FINGER CNC

Open Designs Customize The Future

B-Series Single-Channel Lathe CNC Operation Manual

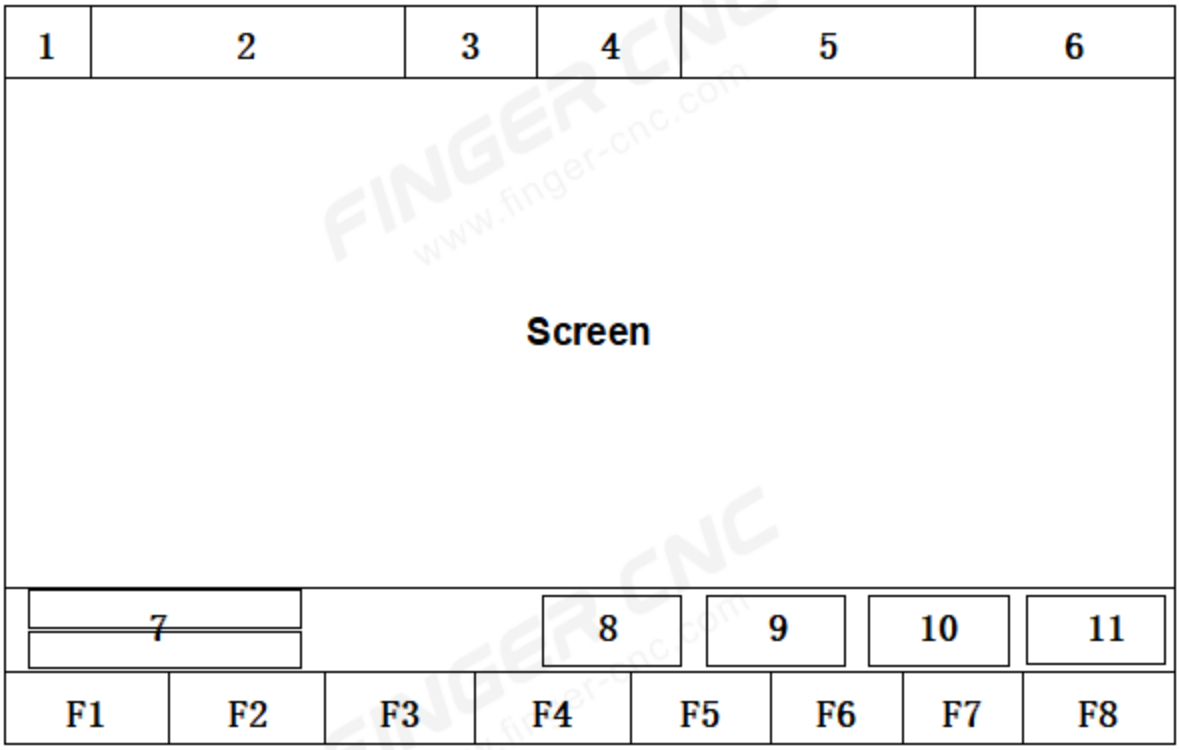
Version No: F202412SLO-CN

Guangzhou Finger Technology Co., Ltd

CONTENTS

Part 1.Function Keys and System Overview

1.1 Introduction to the Main Screen



- **Screen Component Description**

1. Current Workpiece Coordinate System
2. Current Executing Program Name and Jumped Subprogram File Name
3. Current Executing Program Line Number and Jumped Subprogram Line Number
4. Page Title
5. System Date and Time
6. User Permissions
7. Data Input Box/Data setting range
8. Current Status
9. Run Mode
10. Function selection
11. Alarms

1.2 Functional Tree Diagram

1.3 Machine Position

- **Operation Path**

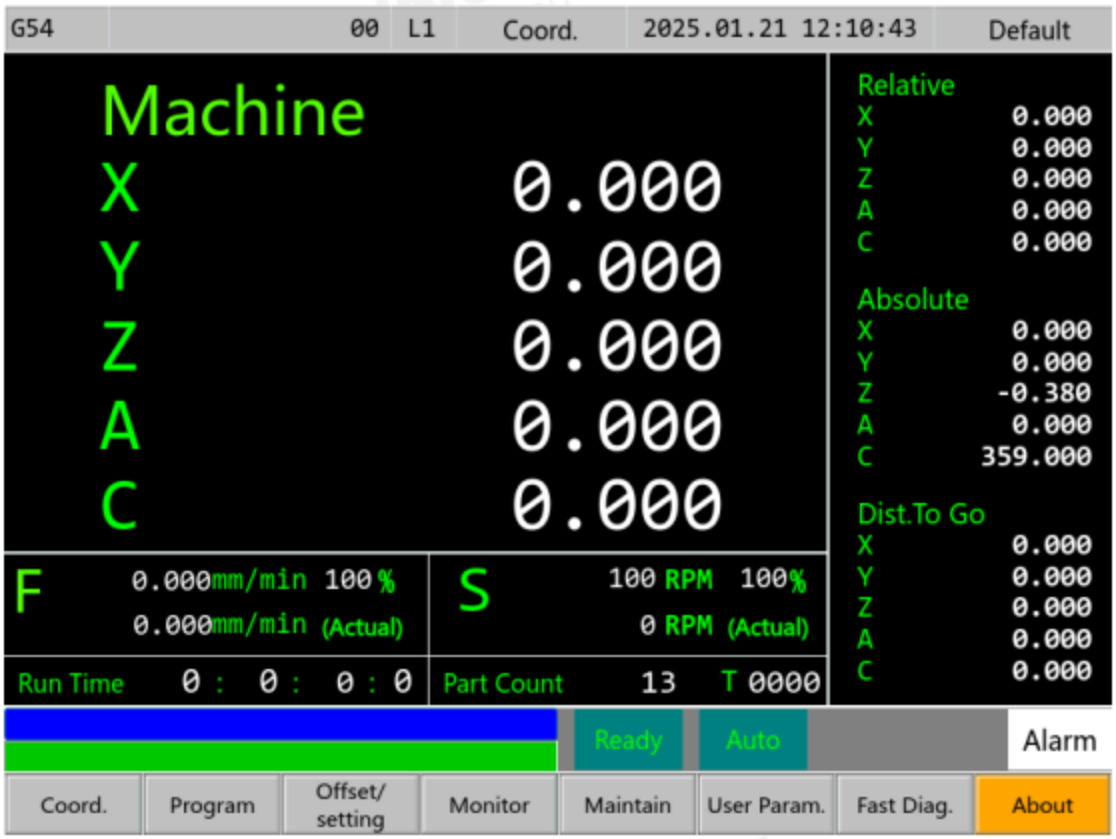
From any page, press the system shortcut key "Position" to enter the "Machine Position"

page. (Applicable to Eight-Key or Ten-Key series).

- **Description**

1. Perform operations on the current machine coordinates.
2. Display commonly used machining information.

1.3.1 Screen Description



1.3.1.1 Coordinate Display

1. This screen simultaneously displays four types of coordinates.
2. You can use the F1 [Machine Coordinates] button → F1 [Coordinate Switch] to switch the coordinate type displayed by the current main coordinates.

1.3.1.2 F (Feedrate)

1. Displays the user-set feedrate.
2. Displays the user-set feedrate speed percentage (F value multiplier).
3. Displays the feed rate of the actual machining.

1.3.1.3 S (Spindle Speed)

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

1.3.1.4 Processing Time

Displays the elapsed time of the current program execution.

1.3.1.5 Workpiece Count

Displays the number of times the program has been executed.

1.3.1.6 T (Tool Number and Tool Compensation)

Displays the tool number and tool compensation used in the current machining. The higher two digits represent the tool number, and the lower two digits represent the tool compensation. The tool number changes only for non-fixed tooling, while the tool compensation represents the real-time tool compensation.

1.3.2 Coordinate Switching

- **Operation Path**

"Machine Position" page → Press F1 for "Machine Coordinates" → Press F1 for "Coordinate Switching".

- **Description**

This button allows you to switch to the machine coordinates screen and change the display order of the coordinate types.

1.3.3 1/2 Coordinates

- **Operation Path**

"Machine Position" page → F1 [Machine Coordinates] → F2 [1/2 Coordinates].

- **Description**

1. Divides the values of corresponding axis relative coordinates by 2.
2. When used in conjunction with the "Relative Coordinate Setting" function, it allows for quickly obtaining the midpoint coordinates between any two points.

- **Operation Method**

"Machine Position" page → F1 [Machine Coordinates] → Enter the desired axis in the data input box → F2 [1/2 Coordinates].

- **Operation Example**

1. Current relative coordinate of the X-axis is 10.000.
2. Enter "X" in the data input box (no need to press the Enter key).
3. Press F1 [1/2 Coordinates] key.
4. The X-axis relative coordinate display value will change to 5.000.

1.3.4 Zero Relative Coordinate

- **Operation Path**

[Machine Position] page→F1 [Machine Coordinate]→F3 [Zero Relative Coordinate].

- **Description**

Set the corresponding axial relative coordinates to 0.

- **Operation Method**

Enter the desired axis symbol and coordinate value in the input box, then press F3 [Zero Coordinate Restore].

- **Operation Example**

1. The current relative coordinate for the X-axis is 10.000.

2. Enter "X0.000" in the input box on the screen.
3. Press F3 [Zero Coordinate Setting].
4. The X-axis relative coordinate display value will be changed to 0.000.

1.3.5 Zero All Relative Coordinate

- **Operation Path**

[Machine Position] page→F1 [Machine Coordinate]→F4 [Zero All Relative Coordinate].

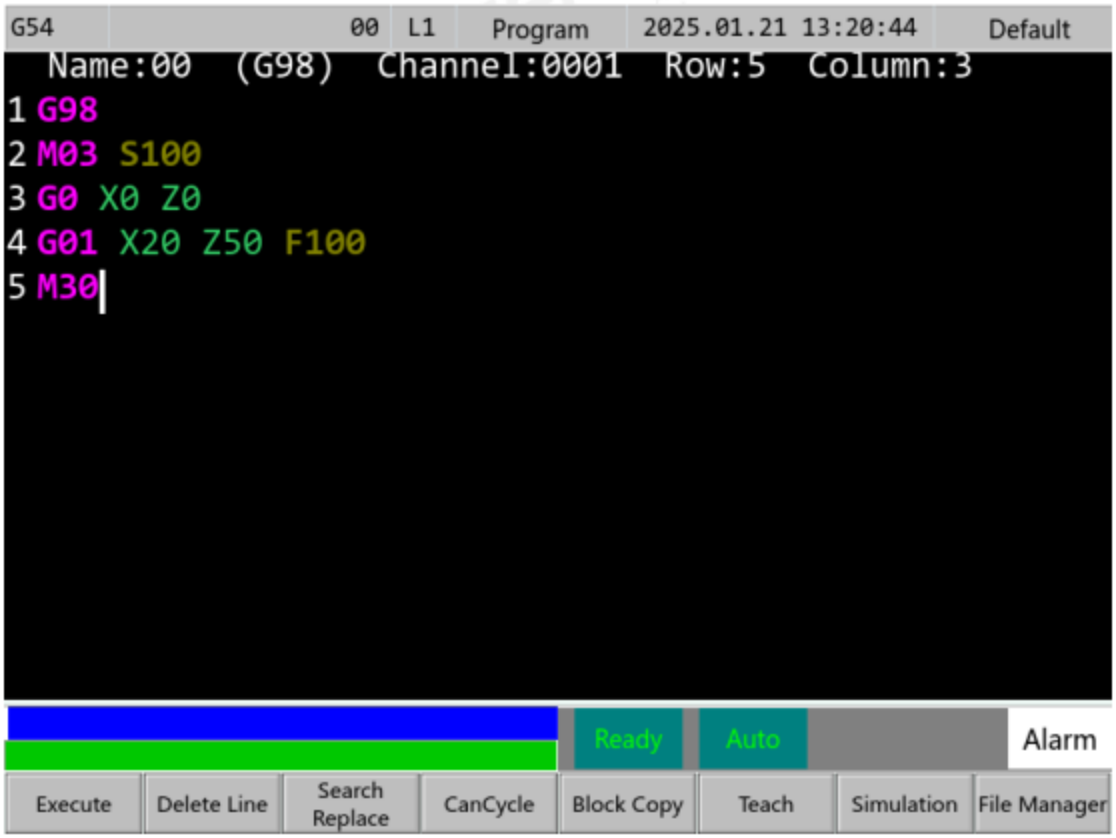
- **Description**

Set all axial coordinates to 0.000.

- **Operation Method**

Press F4 [Zero All Relative Coordinate]，Set all axial coordinates to 0.000.

1.4 Program Editing



- **Operation Path**

1. Path 1: Use the system panel shortcut key [Edit/Prog Select] to switch the page to "Program Edit".
2. Path 2: "Program Select" page → system panel shortcut key [Enter] → "Program Edit" page.
3. Path 3: "Machine Position" page -> F2 [Prog Edit] -> "Program Edit" page.

- **Description**

Used for editing operations on the machining program.

- **Operation Method**

1. Use the directional keys [↑] [↓] [←] [→] to move the cursor.
2. Use [Page Up] and [Page Down] to scroll up and down.
3. Use the system panel shortcut key [Edit/Prog Select] to quickly switch between the "Program Edit" and "Program Select" pages.

1.4.1 Loading Machining

- **Operation Path**

"Program Edit" page → F1 [Load Program].

- **Description**

Use this button to designate the edited program as a machining program and switch the page to the "Machining Monitoring" page.

- **Note**

This button is invalid if the program is currently being executed.

1.4.2 Line Number Search

- **Operation Path**

"Program Edit" page →[>>]→F1 [Line Search].

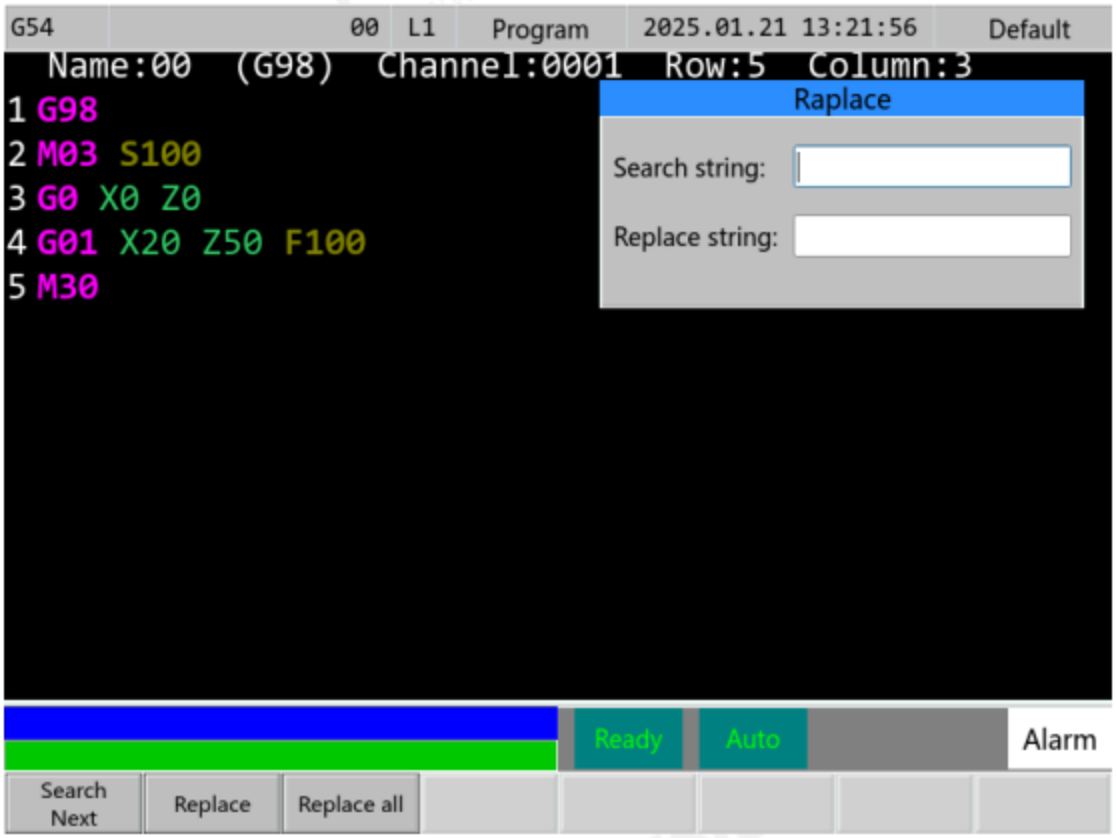
- **Description**

When the machining program is long, this function allows direct positioning of the cursor to the desired line number.

- **Operation Description**

1. Press F1 [Line Search] to open the input dialog box.
2. Enter the line number to jump to in the input area of the dialog box.
3. Press F1 [Confirm], and the cursor will automatically move to the specified line number.
4. Press F2 [Cancel] to exit the jump function and close the input dialog box.

1.4.3 Search/Replace



- **Operation Path**

"Program Edit" page → F3 [Search/Replace].

- **Description**

1. This button allows you to search for or replace characters in the current edited program.
2. After pressing the [Search/Replace] button, a sidebar and a toggle button group will appear on the right side. You can enter the characters to be searched in the "Search string" input box and the characters to replace in the "Replace String" input box.

- **Operation Description**

Using the arrow keys [↑][↓]on the current page allows the cursor to switch between the [Search string] and [Replace string] input boxes.

1.4.3.1 Close

- **Operation Path**

"Program Edit" page → F3 [Search/Replace] → [<<].

- **Description**

When you no longer need to perform search/replace operations, press this button to close the right sidebar and return to the button group.

1.4.3.2 Switch Input Fields

- **Operation Path**

"Program Edit" page → F3 [Search/Replace]→arrow key [↑][↓].

- **Description**

This function allows you to switch the cursor position between the "Search" and "Replace" input fields.

1.4.3.3 Search

- **Operation Path**

"Program Edit" page → F3 [Search/Replace]→F1 [Search Next].

- **Description**

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

1.4.3.7 Replace

- **Operation Path**

"Program Edit" page → F3 [Search/Replace] → F2 [Replace].

- **Description**

This button is used to search for the characters entered in the "Find" input box and replace them with the characters entered in the "Replace" input box in the code editor. Pressing

this button once replaces one occurrence of the characters.

1.4.3.8 Replace All

- **Operation Path**

"Program Edit" page → F3 [Find/Replace] → F3 [Replace All].

- **Description**

This button is used to search for the characters entered in the "Find" input box and replace all occurrences of them with the characters entered in the "Replace" input box in the code editor. Pressing this button replaces all occurrences of the characters.

1.4.4 Program Selection

- **Operation Path**

"Program Editor" page → F8 [File Manager].

- **Description**

Used for the management of machining documents. Please refer to section 1.5 for an introduction to the "Program Selection" functionality.

1.4.5 Graphical Simulation

- **Operation Path**

"Program Editor" page → F7 [Graphical Simulation].

- **Description**

1. This button allows you to preview the machining path for the currently edited program.
2. It has the functionality to check for program errors.
3. The default display range is the maximum range of the entire program coordinates. In other words, the graphics displayed in the graphical simulation are the largest graphics that can be displayed on the plotting frame.

1.4.6 Restart from Current Line

- **Operation Path**

"Program Editor" page → [>>] →F2 [Restart from Current Line].

- **Description**

1. This button allows you to select the starting position for restarting.
2. After this function is enabled, it will automatically switch to the machining monitoring interface and switch to automatic mode.

- **Note**

The prerequisite for using this function is that the currently edited program is the current machining program.

1.4.7 Delete Line

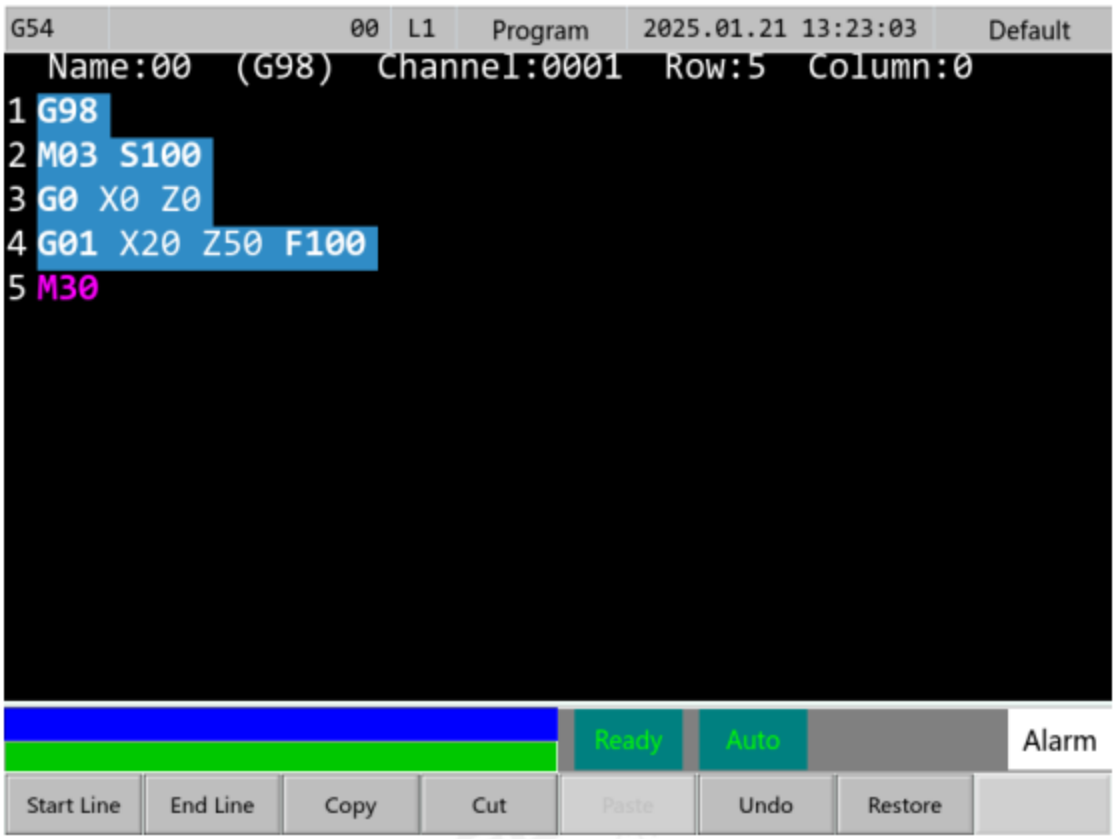
- **Operation Path**

"Program Editor" page → F2 [Delete Line].

- **Description**

Deletes the line where the cursor is located.

1.4.8 Block Copy



- **Operation Path**

"Program Editor" page → F5 [Block Copy].

- **Description**

Selects a block of program code for use with the "Copy," "Cut," and "Paste" functions.

- **Method Description**

Press the [Block Copy] button, and use the arrow keys (↑, ↓, ←, →) to move the cursor and select the desired block of code to copy or cut. The selected block will be highlighted in blue.

1.4.9 Cancel Block Copy

- **Operation Path**

[Block Copy] Button group→F2 [End line].

- **Description**

Close the block copy feature.

- **Method Description**

Press the [End line] button. At this point, block copy is disabled, and any previously selected blocks will be deselected. The [Copy] and [Cut] function buttons are inactive.

1.4.10 Copy

- **Operation Path**

[Block Copy] Button group → F1 [Start Line] →F3 [Copy].

- **Description**

Copy the selected content of the "Block Selection" for use with the "Paste" function.

1.4.11 Cut

- **Operation Path**

[Block Copy] Button group → F1 [Start Line] →F4 [Cut].

- **Description**

Cut the selected content of the "Block Copy". The cut block will be removed and can be pasted using the "Paste" function.

1.4.12 Paste

- **Operation Path**

[Block Copy] Button group →F5 [Paste].

- **Description**

Paste the content that has been "Copied" or "Cut" to the current cursor position in the program.

1.4.13 Undo

- **Operation Path**

[Block Copy] Button group →F6 [Undo].

- **Description**

Undo the previous operation step. You can undo multiple steps and go back to the first step in program editing.

1.4.14 Restore

- **Operation Path**

[Block Copy] Button group →F7 [Restore].

- **Description**

Restore the previously undone operation. You can restore multiple steps and go forward to the last step in program editing.

1.4.15 Teach

- **Operation Path**

"Program Editor" page →F6 [Teach].

- **Description**

1. The "Teach" function allows you to input the current machine coordinate values into the program being edited using "MPG," "Manual," or "Jog" to move the machine to the desired position before using the function.
2. It simplifies the programming process by allowing you to teach the program based on the machine's movement without manual input.
3. The "Teach" function includes "Rapid Positioning Teach," "Linear Cutting Teach," "Arc Cutting Teach," "Cancel Arc Midpoint," and "Point Coordinate Teach."
4. The "Rapid Positioning Teach" function generates a rapid positioning G-code to the current program coordinate, which can be adjusted based on the actual machining needs.
5. The "Linear Cutting Teach" function generates a linear cutting G-code to the current program coordinate, which can be adjusted based on the actual machining needs.
6. The "Arc Cutting Teach" function generates an arc cutting G-code. The arc starting point is the program coordinate before activating the "Arc Cutting Teach" function, the

arc midpoint is the program coordinate when the axis moves to the midpoint, and the arc endpoint is the coordinate when the axis moves to the endpoint. The "Cancel Arc Midpoint" function clears the recorded midpoint coordinate.

7. The "Point Coordinate Teach" function teaches the current coordinate.

1.4.16 Code Help

- **Operation Path**

"Program Editor" page →[>>]→F3 [Open Code Help].

- **Description**

This feature is designed to assist users in writing code by providing help for less frequently used code instructions. It allows users to access code references directly on the controller, serving as a programming manual stored within the controller.

- **Operation Path**

"Program Editor" page →[>>]→F4 [Close Code Help].

