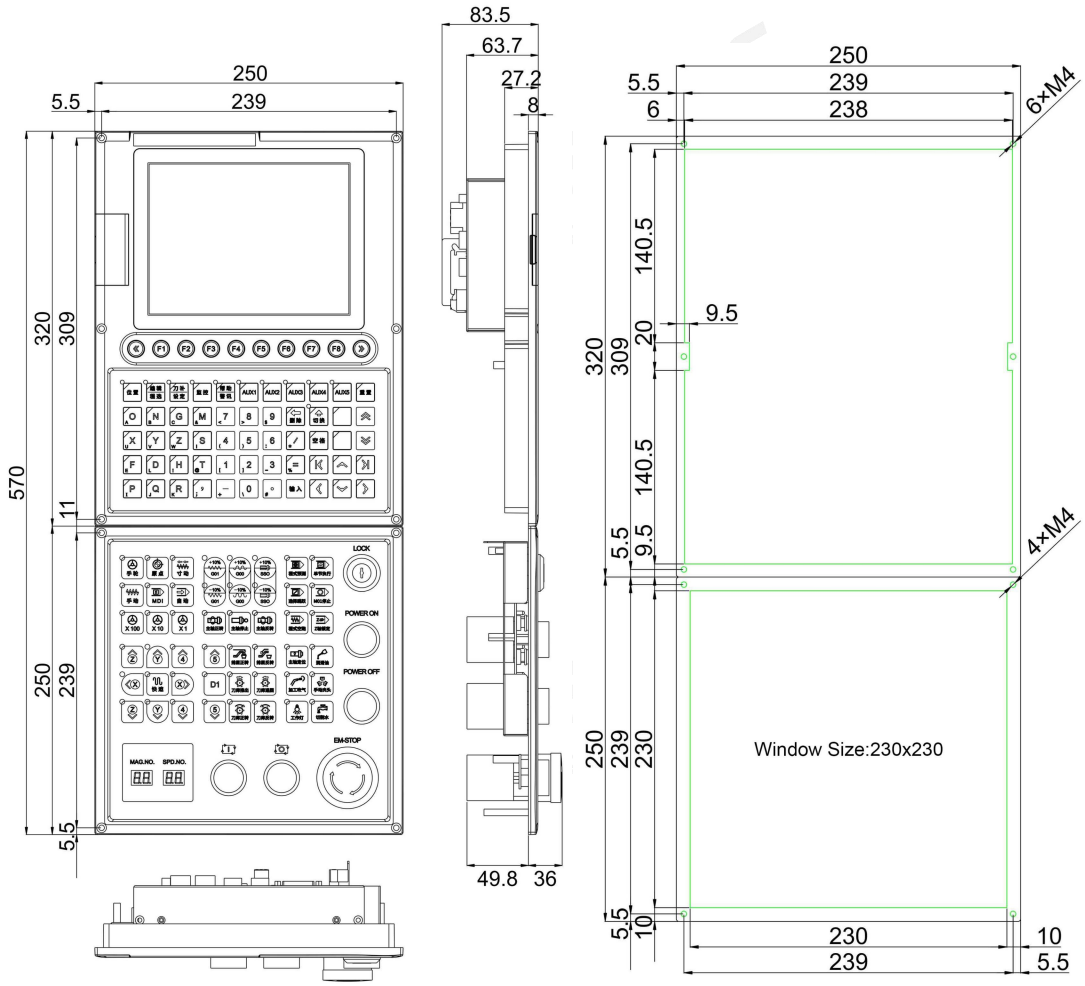
No.	Feature And Function Introduction
1	USB, RS232 communication port
2	F1-F8 function selection buttons
3	Quick menu key
4	Auxiliary Panel
5	Speed ratio selection
6	Auxiliary Panel②
7	Program lock
8	Power on switch
9	JOG feedrate switch (slow, fast)
10	Other operation aera
11	Power off switch
12	Cycle start and stop buttons
13	Emergency-stop switch
14	Normal editing and operation aera



Rear View

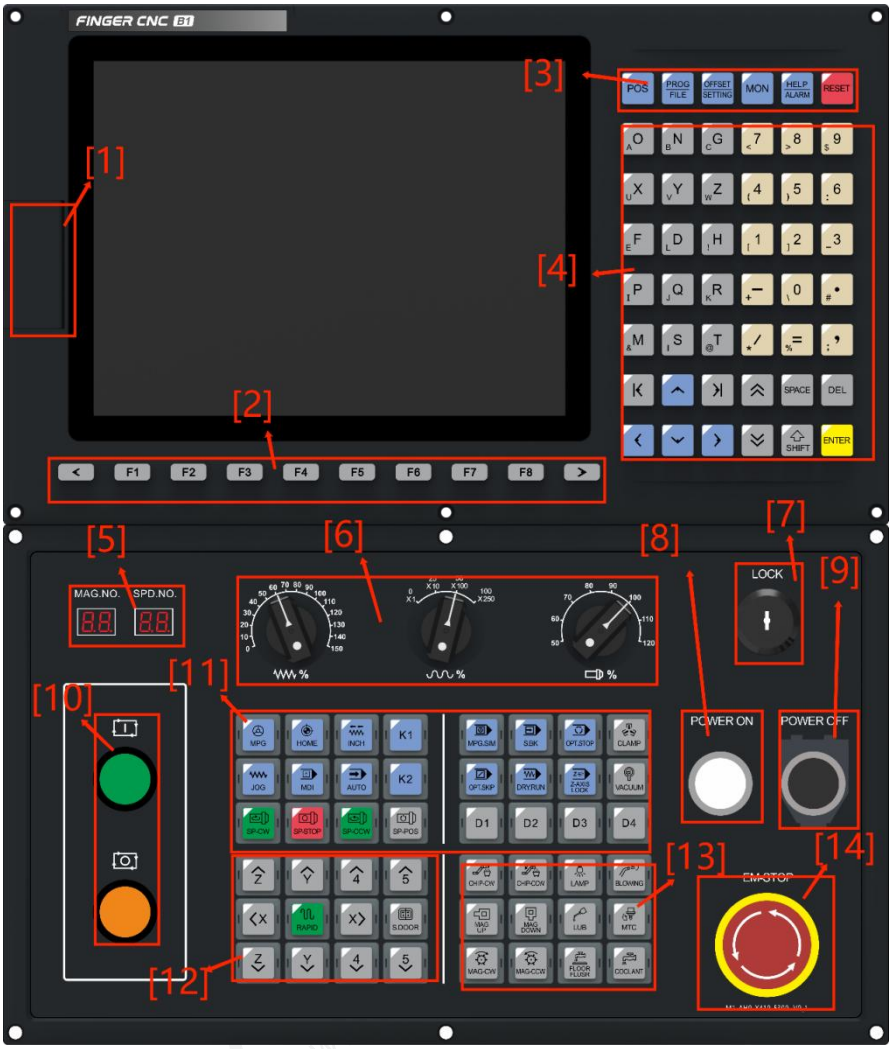
No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port
9	HDMI interface for external displayer connection
10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

