FINGER CNC

Open Designs Customize The Future

B Series
Dual-Channel Lathe
Operation Manual

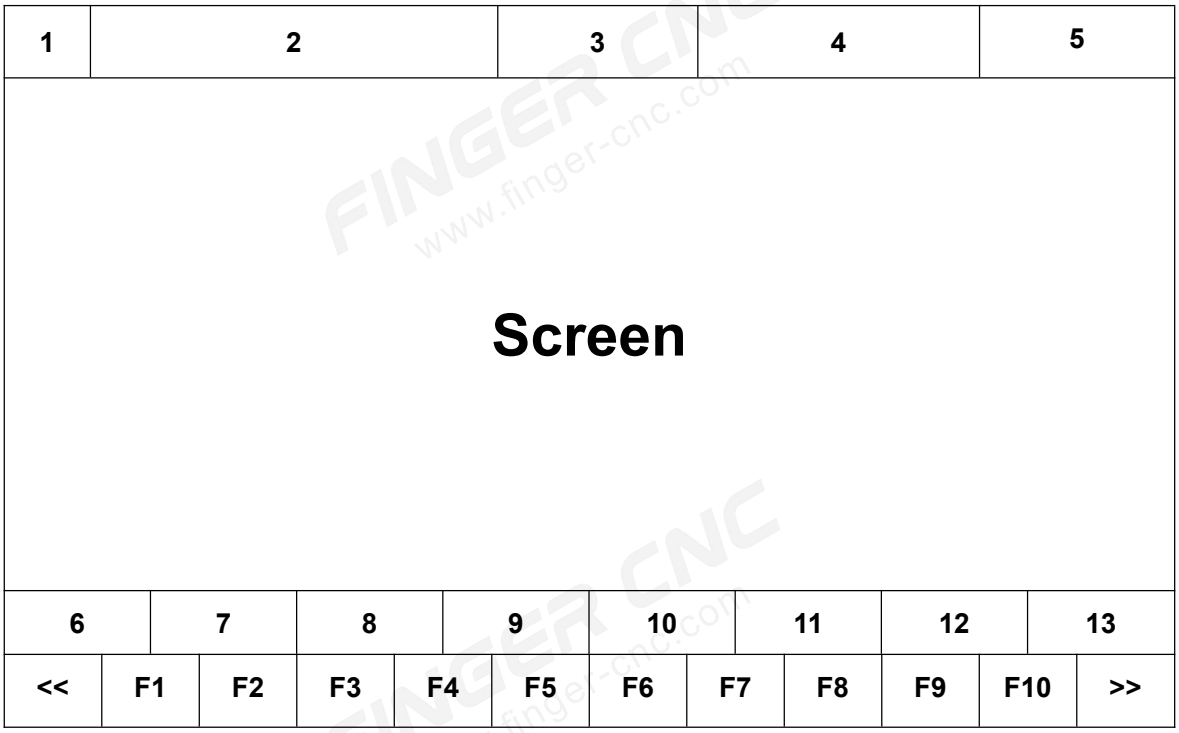
Version Number:F202410DLO-EN

Guangzhou Finger Technology Co., Ltd.

CONTENTS

1.Function Keys and System Overview

1.1Main Screen Introduction



- **Screen Element Description**

1、Current Workpiece Coordinate System

2、Current Channel Selection Status

3、Current Page Title

4、Current System Time

5、Current System Operation Permission Level

6、MPG Mode Axis Display for Channel 1

7、Channel 1 Operating Status

8、Channel 1 Operating Mode

9、Channel 1 Alarm Status

10、MPG Mode Axis Display for Channel 2

11、Channel 2 Operating Status

12、Channel 2 Operating Mode

13、Channel 2 Alarm Status

14、F1-F10 Function Keys (Depending on System Model)