- **Description**

Closes the code help page, exiting the code help mode.

1.4.17 Input method switching

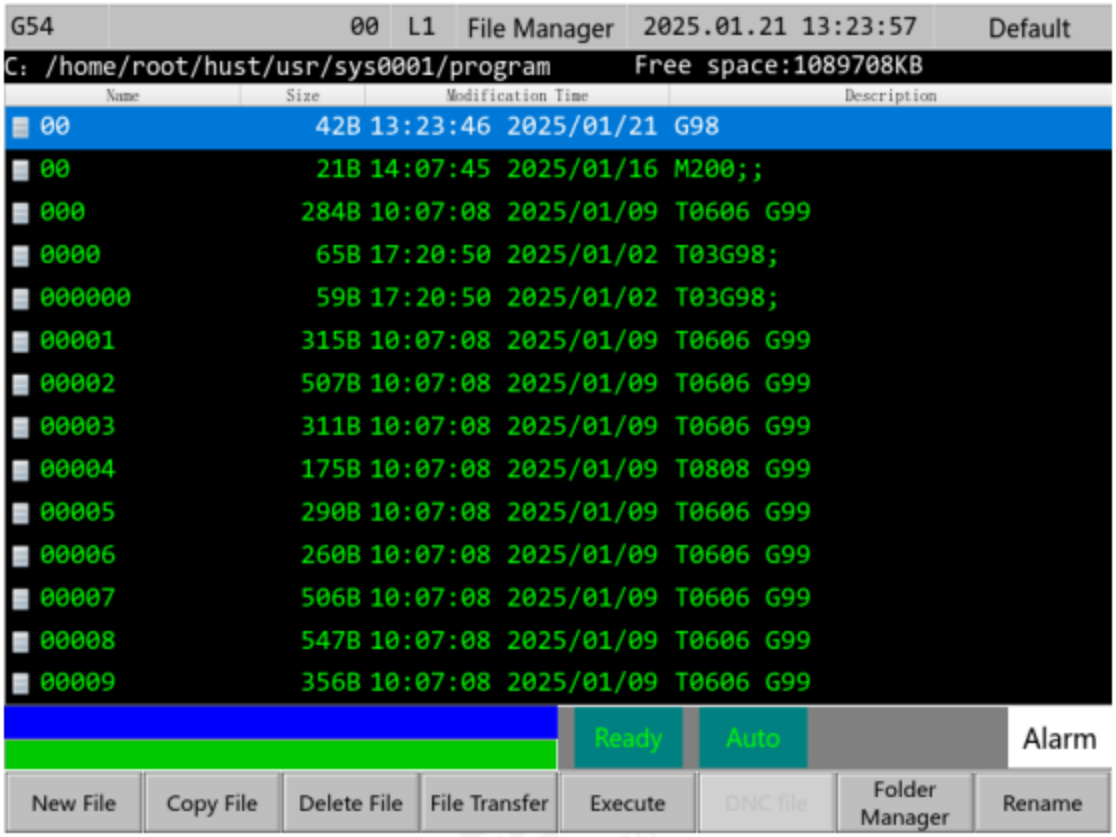
- **Operation Path**

"Program Editor" page →System panel [Switch] button→Click/Double Click/Long Press.

- **Description**

The program editing page can switch between English case input and Chinese input, making it convenient for customers to add comments and understand the program. The system defaults to uppercase letter input. Double click the【 Switch 】button to switch to lowercase letter input, and long press the 【 Switch 】 button to perform Chinese input.

1.5 Program Selection



- **Operation Path**

1. Path 1: Use the system panel shortcut key "Edit/Prog Sel" to switch to the "Program Selection" page.
2. Path 2: "Program Edit" page → F8 [File Manager] → "Program Selection" page.

- **Description**

Used for managing the program documents.

- **Operation Method**

1. Use the arrow keys (↑, ↓) to move the cursor and select the program document.
2. Use the page up (⩓) and page down (⩔) keys to scroll up and down.
3. Use the system panel "Input" key to specify the currently selected program document as the editing document and open it for editing.

1.5.1 Create New File

- **Operation Path**

"Program Selection" page → F1 [Create New File].

- **Description**

Used to create a new machining program file.

- **Operation Method**

1. Press the [New File] button, and a dialog box will appear asking for the file name of the new file.
2. Press F1 [OK] button to create the new file.

- **Note**

If no file extension is provided in the file name, the default extension is ".CNC".

1.5.2 Search

- **Operation Path**

"Program Selection" page → F7 [Folder Manager]→F7 [Search].

- **Description**

Used to search for file documents by file name.

- **Operation Method**

1. Press the [Search] button to open a dialog box and enter the file name to be searched.
2. Press F1 [Search] button to search for file documents in the file manager that match the entered file name.

1.5.3 Rename

- **Operation Path**

"Program Selection" page → F8 [Rename].

- **Description**

Used to rename machining program files.

- **Operation Method**

1. Use the up and down arrow keys to move the cursor and select the program

document to be renamed. Press F8 to open the rename dialog box and enter the new file name.

2. Press F1 [Confirm] to complete the renaming process and close the dialog box.
3. Press F2 [Cancel] to exit the renaming process and close the dialog box.

1.5.4 Copy File

- **Operation Path**

"Program Selection" page → F2 [Copy File].

- **Description**

This button is used to copy the currently selected program document at the cursor.

- **Operation Method**

Use the arrow keys [↑][↓] to move the cursor and select the program document that you want to copy，Press F2 [Copy File] button to bring up the naming dialog box for the copied file, and then press the F1 [Confirm] button to successfully copy the current document.

1.5.5 Delete File

- **Operation Path**

"Program Selection" page → F3 [Delete File].

- **Description**

This button is used to delete program documents.

- **Operation Method**

1. Press F3 [Delete File] button, and a selection box will pop up on the left side. Use the arrow keys [↑] [↓] to move the cursor and select the program document that needs to be deleted
2. Press F1 [Select] button, select the file to be deleted, press F4 [Delete] to pop up a dialog box confirming the deletion of the file, and press F1 [OK] to delete the file. Press F2 [Cancel] to cancel the deletion.

- **Note**

If the selected program document is currently being edited or loaded for processing, it cannot be deleted and the deletion will fail.

1.5.6 Edit

- **Operation Path**

"Program Selection" page → use the "Input" button on the system panel to specify the program document pointed to by the current cursor as the editing file, and open it for editing.

- **Description**

This button is used for editing program documents. Please refer to section 1.4 "Program Editing" for more information.

1.5.7 Load machining

- **Operation Path**

"Program Selection" page →F5 [Execute].

- **Description**

This button is used to designate the selected program document as the machining program and switch the page to the "Machining Monitoring" page.

- **Note**

If the program is currently executing, this button is ineffective and cannot be used.

1.5.8 Open folder

- **Operation Path**

"Program Selection" page →Use the arrow keys [↑] [↓] to move the cursor and select the open folder, then press the [Input] button on the system panel.

- **Description**

This button is used to open the folder of the currently selected cursor in the "Program Selection" page.

1.5.9 Return To Upper Folder

- **Operation Path**

"Program Selection" page →F8 [Return To Upper Folder].

- **Description**

This button is used to go back to the parent directory of the current file in the "Program Selection" page.

1.5.10 Create New Folder

- **Operation Path**

"Program Selection" page →F7 [Folder Manager]→F1 [New Folder].

- **Description**

Use this button to create a new folder in the current file directory.

- **Operation Method**

1. Press F1 [New Folder], and an input dialog box will appear.
2. Enter the name of the folder, press F1 [Confirm], and the creation will be completed. The dialog box will automatically close.
3. Press F2 [Cancel] to cancel the creation. The dialog box will automatically close.

1.5.11 File Transfer

- **Operation Path**

"Program Selection" page →F4 [File Transfer].

- **Description**

This function allows for the transfer of machining program files and system files using a USB flash drive. Please refer to section 1.8.2 "File Transfer" for more information.

1.5.12 Macro Program File / Machining Program File

- **Operation Path**

"Program Selection" page →F7 [Folder Manager]→F6 [Macro file]

- **Description**

1. Clicking on F6 [Macro Program Files] will enter the macro program files, and the F6 key will change to [Machining Program Files].
2. Clicking on the [Macro Program Files] button allows you to view the macro program files (with the extension .MACRO) stored in the system memory. The file format for creating new files on this page remains as CNC files (with the .CNC extension).
3. Clicking on the [Machining Program Files] button allows you to view the machining program files (with the .CNC extension) stored in the system memory, and the F6 key will change back to [Macro Program Files].

1.6 Tool Compensation/Setting

- **Operation Path**

1. Path 1: Use the system panel shortcut key "Tool Compensation/Setting" to switch to the "Wear Compensation" page.

2. Path 2: "Machine Position" page → F3 "Offset/Setting" → "Wear Compensation" page.

- **Description**

1. When entering this page, the default display is the "Wear Compensation" page.

- **Operation Method**

1. Use the arrow keys (↑, ↓, ←, →) to move the cursor.
2. Use the Page Up and Page Down keys to scroll up and down.
3. When inputting values in the tool compensation table, directly entering a number represents setting length compensation/wear compensation value; entering the corresponding axis name followed by a number represents tool compensation operation.

1.6.1 Tool Setting

- **Description**

Tool Setting is used to configure the tool tip size, where the actual tool tip size is calculated as the sum of the tool tip radius and tool wear.

- **Parameter Description**

1. Tool Tip Direction: Sets the direction of the tool tip during machining.
2. Tool Tip Radius: Specifies the tool tip radius compensation for G41/G42.
3. Tool Wear: Adjusts the tool tip radius for smaller sizes in G41/G42.
4. Tool Tip Direction: There are eight available tool tip directions provided by the software. Select the appropriate direction based on the machining requirements. Please refer to the "G41/G42 Programming Manual" for more information.

1.6.2 Wear Compensation

- **Operation Path**

1. Path 1: Use the system panel shortcut key [Tool Compensation/Setting] to switch to the "Wear Compensation" page.
2. Path 2: "Machine Position" page → F3 [Offset/Setting] → [Wear Compensation] page.

- **Description**

1. Set the wear compensation value for the tool.
2. Tool compensation = Tool Length + Tool Wear.

- **Parameter Description**

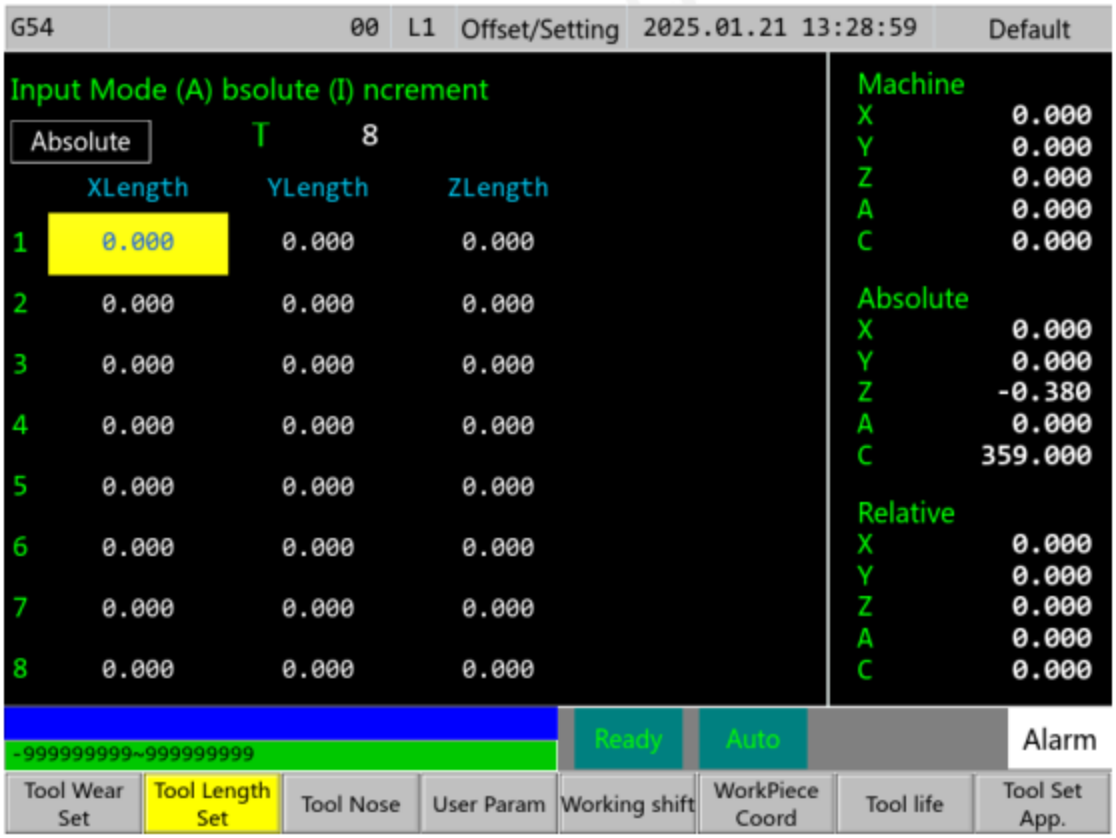
Set the maximum/minimum value for wear compensation.

- **Note**

1. When setting the tool length compensation, the wear compensation value for the corresponding axis will be automatically set to zero.
2. If the wear compensation value is modified during the machining process, if it has not

reached the corresponding T-code, the modified value will take effect in real-time. If it has already reached the corresponding T-code, the modified value will take effect in the next execution of that T-code in the program.

1.6.3 Length Compensation



- **Operation Path**

1. Path 1: Use the system panel shortcut key [Tool Compensation/Setting] to switch to the [Length Compensation] page.
2. Path 2: "Machine Position" page → F3 [Offset/Setting] → F2 [Length Compensation] page.

- **Description**

1. Set the tool length compensation value.
2. Tool compensation = Tool length + Tool wear.

- **Note**

1. When setting the tool length compensation, the corresponding axis's wear compensation value will be automatically cleared.

2. The length compensation value cannot be modified during machining.

1.6.6 Absolute Input

- **Operation Path**

"Wear Compensation" or "Length Compensation" page →Enter character "A" in the data input box at the cursor and then press the [Input] button on the system panel.

- **Description**

This button can be used to set the input method for the table input area. The input character A represents absolute value input.

1.6.7 Incremental Input

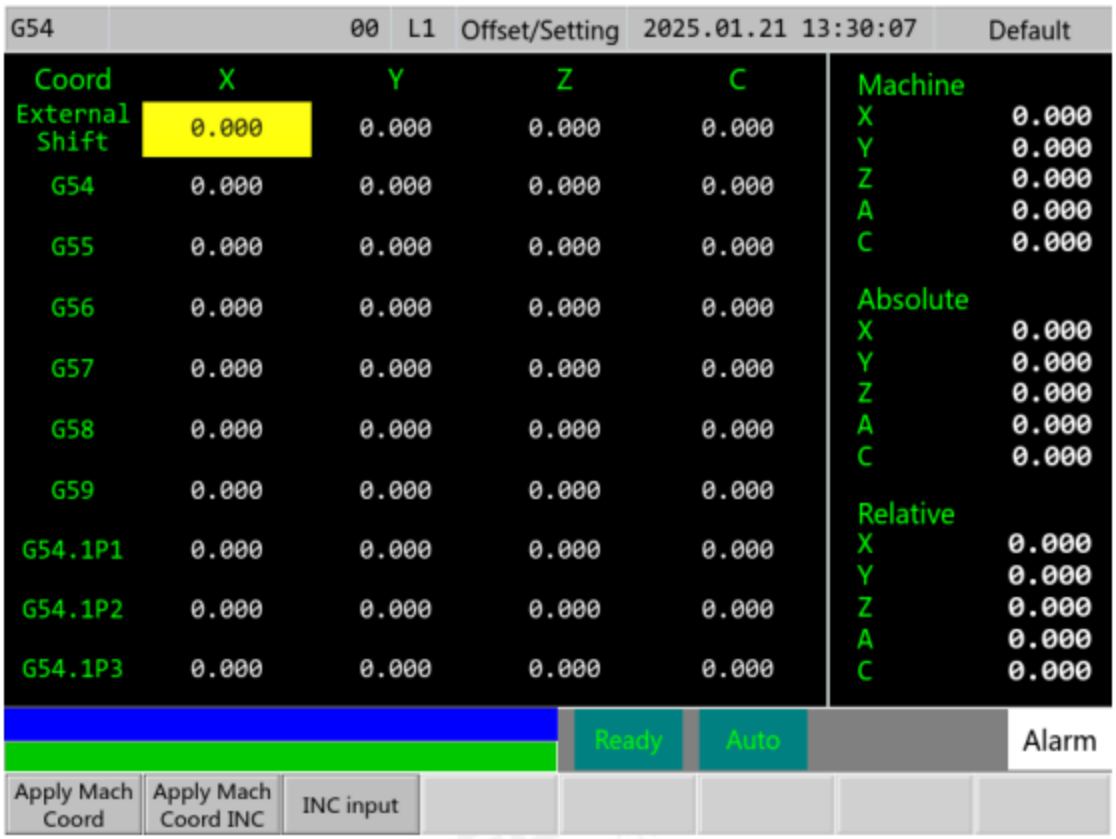
- **Operation Path**

"Wear Compensation" or "Length Compensation" page →Enter the character "I" in the data input box at the cursor, and then press the [Input] button on the system panel.

- **Description**

This button can be used to set the input method for the table input area， The input character "I" represents an incremental input value, indicating an increase in the input value based on the original one.

1.6.8 Workpiece Coordinate



- **Operation Path**

"Wear Compensation" or "Length Compensation" page →F6 [Workpiece Coordinate].

- **Description**

1. This button allows you to switch to the "Workpiece Coordinate System" page for setting the workpiece coordinate system.
2. If G54.1P1-G54.1P48/G54-G59 is not set in the program, the default coordinate system is G54.
3. External workpiece coordinate system applies to all G54.1P1-G54.1P48/G54-G59 coordinate systems.

- **Operation Method**

1. Use the arrow keys [↑] [↓] [←] [→] to move the cursor.
2. Use the [Page Up] and [Page Down] keys to scroll up and down.
3. You can directly input values or input axis names followed by values to perform workpiece coordinate system alignment.

- **Note**

After setting the workpiece coordinate system, it is necessary to re-align the tool unless it is for overall offsetting.

1.6.8.1 Apply Mechanical Coordinate Teaching

- **Operation Path**

"Work Coordinate System" page →F1 [Apply Mechanical Coordinate Teaching].

- **Description**

Press the mechanical coordinate teaching button, and the system will automatically input the current mechanical coordinates into the cursor position.

1.6.8.2 Apply Mechanical incremental teaching

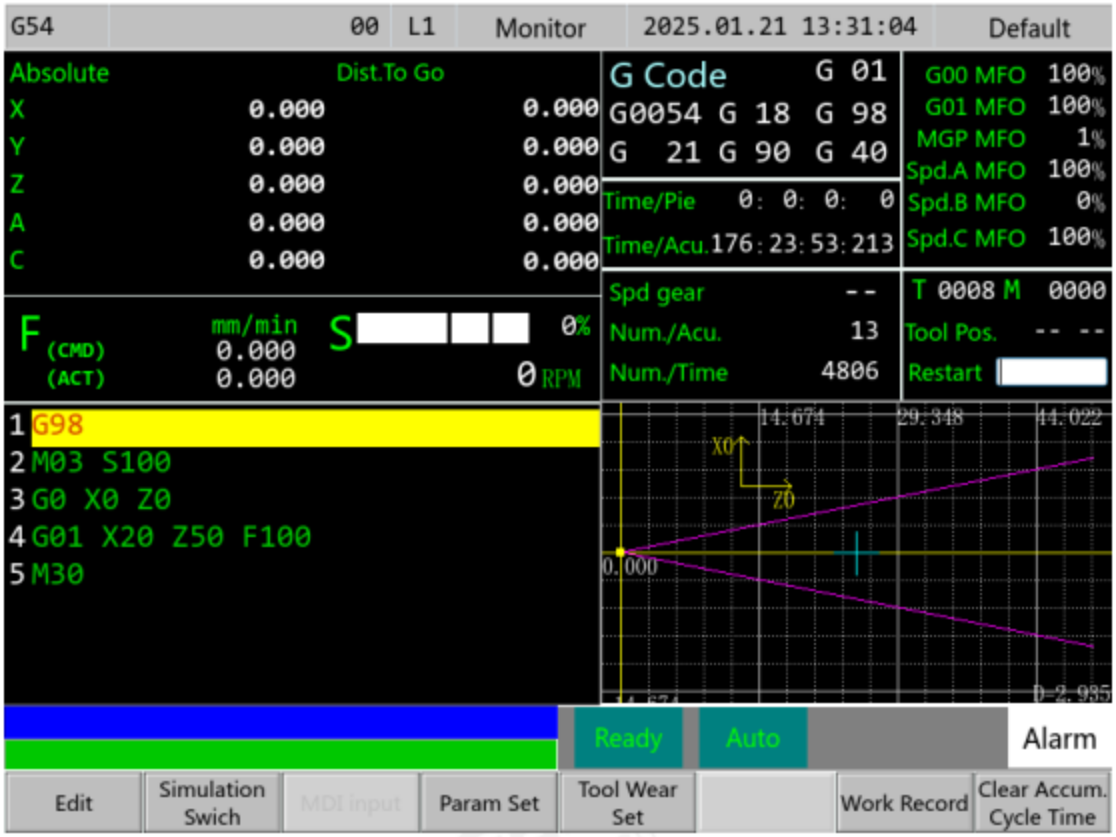
- **Operation Path**

"Work Coordinate System" page →F2 [Apply Mechanical incremental teaching].

- **Description**

Move the cursor up and down to select the desired axis, enter a value in the input box, and press F2 [Apply Mechanical Incremental Teaching]. At this point, set the program coordinates corresponding to the axis as the input value.

1.7 Machining Monitoring



- **Operation Path**

1. Path 1: Use the system panel shortcut key "Monitoring" to switch to the "Process Monitoring" page.
2. Path 2: "Machine Position" page → F4 "Process Monitoring" → "Process Monitoring" page.

- **Description**

This page provides essential information for monitoring during the machining process.

1.7.1 Screen Description

1.7.1.1 Machine Control Area

The Machine Control Area displays the current machine information

1. Program Coordinates
2. Remaining Distance
3. Feed Rate

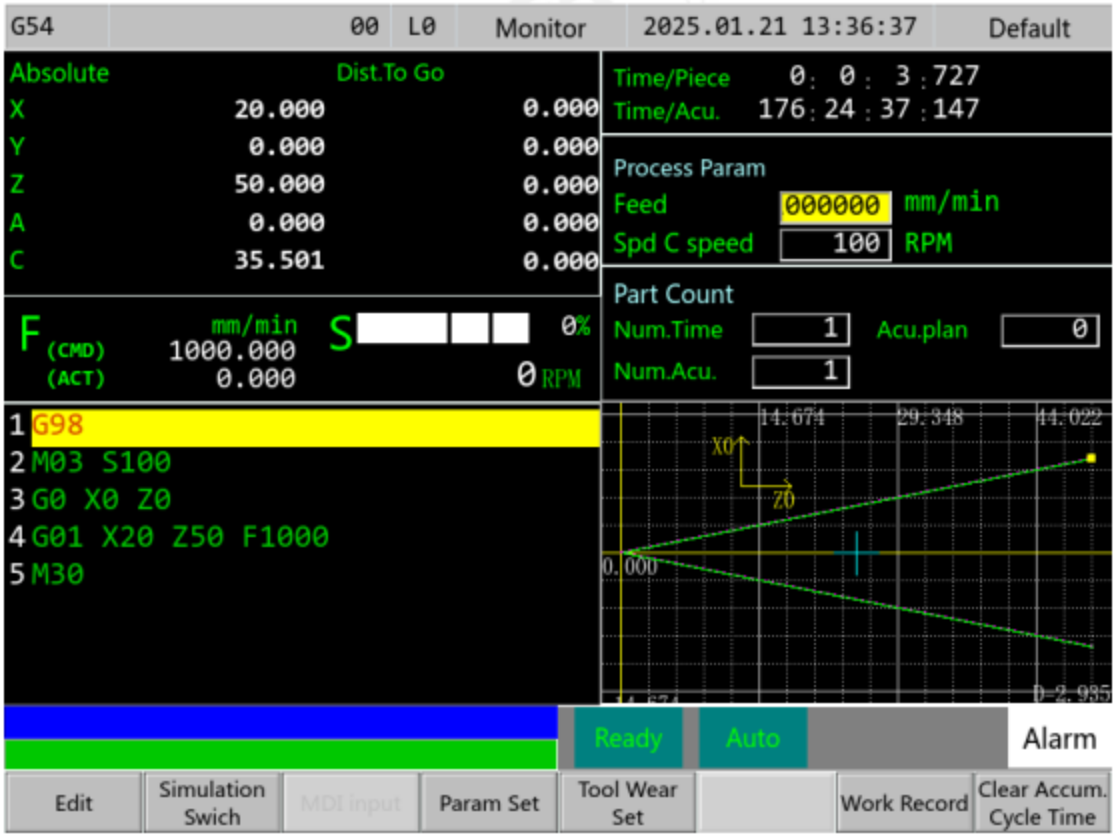
4. Spindle Speed

1.7.1.2 Program Code Monitoring Area

The program code monitoring area displays the current program being executed.

The yellow cursor indicates the currently executing line of the program.

1.7.1.3 Processing Information Display Area



- **Description**

1. This area overlaps with the "Processing Information Setting" area.
2. Pressing the F4 key [Processing Information Setting] allows for display switching.

- **Screen Description**

1. G-Code Status:
   - ❖ Displays the currently active G-code being executed by the system.
2. Current Processing Time:
   - ❖ Displays the elapsed processing time for the current workpiece.
   - ❖ When the program restarts execution, the time is recalculated.
3. Accumulated Working Time:

❖ Represents the total duration of processing since the first execution of the program until the current moment.