- **Installation dimensions and hole positions**



4.4.4 B0x-10.4 inch controller (horizontal)

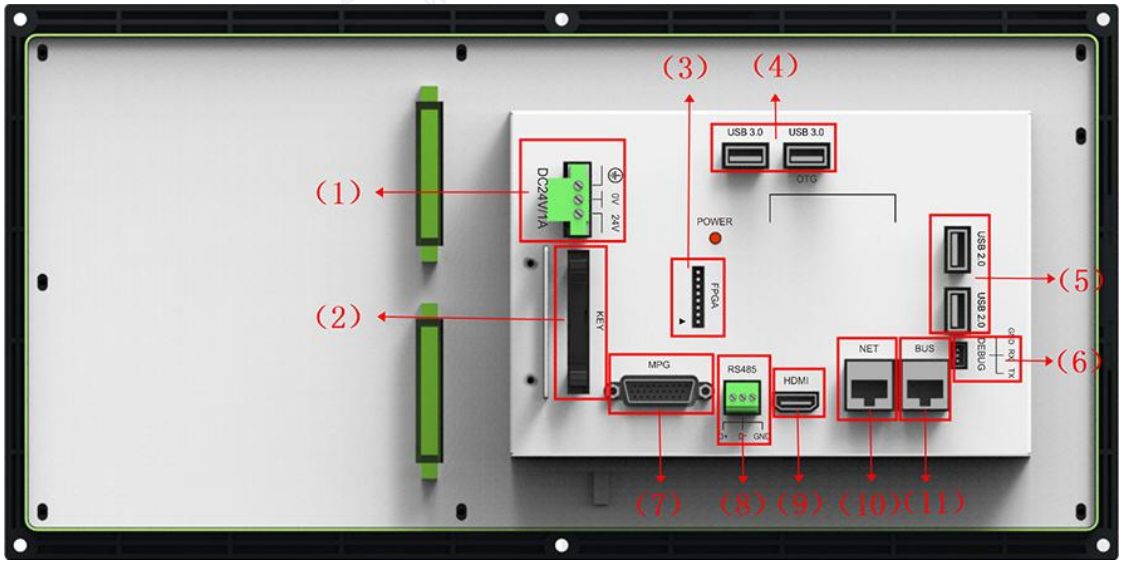
- **Interface description**



Front View

No.	Feature And Function Introduction
1	USB communication port
2	F1-F8 function selection buttons
3	Quick menu key
4	Normal editing and operation aera
5	Display Tool Number
6	Speed ratio selection
7	Program lock
8	Power on switch
9	Power off switch

10	Cycle start and stop buttons
11	Auxiliary Panel
12	JOG feedrate switch (slow, fast)
13	Other operation aera
14	Emergency-stop switch

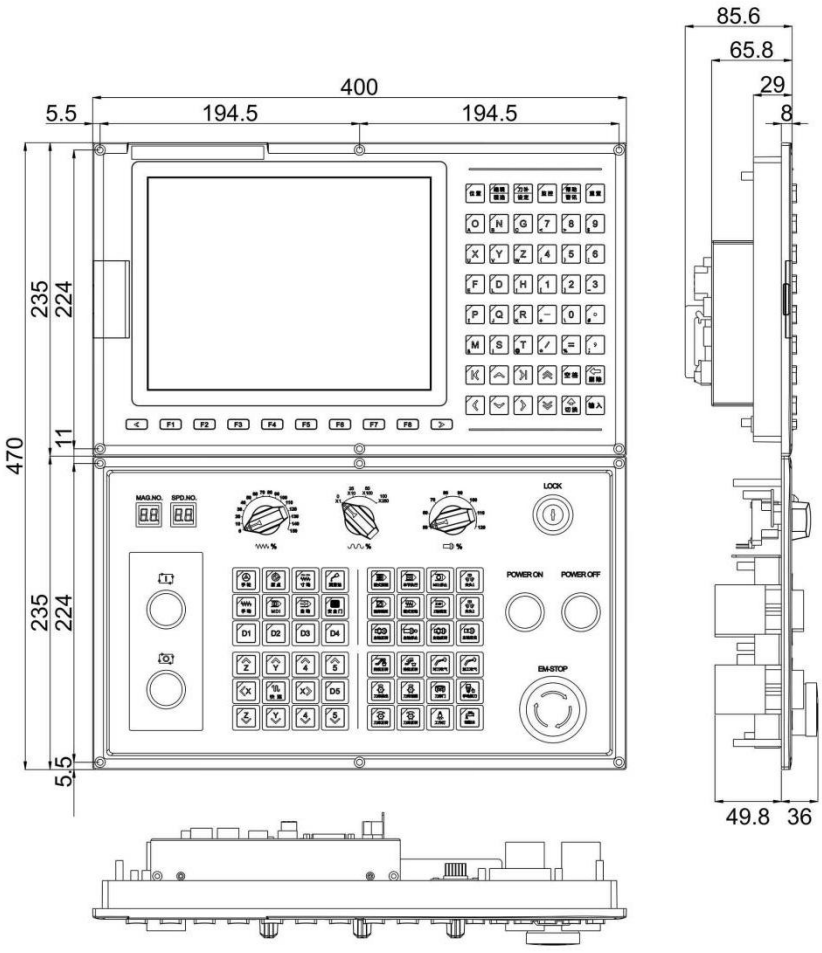


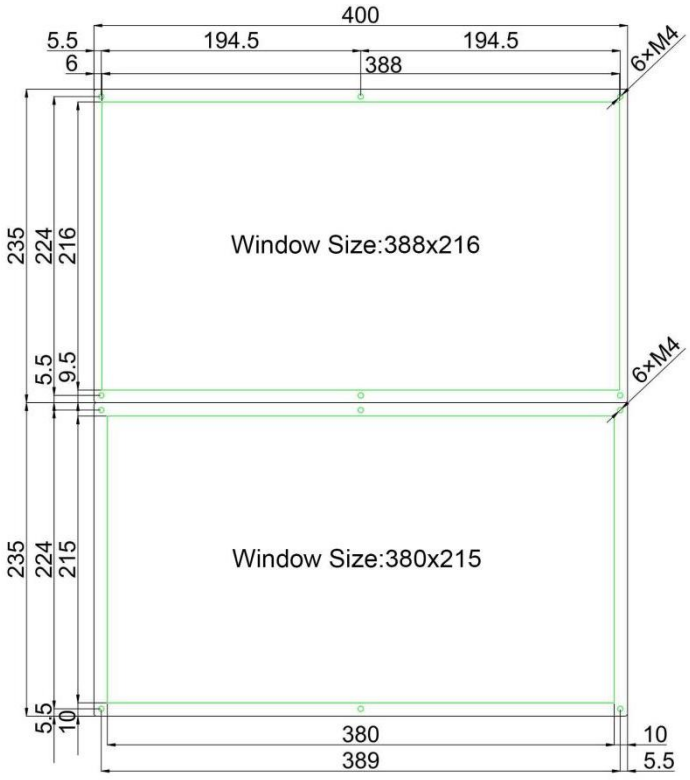
Rear View

No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port

9	HDMI interface for external displayer connection
10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

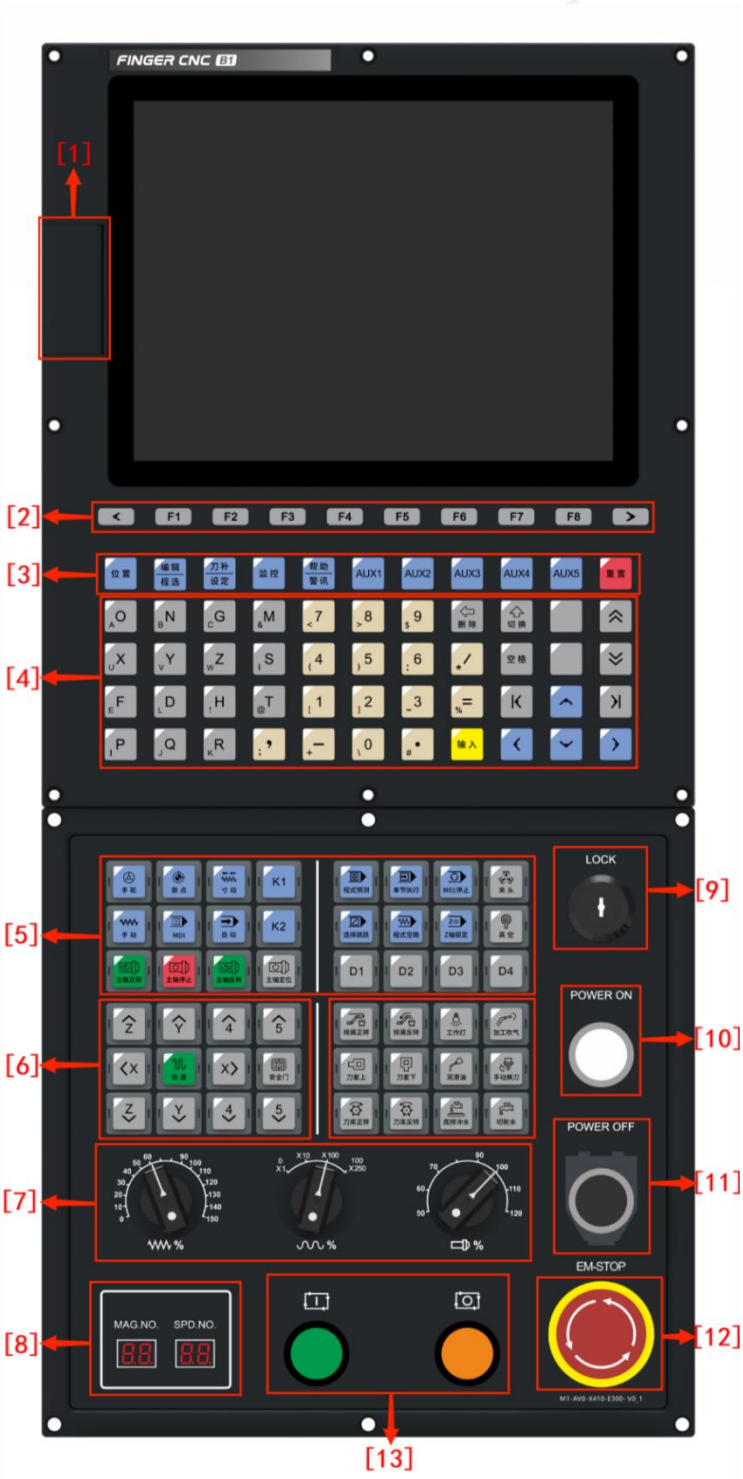
- **Installation dimensions and hole positions**