1.2 Function Tree Diagram

1.3 Machine Position

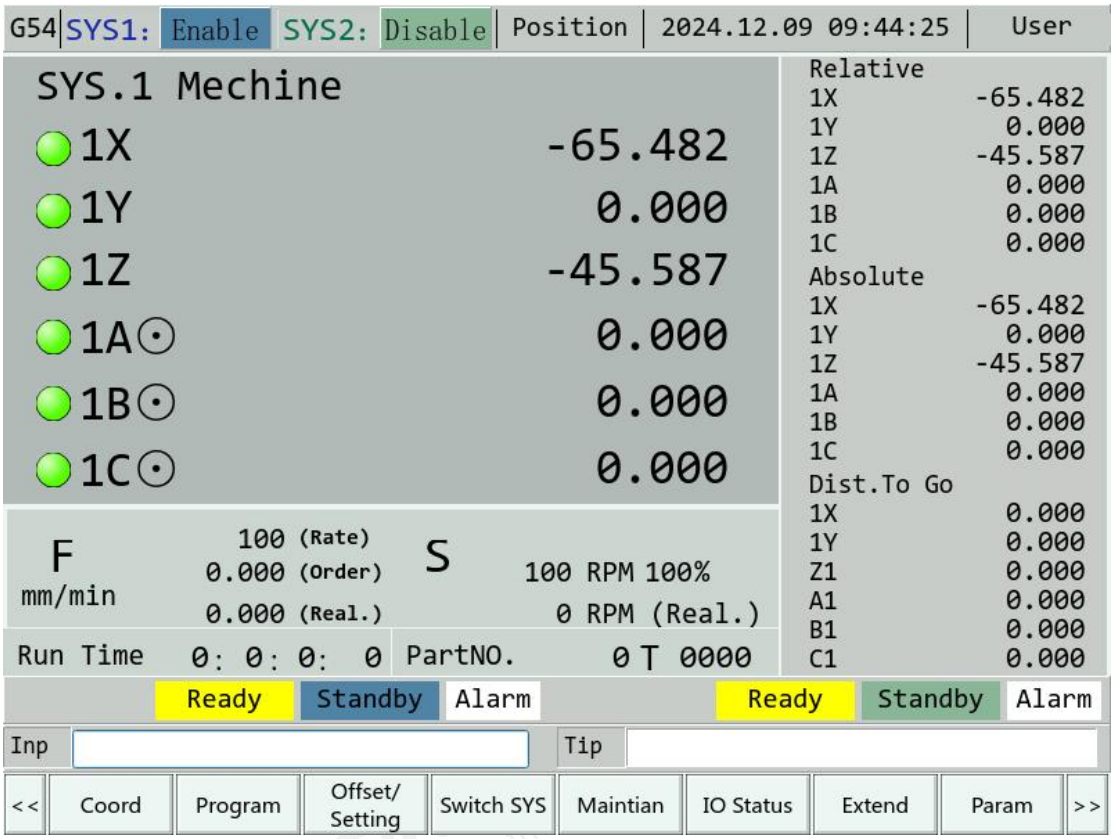
- **Operation Path**

In any page, press the system shortcut key [Position] to enter the "Machine Position" page. Use the [Channel Switch] key to toggle between channels (for eight-key and ten-key series).

- **Description**

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

1.3.1 Screen Description



1.3.1.1 Coordinate Display

1. This screen simultaneously displays four types of coordinates.
2. The [Coordinate Switch] key allows toggling the primary coordinate display type of

the current channel among machine coordinates, relative coordinates, program coordinates, and remaining distance.

1.3.1.2 F (Feed Rate)

1. Displays the user-defined feed rate for the current channel.
2. Displays the user-defined feed rate percentage multiplier (F-value scaling).

1.3.1.3 S (Spindle Speed)

1. Displays the user-defined spindle speed for the current channel.
2. Displays the actual feedback spindle speed of the current channel.
3. Displays the user-defined feed rate percentage multiplier (spindle scaling).