❖ The "Clear Accumulated Time" button can be used to reset the accumulated processing time.

4. Magnification:

❖ G00 MFO

❖ G01 MFO

❖ MPG MFO

❖ SPN MFO

5. Accumulated Completion:

❖ Indicates the total number of workpieces processed by the machine.

❖ The system does not automatically reset this count. It can be manually reset by entering zero in the input field provided in the "Processing Information Setting" for the "Accumulated Completion" item.

6. Current Completion:

❖ Displays the number of workpieces completed in the current processing.

❖ It is reset to zero when the processing program is executed for the first time.

7. Tool Information:

❖ T (4-digit display)

❖ The higher two digits indicate the current executing tool number.

❖ The lower two digits indicate the current tool compensation number.

❖ The tool turret displays the current tool number.

8. M-Code:

❖ Displays the currently executing M-code.

9. Restarting from a Specific Line:

❖ Allows setting the starting line for restarting the program.

❖ N: Specifies the starting line as the N-th line.

❖ N=0 means starting from the first line.

❖ If N exceeds the maximum number of lines, the system will issue an alarm.

1.7.1.4 Work Setting Display Area

- **Description**

1. This area overlaps with the "Processing Information Area".
2. Use the "Processing Information Setting" button to switch the display.

- **Screen Description**

1. Displays processing time and accumulated time.
2. Feedrate Setting (Note: Currently for display only, not editable)
   - ❖ Set the processing feedrate.
3. Spindle Speed Setting
   - ❖ Set the spindle speed.
4. Accumulated Completed
   - ❖ Set the total number of workpieces processed by the machine.
   - ❖ The system does not automatically reset, manual reset is required.
5. Current Completed
   - ❖ Set the number of workpieces for the current processing.
   - ❖ When changing the processing program, the workpiece count is reset.
   - ❖ Requires M-code to execute increment by 1. When the workpiece count reaches the desired count, it triggers a pause. (M15 is used to increment the count, M16 is used to reset the count).
6. Planned for Current Run
   - ❖ Set the upper limit of the number of workpieces to be processed.
   - ❖ When the workpiece count reaches the desired count, a dialog box will pop up to indicate that the desired count has been reached, and the processing will be paused.

1.7.1.5 Graphical Simulation Display Area

- ❖ **Description**

1. This area shows the actual tool path of the current machining program.
2. You can toggle the display of the graphical simulation by pressing the F2 key.

1.7.2 Load Program Editing

- **Operation Path**

"Processing Monitoring" page → F1 [Load Program Editing].

- **Description**

Load the currently running program into the code editor and switch the screen to the "Program Editing" page.

- **Note**

If you press this button while a program is executing, the screen will switch to the "Program Editing" page, but editing of the program will not be possible.

1.7.3 Graphic Simulation Display

- **Operation Path**

"Processing Monitoring" page → F2 [Graphic Simulation Display].

- **Description**

1. This function is used to toggle the display of the graphic simulation.
2. The graphic adjustment can only be performed when the graphic simulation display is active.
3. For detailed instructions, please refer to section 1.7.8 "Graphic Adjustment".

1.7.4 MDI Input

- **Operation Path**

"Processing Monitoring" page → F3 [MDI Input].

- **Description**

This feature allows you to edit MDI programs for execution.

- **Operation Method**

1. Switch the mode to "MDI" mode.
2. Press the "MDI Input" button, which will prompt an editing dialog box to appear.
3. In the editing dialog box, edit the program as needed. Press the "Confirm" button to

automatically load the edited program into the machining storage area.

4. Press the "Start" button to execute the MDI program for machining.

- **Note**

This button is only functional in "MDI" mode. You can switch to MDI mode by using the "MDI" button on the auxiliary panel.

1.7.5 Processing Information Settings

- **Operation Path**

"Monitoring" page → F4 [Processing Information Settings].

- **Description**

This function allows you to toggle between displaying "Processing Information" and "Processing Settings".

1.7.6 Program Restart

- **Operation Path**

1. "Processing Monitor" page → the system defaults the cursor to the "Processing Information" area and enters the number of lines in the "Restart" input box.
2. "Program Editing" page → [>>] → F2 [Resume from Current Line].

- **Description**

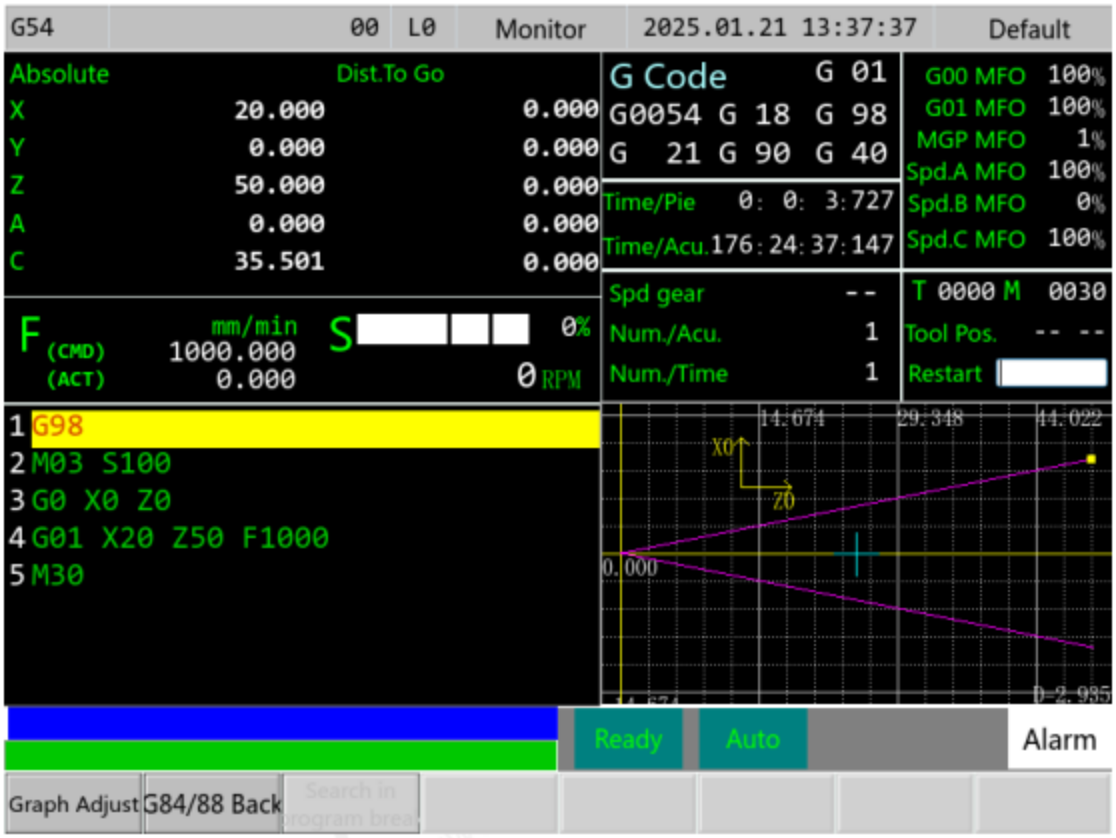
Used to specify a specific block in the program as the resume block, allowing execution to start from that block during processing.

- **Operation Method**

1. Switch to "Auto Execution" mode. In the "Processing Information" area, enter the block number for resume and press the [Start] button to start execution from the specified block.
2. Switch to "Auto Execution" mode. In the editing page, move the cursor to the desired resume position, press [>>]→F2 [Resume from Current Line], and the page will automatically switch to the monitor page. Press the [Start] button to start execution

from the specified block.

1.7.7 Graph Adjustment



- **Operation Path**

"Processing Monitoring" page → [>>]→F1 [Graph Adjustment].

- **Description**

Allows for appropriate adjustments to the graphic simulation.

1.7.7.1 Plane Selection

- **Operation Path**

"Processing Monitoring" page →[>>]→F1 [Graph Adjustment]→F3 [Plane Selection].

- **Description**

Allows for switching between different planes for graphic simulation display.

1.7.7.2 Zoom In

- **Operation Path**

"Processing Monitoring" page →[>>]→F1 [Graph Adjustment]→F1 [Zoom In].

- **Description**

Allows for zooming in on the trajectory of the graphic simulation.

1.7.7.3 Graph Reset

- **Operation Path**

"Processing Monitoring" page →[>>]→F1 [Graph Adjustment]→F2 [Graph Reset].

- **Description**

Allow image simulation to be restored to its initial state

1.7.7.4 Viewport Adjustment

- **Operation Path**

"Processing Monitoring" page →[>>]→F1 [Graph Adjustment]→F1 [Zoom In]→arrow key [↑] [↓] [←] [→].

- **Description**

This function allows for moving the viewport of the graphic simulation.

1.7.7.5 Overall Drawing Settings

- **Operation Path**

"Processing Monitoring" page →[>>]→F1 [Graph Adjustment]→F8 [Overall Drawing Settings].

- **Description**

This function is used to configure settings for drawing, such as color settings for drawings, layer colors, and background colors.

1.7.8 To clear accumulated time

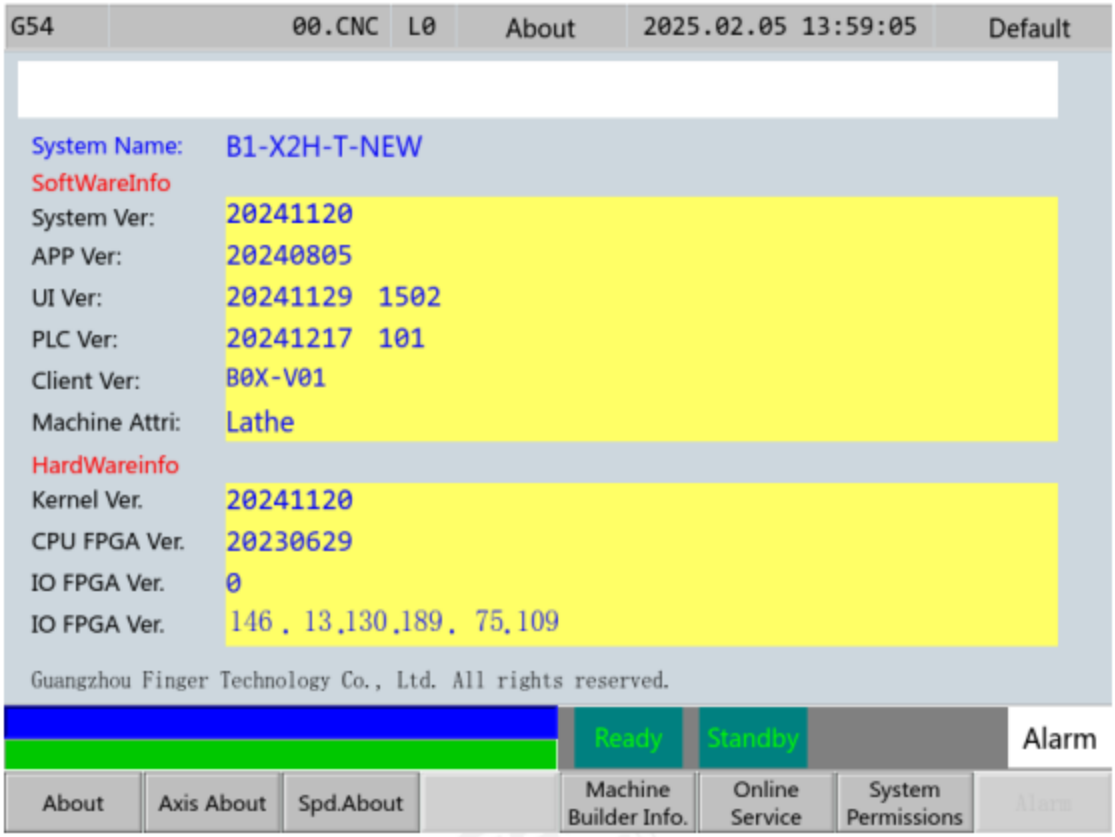
- **Operation Path**

"Processing Monitoring" page →F8 [dear accumulated time].

- **Description**

This function is used to reset the accumulated processing time to zero.

1.8 System Information/Maintenance

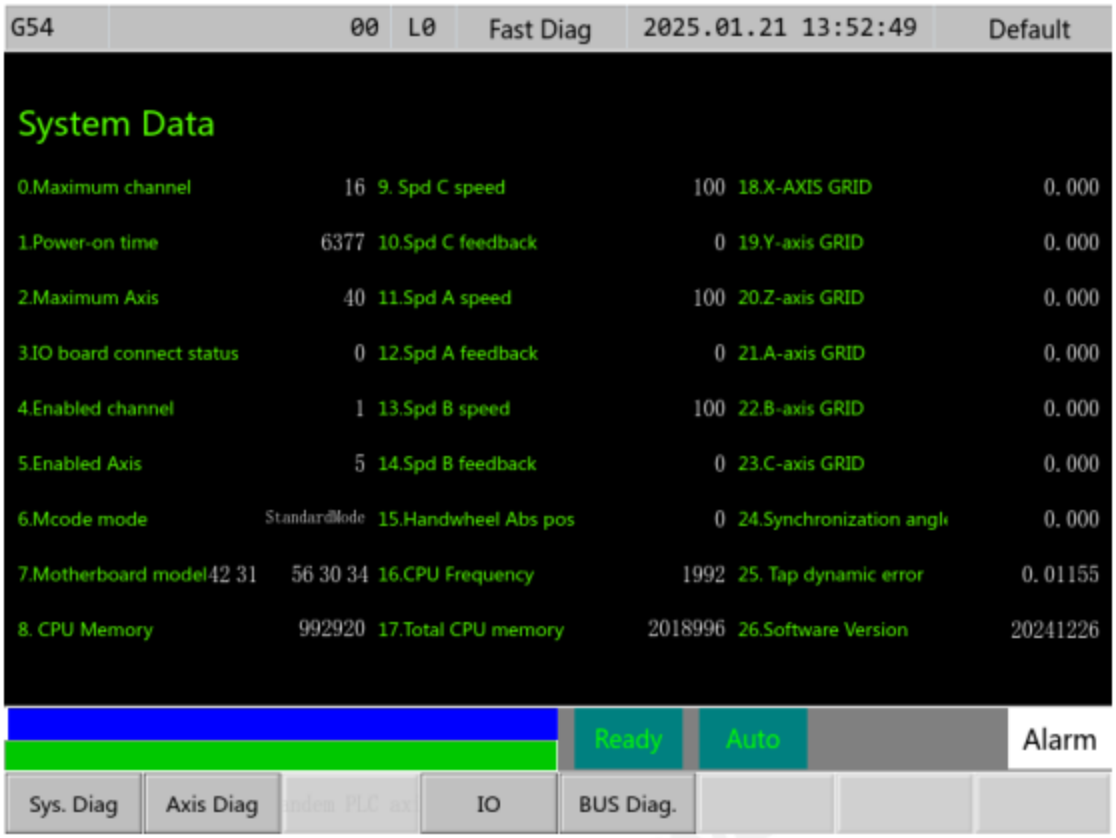


- **Operation Path**

"Position" page→F8 [Version].

- **Description**

This page can display system information.



- **Operation Path**

"Position" page→F7 [Fast Diagnosis].

- **Description**

This page can display system information

1.8.1 Axis Oscilloscope

- **Operation Path**

"Fast Diagnosis" page→F2 [Axis Diagnosis].

- **Description**

This function can monitor the command and feedback waveforms of each axis.

1.8.1.1 Channels

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channels].

- **Description**

This function allows you to adjust the position and display mode of the waveforms for the four channels of the oscilloscope.

1.8.1.1.1 PB1

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1].

- **Description**

This function allows you to adjust the position and display mode of Channel 1 waveform on the oscilloscope. The adjustment method for Channels 2, 3, and 4 is the same as Channel 1. Only Channel 1 will be explained here.

1.8.1.1.1.1 PB1Up

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F1 [PB1Up].

- **Description**

Pressing this button allows you to move the waveform of the oscilloscope upward.

1.8.1.1.1.2 PB1Down

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F2 [PB1Down]

- **Description**

Pressing this button allows you to move the waveform of the oscilloscope downward.

1.8.1.1.1.3 PB1 Zoom In

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F3 [PB1 Zoom In].

- **Description**

Pressing this button allows you to zoom in the vertical axis of the oscilloscope waveform.

1.8.1.1.1.4 PB1 Zoom Out

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F4 [PB1 Zoom Out].

- **Description**

Pressing this button allows you to zoom out the vertical axis of the oscilloscope waveform.

1.8.1.1.1.5 PB1 Standard Position

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F5 [PB1 Standard Position].

- **Description**

Pressing this button adjusts the position of the oscilloscope waveform to the standard position.

1.8.1.1.1.6 PB1 Standard Scale

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F6 [PB1 Standard Scale].

- **Description**

Pressing this button adjusts the size of the oscilloscope waveform to the standard scale.

1.8.1.1.1.7 PB1 Show/Hide

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F7 [Show/Hide].

- **Description**

Pressing this button toggles the display of the waveform for the PB1 channel on/off.

1.8.1.1.1.8 PB1 Reverse/Normal

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→F8 [Reverse/Normal].

- **Description**

Pressing this button can switch the oscilloscope waveform between reverse/non reverse.

1.8.1.1.1.9 PB1 Linear connection/Nonlinear connection

- **Operation Path**

F2 [Axis Diagnosis]→F1 [Channel]→F1 [PB1]→[>>]→F1 [Linear connection/Nonlinear connection].

- **Description**

Pressing this button can switch the oscilloscope waveform between linear/nonlinear.

1.8.1.2 Time

- **Operation Path**

F2 [Axis Diagnosis]→F2 [Time].

- **Description**

This function is used to adjust the time period of the waveform displayed in the current window of the oscilloscope, achieving horizontal waveform adjustment.

1.8.1.2.1 Horizontal Zoom

- **Operation Path**

F2 [Axis Diagnosis]→F2 [Time]→F1 [Horizontal Zoom].

- **Description**

This button adjusts the horizontal zoom of the oscilloscope waveform.

1.8.1.2.2 Horizontal Shrink

- **Operation Path**

F2 [Axis Diagnosis]→F2 [Time]→F2 [Horizontal Shrink].

- **Description**

This button adjusts the horizontal shrink of the oscilloscope waveform.

1.8.1.2.3 Standard Scale

- **Operation Path**

F2 [Axis Diagnosis]→F2 [Time]→F3 [Standard Scale].

- **Description**

This button adjusts the horizontal size of the oscilloscope waveform to the standard scale.

1.8.1.3 Overall Settings

- **Operation Path**

F2 [Axis Diagnosis]→F3 [Overall Settings].

- **Description**

This function is used to set the controller, axis, and pulse type for the four channels of the oscilloscope. The settings for all four channels are the same. Here, we will only describe

the settings for channel 1 of the oscilloscope.

- **Operation Method**

Press the [Overall Settings] button to automatically display the settings table.

1. In the PB1 row, enter 1 for the channel, X for the axis, and Command for the type.
2. (Axis options include: X, Y, Z, A, B, C, X1, Y1, Z1, A1, B1, C1, ..., X5, Y5, Z5, A5, B5, C5)
3. (Type options include: Command, Feedback, Voltage)
4. Press the [Apply] button to set the oscilloscope channel.
5. In this way, the waveform for channel 1 of the oscilloscope is set to the command pulses of system channel 1 in the X axis.
6. You can also set the display of the grid, probe, horizontal layout, vertical layout, waveform inversion, waveform linearity, waveform scale, zero position, and color.

1.8.1.4 Clear

- **Operation Path**

F2 [Axis Diagnosis]→F4 [Clear].

- **Description**

This button is used to erase all waveforms in the oscilloscope window.

1.8.1.5 Pause Oscilloscope

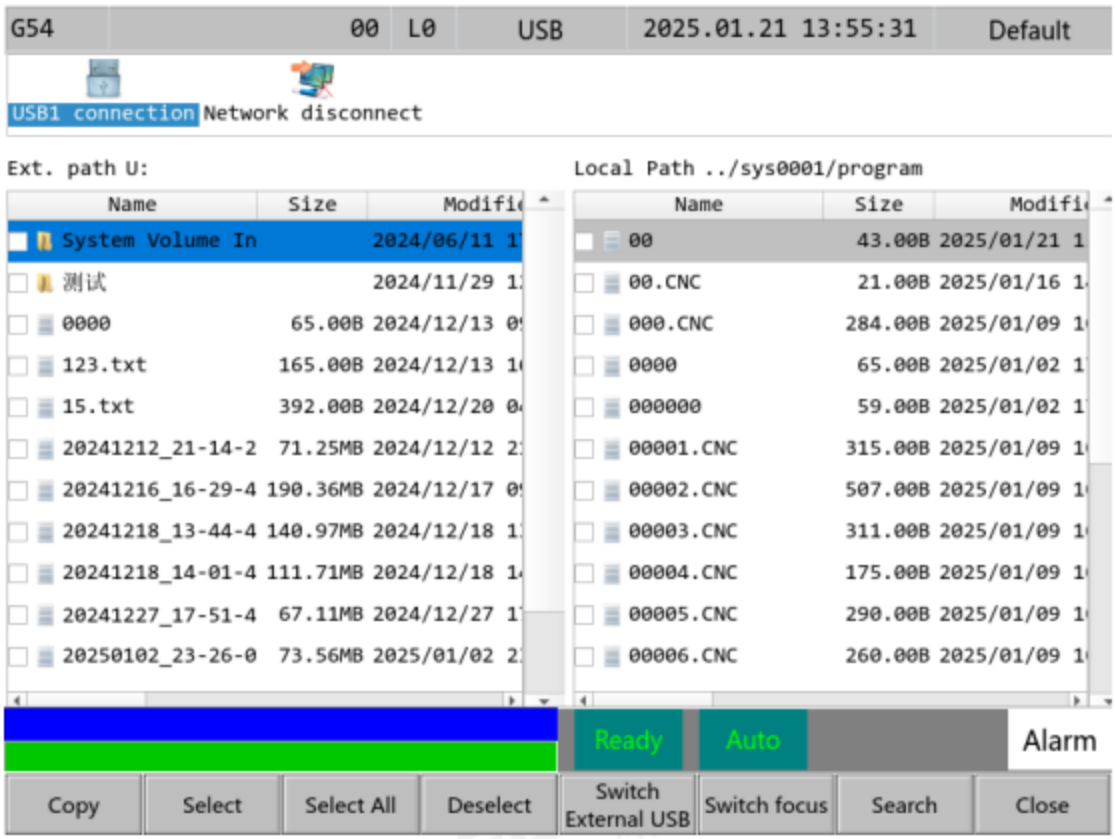
- **Operation Path**

F2 [Axis Diagnosis]→F5 [Pause Oscilloscope].

- **Description**

Pressing this button stops the waveform from refreshing, allowing you to view and analyze the current waveform.

1.8.2 File Transfer



- **Operation Path**

"Program Selection" page→F4 [File Transfer].

- **Description**

This function is used for file sharing, file input/output between the controller and external devices such as USB drives and computers.

1.8.2.1 USB/CNC switch

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→F6 [Switch Focus].

- **Description**

This is used to switch the focus between the controller's file selector and the external device's file selector. The cursor can only be moved when the focus is on the corresponding file selector.

1.8.2.2 Open Folder

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→ [>>]→F7 [Open Folder].

- **Description**

This is used to open the currently selected folder by the cursor.

1.8.2.3 Return To Upper Layer

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→ [>>]→F8 [Return To Upper Layer].

- **Description**

Navigate to the parent directory of the current file.

1.8.2.4 Search

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→F7 [Search].

- **Description**

This function is used to search for files.

- **Operation Method**

1. Press the [Search] button to open the input dialog box.
2. Enter the filename of the file you want to search for.
3. Press F1 [Confirm] to perform the search.
4. Press F2 [Cancel] to abort the search.

1.8.2.5 Select

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→F2 [Select].

- **Description**

Selects the file currently highlighted by the cursor，Copy operations can only be performed on selected files.

1.8.2.6 Delete

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→ [>>]→F6 [Delete].

- **Description**

Delete the currently selected file.

1.8.2.7 Rename

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→[>>]→F5 [Rename].

- **Description**

Renames the currently selected file.

1.8.2.8 Create New Folder

- **Operation Path**

"Program Selection" page→F4 [File Transfer]→[>>]→F1 [Create New Folder].

- **Description**

Creates a new folder in the current file directory.