400
5.5
194.5
194.5
85.6
65.8
29
8
470
235
224
11
235
224
5.5
LOCK
POWER ON
POWER OFF
EM-STOP
49.8
36


400
5.5
194.5
194.5
6
388
6×M4
235
224
216
Window Size:388x216
5.5
9.5
6×M4
235
224
215
Window Size:380x215
5.5
10
380
389
10
5.5

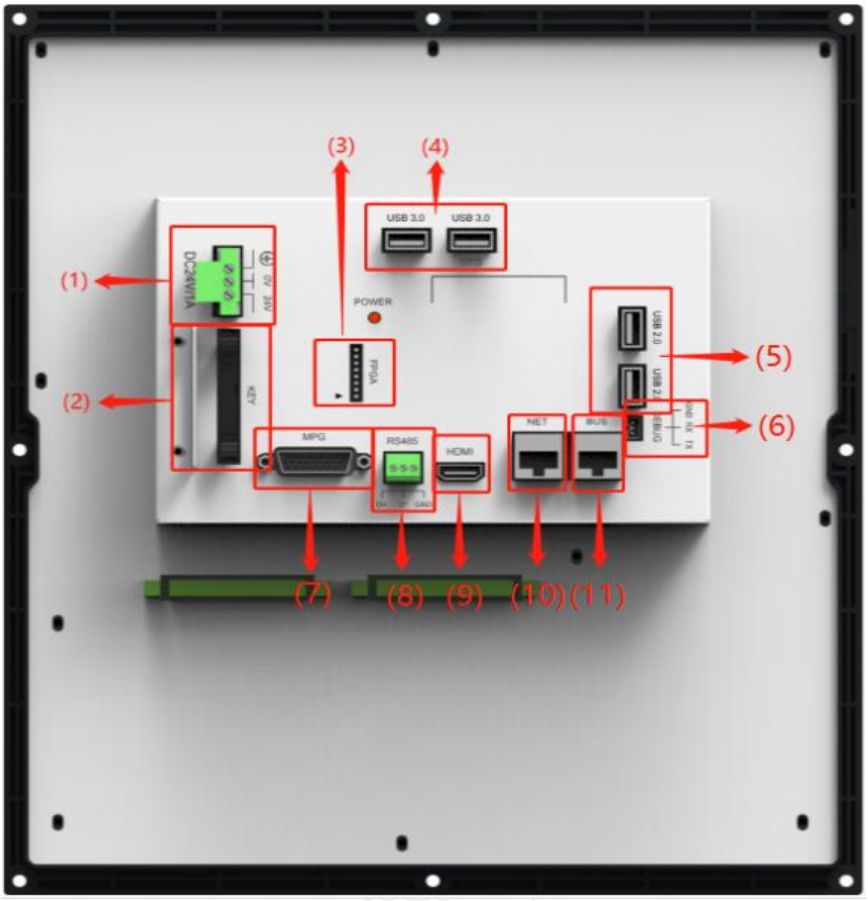
4.4.5 B0x-10.4 inch controller (vertical)

- **Interface description**



Front View

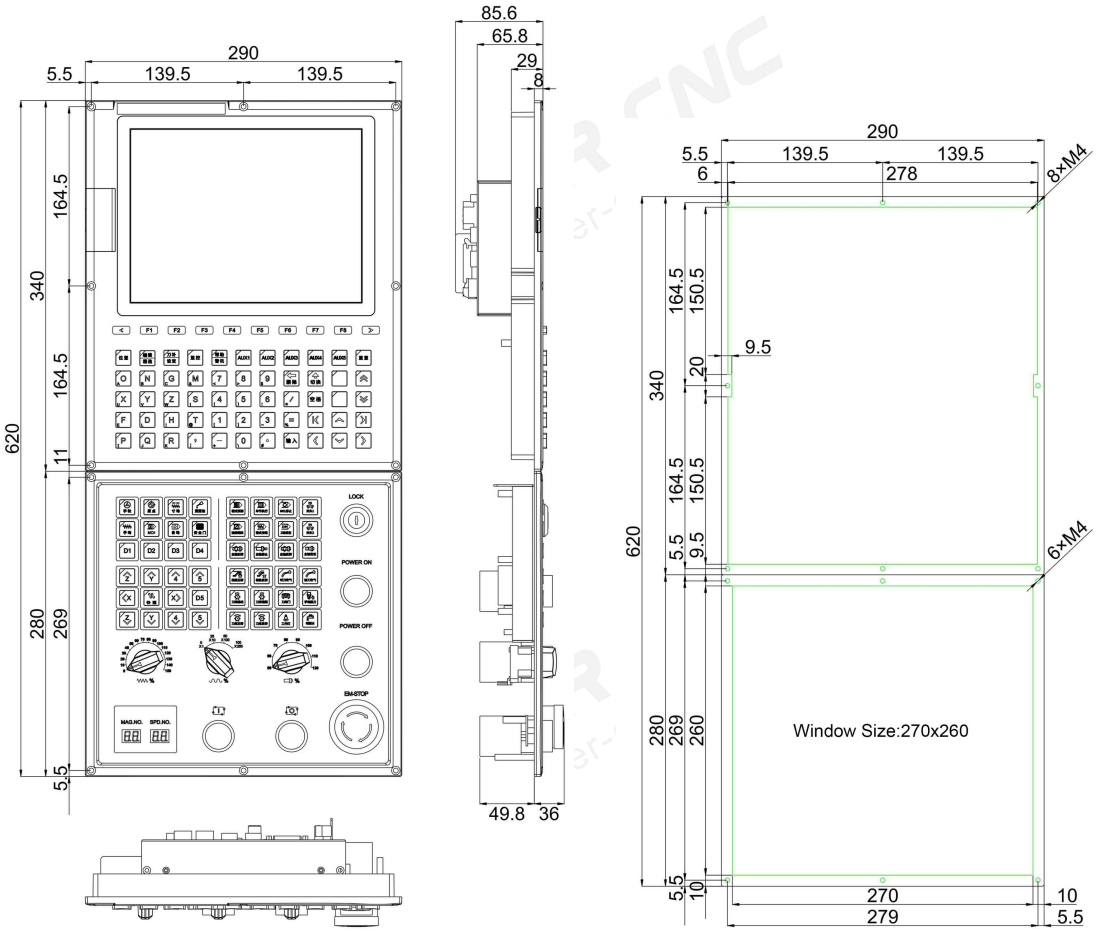
No.	Feature And Function Introduction
1	USB communication port
2	F1-F8 function selection buttons
3	Quick menu key
4	Normal editing and operation aera
5	Auxiliary Panel
6	JOG feedrate switch (slow, fast)
7	Speed ratio selection
8	Display Tool Number
9	Program lock
10	Power on switch
11	Power off switch
12	Emergency-stop switch
13	Cycle start and stop buttons