1.3.1.4 Processing Time

Displays the elapsed program execution time for the current channel.

1.3.1.5 Workpiece Count

Displays the number of parts processed by the current program on the current channel.

1.3.1.6 T (Tool Number and Tool Offset Number)

Displays the tool number and tool offset number used in the current channel's operation. The higher two digits represent the tool number, and the lower two digits represent the tool offset number. For gang tooling, the tool number remains constant, while the last two digits represent the real-time tool offset number.

1.4Coordinate Switching

1.4.1Coordinate Switch

- **Operation Path**

"Machine Position" page → F1 [Machine Coordinates] → F1 [Coordinate Switch]

- **Description**

This key toggles the machine coordinate screen. The display sequence of coordinate types cycles through machine coordinates, relative coordinates, program coordinates, and remaining distance for the current channel.

1.4.2 1/2 Coordinates

- **Operation Path**

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

- **Description**

1. Divides the relative coordinate value of the corresponding axis in the current channel by 2.
2. Used in conjunction with the "Relative Coordinate Setting" function, it quickly determines the midpoint coordinate between any two points.

- **Operation Method**

"Machine Position" page → F1 [Machine Coordinates] → Enter the axis to be set in the data input field → F2 [1/2 Coordinates].

- **Operation Example**

1. The current relative coordinate of the X-axis in the channel is 10.000.
2. Enter "X" in the data input field (no need to press the enter key after input).
3. Press F2 [1/2 Coordinates].
4. The relative coordinate of the X-axis will be updated to 5.000.

1.4.3 Relative Coordinate Setting

- **Operation Path**

"Machine Position" page → F1 [Machine Coordinates] → F3 [Relative Coordinate Setting].

- **Description**

Sets the relative coordinate of the corresponding axis to any value.

- **Operation Method**

Enter the axis symbol and coordinate value in the input field, then press F3 [Relative Coordinate Setting].

- **Operation Example**

1. The current relative coordinate of the X-axis in the channel is 10.000.
2. Enter "X0.000" in the input field.
3. Press F3 [Relative Coordinate Setting].
4. The relative coordinate of the X-axis will be updated to 0.000.

1.4.4 Relative Coordinate Reset

- **Operation Path**

"Machine Position" page → F1 [Machine Coordinates] → F4 [Relative Coordinate Reset].

- **Description**

Resets the relative coordinates of the corresponding axes to zero.

- **Operation Method**

Press F4 [Relative Coordinate Reset] to clear the values of all relative coordinates in the current channel.