1.8.2.9 External Device Switch

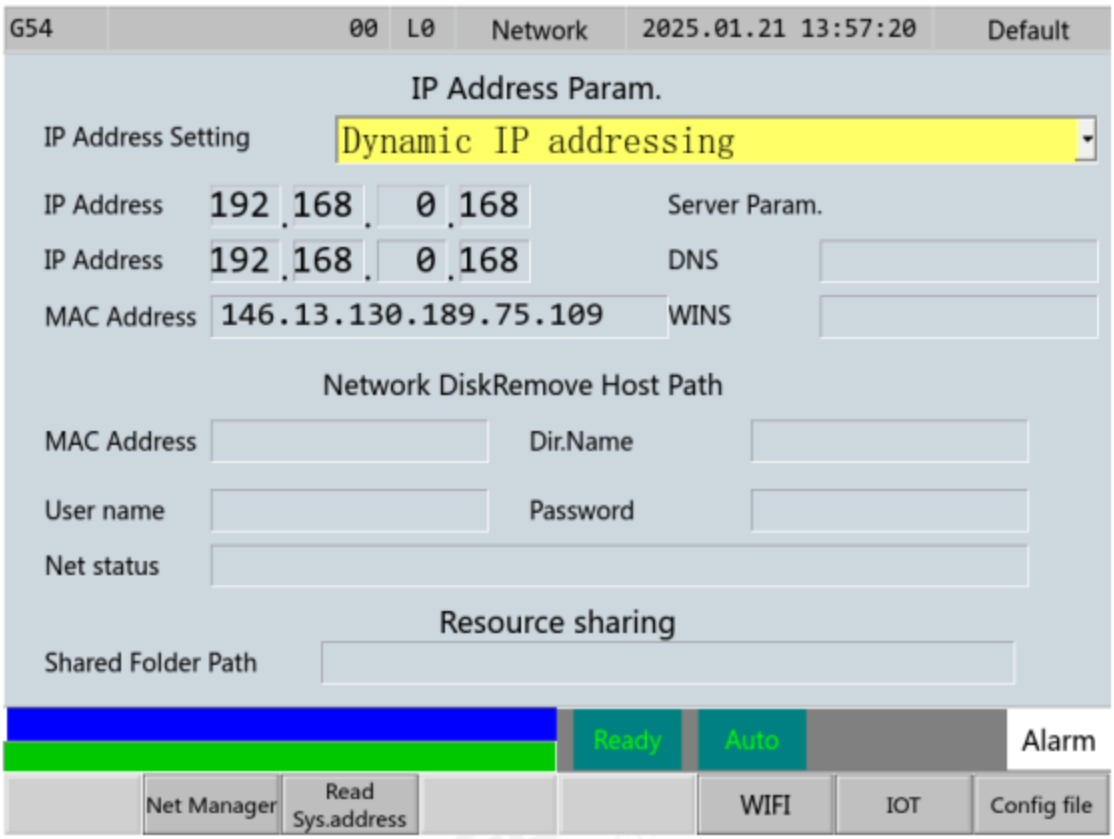
- **Operation Path**

"Program Selection" page→F4 [File Transfer]→F5 [External Device Switch].

- **Description**

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

1.8.3 Network Settings



- **Operation Path**

"Position" page→F5 [Maintenance]→F2 [Network].

- **Description**

This function is used for configuring controller IP settings, physical address monitoring, time settings, etc.

1.8.3.2 Network Management

- **Operation Path**

"Position" page→F5 [Maintenance]→F2 [Network]→F2 [Network Management].

- **Description**

This function is used for reading and configuring the controller's IP address.

1.8.3.2.1 Read IP

- **Operation Path**

"Position" page→F5 [Maintenance]→F2 [Network]→F2 [Network Management].

→F1 [Read IP].

- **Description**

This function is used to read the controller's IP address.

1.8.3.2.2 Set IP

- **Operation Path**

"Position" page→F5 [Maintenance]→F2 [Network]→F2 [Network Management]

→F2 [Set IP].

- **Description**

This function is used to set the controller's IP address.

- **Operation Method**

1. To set the IP address, enter the corresponding IP address in the IP Address input field on the page.
2. Press [Set IP] to set the controller's IP address to the value entered in the input field.

1.8.3.3 Read MAC Address

- **Operation Path**

"Position" page→F5 [Maintenance]→F2 [Network]→F3 [Read MAC Address].

- **Description**

This function is used to read the controller's network card MAC address.

1.8.3.4 Set Time

- **Operation Path**

"Position" page→F5 [Maintenance]→F5 [Sys. Settings]→F6 [Time Setting].

- **Description**

This function is used to set the system interface display time.

1.8.4 Operation record

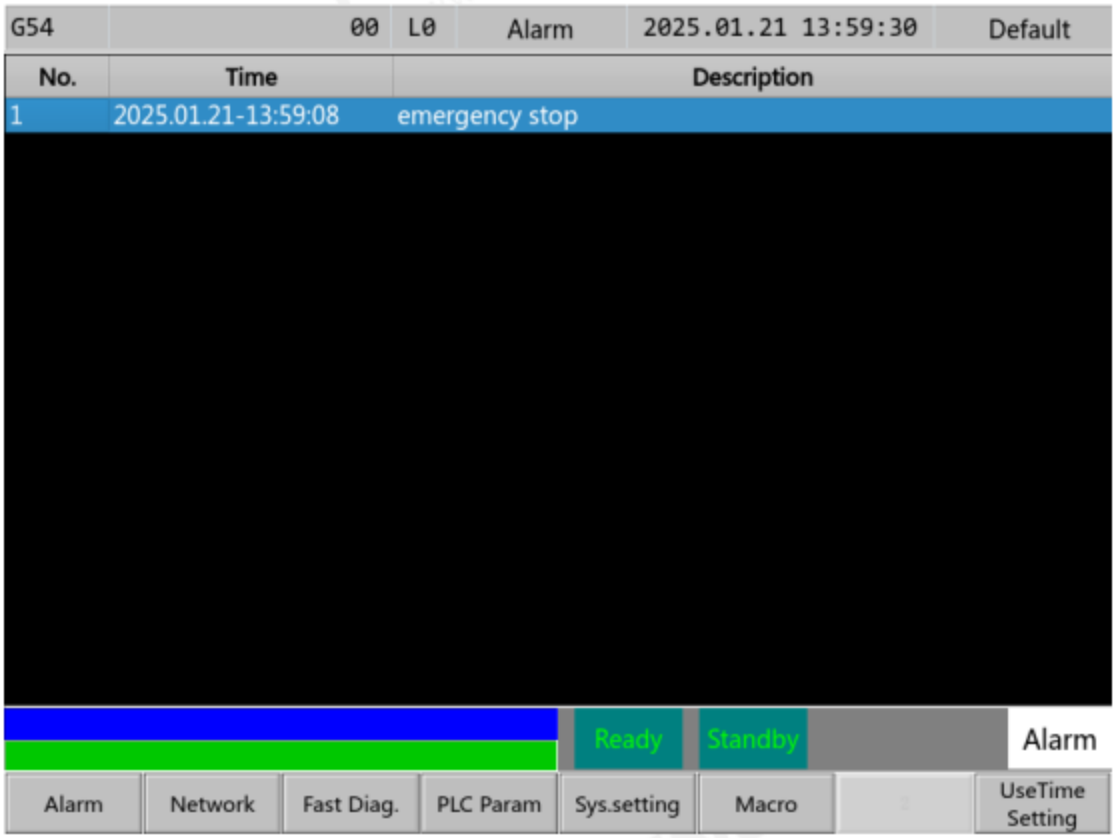
- **Operation Path**

"Position" page→[>>]→F2[Diagnosis]→F4[Operation record].

- **Description**

This function is used to record the user's operational steps. It can be used to trace improper operations that may have caused issues, facilitating troubleshooting and reproducing on-site problems.

1.8.5 Alarm



- **Operation Path**

1.Use the system panel shortcut key [Help/Alarm] to switch the page to the "Alarm Monitoring" page.

2."Position" page→F5 [Maintenance]→F1 [Alarm].

- **Description**

This page is used to view the current alarms and alarm history of the controller.

1.8.5.1 Current Alarms

- **Operation Path**

1. "Alarm Monitoring" page→press F1 [Pending Alarm].
2. Use the up and down arrow keys to move the cursor.

- **Description**

This function is used to view the currently occurring alarms in the controller.

1.8.5.2 Alarm History

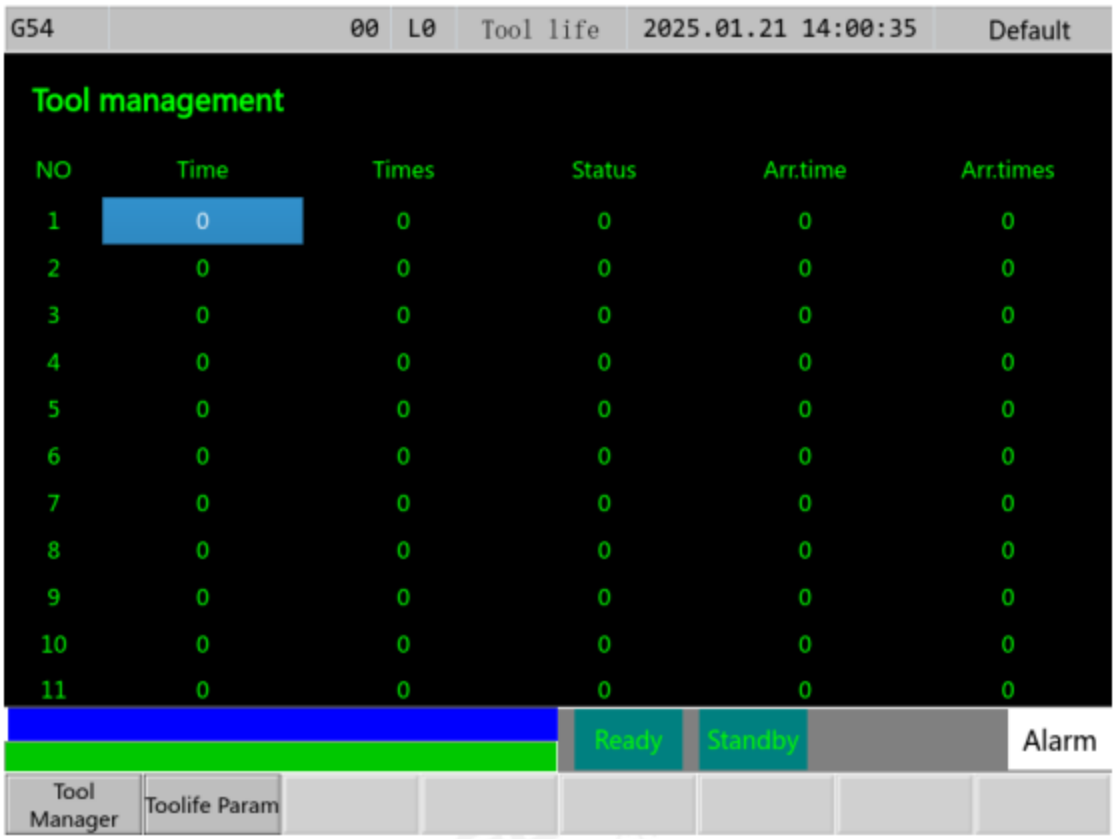
- **Operation Path**

"Alarm Monitoring" page→F2 [History Alarm].

- **Description**

1. This function is used to view the recent alarms that have occurred in the controller. The alarms are arranged in chronological order from top to bottom, with the older alarms displayed towards the top.
2. Use the up and down arrow keys to move the cursor.

1.8.6 Tool Life Management



- **Operation Path**

1.Use the system panel shortcut key [Offset/Setting]→F7 [Tool life].

2."Position" page→F3 [Offset/Setting]→F7 [Tool life].

- **Description**

This function is used for tool management, including tool management, group management, tool parameter settings, etc.

1.8.6.1 Tool Management

- **Operation Path**

"Tool Life" page→F1 [Tool Management].

- **Description**

1. Used for tool management, including the current usage time, current usage count, life management status, arrival time setting, arrival count setting, wear compensation code setting, length compensation code setting, tool number setting, etc.

2. Use the up, down, left, and right arrow keys to move the cursor.
3. Use [⩓] and [⩔] to scroll up and down.

1.8.6.2 Tool Life Parameter

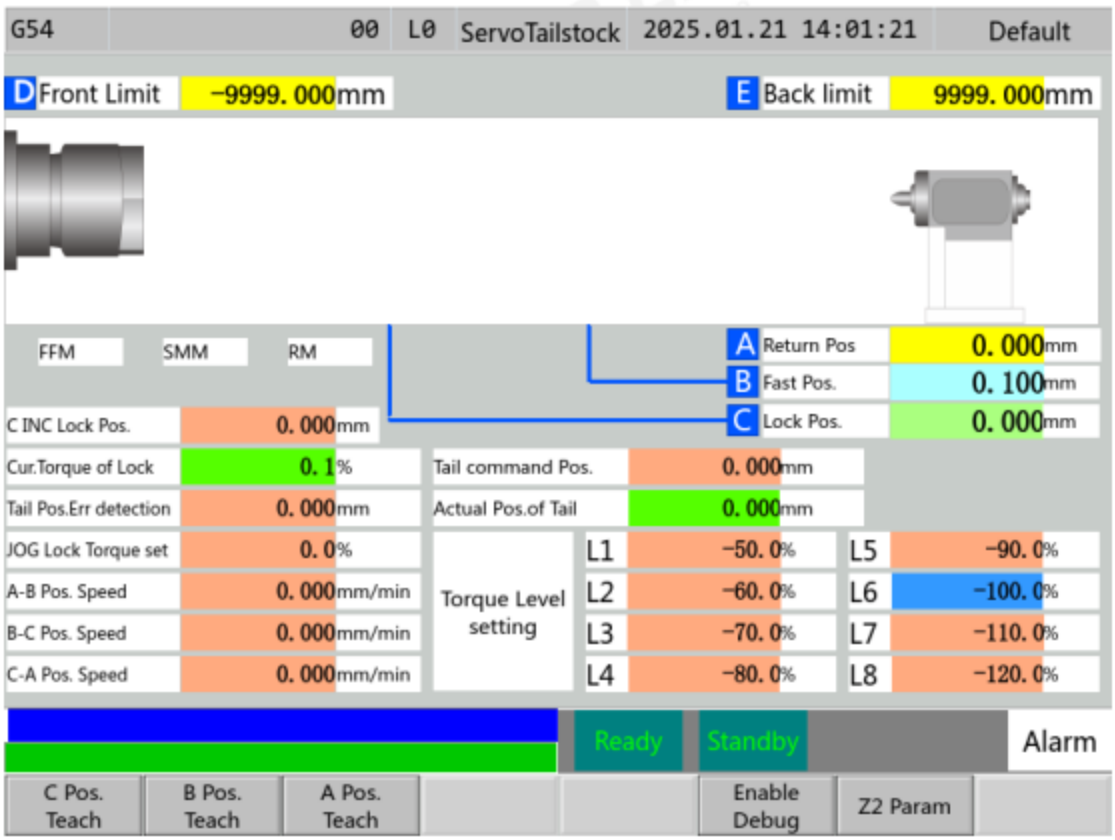
- **Operation Path**

"Tool Life" page→F2 [Tool Life Parameter].

- **Description**

1、Set tool parameters. Including whether to enable tool life management, tool life counting method, and setting the effective number of tool parameter groups.

1.8.7 Servo Tailstock



- **Operation Path**

"Position" page →[>>]→F5 [Extend]→F3 [Servo Tailstock].

- **Description**

1. Regular Tailstock: The tailstock cylinder is actuated by system outputting O point, and the tailstock screw moves forward or backward.
2. Servo Tailstock: Axis-controlled tailstock, the tailstock motor Z2 axis controls the forward or backward movement of the tailstock, allowing segmented speed control of the tailstock, real-time monitoring of the tailstock position coordinates, limiting the tightening torque of the workpiece, and automatic alarm by the system when the torque exceeds, effectively preventing workpiece damage caused by improper tightening.

- **Operation Method**

1. Point A: Return to the final position, stop when reaching this position during the tailstock retracting motion.
2. Point B: Fast move forward position, the boundary between the fast feed state and the slow feed state.
3. Point C: Tightening position, reaching this position indicates the workpiece is tightened.

1.8.8 Custom Jump

- **Operation Path**

"Position" page →F5 [Maintenance]→F6 [MACRO].

- **Description**

This function is used to set up the configuration table for macro jumps, for use by developers.

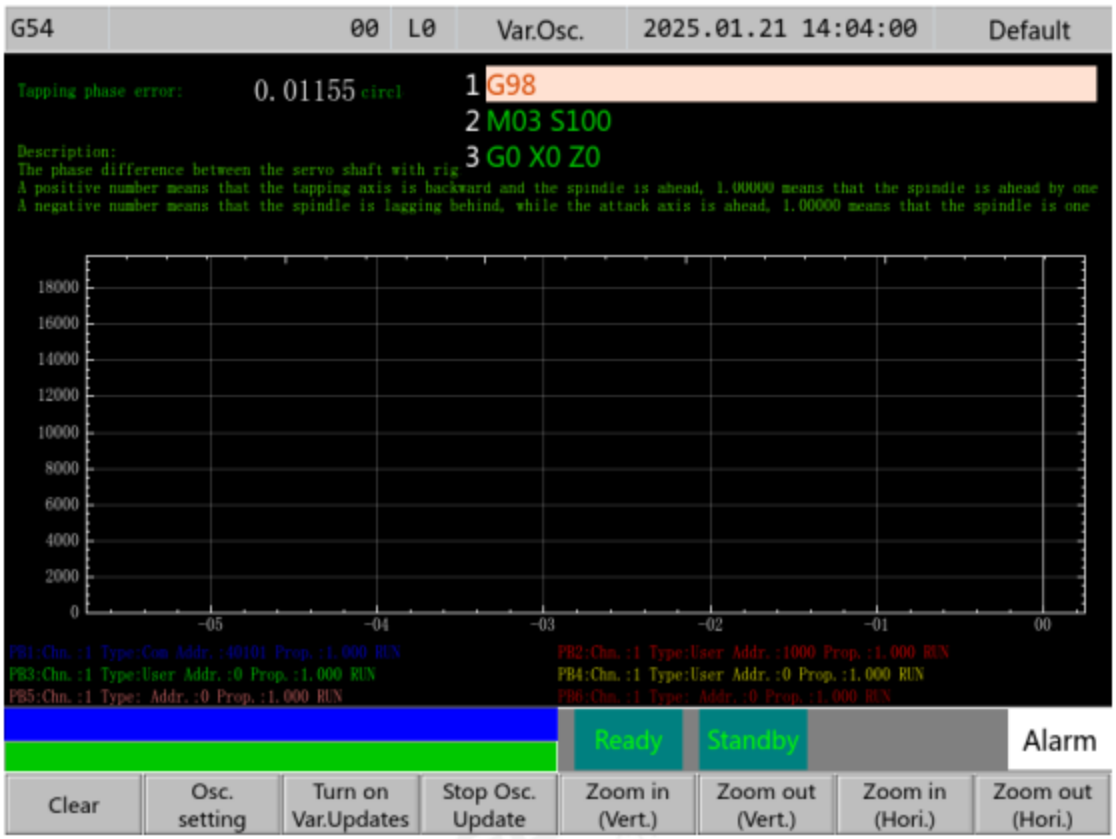
- **Operation Method**

1. Set the required letters, values, jump types, file names to jump to, priority, whether it is modal, and whether to enable the jump in the configuration table.
2. Press F2 [Save List] to automatically generate the configuration file.

- **Note**

After setting up the configuration table and saving it, a reboot is required for the changes to take effect.

1.8.9 Variable Oscilloscope



- **Operation Path**

"Position" page → [>>] → F5 [Extend]→F4 [Variable Oscilloscope].

- **Description**

This function is used to monitor changes in various variables and phase errors in each axis ofthe tooth

1.8.9.1 Clear

- **Operation Path**

"Variable Oscilloscope" page → F1 [Clear].

- **Description**

This function is used to clear the waveform displayed in the oscilloscope.

1.8.9.2 Oscilloscope Settings

- **Operation Path**

"Variable Oscilloscope" page → F2 [Oscilloscope Settings].

- **Description**

This function is used to configure the interface display, data settings, and channel settings of the variable oscilloscope.

1.8.9.3 Turn On Variable Update

- **Operation Path**

"Variable Oscilloscope" page → F3 [Turn On Variable Update].

- **Description**

This function is used to enable continuous updating of variables, keeping the oscilloscope waveform refreshed.

1.8.9.4 Stop Oscilloscope Update

- **Operation Path**

"Variable Oscilloscope" page → F4 [Stop Oscilloscope Update].

- **Description**

This function is used to stop the update of the oscilloscope, allowing observation of the waveform and data at that moment.

1.8.9.5 Vertical Zoom In

- **Operation Path**

"Variable Oscilloscope" page → F5 [Vertical Zoom In].

- **Description**

This function is used to zoom in the vertical data units of the oscilloscope.

1.8.9.6 Vertical Zoom Out

- **Operation Path**

"Variable Oscilloscope" page → F6 [Vertical Zoom Out].

- **Description**

This function is used to zoom out the vertical data units of the oscilloscope.

1.8.9.7 Horizontal Zoom In

- **Operation Path**

"Variable Oscilloscope" page → F7 [Horizontal Zoom In].

- **Description**

This function is used to zoom in the horizontal data units of the oscilloscope.

1.8.9.8 Horizontal Zoom Out

- **Operation Path**

"Variable Oscilloscope" page → F8 [Horizontal Zoom Out].

- **Description**

This function is used to zoom out the horizontal data units of the oscilloscope.

1.8.9.9 Start Tapping Phase Error Detection

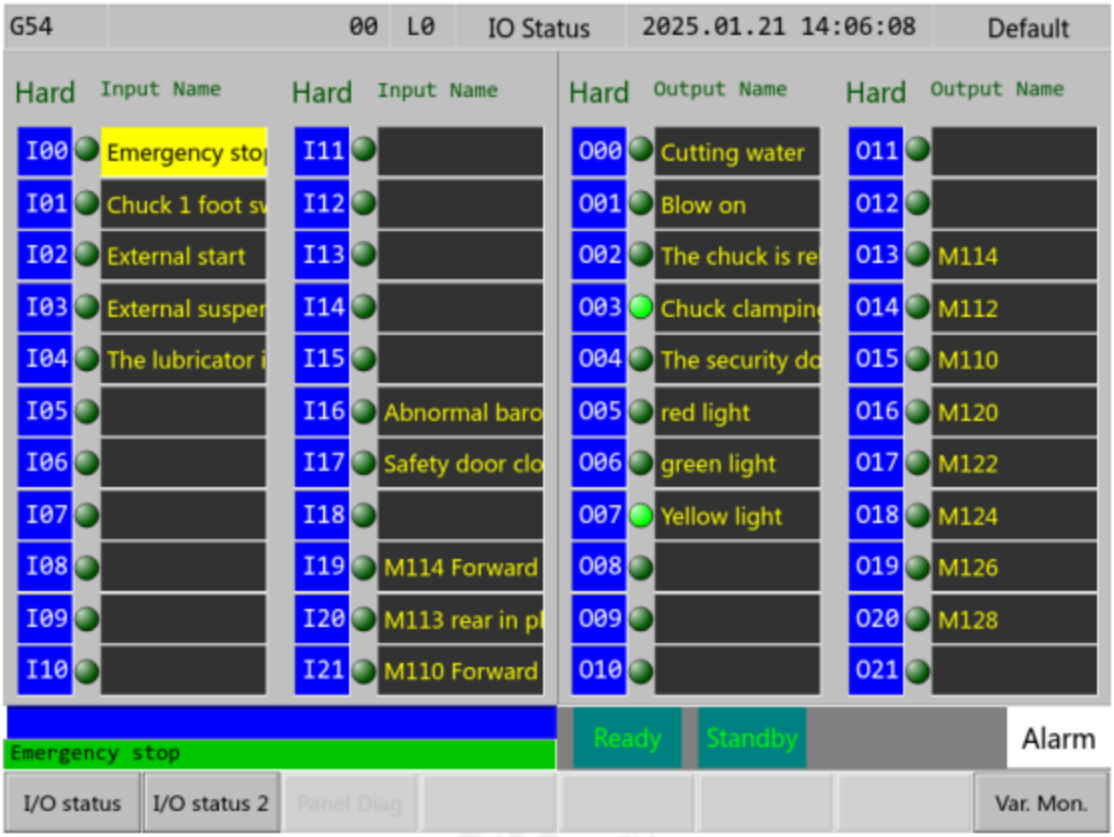
- **Operation Path**

"Variable Oscilloscope" page →[>>]→F1 [Start Tapping Phase Error Detection].

- **Description**

This function is used to detect the phase error of each axis during tapping.

1.9 Status Monitoring



- **Operation Path**

"Position" page → F7 [Fast Diagnosis]→F4 [IO].

- **Description**

This function is used to monitor the software IO, IOCSA, macro variables, auxiliary panel status, and variables of the controller.

1.9.1 I/O Status

- **Operation Path**

"Position" page → F7 [Fast Diagnosis]→F4 [IO] →F1 [I/O Status].

- **Description**

This function is used to switch the IO point status page of the controller.

1.9.2 I/O Status2

- **Operation Path**

"Position" page → F7 [Fast Diagnosis]→F4 [IO] →F2 [I/O Status2].

- **Description**

This function is used to switch the IO point status page of the controller.

1.9.3 IOCSA

- **Operation Path**

"Position" page →[>>]→F1 [PLC Status].

- **Description**

This function is used to monitor the software IOCSA status of the controller.

1.9.4 Variable Monitoring

- **Operation Path**

"Position" page → F7 [Fast Diagnosis]→F4 [IO] →F8 [Variable Monitoring].

- **Description**

This function is used to monitor the status of system variables of the controller.

1.10 Parameter Settings

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings] .

- **Description**

This function allows you to set the control parameters and user parameters for the controller's axes.

- **Operation Method**

1. Use the directional keys [↑] [↓] [←] [→] to move the cursor.
2. Use [⩓] [⩔] to scroll up and down.
3. Use the [Input] button to enter values.

- **Note**

After modifying parameters, press the [Reset] button to apply the changes.

1.10.1 All parameter

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F1 [All parameter].

- **Description**

This button can open the directory page and select the corresponding parameter modification.

1.10.2 Absolute value setting

- **Operation Path**

1："Position" page → [>>]→F3 [Parameter Settings]→F1 [All parameter]→Use arrow key [↑] [↓] [←] [→]Move the cursor to find [Serial parameter]→F1 [Normal Axis]→F7 [Absolute setting].

2："Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F7 [Absolute setting].

- **Description**

This button can switch to the absolute value parameter page and make changes.