Rear View

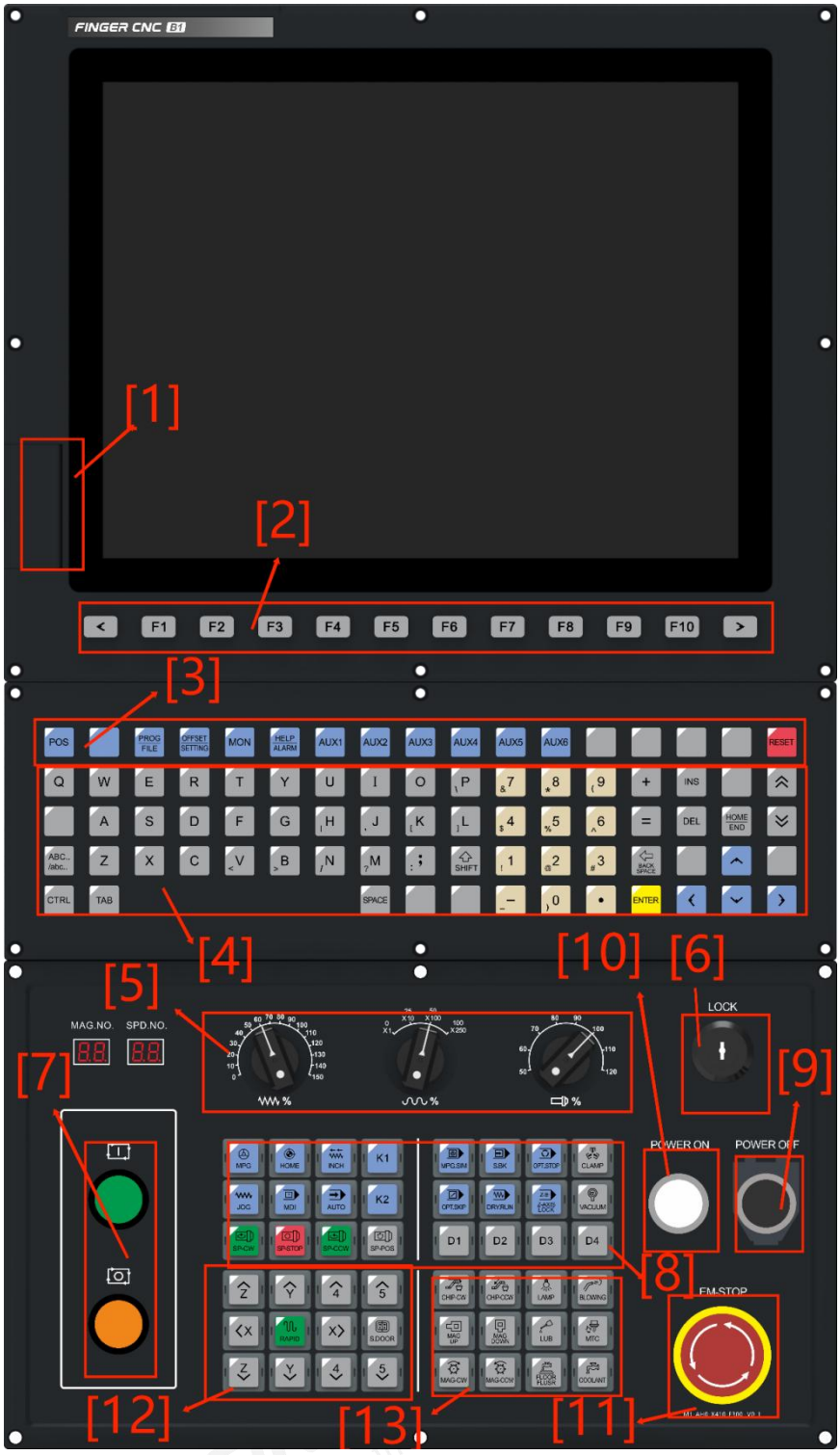
No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port
9	HDMI interface for external displayer connection
10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