- **Operation Example**

1. The current relative coordinates in the channel are:

◆ X-axis: 10.000

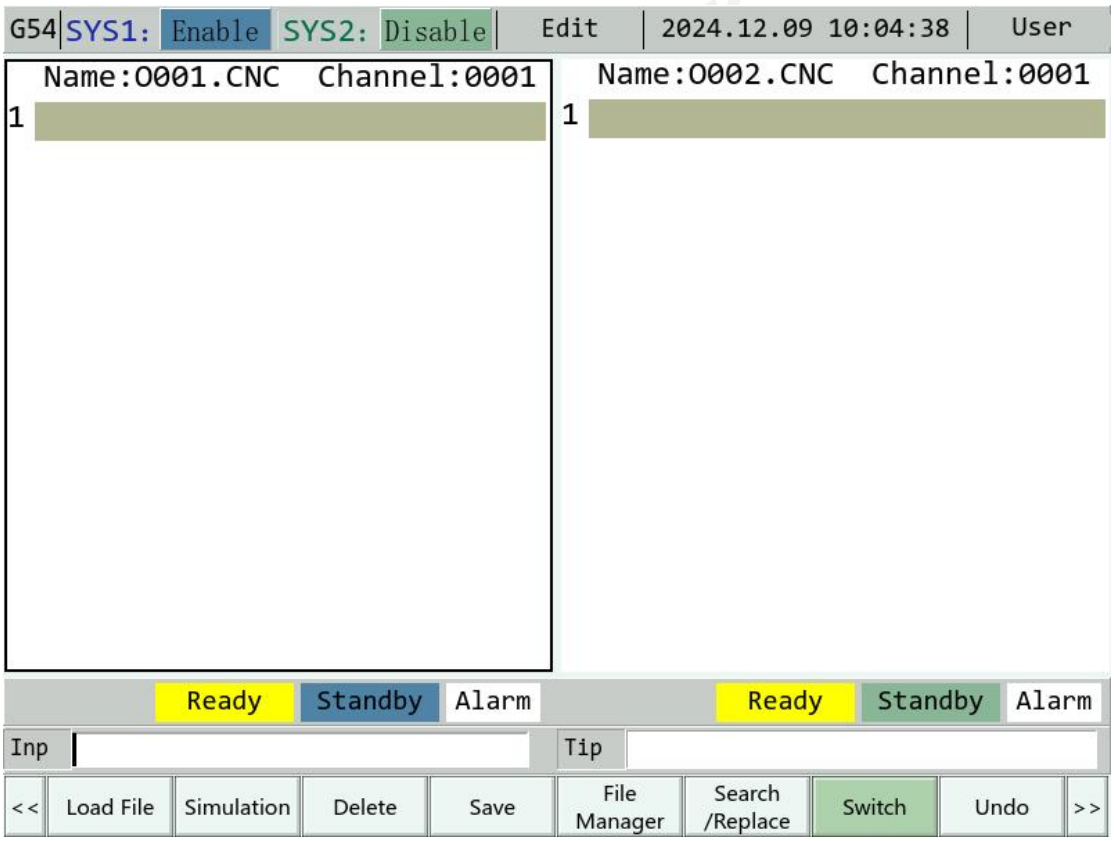
◆ Y-axis: 9.999

◆ Z-axis: 5.000

2. Press F4 [Relative Coordinate Reset].

3. The relative coordinates for X, Y, and Z axes will be updated to 0.000.

1.5 Program Editing



- **Operation Paths**

Path 1: Use the system panel shortcut key [Edit/Program Select] to switch to the "Program Editing" page.

Path 2: From the "Program Selection" page, press F5 [Load to Editor 1] or F6 [Load to Editor 2] to enter the "Program Editing" page.

Path 3: From the "Machine Position" page, press F2 [Program Edit] to access the "Program Editing" page.

- **Description**

The "Program Editing" page is used to edit machining programs. Typically, Editor 1 is for editing Channel 1 programs, and Editor 2 is for Channel 2 programs.

- **Operation Instructions**

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

2、Use [⩓] [⩔] for page up and page down.

3、Use [Line Start] [Line End] to quickly move the cursor to the beginning or end of the current line.

4、Use the system panel shortcut key [Edit/Program Select] to toggle between the "Program Editing" and "Program Selection" pages.

5、Use the [Switch Focus] button to switch between Editor 1 and Editor 2 for editing different programs.

1.5.1 Load Machining Program

- **Operation Path**

From the "Program Editing" page, press F1 [Load Machining Program].

- **Description**

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

The program in Editor 1 is loaded into Channel 1.

The program in Editor 2 is loaded into Channel 2.

- **Note**

If a program is currently running, this button will be disabled, and an error message will be displayed.

1.5.2 Graphic Simulation

- **Operation Path**

From the "Program Editing" page, press F2 [Graphic Simulation].

- **Description**

1、This button allows you to preview the machining path of the currently edited program.

2、It also provides a function to check for program errors.

3、The preset display range covers the maximum coordinate range of the entire program. In other words, the graphic simulation will show the largest possible image that can fit

within the drawing frame.

1.5.3 Delete Line

- **Operation Path**

From the "Program Editing" page, press F3 [Delete Line].

- **Description**

This button deletes the program content in the current editor, at the line where the cursor is located.