1.10.2.1 Set X-Axis Coordinate to Zero

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F7 [Absolute setting]→F1 [Absolute zero setting]→F1 [Set X]

- **Description**

1. This button sets the X-axis coordinate to zero.

1.10.2.2 Set Y-Axis Coordinate to Zero

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F7 [Absolute setting]→F1 [Absolute zero setting]→F2 [Set Y]

- **Description**

1. This button sets the Y-axis coordinate to zero.

1.10.4.3 Set Z-Axis Coordinate to Zero

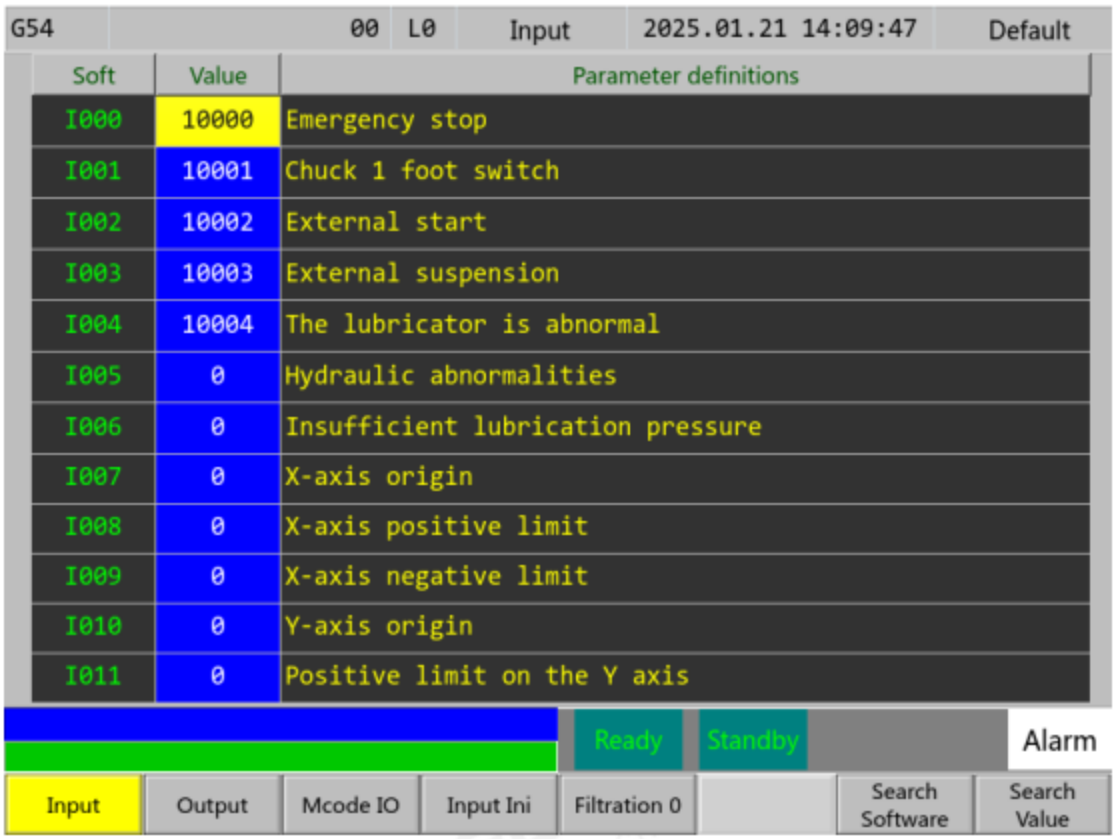
- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F7 [Absolute setting]→F1 [Absolute zero setting]→F3 [Set Z]

- **Description**

1. This button sets the Z-axis coordinate to zero.

1.10.3 IO Redefinition



- **Operation Path**

1."Position" page → [>>]→F3 [Parameter Settings]→F1 [All parameter]→Use arrow key [↑] [↓] [←] [→]Move the cursor to find [IO Redefinition].

2."Position" page → [>>]→F3 [Parameter Settings]→F8 [debug parameter]→F1 [IO Redefinition].

- **Description**

This function allows you to set the corresponding hardware IO signals for the PLC software IO signals, making it easier for customers to wire.

- **Note**

1. Multiple software output points cannot be assigned to the same hardware output point, or the system will generate an alarm. A single software input point can be assigned to one or more hardware input points.
2. After setting the parameters, press the [Reset] button, and then restart the machine

for the changes to take effect.

1.10.3.1 I-Point Configuration Page

- **Operation Path**

"IO Redefinition" page→F1 [Input].

- **Description**

This button switches to the I-Point Configuration page where you can redefine the I-points.

1.10.3.2 O-Point Configuration Page

- **Operation Path**

"IO Redefinition" page→F2 [Output].

- **Description**

This button switches to the O-Point Configuration page where you can redefine the O-points.

1.10.3.3 Custom M-Code Redefinition

- **Operation Path**

"IO Redefinition" page→F3 [Mcode IO].

- **Description**

This button switches to the Custom M-Code Redefinition page where you can redefine the IO points for custom M-code functions.

1.10.3.4 Simple mode

- **Operation Path**

"IO Redefinition" page→F5 [Simple mode].

- **Description**

This feature will display the IO functions used and hide the IO functions not used.

1.10.3.5 Restore Default Settings for I-Points

- **Operation Path**

"IO Redefinition" page→F1 [Input] →F4 [Input initialization].

- **Description**

This function restores the I-point settings to their factory default values. Use with caution!

1.10.3.6 Restore Default Settings for Output Points

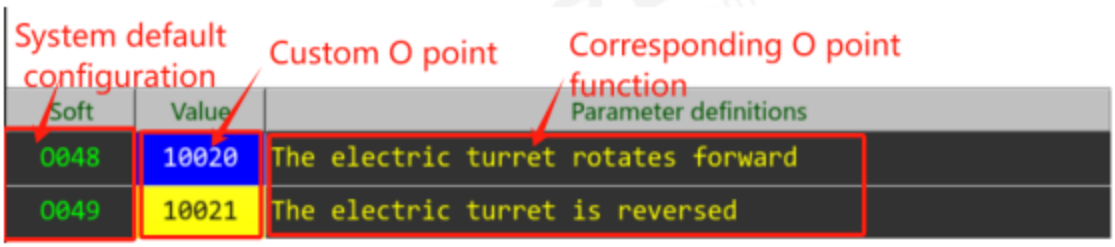
- **Operation Path**

"IO Redefinition" page→F1 [Output] →F4 [Output initialization].

- **Description**

This function restores the O-point settings to their factory default values. Use with caution!

1. IO Redefinition Application and Examples:
2. When a single hardware IO point is damaged, it can be redefined to another location using IO redefinition.
3. If the hardware IO points are limited and additional functionality beyond the software IO is required, it can be achieved through IO redefinition. Taking an electric turret as an example: Define the positive rotation of the electric turret's O048 as O12, which means the actual output on the IO board is O12 and it is configured as normally open. Define the reverse rotation of the electric turret's O049 as O13, which means the actual output on the IO board is O13 and it is configured as normally closed. After the configuration is completed, IO points O12 and O13 on the IO board will correspond to the turret functionality.

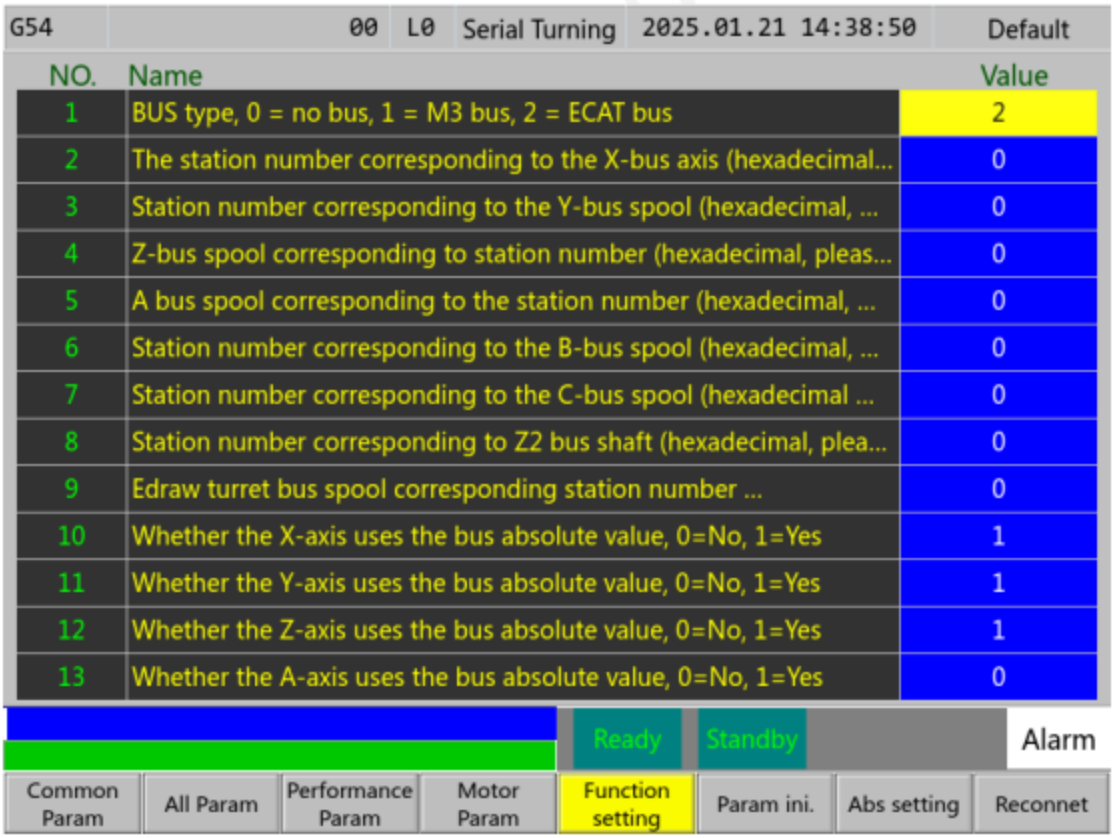


- **Additional Explanation**

In the IO Redefinition feature, the software IO points are pre-configured by the system,

and users can redefine them by entering values in the "Setting Value" column. The input format for redefining IO points is 10000+ the desired IO point number.

1.10.4 Bus Parameters



- **Operation Path**

1："Position" page → [>>]→F3 [Parameter Settings]→F1 [All parameter]→Use arrow key [↑] [↓] [←] [→]Move the cursor to find [Serial parameter]→F1 [Normal Axis]→F5 [Function setting].

2："Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F5 [Function setting].

- **Description**

This function is used to set parameters related to the bus.

- **Operation Method**

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

2. Use the scroll keys [⩓] [⩔] to scroll up and down.
3. Use the [Input] button to enter input.

1.10.4.1 Bus Monitoring

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→[>>]→F1 [Bus Monitoring].

- **Description**

This function is used to monitor the information of bus hardware axis commands, feedback, and torque.

1.10.4.2 Device Parameters

- Operation Path

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis]→F1 [common parameters].

- Description

This function is used to set the driver device parameters.

- Note

This button is only effective when the bus function is enabled and bus hardware axes are set.

1.10.4.3 Load All Drive Parameters

- **Operation Path**

"Position" page → [>>]→F3 [Parameter Settings]→F7 [Serial parameter]→F1 [Normal Axis].

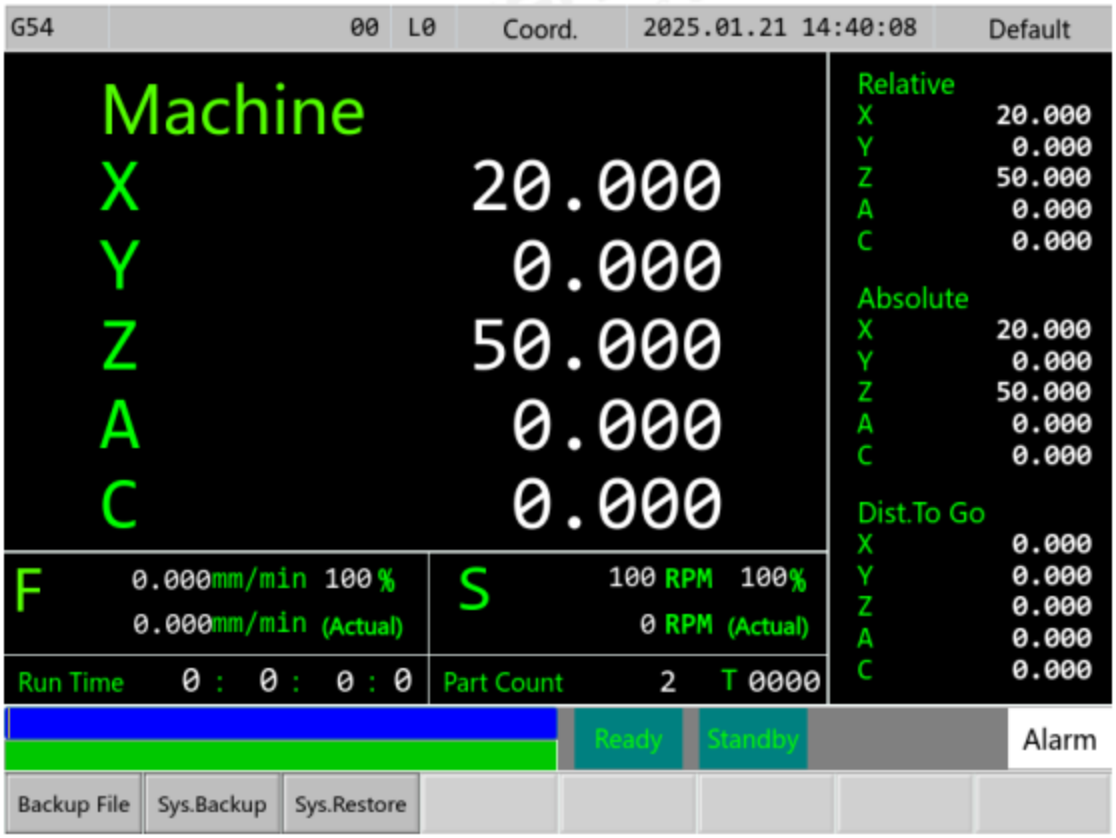
- **Description**

This function is used to set the driver device parameters.

- **Note**

This button is only effective when the bus function is enabled and bus hardware axes are set.

1.11 System Management



- **Operation Path**

"Position" page →[>>]→F4 [System admin].

- **Description**

This function is used for backup package management, data backup, data restoration, and factory reset.

- **Note**

The System Management page requires a password to enter, and the default password is "520".

1.11.1 Backup File

- **Operation Path**

"Position" page →[>>]→F4 [System admin]→F1 [Backup File].

- **Description**

This function is used to manage backup packages of the controller. It allows adding, deleting, editing comments, and transferring backup packages to a USB drive. For detailed instructions, please refer to the "Controller Backup" document.

1.11.2 System Backup

- **Operation Path**

"Position" page →[>>]→F4 [System admin]→F2 [System Backup].

- **Description**

This function is used to back up controller-related data and convert it into package format. For detailed instructions, please refer to the "Controller Backup" document.

1.11.3 System Restore

- **Operation Path**

"Position" page →[>>]→F4 [System admin]→F3 [System Restore].

- **Description**

This function is used to restore the backed-up controller data to the system, updating the system with the restored data. For detailed instructions, please refer to the "Controller Backup" document.

1.12 Load Monitoring/Rate Monitoring

- **Operation Path**

"Position" page →F4 [Monitor]→F4 [Parameter set].

- **Description**

This function is used to switch the display interface. Pressing this button will show the load rate for each axis. Pressing it again will display feed rate (F), spindle speed (S), machining time, workpiece count, and tool number (T).

Part 2.Mechanical Control Panel Description

2.1 Panel Operation Buttons

- **Power On**

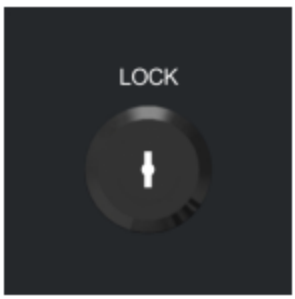
Turning on the controller power allows the machine to operate.

- **Power Off**

Turning off the controller power disables machine operation.

- **Emergency Stop**

In the event of a safety issue or machine malfunction, press this button to immediately stop the machine's motion. The controller will sound an alarm, and the control of I/O components will be determined based on the specific configuration.



- **Program Lock**

When the program lock function is enabled, the program cannot be edited.

- **Auto Mode**

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

- **Home Mode**

This mode allows the axes to perform homing actions, returning to the set zero position or mechanical zero position.

- **MDI Mode**

This mode is used to execute immediate programming without the need for executing a machining file.

- **Manual Mode (JOG)**

This mode is used for manually driving each axis. Before operating, select the corresponding axis.

- **MPG Mode**

This mode allows for axis control using a handwheel. Before operating, select the corresponding axis.

- **Jog Mode**

This mode allows for incremental motion of each axis. Before operating, select the corresponding axis.

- **Program Preview**

This feature allows the use of the handwheel to simulate the execution of the program in Auto Mode/MDI Mode. It helps in checking the correctness of the machining path.

- **Single Block Execution**

This feature allows for step-by-step execution of the machining program in Auto Mode/MDI Mode.

- **Skip Selection**

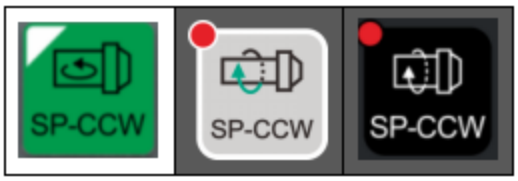
This feature allows for skipping program blocks containing the "/1" character during automatic program execution.

- **Stop Selection (M01)**

This feature allows for pausing the program execution at the M01 block during Auto Mode/MDI Mode.

- **Spindle Forward**

Starts the spindle rotation in the forward direction.

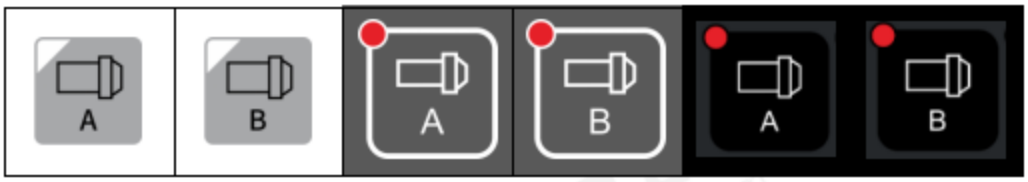


- **Spindle Reverse**

Starts the spindle rotation in the reverse direction.

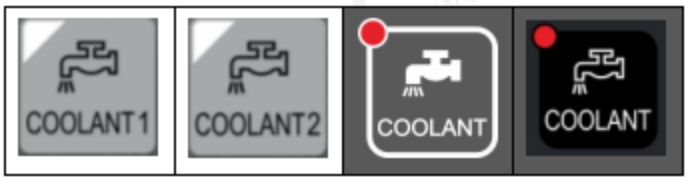
- **Spindle Stop**

Stops the spindle rotation



- **Spindle A/B**

This feature controls the forward rotation of spindle A/B.



- **Coolant On/Off**

Manually turns on/off the coolant.

- **Lubrication Oil**

Manually triggers a lubrication oil cycle.

- **Tool Changer Forward/Reverse**

This feature is used to control the forward/reverse movement of the tool changer.

- **Clamp/Unclamp**

This feature controls the loosening/clamping of the tool holder.

- **Work Light**

This feature controls the work light.

- **Air Blower**

This feature controls the air blower.

- **Spindle Brake**

This feature controls the spindle brake.

2.2 System Text Button Description

- Position: Switch to Machine Position page.
- Edit/Program Select: Used to switch between Program Editing page and Program Selection page.
- Tool Compensation/Settings: Used to switch between Tool Wear Compensation and Length Compensation pages.
- Monitoring: Switch to Machining Monitoring page.
- Help/Alarm: Used to switch between Help page and Alarm Monitoring page.
- A-Z Keys: Letter keys, total of 26 keys.
- 0-9 Keys: Numeric keys.
- Input: Used for inputting letters, numbers, symbols, etc.
- Delete: Backspace key for deleting.
- Space: Inserts a space.
- Reset: Generally used to stop the program, apply parameter changes, clear alarms after modifying parameters, etc.
- Home/End: Moves the cursor to the beginning or end of the line.

- Toggle: Pressing this key allows inputting characters located on the lower left corner of numeric and letter keys.
- Start: Starts program execution.
- Other commonly used keys: Not described individually.

Part 3. Operation

3.1 System Status

In the system, there are different status prompts based on the system's current state. Here are the trigger conditions for various states

3.1.1 Not Ready

The system imposes different restrictions based on different alarms.

- **Trigger Conditions**

1. When the system encounters an alarm.
2. When any axis has not been homed.

3.1.2 Ready State

The system is now ready for various operations.

- **Trigger Conditions**

1. The system alarm has been cleared, and the axes have returned to their home positions.
2. While in the machining/pause state, pressing the "Reset" button will change the system status to "Ready."

3.1.3 Machining

The system is currently executing a machining program.

- **Trigger Conditions**

When the system is in the "Ready" state and starts executing a machining program, the system state switches to "Machining".

3.1.4 Pause

The program is paused during the machining process.

- **Trigger Conditions**

When the system is in the "Machining" state, triggering a pause action will switch the system state from "Machining" to "Pause".

- **Note**

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

3.2 Alarm State

Indicates whether the system currently has any alarms.

3.2.1 Alarm Blinking

- **Trigger Conditions**

When there is any alarm present in the system.

3.2.2 Alarm Not Blinking

- **Trigger Conditions**

When there are no alarms present in the system.

3.3 Machine Preparation

3.3.1 Manual Function

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 describe each of these four functions.

3.3.1.1 Manual Continuous Feed

- **Description**

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

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. The operation mode is switched to "Manual Mode."

- **Operation Method**

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

3.3.1.2 Manual Rapid Feed

- **Description**

1. Drives the axis to continuously and rapidly move in one direction using the feed rate set by G00.
2. Can drive multiple axes simultaneously.

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. All axes have returned to the home position.
3. The operation mode is switched to "Manual Mode."

- **Operation Method**

1. Press and hold the "Rapid" button until the button light is on to activate rapid feed.
2. Simultaneously click on the axis buttons "X↑, X↓, Y↑, Y↓, Z→, Z←" to control the corresponding axis for continuous rapid movement.
3. The feed rate is adjusted based on the maximum feed rate set by G00.
4. The acceleration/deceleration time is set through parameters.

3.3.1.3 Jog Feed

- **Description**

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

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. The operation mode is switched to "Jog Mode."

- **Operation Method**

1. Click on the axis buttons "X↑, X↓, Y↑, Y↓, Z→, Z←" to control the corresponding axis for movement.

2. Each click triggers a single movement.
3. The distance for each jog movement is set as follows:
   - ❖ X1: Moves 0.001mm per jog.
   - ❖ X10: Moves 0.010mm per jog.
   - ❖ X100: Moves 0.100mm per jog.
   - ❖ This setting is shared with the handwheel multiplier selection.

3.3.1.4 Mpg Feed

❖ **Description**

Drives the axis to move continuously in one direction.

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. The operation mode is switched to "Handwheel Mode."

- **Operation Method**

1. Determine the axis to be driven by either switching the axis selection on the handwheel or clicking the axis selection buttons on the auxiliary panel.
2. Rotate the handwheel to drive the corresponding axis. The direction of rotation determines the direction of axis movement.
3. The distance for each handwheel rotation is set as follows:
   - ❖ X1: Moves 0.001mm per rotation.
   - ❖ X10: Moves 0.010mm per rotation.
   - ❖ X100: Moves 0.100mm per rotation.
   - ❖ This rotation multiplier setting is shared with the jog distance selection.

3.3.2 Program Processing

3.3.2.1 Automatic Execution

❖ **Description**

Executes the "Processing Program" for machining.

- **Operation Conditions**

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

❖ **Operation Method**

1. Click the "Auto" button to activate it. The button light will turn on.
2. Click the "Start" button to execute the current processing program.
3. The system status changes from "Ready" to "In Operation."
4. Once the program execution is complete, the system status switches back to "Ready."

3.3.2.2 Single Block Execution (MDI)

- **Description**

Executes [MDI] machining.

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. The operation mode is switched to "MDI" mode.

- **Operation Method**

1. Click the "MDI" button to activate it. The button light will turn on.
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 execution is complete, the system status switches back to "Ready."

3.3.3 Homing Mode

All coordinate settings required for machining are based on the machine origin. Therefore,

after powering on the controller, it is necessary to perform the origin return action to determine the machine origin.

- **Operation Conditions**

1. The system has not triggered any severe alarms such as emergency stop or axis abnormalities.
2. The operation mode is switched to "Origin" mode.

- **Operation Method**

1. Click the axis keys "X↑, X↓, Y↑, Y↓, Z→, Z←" to move the corresponding axis towards its origin position.
2. Once the axis completes the origin return action and comes to a stop, the system will zero the mechanical coordinates of that axis.
3. The method, direction, and speed of the origin return action can be set through parameters.

- **Note**

During the origin return process, the software limit for each axis is ineffective.

3.4 Tool Preparation

When editing a machining program, the description is based on the relationship between the tool tip and the workpiece, without considering the actual position of the tool or the differences in tool lengths. Therefore, the offset settings for each tool need to be measured and input into the controller before machining.

The system provides the following four correction methods: tool length compensation, tool wear compensation, tool radius compensation, and workpiece coordinate offset.

- **Tool Length Compensation**

Enable the tool number using T code, and the tool number corresponds to the compensation number.

- **Tool Wear Compensation**

Enable wear compensation using T code. This is commonly used for fine adjustments due

to tool wear, thermal expansion/contraction, or tool length correction.

- **Tool Radius Compensation**

Enable the tool number using T code, and then enable tool radius compensation using G41/G42 in the program.

- **Workpiece Coordinate Offset**

Use G54.1P1-G54.1P48, G54-G59 for coordinate system settings.