- **Installation dimensions and hole positions**



4.4.6 B0x-15 inch controller (vertical)

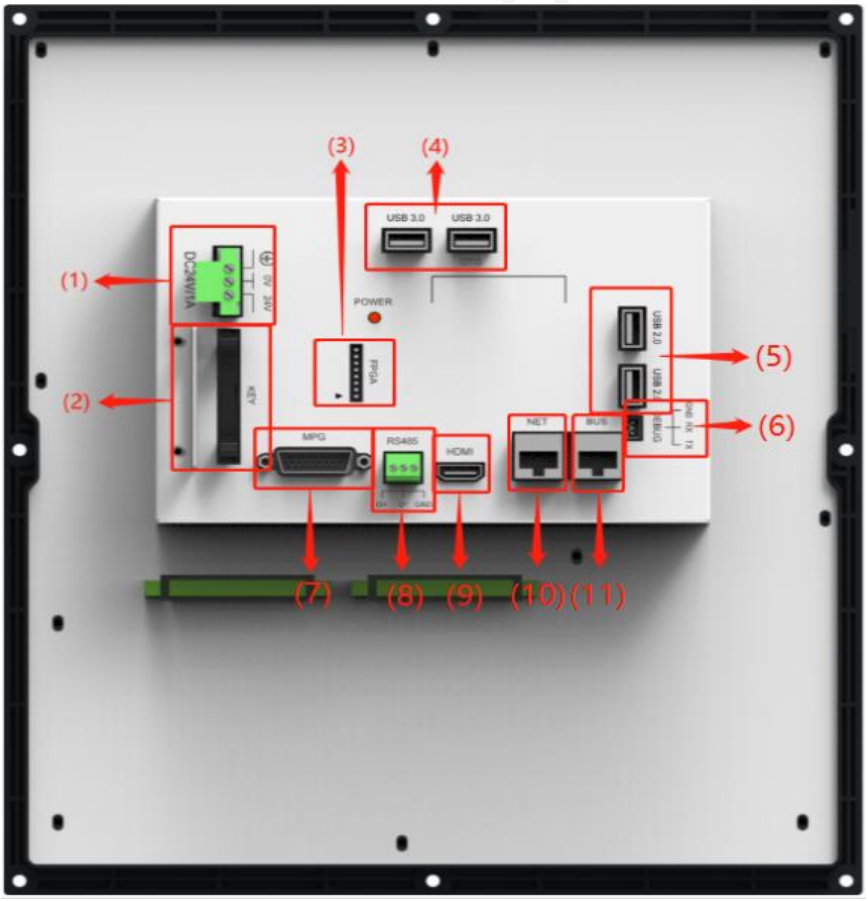
- **Interface description**



Front View

No.	Feature And Function Introduction
1	USB communication port

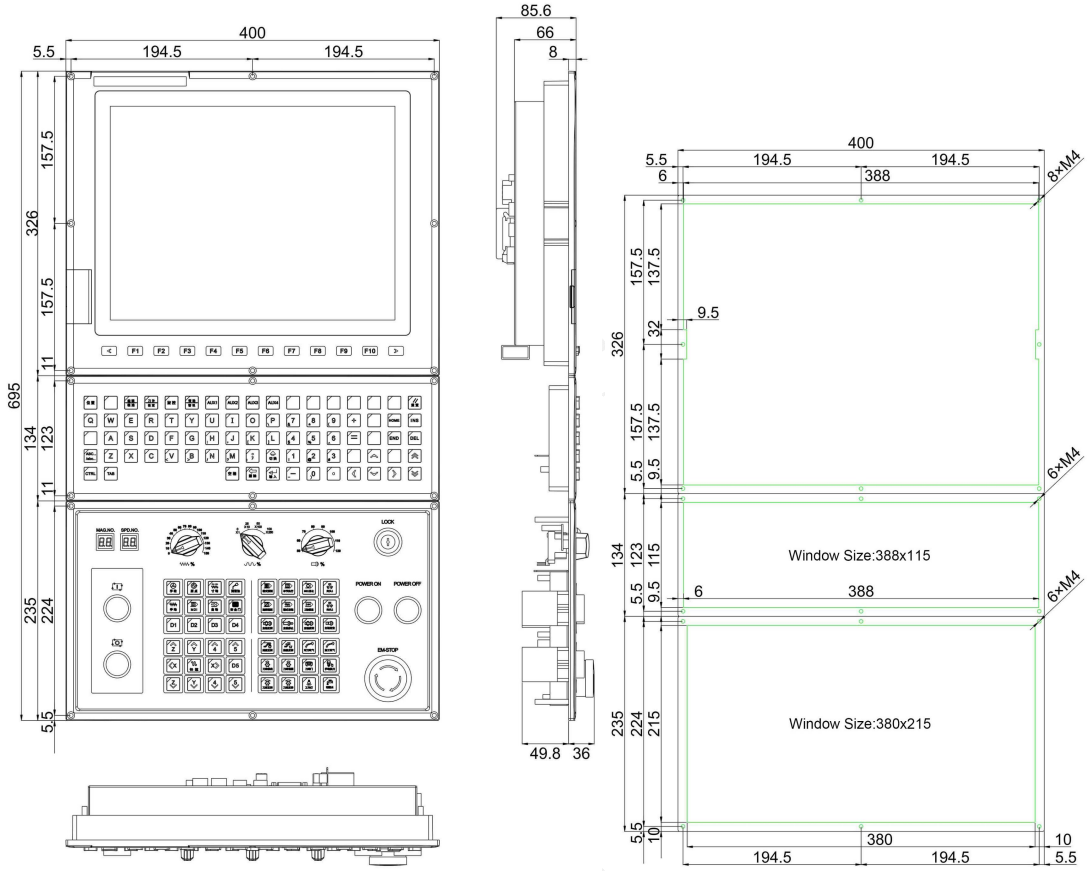
2	F1-F10 function selection buttons
3	Quick menu key
4	Normal editing and operation aera
5	Speed ratio selection
6	Program lock
7	Cycle start and stop buttons
8	Auxiliary Panel
9	Power off switch
10	Power on switch
11	Emergency-stop switch
12	JOG feedrate switch (slow, fast)
13	Other operation aera



Rear View

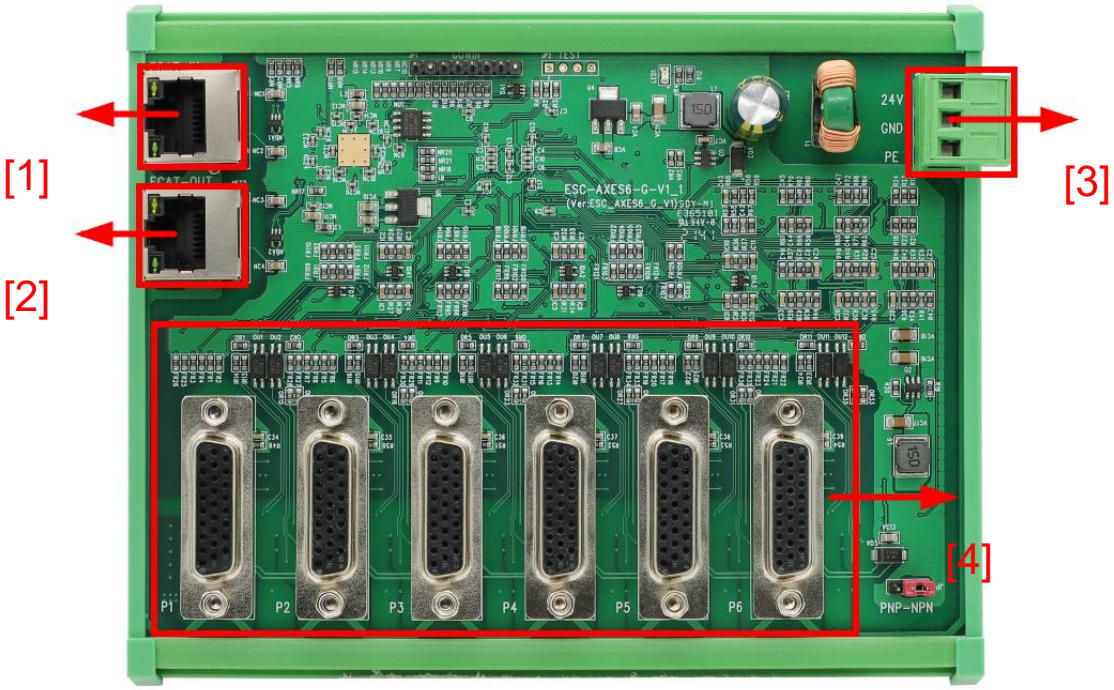
No.	Interface Introduction for Back
1	DC 24V power supply port
2	System keypad port
3	Firmware updating port
4	USB 3.0 port
5	USB 2.0 port
6	Debug port
7	MPG port
8	RS485 communication port
9	HDMI interface for external displayer connection
10	Local aera network supports OPC-UA,Modbus TCP,Ftp,etc..
11	Field Bus interface supports EtherCAT/MIII

- **Installation dimensions and hole positions**



4.4.7 ESC-AXES6-G-V1 (Six-Axis Axis Card)

- **Interface description**

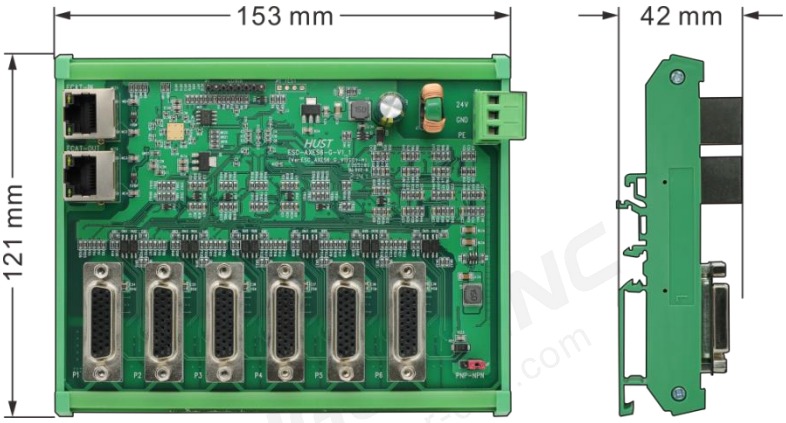


Front View

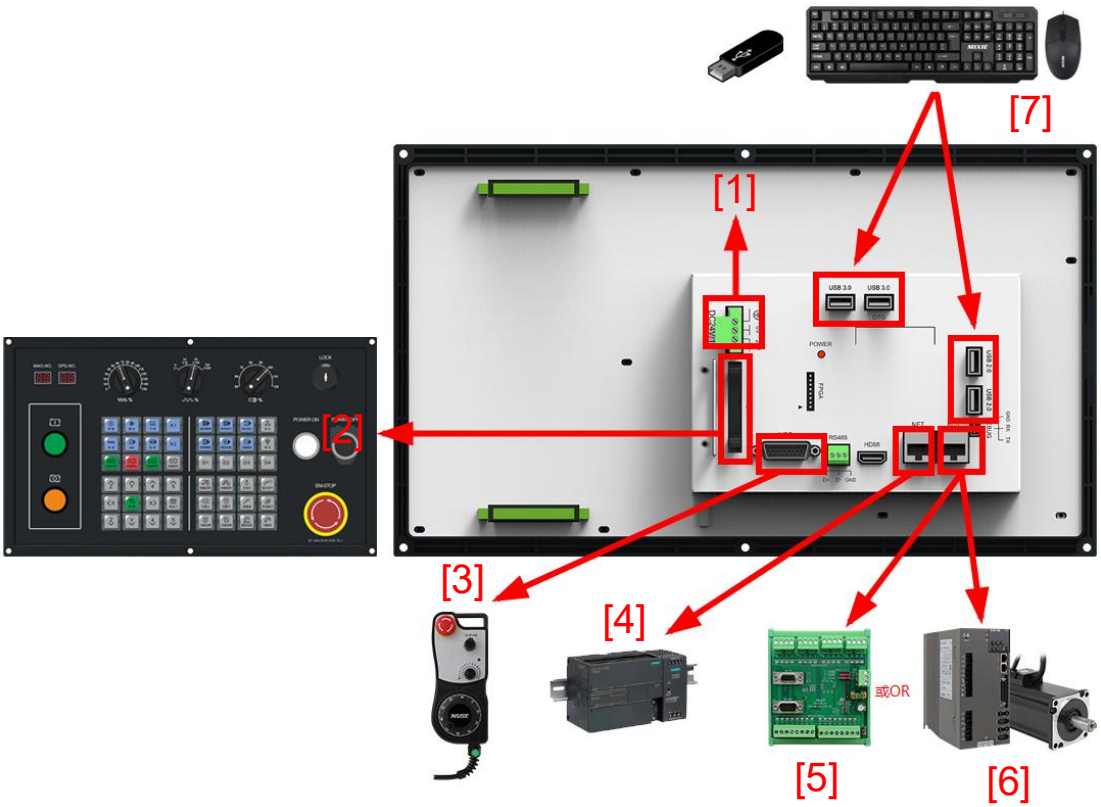
No.	Feature And Function Introduction
1	EtherCAT-in socket
2	EtherCAT-out socket
3	24V power supply
4	P1-P6 socket

Note:Axis port definition details, please refer to section 4.6 Interface Definition.