1.5.4 Save Program

- **Operation Path**

From the "Program Editing" page, press F4 [Save Program].

- **Description**

This button saves the current program content to a file, preventing data loss.

1.5.5 Program Selection

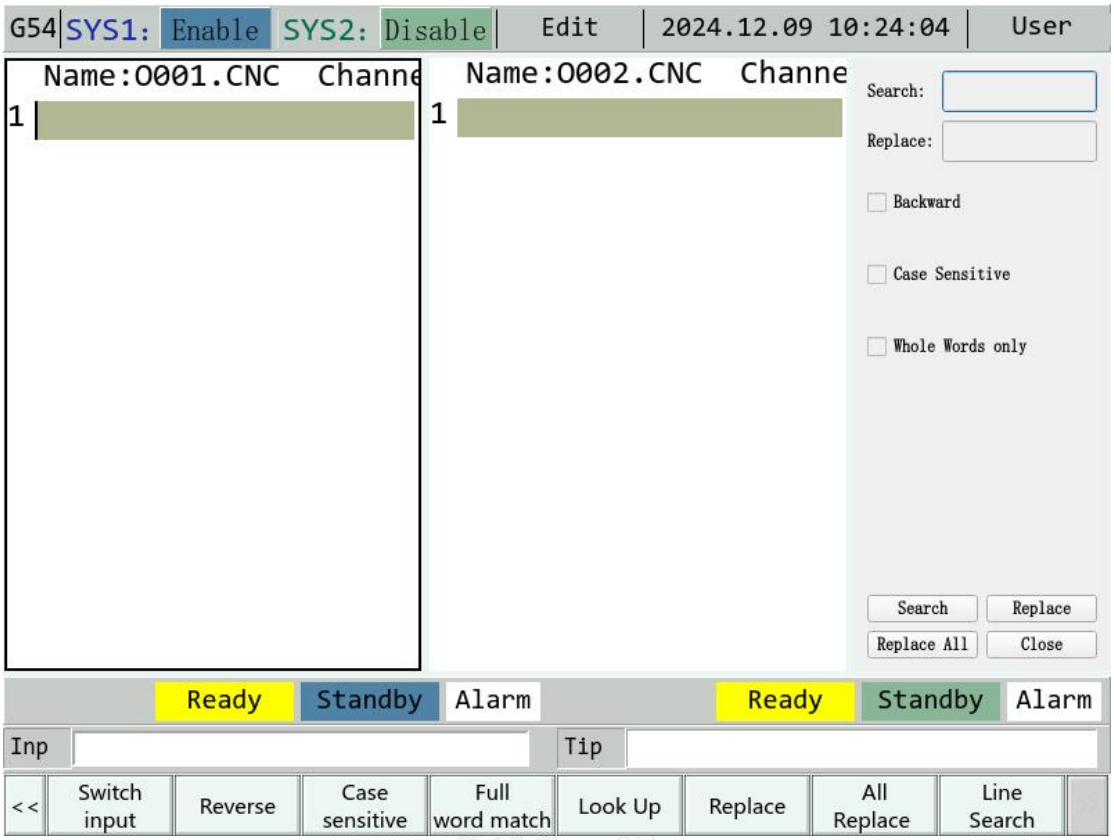
- **Operation Path**

From the "Program Editing" page, press F5 [File Management].

- **Description**

This function is used for managing machining documents. Please refer to section 1.6 for the "Program Selection" feature details.

1.5.6 Find/Replace



- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace].

- **Description**

1、This button allows you to search or replace characters in the program content of the current editor, where the cursor is focused.

2、After pressing [Find/Replace], a sidebar will appear along with toggle buttons. You can enter the character to search in the "Find" input box, and the replacement character in the "Replace" input box.

- **Operation Instructions**

Use the [Switch Input Box] button to toggle between the "Find" and "Replace" input boxes.

1.5.6.1 Close

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → [<<].