3.4.1 Tool Length Compensation

When editing a machining program, the description is based on the relationship between the tool tip and the workpiece, without considering the actual position of the tool or the differences in tool lengths. Therefore, it is necessary to measure and input the length compensation for each tool into the controller before machining.

- **Operation Method**

Press the "Tool Compensation/Setting" shortcut key twice to enter the tool length compensation page. Alternatively, you can go to the "Machine Position" page and press the "Offset/Setting" button, followed by the "Length Compensation" option to enter the tool length compensation page.

1. To change the values, use the page up and page down keys (⩓, ⩔) or the arrow keys (↑, ↓, ←, →) to move the cursor to the desired position.
2. There are three input methods available:

❖ Enter character A at the cursor and press the [Enter] button. Absolute input will be displayed in the upper left corner, and the cursor will move to the corresponding position to directly input numerical values; Generally, absolute values are used to input tool tip radius compensation or tool length correction.

❖ Enter the character I at the cursor and press the [Enter] button. Incremental input will be displayed in the upper left corner. Move the cursor to the corresponding position and directly enter the value to be accumulated. The value at that position will be equal to the current value plus the input value; Generally, incremental values are used to input tool radius wear compensation or tool wear correction.

❖ Use "X***" or "Z***" format for teaching input, where X or Z can be replaced with other axis names (e.g., Y, X1, X2), and *** represents the taught value. This method is commonly used for tool length compensation. (This process is also known as tool setting.)

3. The calculation formulas for compensation are as follows:

   ❖ (Tool tip radius compensation + Tool radius wear compensation) represents the actual G41/G42 compensation amount.

   ❖ (Tool length compensation + Tool wear compensation) represents the actual tool compensation amount.

   ❖ When using teaching input, the cursor does not need to be moved to the correct axis position. Just move it to the correct tool number for input. The controller will fill in the values according to the taught axis names.

   ❖ When teaching input is completed, the corresponding wear compensation values will be cleared. For example, if teaching input for tool number 1 is performed on the X-axis, the X-axis wear compensation value for tool number 1 will be cleared to zero.

3.4.2 Tool Length Measurement

After the T code is executed, tool compensation takes effect. Therefore, the operator must set the tool length data before cutting.

❖ **Before measuring, it is necessary to confirm the following**

1. Verify that the external offsets are correct.
2. Verify that the workpiece coordinate system offsets are correct.

3.4.2.1 Z-Axis Tool Length Measurement

Press the "Tool Compensation/Setting" shortcut key twice to enter the tool length compensation page, or on the "Machine Position" page, press "Offset/Setting," then press "Tool Length Compensation" to enter the "Tool Length Compensation" page.

● **Operating Steps**

1. Move the cursor to the tool number for which you want to modify the compensation amount, for example: Tool 1.
2. Use manual operation to move the tool until the tool tip contacts the Z-axis work zero

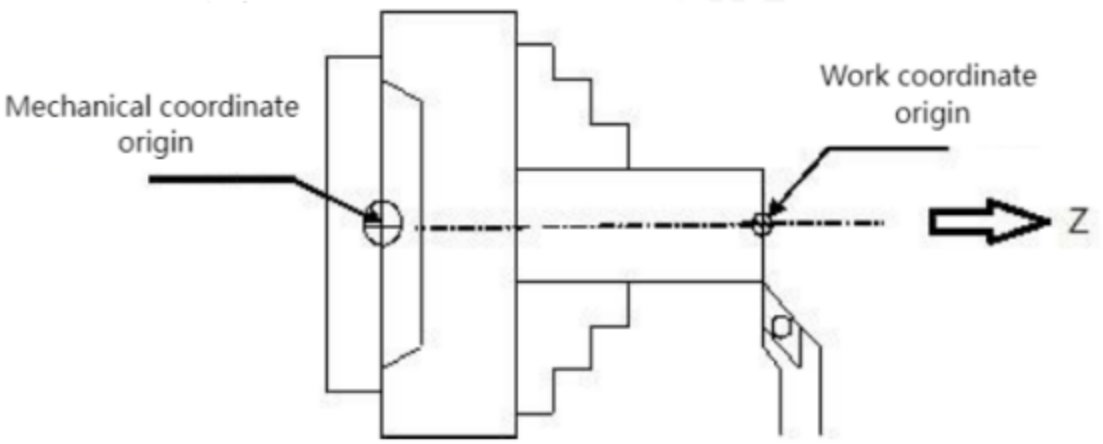
point, which is the origin of the Z-axis program coordinates. Use teaching input "Z0" to set the current position as the Z-axis work zero point.

- **Z-axis cutting measurement input**

It is usually difficult to manually move the tool tip to the Z-axis work zero point. Therefore, cutting is often used for measurement.

- **Operation Example**

1. Clamp the round bar in the spindle.
2. Manually move the tool tip above the end face of the workpiece to ensure that it can cut into the workpiece along the X-axis direction.
3. Start the spindle rotation and manually perform turning along the X-axis.
4. Retract the tool along the X-axis without moving the Z-axis.
5. Use teaching input "Z0." The end face that has been cut is now set as the Z-axis work zero point.



- **Description**

When teaching input is completed, the corresponding wear compensation value will be cleared. For example, if teaching input for the Z-axis length setting is used for Tool 1, the Z-axis wear compensation for Tool 1 will be cleared to zero.

3.4.2.2 X-Axis Tool Length Measurement

- **X-axis cutting measurement input**

It is usually difficult to manually move the tool tip to the X-axis work zero point. Therefore,

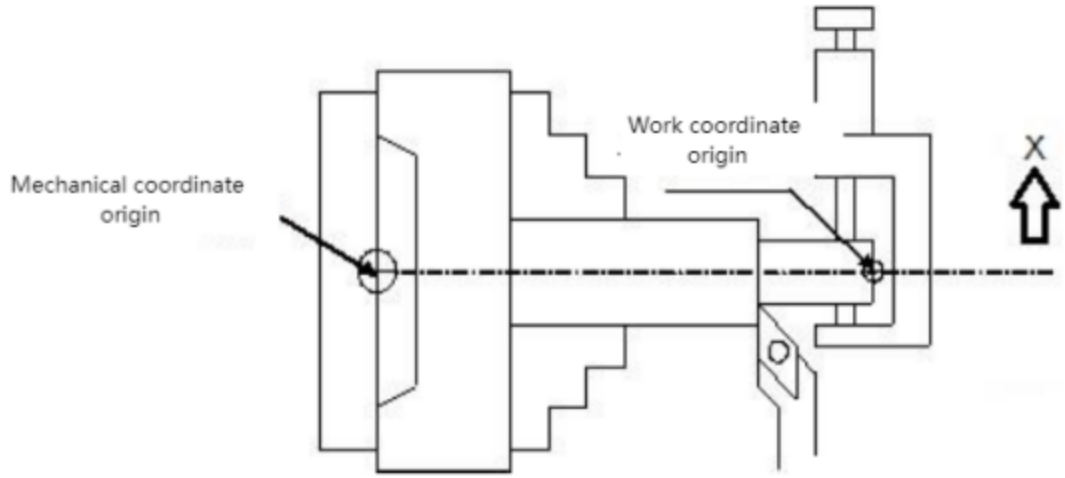
cutting is often used for measurement.

- **Operating Steps**

1. Press the "Tool Compensation/Setting" shortcut key twice to enter the tool length compensation page, or on the "Machine Position" page, press "Offset/Setting," then press "Tool Length Compensation" to enter the tool length compensation page.
2. Move the cursor to the tool number for which you want to modify the compensation amount, for example: Tool 1.
3. Use manual operation to move the tool until the tool tip contacts the X-axis work zero point, which is the origin of the X-axis program coordinates. Use teaching input "X0" to set the current work zero point position.

- **Operation Example**

1. Clamp a round bar with a diameter of 10mm in the spindle.
2. Manually move the tool to ensure that the actual distance between the tool tip and the center of the workpiece is less than 5mm, ensuring that the tool can cut into the workpiece along the Z-axis direction.
3. Start the spindle rotation and manually perform turning along the Z-axis.
4. Retract the tool along the Z-axis without moving the X-axis.
5. Measure the diameter "D" of the workpiece using a caliper after cutting.
6. Use teaching input to enter the measured value as "X+/-D." For example, if the measured workpiece diameter is 9mm, enter "X9." If using the X-radius axis, enter "X4.5."
7. The positive/negative sign is determined by the front or rear turret of the machine. If the tool tip is located at X+ (on the centerline), the symbol is "+". If the tool tip is located at X-, the symbol is "-".



- Description

When teaching input is completed, the corresponding wear compensation value will be cleared. For example, if teaching input for the X-axis length setting is used for Tool 1, the X-axis wear compensation for Tool 1 will be cleared to zero.

3.4.3 Tool Wear Compensation

1. The tool wear setting is commonly used for fine adjustments of tool wear, thermal expansion/contraction, or tool length correction. After the lathe completes the tool length setting, the tool length value is not modified arbitrarily. Instead, the tool wear setting is used to fine-tune the cutting dimensions.
2. When teaching input for the tool length setting is used, the tool wear is automatically set to zero. Based on test cutting and measurement of the machining results, if there is an error, the tool wear compensation is used to compensate for the tool length.
3. Actual tool length = Tool length + Tool wear.

- **Operation Method**

1. Press the shortcut key "Tool Comp./Setting" to enter the tool wear compensation page, or press "Offset/Setting" on the "Machine Position" page to enter the tool wear compensation page.
2. Use the page up/down keys or arrow keys to move the cursor to the desired value.

- **Operation Method**

1、Enter character A at the cursor and press the [Enter] button. Absolute input will be displayed in the upper left corner, and the cursor will move to the corresponding position to directly input numerical values.

2、Enter the character I at the cursor and press the [Enter] button. Incremental input will be displayed in the upper left corner. Move the cursor to the corresponding position and directly enter the value to be accumulated. The value at that position will be equal to the current value plus the input value.

1. Positive/negative sign +/-: The input value can be preceded by the sign "+/-".
2. The positive/negative sign determines the compensation direction.
3. If the tool tip needs to be adjusted in the positive direction, enter the positive sign "+".
4. If the tool tip needs to be adjusted in the negative direction, enter the negative sign "-".

- **Example**

If the machining result is larger than the programmed value by 10um and the tool tip needs to be adjusted in the X- direction, input "-0.010" for the tool wear value. This will add "-10um" to the current tool wear value. In the next cutting operation, the tool tip cutting path in the diameter direction will be offset by 10um towards the X- direction.

- **Parameters**

The maximum value for tool wear compensation can be set through parameters to avoid collision caused by input errors.

3.4.4 Tool Tip Radius Compensation

Because the tool tip is circular, only the precise length of the tool can be measured. To compensate for any errors in the tool tip size, the tool tip radius is used.

- **Operation Method**

Press the shortcut key "Tool Offset/Setting" to enter the tool tip radius wear compensation page, or press "Offset/Setting" on the "Machine Position" page to enter the tool tip radius wear compensation page, and then press "Length Compensation" to switch to the tool tip

radius compensation page.

Use the page up/down keys or arrow keys (up/down/left/right) to move the cursor to the desired value.

- **The following information about the tool tip is available**

1. Tool Tip Radius: The radius of the tool tip.
2. Tool Tip Radius Wear: The wear on the tool tip radius. The actual tool tip size is calculated as Tool Tip Radius + Tool Tip Radius Wear.
3. Tool Tip Direction: The direction of the tool tip. There are eight options for tool tip direction. Depending on the shape of the tool tip (please refer to the machine programming manual for more detailed information on G41/G42), use G41/G42 to activate tool tip compensation.

3.5 Program Preparation and Execution

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

3.5.1 Set Processing File

- **Operating Conditions**

Any mode except for "MDI" mode.

- **Procedure**

1. Specify the current editing program as the machining program.
2. Switch to the "Program Edit" page.
3. Click on "Load Machining" to switch to the "Machining Monitoring" page.
4. The program file will be designated as the machining program file.
5. Specify a program file from the file management as the machining program file.
6. Switch to the "Program Select" page.
7. Move the cursor to the desired program to be loaded for machining.

8. Click on "Load Machining" to switch to the "Machining Monitoring" page.
9. The program will be designated as the machining program.

- **Verification**

1. The successful specification of the machining program can be confirmed in the following two ways:
2. The current machining program name displayed at the top of the screen.
3. The program content displayed on the "Machining Monitoring" page.

3.5.2 Graphic Simulation

The system provides a convenient program content simulation function, which allows easy simulation of the machining path after editing the program. This function includes program checking capabilities, helping users quickly identify syntax errors or unreasonable actions in the machining program. It is recommended to use this function to check the program content after editing.

- **Procedure**

1、On the "Program Select" page, select the program to be edited.

2、After editing the program,"Program Editor" page→F7 [Graphic simulation]

3、Scan the content of the program.

4、Once the scanning is complete, the system will start the graphic simulation based on the program content until the entire program is simulated.

- **Description**

❖ In the simulation graphics:

1. Solid lines represent the cutting path.
2. Dotted lines represent the movement path.

❖ If there are any syntax or content errors detected during the program scanning process, the system will generate an alarm message and indicate the error line number.

3.5.3 Test Processing

3.5.3.1 Program Prediction (MPG Simulation)

This section explains how to use the MOG simulation for machining testing.

- **Operating Conditions**

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

- **Procedure**

1. Start the program execution for machining.
2. Press the "Program Prediction" button on the panel.
3. Use the MPG to perform the machining.
4. Rotate the MPG clockwise to advance to the next line of the program for machining.
5. Rotate the MPG counterclockwise to go back to the previous line of the program for machining. This function is also known as "Manual Pulse Generator retreat".

- **Verification**

1. You can confirm the successful activation of the MPG simulation through the following:
2. The "Program Prediction" indicator light on the auxiliary panel is lit.
3. Once the MPG simulation is enabled during program execution, the machine will immediately decelerate to 0 until the MPG is operated or the MPG simulation is canceled.

3.5.3.2 Single Block Execution

This section explains how to use the single block mode to execute program machining.

- **Operating Conditions**

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

- **Procedure**

1. Start the program execution for machining.
2. Press the "Single Block" button on the panel.
3. The system will decelerate after completing the current block until it reaches 0,

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 completing the next block.

3.5.4 Machining Monitoring

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

- **Operating Conditions**

None

- **Description**

1. Total Workpieces:

❖ Represents the cumulative total of workpieces processed by the machine.

2. Required Workpieces:

❖ When a single machining program specifies a required number of workpieces, if the program continues to run with M99, it will pause and issue a notification when the required number of workpieces is reached.

3. Workpiece Count:

❖ Continuously accumulates when the program is set to continue machining with the M99 command.

❖ The workpiece count can be reset to 0 under the following circumstances:

✓ When the required workpiece count is reached.

✓ When changing the machining program.

✓ When modifying the required workpiece count to a value lower than the current workpiece count.

✓ When executing the M16 command.

3.6 System Alarm Handling

In order to prevent hazards to personnel and machine safety due to incorrect 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 will explain how to access alarms and the troubleshooting methods.

3.6.1 Emergency Stop

When a machine malfunction or unexpected action occurs, posing a risk to personal or machine safety, pressing the emergency stop button will immediately halt the machine's operation. Once pressed, the emergency stop button remains locked. The unlocking procedure may vary depending on the manufacturer, but typically involves rotating the button. This button interrupts the machine's actions, and before unlocking, the cause of the fault must be addressed.

3.6.2 Alarm Display

Alarms are categorized as current alarms and alarm history. Refer to section 1.8.8 "Alarms" for more information on their display.

3.6.2.1 Current Alarms

1. Shows the current status of system alarms.
2. When an alarm occurs, the controller displays a pop-up window with the details of the current alarm.
3. Pressing the return key [<<]cancels the pop-up window.
4. If the alarm is not resolved, clicking "Reset" will display the alarm window again.
5. Switching to the "Alarms" page will automatically display the current alarms.

3.6.2.2 Alarm History

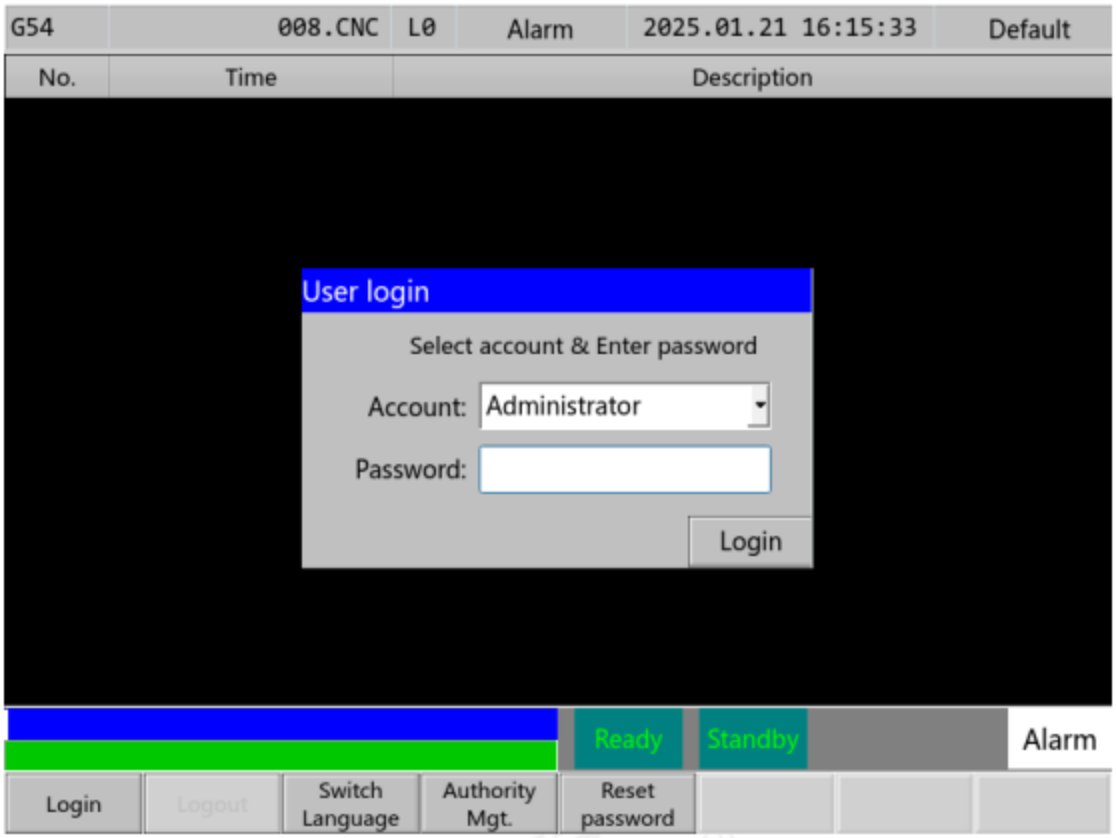
1. Displays past alarms that have occurred in the system, allowing identification of the possible causes.
2. Switch to the "Alarms" page and click "Alarm History" to view the alarm history.
3. When multiple alarms are present, they are listed in chronological order, with the most recent alarms at the top.

Part 4. Access Management

4.1 Parameter and System Protection

To protect the system's special functions and operations, access is restricted to authorized personnel with passwords. This prevents accidental misuse of the system's special functions, which could lead to system malfunctions.

4.1.1 Password Login



- **Operation Path**

Path 1: "Position" page →F5 [Maintain]→[>>]→F1 [Login].

- **Description**

This function is used to enter a password and gain access to specific operations.

- **Operation Method**

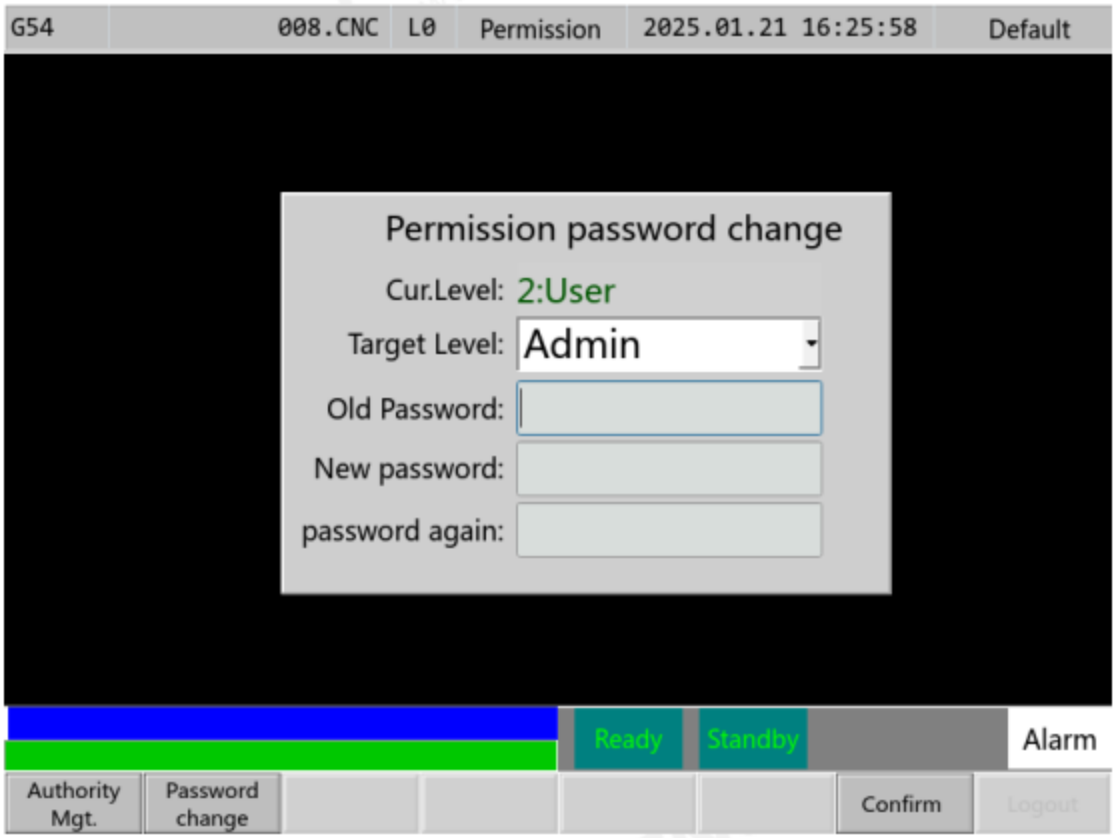
1. Use the numerical keypad on the system panel to enter the password.

2. Use the [↑] and [↓] keys to navigate to the "Authorization Selection" and press [Enter] to confirm the access level.

- **Note**

If an incorrect password is entered, the system will display a "Login failed!!! Incorrect password" message.

4.1.2 Password Modification



- **Operation Path**

"Position" page →F5 [Maintain]→[>>]→F5 [Reset password].

- **Description**

This function is used to modify the authorization password.

- **Operation Method**

1. Use the numerical keypad on the system panel to enter the original password and the new password.

2. Click on F7 [Confirm Password Modification] to confirm the password modification.

- **Note**

The original password and the new password cannot be the same. Otherwise, a message saying "New password cannot be the same as the original password" will be displayed.

4.1.2.1 Move Down

- **Operation Path**

"Password " page → 【∨】 Key.

- **Description**

This function is used to switch the input box and move the focus downwards.

4.1.2.2 Move Up

- **Operation Path**

"Password " page → 【∧】 Key.

- **Description**

This function is used to switch the input box and move the focus upwards.

4.1.2.3 Confirm Password Modification

- **Operation Path**

"Password " page → F7 [Confirm Password Modification].

- **Description**

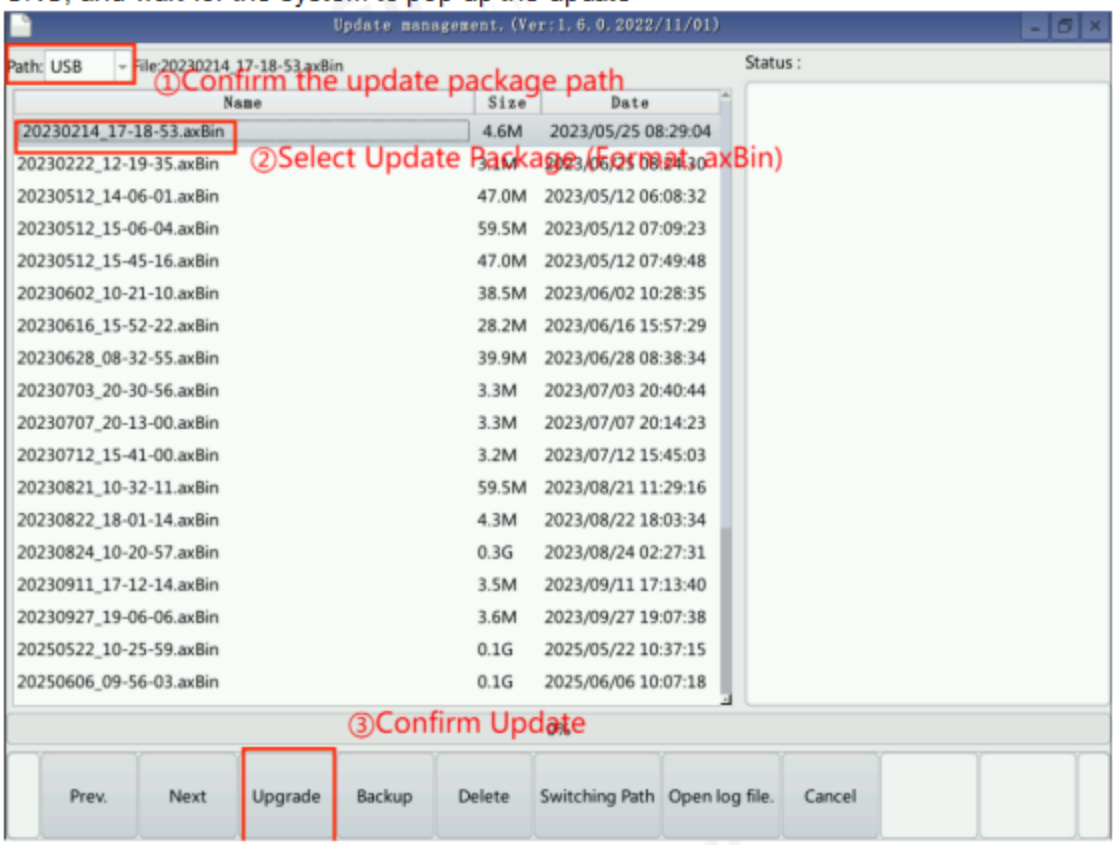
This function is used to confirm the password modification.