- **Installation dimensions and hole positions**



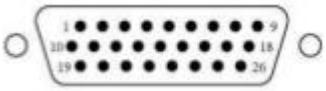
4.5 Simplified Wiring



No.	Feature And Function Introduction
1	24V power supply
2	Auxiliary panel
3	Handwheel
4	PLC
5	IO board
6	Bus servo
7	USB drive/keyboard/mouse

4.6 Interface Definition

- **B Series MPG**

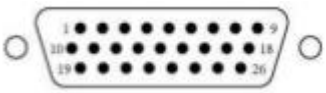


PIN	1	2	3	4	5	6	7	8	9
Definition	A1	B1	A2	B2	G31-IO			GND	5V
Hardware Definition	MPGIN0	MPGIN1	MPGIN2	MPGIN3	G31IN/O			GND-CN	VCC-CN
PIN	10	11	12	13	14	15	16	17	18
Definition							G31-I1		
Hardware Definition							G31IN/1		
PIN	19	20	21	22	23	24	25	26	
Definition	I456	I457	I458	I459	I460	I461	24V	GND-CN	
Hardware Definition	MPGI0	MPGI1	MPGI2	MPGI3	MPGI4	MPGI5	+24V-CN	GNDX-CN	

***6/7/8 pins GND are the 5V reference ground.**

***Pin 26 GND-CN is the I-point/G31 reference ground.**

- **EtherCAT module axis**

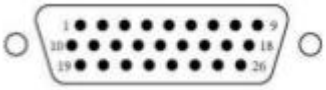


PIN	9	8	7	6	5	4	3	2	1
Definition	5V	GND	VCMD	Z-	Z+	A-	A+	B-	B+
Hardware Definition	VCC-CN	GND-CN	VCND0	GRD1	GRD0	ENC3	ENC2	ENC1	ENC0
PIN	18	17	16	15	14	13	12	11	10
Definition	OUT-*	IN-*							ADC
Hardware Definition	AXO1	AXIN1							ADIN0
PIN	26	25	24	23	22	21	20	19	
Definition	GND-CN	24V-CN	OUT-*	IN-*	CCW-	CCW+	CW-	CW+	
Hardware Definition	GNDX-CN	+24V-CN	AXO0	AXIN0	PC3	PC2	PC1	PC0	

***Pin 8 GND is for 5V/VCMD/ADC ground.**

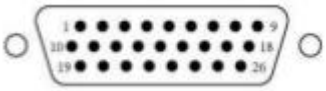
***Pin 26 GND-CN is for I point ground.**

- **Spindle voltage command control — in conjunction with a variable frequency drive**



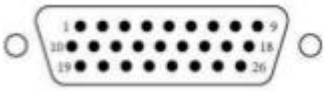
PIN	9	8	7	6	5	4	3	2	1
Definition	5V	GND	VCMD	Z-	Z+	A-	A+	B-	B+
PIN	18	17	16	15	14	13	12	11	10
Definition	forword								
PIN	26	25	24	23	22	21	20	19	
Definition	GND-CN	24V	reversal	ALM					

- **Spindle voltage command control — in conjunction with a servo drive.**



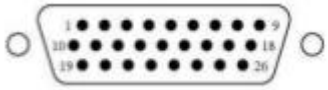
PIN	9	8	7	6	5	4	3	2	1
Definition	5V	GND	VCMD	Z-	Z+	A-	A+	B-	B+
PIN	18	17	16	15	14	13	12	11	10
Definition									
PIN	26	25	24	23	22	21	20	19	
Definition		24V	SVO	ALM					

- **Spindle pulse command control — in conjunction with a servo drive — asynchronous motor**



PIN	9	8	7	6	5	4	3	2	1
Definition				Z-	Z+	A-	A+	B-	B+
PIN	18	17	16	15	14	13	12	11	10
Definition	Position control								
PIN	26	25	24	23	22	21	20	19	
Definition	GND-CN	24V	Speed control	ALM	CCW-	CCW+	CW-	CW+	

- **Spindle pulse command control — in conjunction with a servo drive — servo motor**



PIN	9	8	7	6	5	4	3	2	1
Definition				Z-	Z+	A-	A+	B-	B+
PIN	18	17	16	15	14	13	12	11	10
Definition									
PIN	26	25	24	23	22	21	20	19	
Definition		24V	SVO	ALM	CCW-	CCW+	CW-	CW+	

4.7 Common Servo Drive Wiring

Note: For detailed pin definitions of the drive, please refer to the corresponding drive manual.

- **Finger C3 Series (Pulse NPN)**

NPN	
ESC-AXES6-G-V1	Finger c3
3（A+）	20(APAO+)
4（A-）	5(APAO-)
1（B+）	21(BPBO+)
2（B-）	6(BPBO-)
5（Z+）	22(CPCO+)

6（Z-）	7(CPCO-)
7（vcmd）analog signal	
8（GND）ground for 10V & 5V	
9（5V+）	
10（TOG）	
11（）	
12（）	
13（）	
14（）	
15（）	
16（）	
17（IN）	
18（OUT）	
19（P+）	1（P+）
20（P-）	16（P-）
21（S+）	2（S+）
22（S-）	17（S-）
23（IN）	32(ALM+)
24（OUT）	18（SON）
25（24V）	31（DICOM）
26（24V GND）	38（DOCOM）

- **Yaskawa (Pulse NPN)**

NPN	
ESC-AXES6-G-V1	Yaskawa Σ-V
3（A+）	33（PAO）

4（A-）	34（/PAO）
1（B+）	35（PBO）
2（B-）	36（/PBO）
5（Z+）	19（PCO）
6（Z-）	20（/PCO）
7（vcmd）analog signal	
8（GND）ground for 10V & 5V	
9（5V+）	
10（TOG）	
11（）	
12（）	
13（）	
14（）	
15（）	
16（）	
17（IN）	
18（OUT）	
19（P+）	7（PULS）
20（P-）	8（/PULS）
21（S+）	11（SIGN）
22（S-）	12（/SIGN）
23（IN）	31ALM+
24（OUT）	40（son）
25（24V）	47（+24Vin）
26（24V GND）	32ALM-

Delta B2 wiring diagram

NPN	
ESC-AXES6-G-V1	Delta B2
3（A+）	21（A+）
4（A-）	22（A-）
1（B+）	25（B+）
2（B-）	23（B-）
5（Z+）	13（Z+）
6（Z-）	24（Z-）
7（vcmd）analog signal	
8（GND）ground for 10V & 5V	
9（5V+）	
10（TOG）	
11（）	
12（）	
13（）	
14（）	
15（）	
16（）	
17（IN）	
18（OUT）	
19（P+）	41（P+）
20（P-）	43（P-）
21（S+）	37（S+）
22（S-）	39（S-）
23（IN）	28(ALM)
24（OUT）	9（SON）