- **Description**

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

1.5.6.2 Switch Input Box

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F1 [Switch Input Box].

- **Description**

Switches the cursor between the "Find" and "Replace" input boxes.

1.5.6.3 Reverse

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F2 [Reverse].

- **Description**

When searching or replacing, this button allows you to choose the search direction.

- **Operation Instructions**

Press this button, and the "Reverse Search" option in the sidebar will be selected. When this option is enabled, pressing "Find" or "Replace" will search or replace characters above the current cursor position. If not selected, the default search direction will be downwards.

1.5.6.4 Case Sensitivity

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F3 [Case Sensitivity].

- **Description**

When searching or replacing, pressing this button will ensure that the letter case is respected when finding characters.

- **Operation Instructions**

Pressing this button will select the "Case Sensitive" option in the sidebar, activating the feature.

1.5.6.5 Whole Word Match

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F4 [Whole Word Match].

- **Description**

When searching or replacing, this button will ensure that only whole words matching the content in the "Find" input box are searched.

- **Operation Instructions**

Pressing this button will select the "Whole Word Match" option in the sidebar, activating the feature.

1.5.6.6 Find

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F5 [Find].

- **Description**

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

1.5.6.7 Replace

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F6 [Replace].

- **Description**

Searches for the characters entered in the "Find" input box within the code editor and replaces them with the characters in the "Replace" input box. (Each press of the button replaces one occurrence of the character.)

1.5.6.8 Replace All

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F7 [Replace All].

- **Description**

Searches for the characters entered in the "Find" input box within the code editor and replaces all occurrences with the characters in the "Replace" input box.

1.5.6.9 Line Number Search

- **Operation Path**

From the "Program Editing" page, press F6 [Find/Replace] → F8 [Line Number Search].

- **Description**

Searches for the line number of the current editor where the cursor is located and jumps the cursor to display it.

1.5.7Switch Focus

- **Operation Path**

From the "Program Editing" page, press F7 [Switch Focus].

- **Description**

Switches the focus between Editor 1 and Editor 2.

1.5.8 Undo

- **Operation Path**

From the "Program Editing" page, press F8 [Undo].

- **Description**

Undoes the last editing operation in the current editor, such as line deletion, copy, paste, etc.