Part 5. Appendix

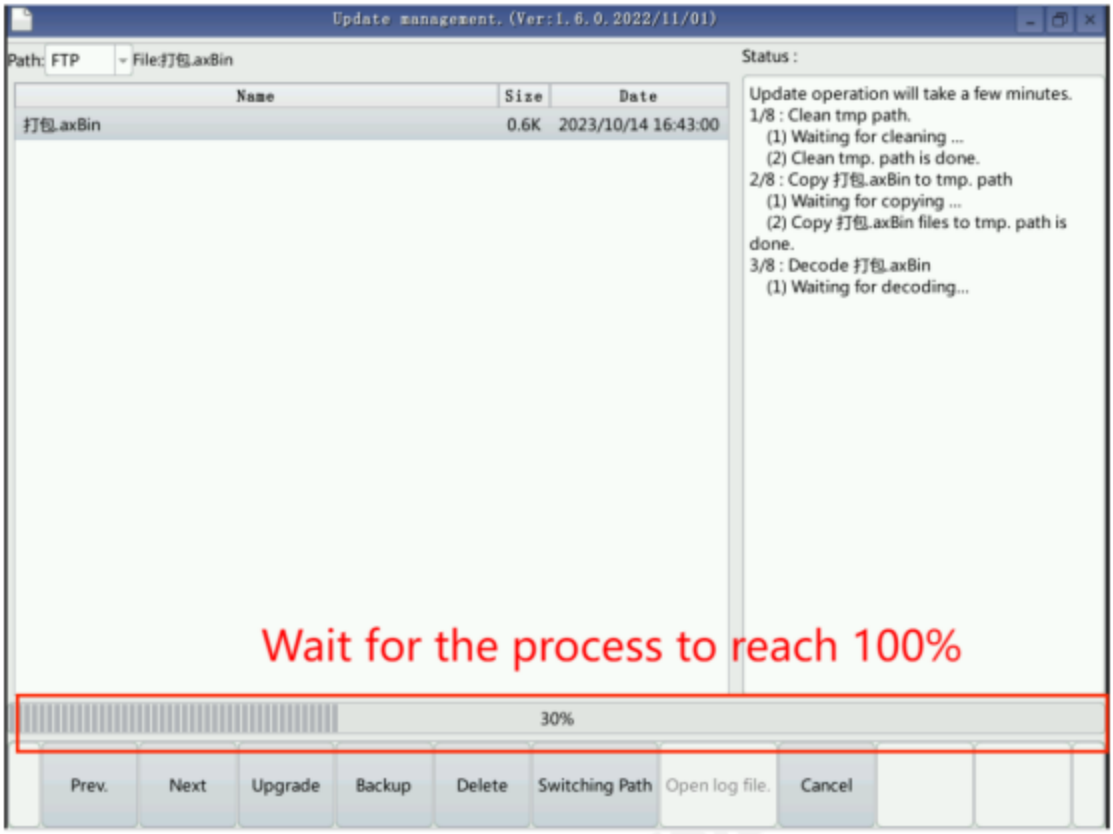
5.1 Software Upgrade/Backup Operation Steps

5.1.1 Software upgrade steps

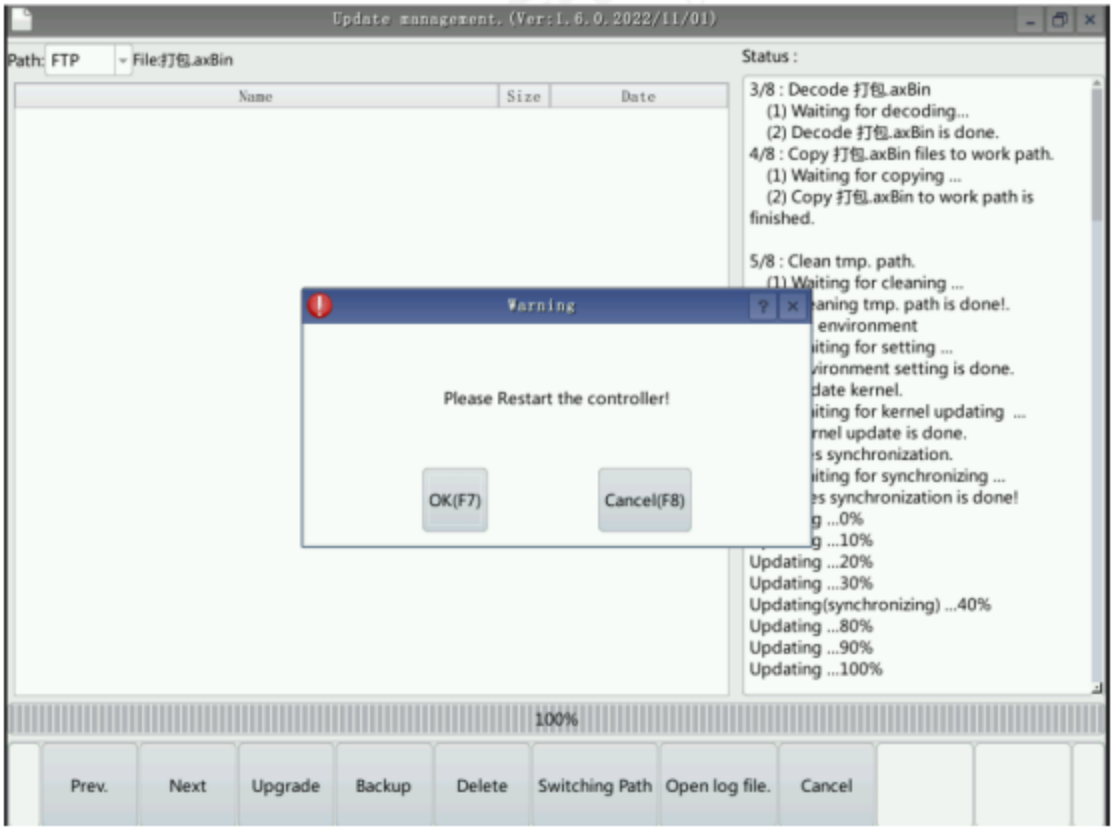
Step 1: Transfer the update package to the USB drive, insert the USB drive into FINGER CNC, and wait for the system to pop up the update



Step 2: During the update

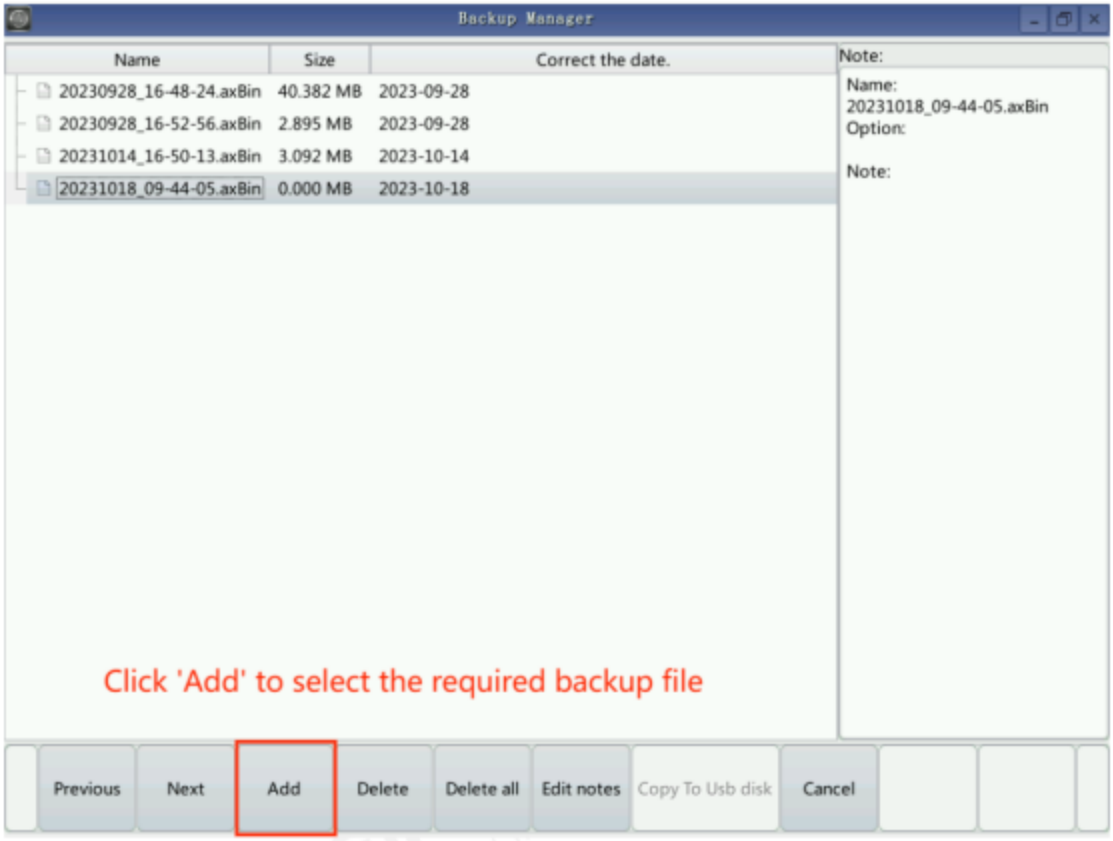


Step 3: Update completed: After the update process reaches 100%, a message 'Please restart the system!' will pop up to complete the update.

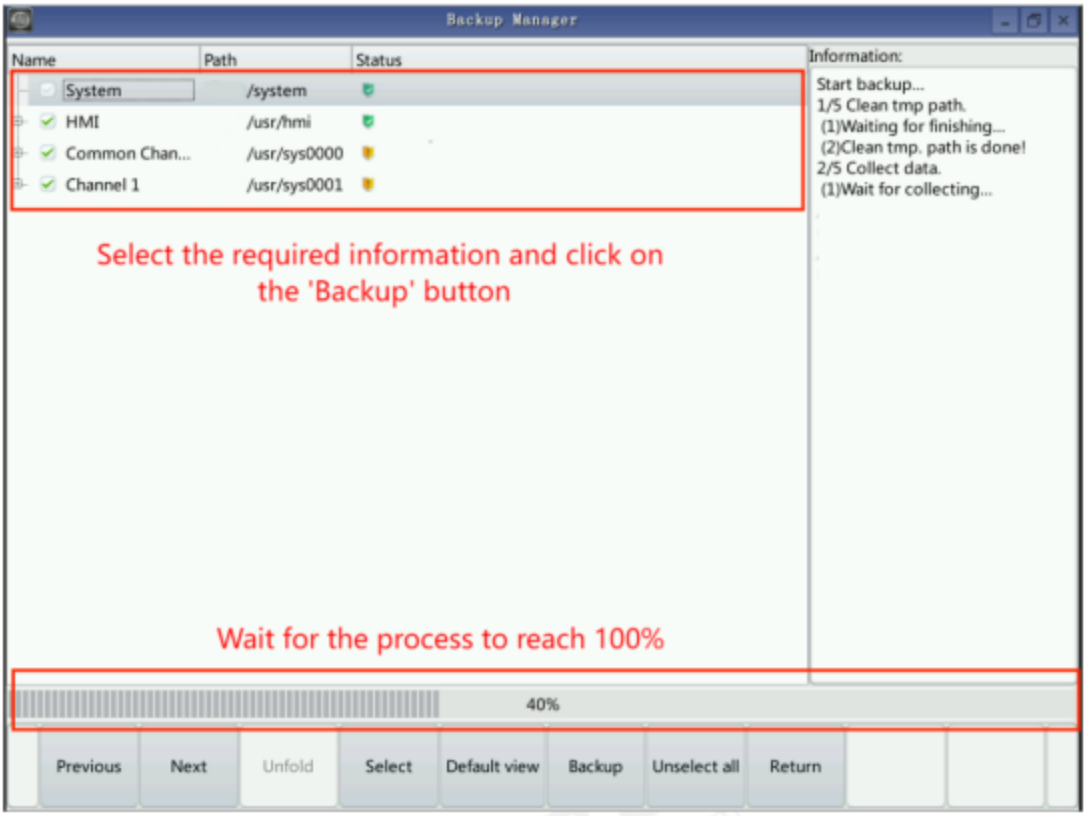


5.1.2 Software backup steps

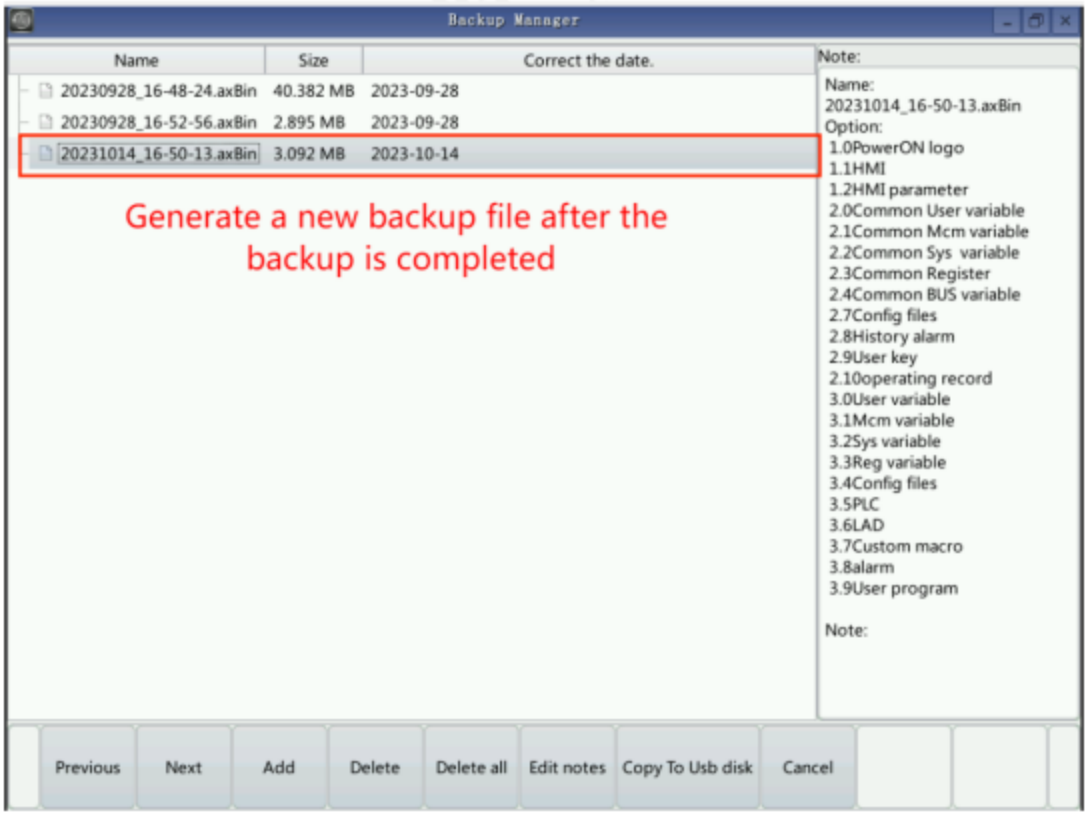
Step 1: Click the backup button to enter the backup



Step 2: Backup file selection and operation



Step 3: Newly generated data files after backup is completed



Part 6. Lathe Wiring Description

6.1 System Installation Environmental Requirements

The B-series controller must be installed and used in the following environment. Using it outside of these ranges may result in abnormal operation.

Ambient temperature	**During Operation**	0℃ to45℃
	During Storage or Transportation	-20℃ to 55℃
Humidity	**Under normal conditions**	Relative humidity less than 80%RH
	For short periods of time	Maximum 95%RH
Vibration limit	**During Operation**	At a frequency of 5Hz, the maximum is 0.075 millimeters
Noise	**During Operation**	The highest voltage pulse between every 0.01 second is 2000 volts/0.1X10-6 seconds
Rate of temperature change		Maximum 1.1°C per minute
Others		In environments with excessive dust, cutting fluids, and organic solvents, please contact our company first.

6.2 Considerations for Controller Box Design

- The controller and auxiliary panel boxes must be sealed to prevent dust ingress.
- The temperature rise inside the box should not exceed 10℃ compared to the surrounding ambient temperature.
- Cable entry and exit points must be sealed.
- To avoid noise interference, the distance between cables, each unit, and the AC power source must be at least 100 millimeters. If there is a magnetic field, the distance between them should be at least 500 millimeters.
- For servo drive installation, please refer to the servo operation manual.

6.3 Box Internal Temperature Design

- The maximum temperature rise inside the box compared to the surrounding ambient temperature should not exceed 10℃. When designing the box, the main considerations are the heat source and the heat dissipation area. For the controller, customers have limited control over the heat source, but the heat dissipation area is an important factor to consider. The allowable temperature rise inside the box can be estimated using the following formulas:
  1. With cooling fans, the allowable temperature rise is 1℃/6W/1m2.
  2. Without cooling fans, the allowable temperature rise is 1℃/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 6W heat source inside the box (4W without cooling fans), the internal temperature will rise by 1℃. 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)

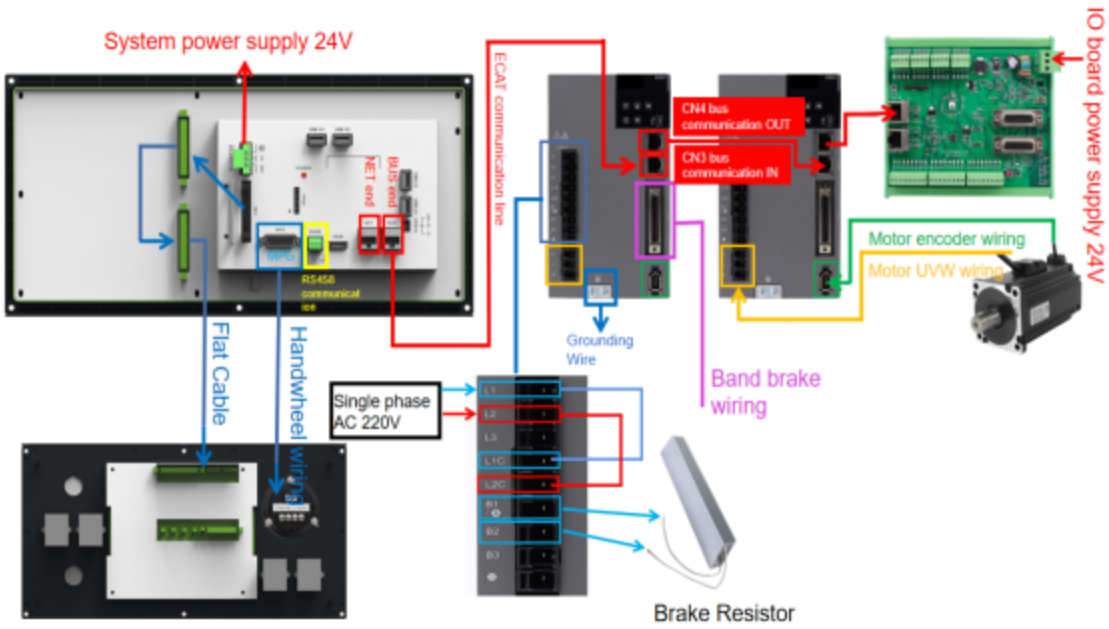
- Heat dissipation area = 2m2.

- Maximum allowable temperature rise inside the box = 10℃.
- Therefore, the maximum allowable heat source inside the box = 6W x 2m2 x 10 = 120W.
- If the heat source inside the box exceeds 120W, cooling devices such as Cooling fins must be added.

Example 2: (Without cooling fans)

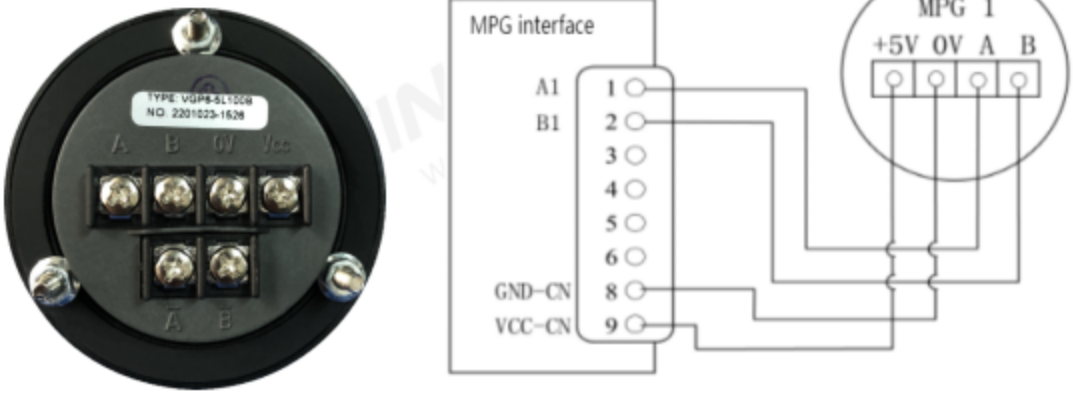
- Heat dissipation area = 2m2.
- Maximum allowable temperature rise inside the box = 10℃.
- Therefore, the maximum allowable heat source inside the box = 4W x 2m2 x 10 = 80W.
- If the heat source inside the box exceeds 80W, fans or cooling devices such as Cooling fins must be added.

6.4 System Wiring

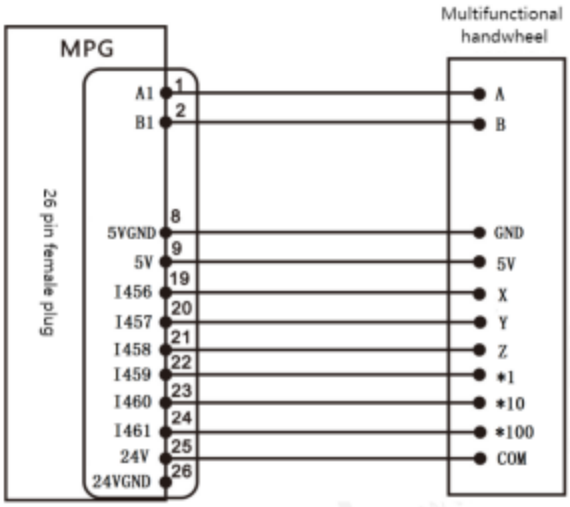


6.5 Manual Pulse Generator Wiring

6.5.1 Wiring of a regular Manual Pulse Generator Wiring



6.5.2 Wiring of 6-Axis Multi-Function Manual Pulse Generator



Note: Is the common terminal ofthe multifunctional handwheel powered by 24V

6.5.3 Wiring of 8-Axis Multi-Function Manual Pulse Generator

1	A1	A（Yellow）
2	B1	B（White）
8	GND-CN	0V（Black）
9	VCC-CN	5V（Red）
19	I456	Axis selection 1 （Brown）
20	I457	Axis selection 2（Orange）
21	I458	Axis selection 4（Light Blue）
22	I459	Axis selection 8(Blue)

23	I460	Magnification 1 （Green/Black）
24	I461	Magnification 12（Blue/Black）
25	24V	L+(Transparent)
26	GNDX-CN	COM(Purple)
		L-(Light Purple)

6.5.4 Diagram of MPG Port Pin Definitions

● MPG

PIN	1	2	3	4	5	6	7	8	9
Definition	A1	B1	A2	B2	G31-IO	GND			5V
PIN	10	11	12	13	14	15	16	17	18
Definition	+12V	-12V	DAC1	DAC2	ADC1	ADC2	G31-I1	CAN D+	CAN D-
PIN	19	20	21	22	23	24	25	26	
Definition	I456	I457	I458	I459	I460	I461	24V	GND-CN	

*** Pins 6/7/8 GND are grounded for 5V/ADC/DAC**

*** Pin 26 GND-CN is grounded for I-point/G31**

6.6 Interface Definitions

- **P1-P6 Servo Axis**

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	OUT-*	IN-*							ADC
PIN	26	25	24	23	22	21	20	19	
Definition	24VGND	24V	OUT-*	IN-*	CCW-	CCW+	CW-	CW+	

Pin 8 (GND) serves as the ground for 5V, VCMD, and ADC.

Pin 7 (VCMD) is an analog signal ranging from -10V to +10V.

- **MPG**

PIN	1	2	3	4	5	6	7	8	9
Definition	A1	B1	A2	B2	G31-IO	GND			5V
PIN	10	11	12	13	14	15	16	17	18
Definition	+12V	-12V	DAC1	DAC2	ADC1	ADC2	G31-I1	CAN D+	CAN D-
PIN	19	20	21	22	23	24	25	26	
Definition	I456	I457	I458	I459	I460	I461	24V	24VGND	

Pins 6, 7, and 8 (GND) are the ground connections for 5V, ADC, and DAC.

Pin 26 (24VGND-CN) can be used as the ground for G31.

- **Voltage Command Control of the Spindle——Combined with Frequency Inverter**

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	Reverse								
PIN	26	25	24	23	22	21	20	19	
Definition	24VGND	24V	Forward	Alarm					

Pin 8 (GND) is the ground for 5V, VCMD, and ADC.

Pin 7 (VCMD) is a 0V~10V analog signal.