25（24V）	11(24V com)
26（24V GND）	14（24V com）

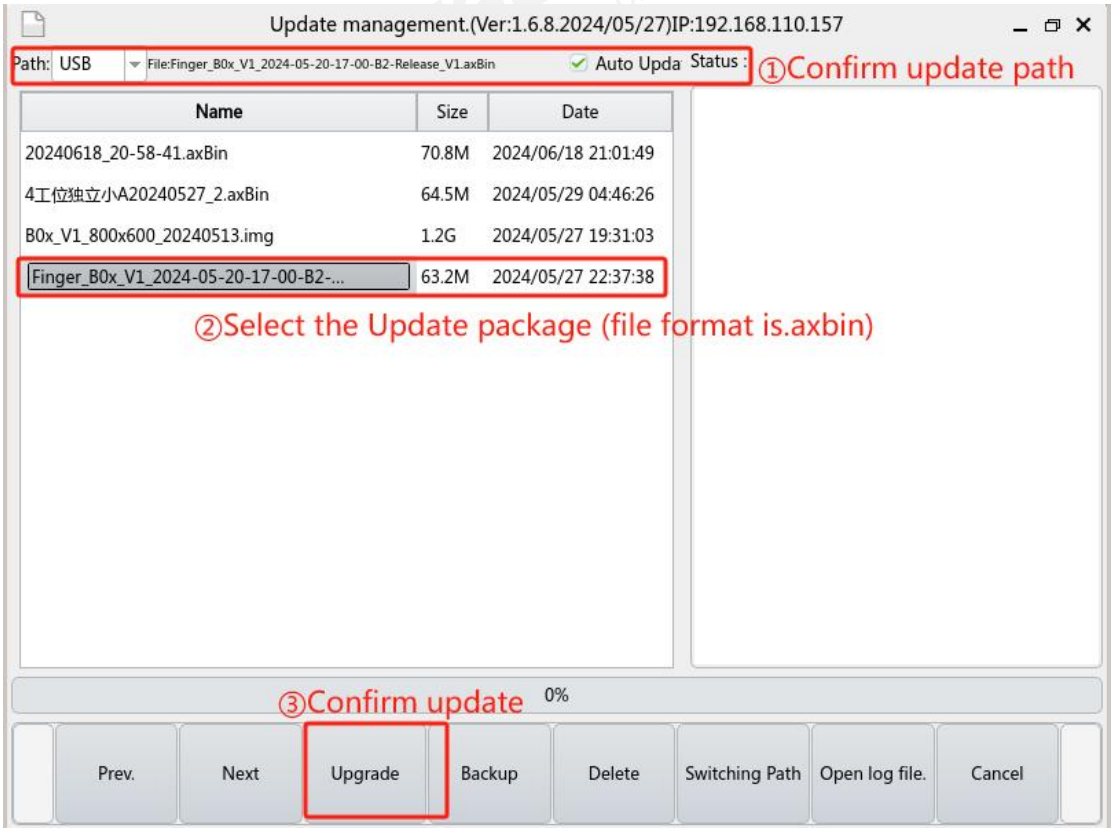
4.8 Wiring precautions

- When wiring the machine, please ensure that the wire ends are properly terminated or soldered.
- If non-standard servo cables are used, it is essential to measure the correctness of all pin connections before conducting the power test. Incorrect wiring can result in abnormal output of the controller's command and may cause controller failure.
- When using an external 24V DC power supply for wiring, it is recommended to use products that are certified for safety and have protection functions to prevent faults due to wiring errors. (Recommended specifications: must comply with the requirements of EN60950&UL1950)
- If MECHATROLINK-III functionality is used, it is advisable to use CAT5e or CAT6 cables for network communication to ensure smooth network connectivity and avoid noise interference.
- Grounding wire instructions:
- The grounding wire size should comply with the regulations of the electrical equipment technical standards, and shorter grounding wires are preferable.
- The grounding wire of the controller must not be connected to the grounding of high-current loads such as welding machines or large motors. They must be grounded separately.

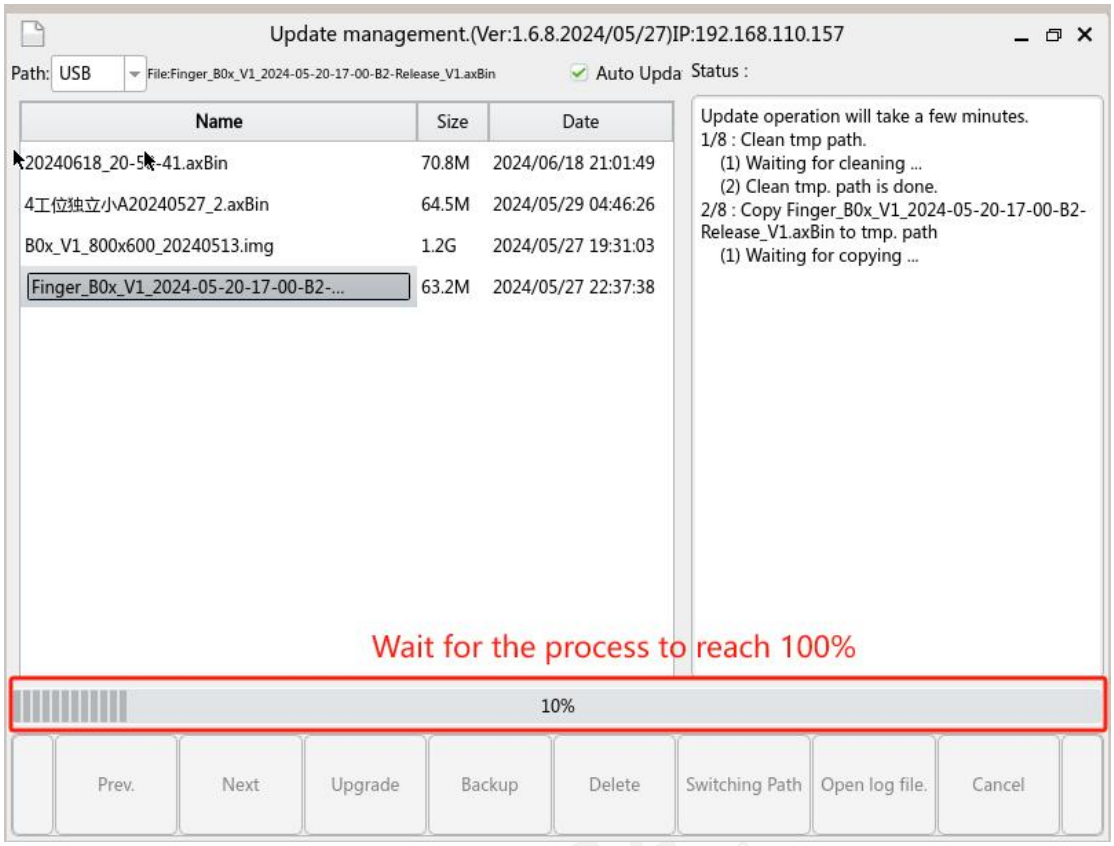
Part 5. Appendix

5.1 Software Upgrade/Backup Operation Steps

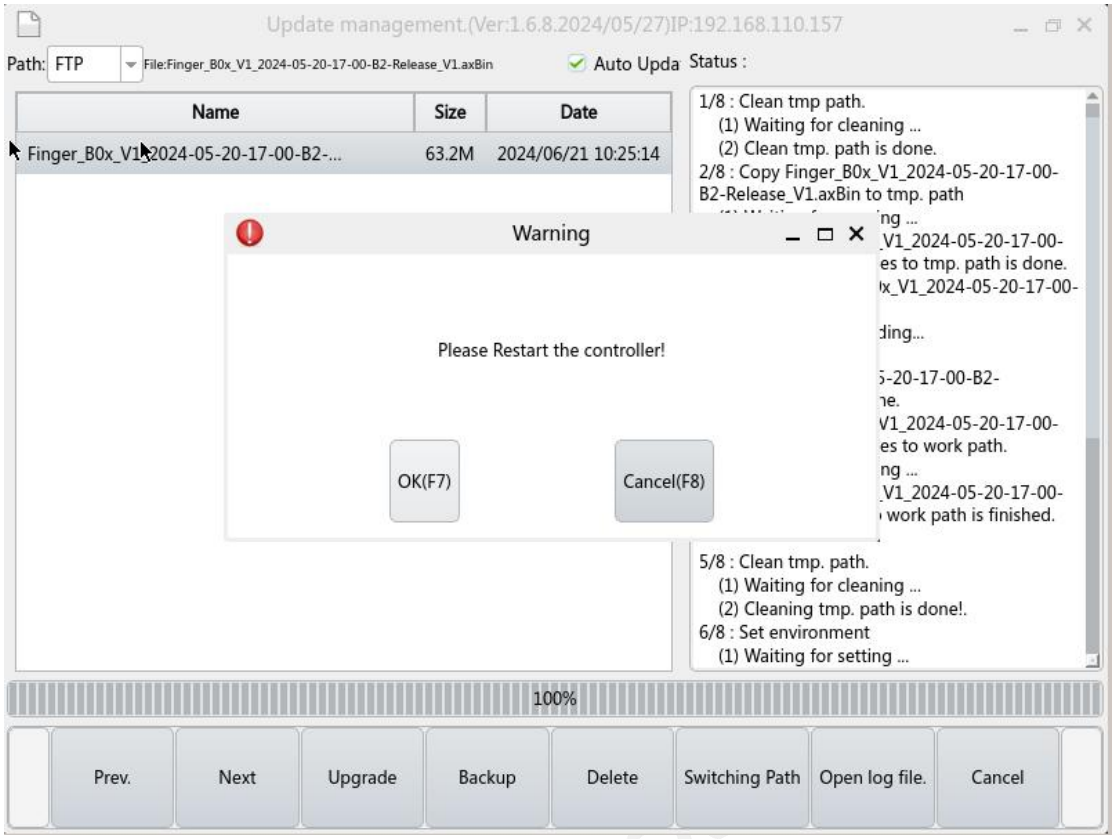
A. Transfer the update package to a USB flash drive and insert it into the FINGER CNC machine. Wait for the system to display the update interface.



During the update process..

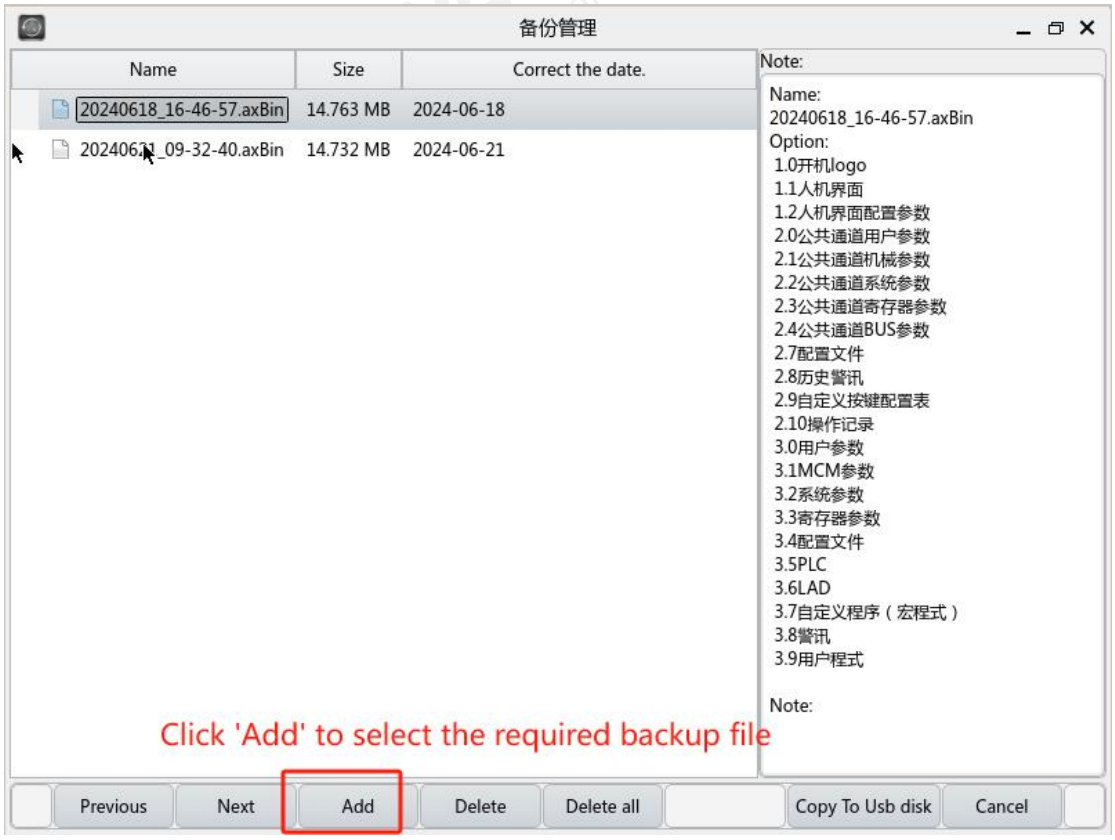


B. Update Completed: After the update process reaches 100%, a prompt will appear saying "Please restart the system!" Restart the system to complete the update.

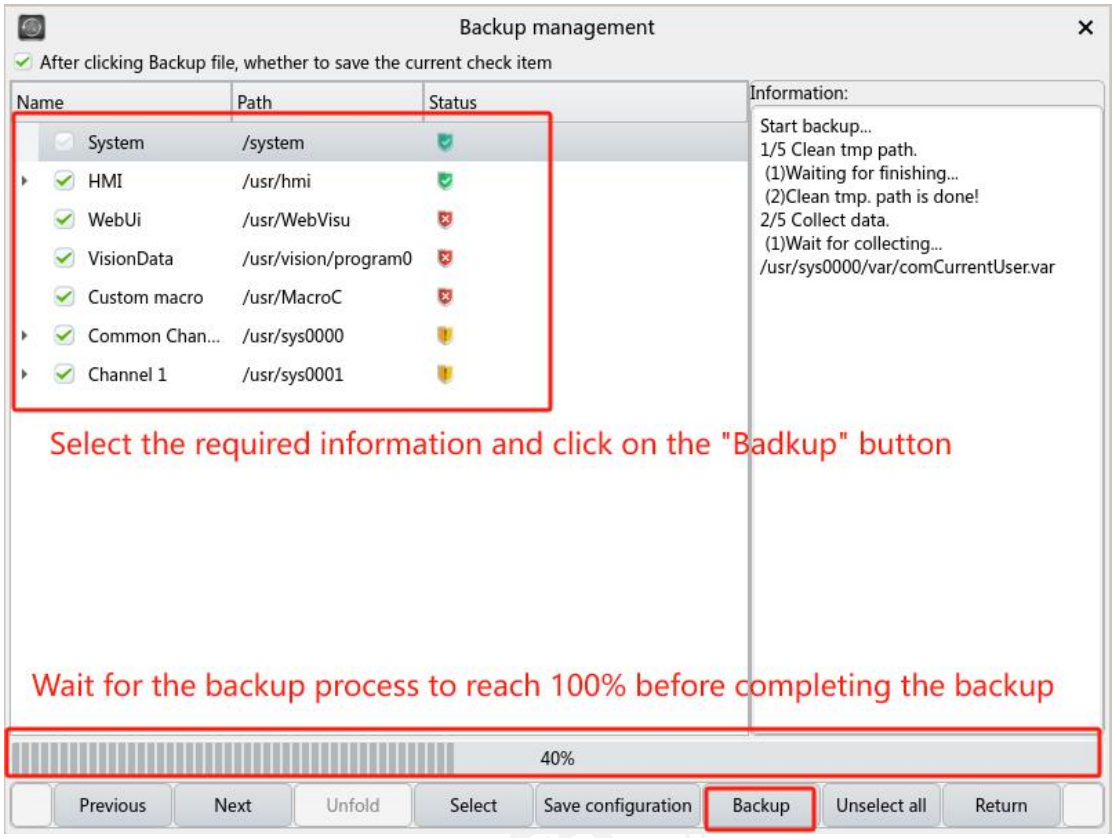


Backup

Click the backup button to enter the backup interface.



C. Select and operate on the backup files.



FINGER CNC

Guangzhou Finger Technology Co.,Ltd

Hotline：020-39389901
Repair Helpline：18127931302
Fax：020-39389903
Postal Code：511495
E-mail：finger@fingercnc.com
Website：www.finger-cnc.com
Address：1F,No. 8, Chengding Street, Zhongcun Street, Panyu District, Guangzhou City, Guangdong Province

Official Website

Official Wechat