- **Voltage Command Control of the Spindle—— Voltage Command Control of the Spindle**

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					
Pin 8: GND for 5V/VCMD/ADC ground. **Pin 7: VCMD for analog signal in the range of -10V to 10V.**									

- **Pulse Command Control of the Spindle——Combined with Servo Drive——Asynchronous Motor**

PIN	9	8	7	6	5	4	3	2	1
Definition		GND		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	24VGND	24V	Speed control	Alarm	CCW−	CCW+	CW−	CW+	

Pin 8: GND for 5V/VCMD/ADC ground.

- **Pulse Command Control of the Spindle——Combined with Servo Drive——Servo Motor**

PIN	9	8	7	6	5	4	3	2	1
Definition		GND		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	24VGND	24V	Enable	Alarm	CCW-	CCW+	CW-	CW+	
Pin 8 GND is the ground for 5V/VCMD/ADC.									

Note：

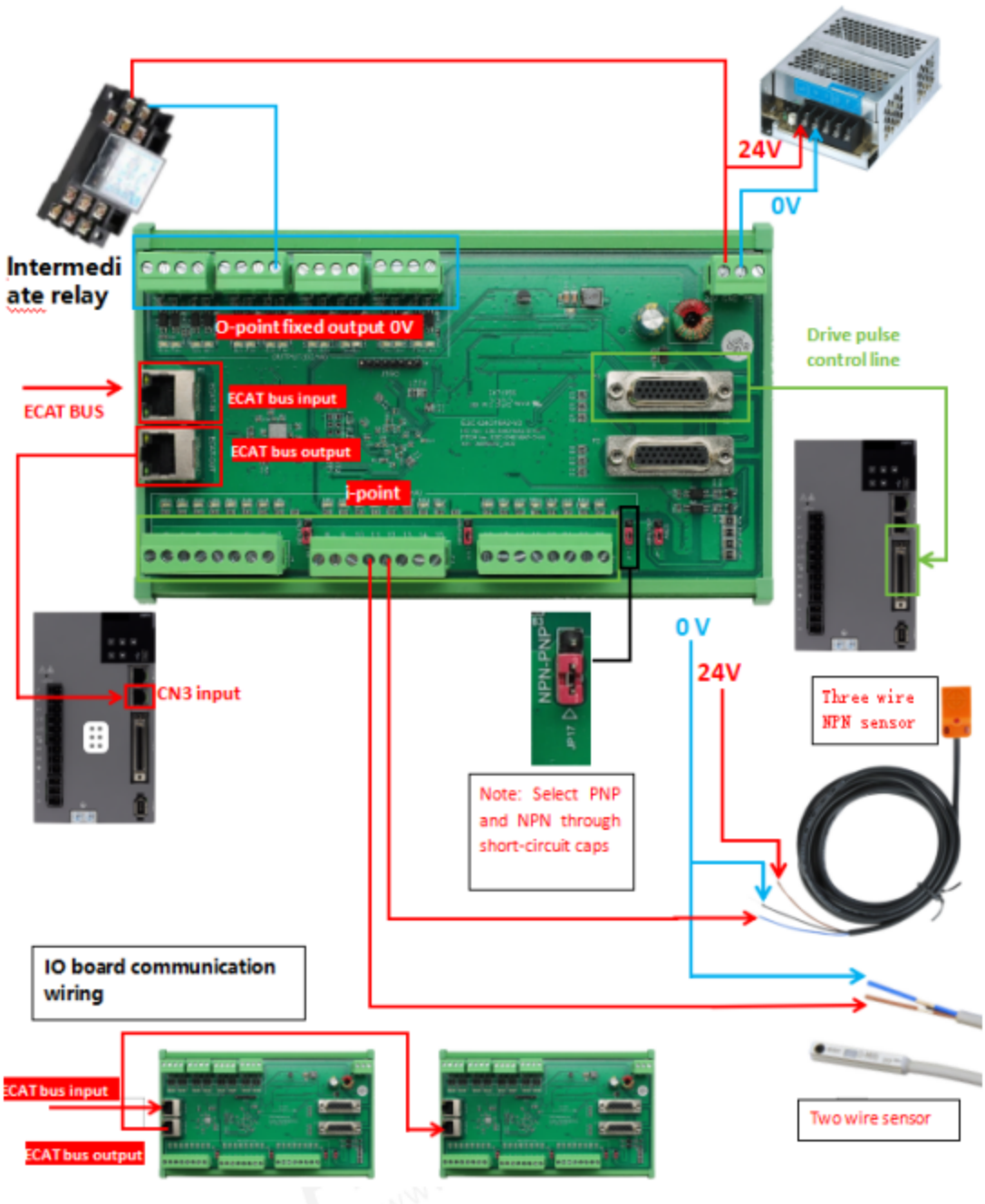
1. The definitions of each servo axis are only different in the IO points, while the rest are the same.
2. IO points in the table marked with "*" indicate IO points.
3. Pin 8 is for 5V power ground, voltage command ground, etc., while pin 26 is for IO interface and 24VGND. It is prohibited to have a common ground between pins 8 and 26.

The IO point definitions in each axis are as follows:

Pinout	P1	P2	P3	P4	P5	P6
23	I384	I386	I388	I390	I392	I394
17	I385	I387	I389	I391	I393	I395
24	O384	O386	O388	O390	O392	O394
18	O385	O387	O389	O391	O393	O395

6.7 IO board wiring

6.7.1 Wiring schematic



6.7.2 Definition of I/O point functions

Definition of I-point	Definition of O-point

I0	Emergency Stop	00	Cutting Fluid
I1	Foot Switch	01	Lubricating Oil
I2	External Start	02	Chuck Release
I3	External Pause	03	Chuck Clamp
I4	Lubrication System Abnormality	04	Work Light
I5	Hydraulic System Abnormality	05	Red Light
I6	Insufficient Lubrication Pressure	06	Green Light
I7	X-Axis Home Position	07	Yellow Light
I8	X-Axis Positive Limit Switch	08	Hydraulic Station Start
I9	X-Axis Negative Limit Switch	09	Spindle Brake Clamp
I10	Y-Axis Home Position	010	X-Axis Brake
I11	Y-Axis Positive Limit Switch	011	Y-Axis Brake
I12	Y-Axis Negative Limit Switch	012	Spindle Torque Reduction
I13	Z-Axis Home Position	013	Air Blowing On
I14	Z-Axis Positive Limit Switch	014	Z-Axis Brake
I15	Z-Axis Negative Limit Switch	015	Spindle Positioning at Stop
I16	Cutting Fluid Abnormality (Reserved)	016	Spindle Brake Release
I17	Reserved	017	Reserved
I18	Reserved	018	Tailstock Advance
I19	Safety Door Signal	019	Tailstock Cushion
I20	Feeder In Position	020	Tailstock Retract
I21	Feeder Low Material	021	Reserved
I22	Feeder Alarm	022	Material Feeder Extend
I23	Spindle Home Signal 1	023	Safety Door Closed
I24	Spindle Home Signal 2	024	Safety Door Open
I25	Tailstock Advance Knob	025	Chuck 2 Clamp
I26	Tailstock Retract Knob	026	Chuck 2 Release
I27	Spindle Brake In Position Signal	027	Feeder Start

I28	Spindle Clamp In Position Signal	O28	Reserved
I29	Spindle Brake Release In Position Signal	O29	Reserved

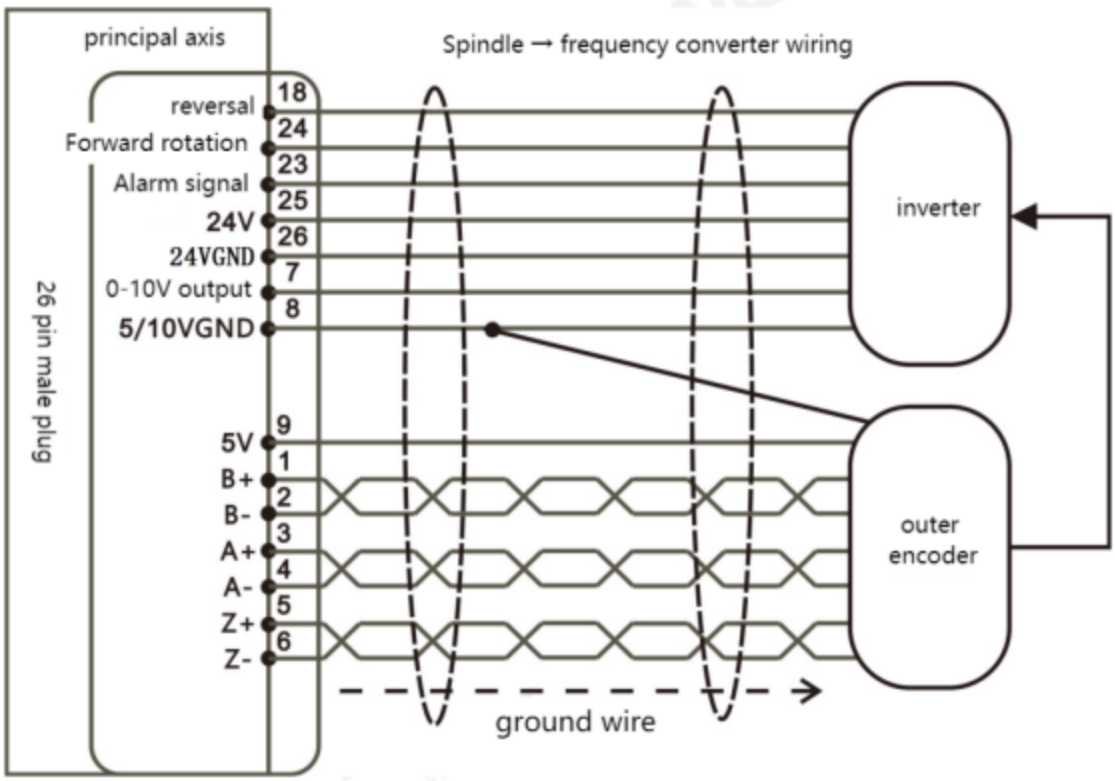
6.8 Spindle Control Wiring

Note: The spindle control wiring methods described below are all PNP.

6.8.1 Voltage Command Type

A. With Variable Frequency Drive

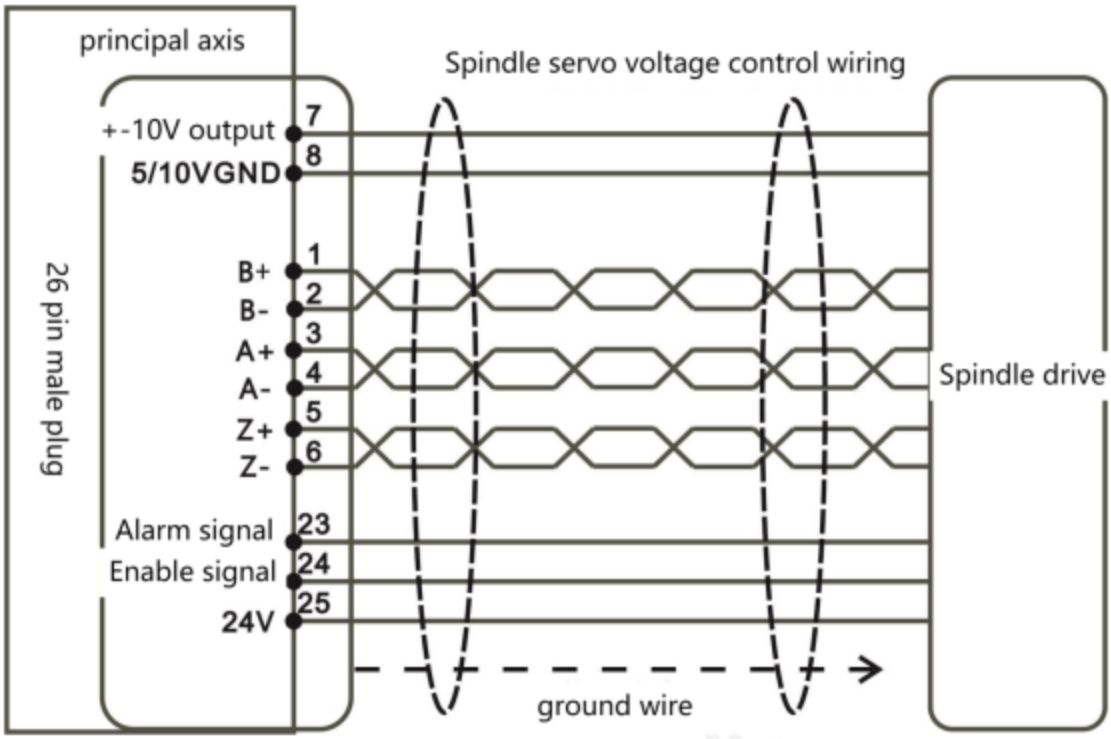
Voltage command control - Open-loop wiring:



Note：For detailed definitions of the driver pins, please refer to the corresponding driver manual.The system axis interface is a 26-pin female connector, so the spindle soldering plug should be a 26-pin male connector.

B. Coupled with servo drive

Spindle voltage command control:

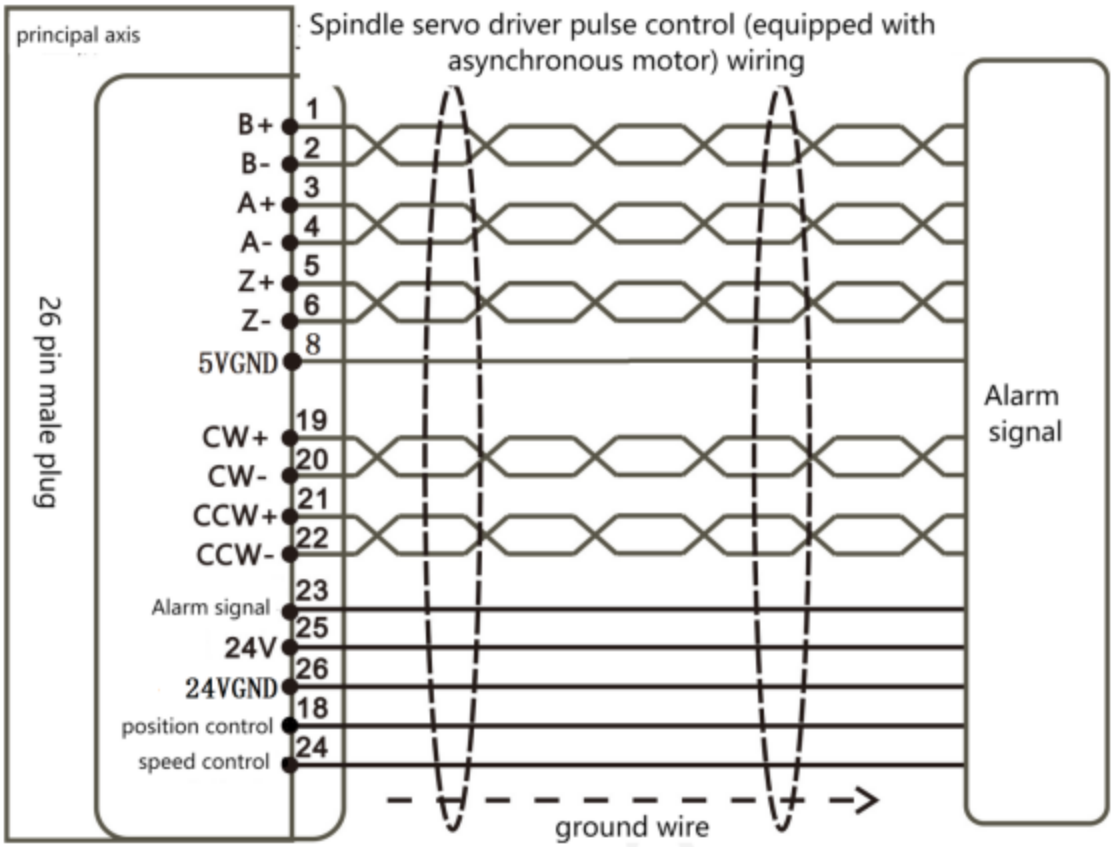


Note：Please refer to the corresponding driver's manual for detailed pin definitions. The system axis connector is a 26-pin female connector, so the spindle wiring plug should be a 26-pin male plug.

6.8.2 Pulse command type

A. Coupled with servo drive - asynchronous motor (e.g., super synchronous spindle)

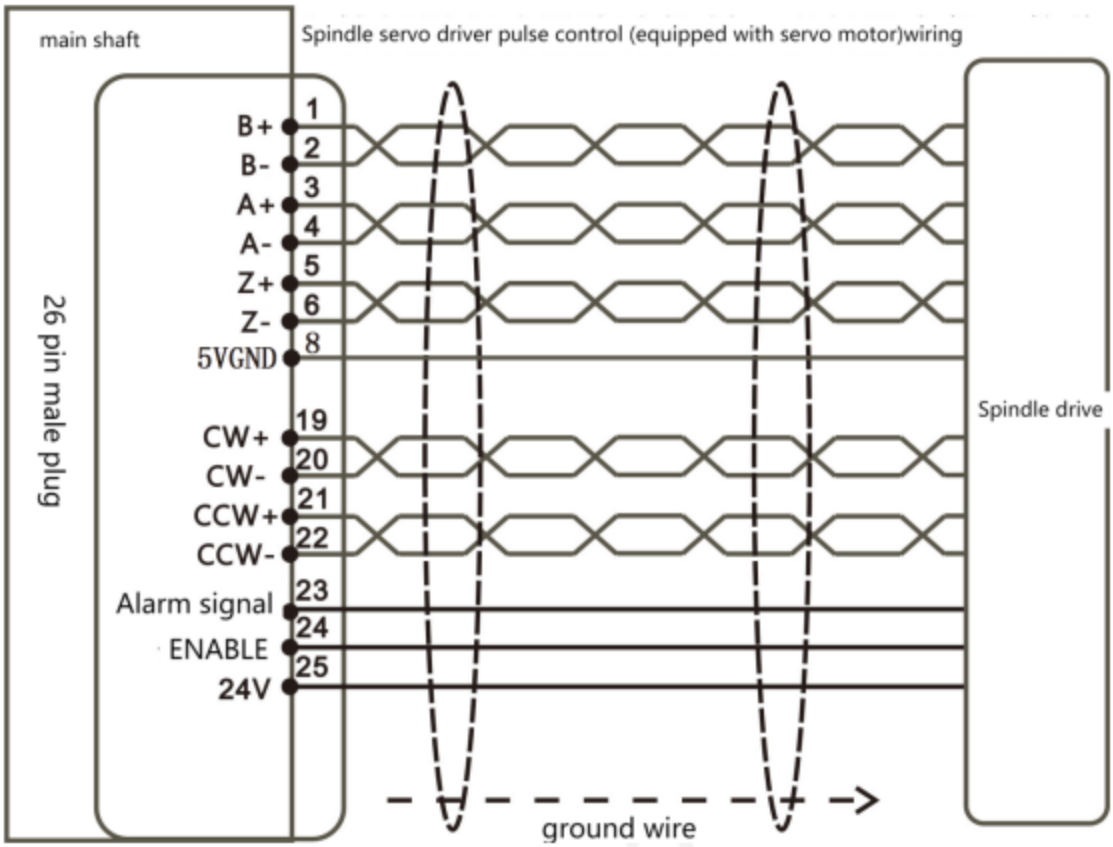
Spindle (servo) position mode, velocity mode wiring:



Note：Please refer to the corresponding drive manual for detailed definitions of the drive pins.The system axis connector is a 26-pin female connector, so the spindle cable connector should be a 26-pin male connector.

B. Pairing with Servo Drive - Servo Motor (e.g., Yasukawa Spindle)

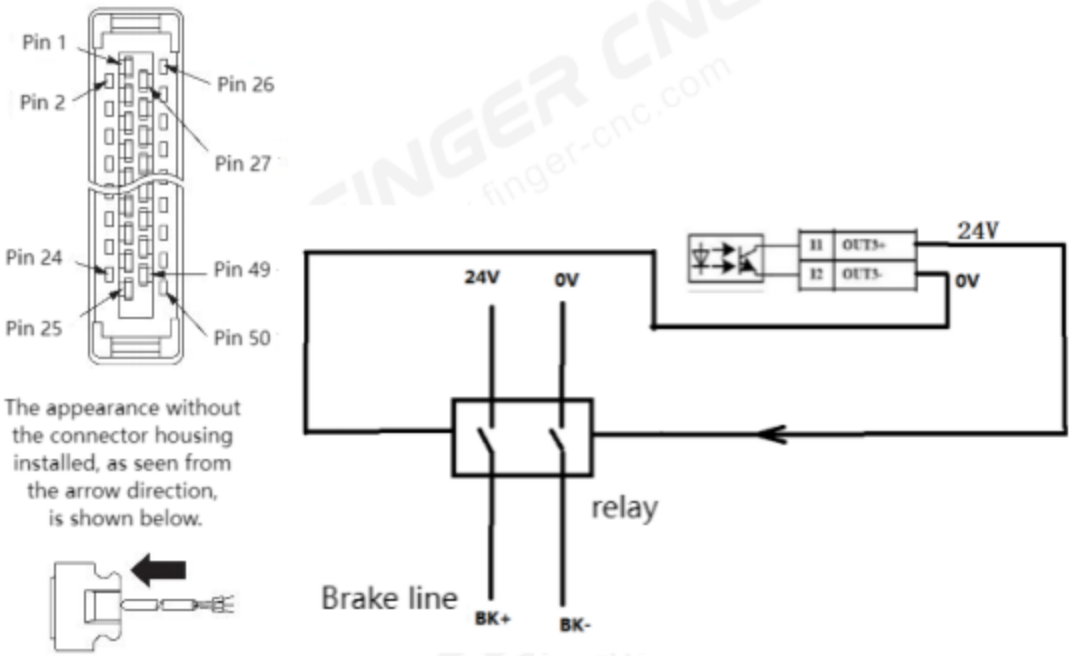
Spindle (Servo) Pulse Command Control:



Note：Please refer to the corresponding drive's user manual for detailed pin definitions. The system axis connector is a 26-pin female connector, so the spindle wire connector should be a 26-pin male connector.

6.9 Drive Motor Brake Wiring Diagram

6.9.1 I7 Drive Motor Brake Wiring Diagram



i7 driver

Parameter: Pn50F=X2XX, the third digit is 2, brake pins are 11 and 12.
Parameter: Pn50F=X6XX, the third digit is 6, brake pins are 36 and 37.

Note： The parameters PA50F for Axis A and PB50F for Axis B in the I7 dual-axis drive should not be set to the same values, otherwise, the motor brake output signal will be ineffective.

6.10 Wiring Precautions

- When wiring the machine, ensure that all wire ends are properly terminated or soldered.

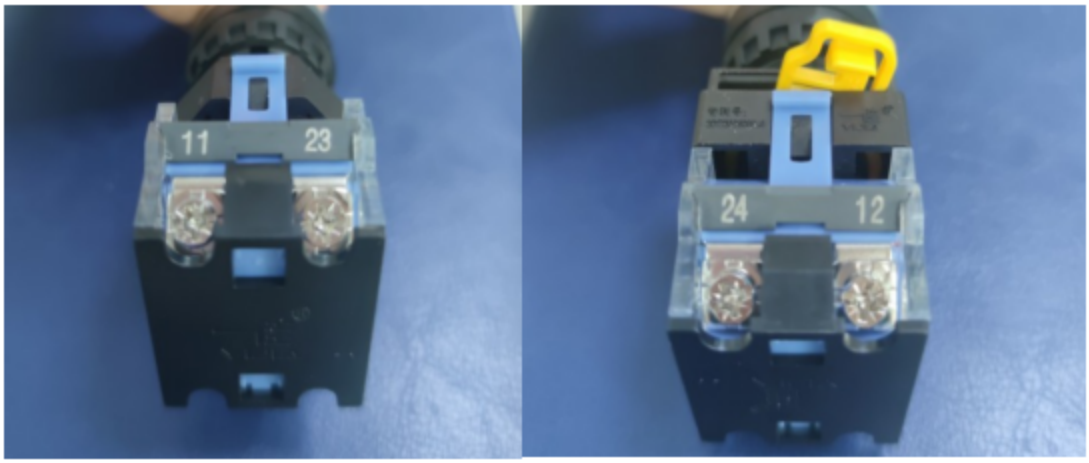
- If non-standard servo cables are used, it is important to measure the correctness of all pin connections before performing any powered tests. Incorrect wiring can lead to abnormal functionality of the controller's command output and may result in controller failure.
- When using an external 24V DC power supply for wiring, use products that have safety certification and built-in protection functions to prevent malfunctions due to wiring errors (recommended criteria: must meet the requirements of EN60950 & UL1950).
- If MECHATROLINK-III functionality is used, it is recommended to use CAT5e or CAT6 cables for network communication to ensure smooth connectivity and avoid noise interference.
- Grounding wire instructions:
- The size of grounding wire should comply with the technical standards of electrical equipment, and shorter grounding wires are preferred.
- The controller's grounding wire should not be connected to high-current loads such as welding machines or high-power motors. They must be separately grounded.

6.11 Three level rotary knob switch inspection

Turn the rotary switch to three different positions and use a multimeter to measure the on/off status of pins 11 and 12, as well as pins 23 and 24, respectively.

Multimeter usage: The picture below shows the continuity test position of the multimeter. When the two test probes of the multimeter measure a circuit that is conductive, the multimeter will give an audible feedback and light up a red light. (When testing for continuity, the probe positions do not distinguish between positive and negative (red and black).)

Picture of rotary switch foot position



Gear

Gear One Gear Two Gear Three

Truth table of on-off states for three-position rotary switch terminals			
	Gear One	Gear Two	Gear Three
11、12 Pin	0	1	1
23、24 Pin	0	0	1
Note：0 is open circuit、1 is Circuit continuity			

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