1.5.9 Program Copy/Paste

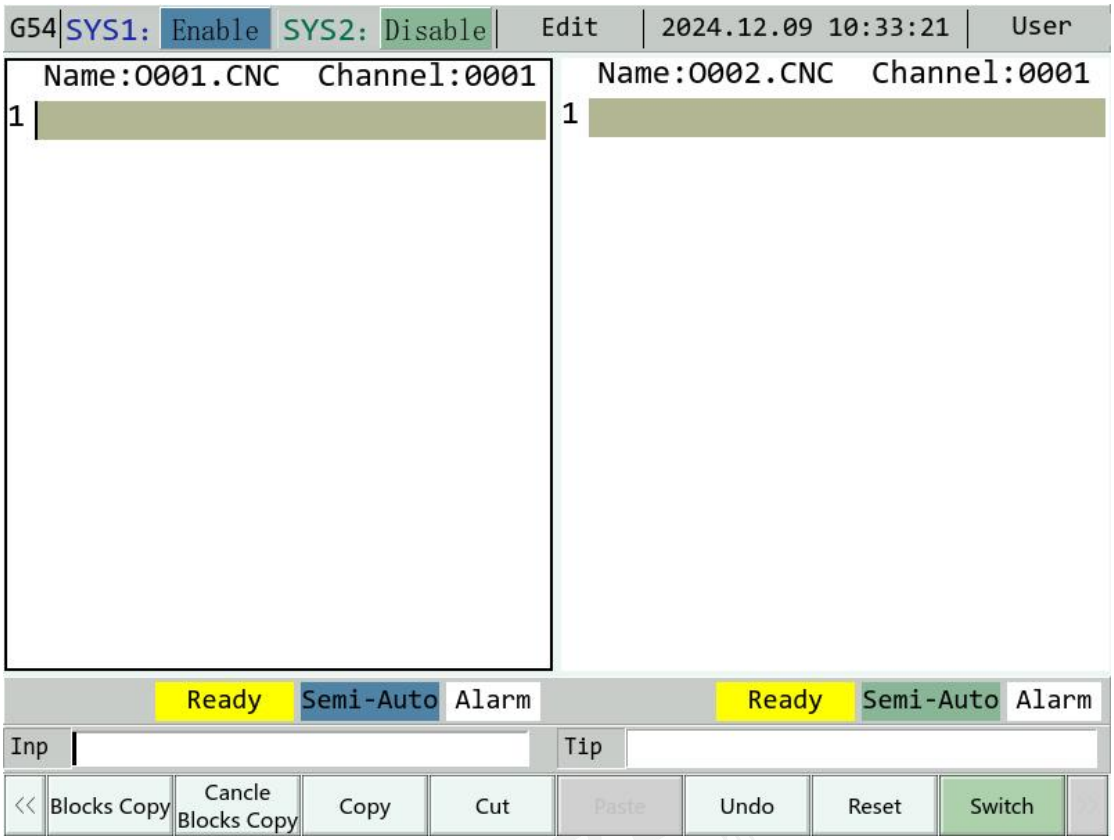
- **Operation Path**

From the "Program Editing" page, press [>>] → F1 [Zoom In Editor].

- **Description**

Performs copy, cut, paste, undo, and redo operations on the character content of the current editor where the cursor is located.

1.5.9.1 Block Selection



- **Operation Path**

From the "Program Editing" page, press [>>] → F1 [Block Selection].

- **Description**

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

- **Operation Instructions**

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

1.5.9.2 Cancel Block Selection

- **Operation Path**

From the "Program Editing" page, press [>>] → F2 [Cancel Block Selection].

- **Description**

Disables block selection functionality.

- **Operation Instructions**

Press the [Cancel Block Selection] button to disable block selection. Any previously selected block will be canceled, and the "Copy" and "Cut" buttons will become inactive.

1.5.9.3 Copy

- **Operation Path**

From the "Program Editing" page, press [>>] → F3 [Copy].

- **Description**

Copies the content of the selected block to be used with the "Paste" function.

1.5.9.4 Cut

- **Operation Path**

From the "Program Editing" page, press [>>] → F4 [Cut].

- **Description**

Cuts the selected block of content, which will be removed and can be used with the "Paste" function.

1.5.9.5 Paste

- **Operation Path**

From the "Program Editing" page, press [>>] → F5 [Paste].

- **Description**

Pastes the content copied or cut from another location to the current cursor position in the

program.

1.5.9.6 Undo

- **Operation Path**

From the "Program Editing" page, press [>>] → F6 [Undo].

- **Description**

Undoes the last editing operation. This can be repeated to undo multiple actions, all the way back to the first step of the program editing.

1.5.9.7 Redo

- **Operation Path**

From the "Program Editing" page, press [>>] → F7 [Redo].

- **Description**

Restores the last undone operation. This can be repeated to redo multiple actions, all the way to the last step of the program editing.

1.5.10 Zoom In Editor

- **Operation Path**

From the "Program Editing" page, press F8 [Zoom In Editor].

- **Description**

Zooms in on the currently focused editor, displaying only that editor on the screen.

1.5.11 Restart from Current Line

- **Operation Path**

From the "Program Editing" page, press [>>] → F7 [Restart from Current Line].

- **Description**

1. Allows selection of the starting point for restarting the program;

2. This function can only be used in automatic mode;
3. After activation, the system will automatically jump to the "Processing Monitoring" page.

- **Notes**

1、The program being edited must be the current processing program.

2、The system will restart from the line where the cursor is located in the focused editor.

1.5.12 Input Method Switching

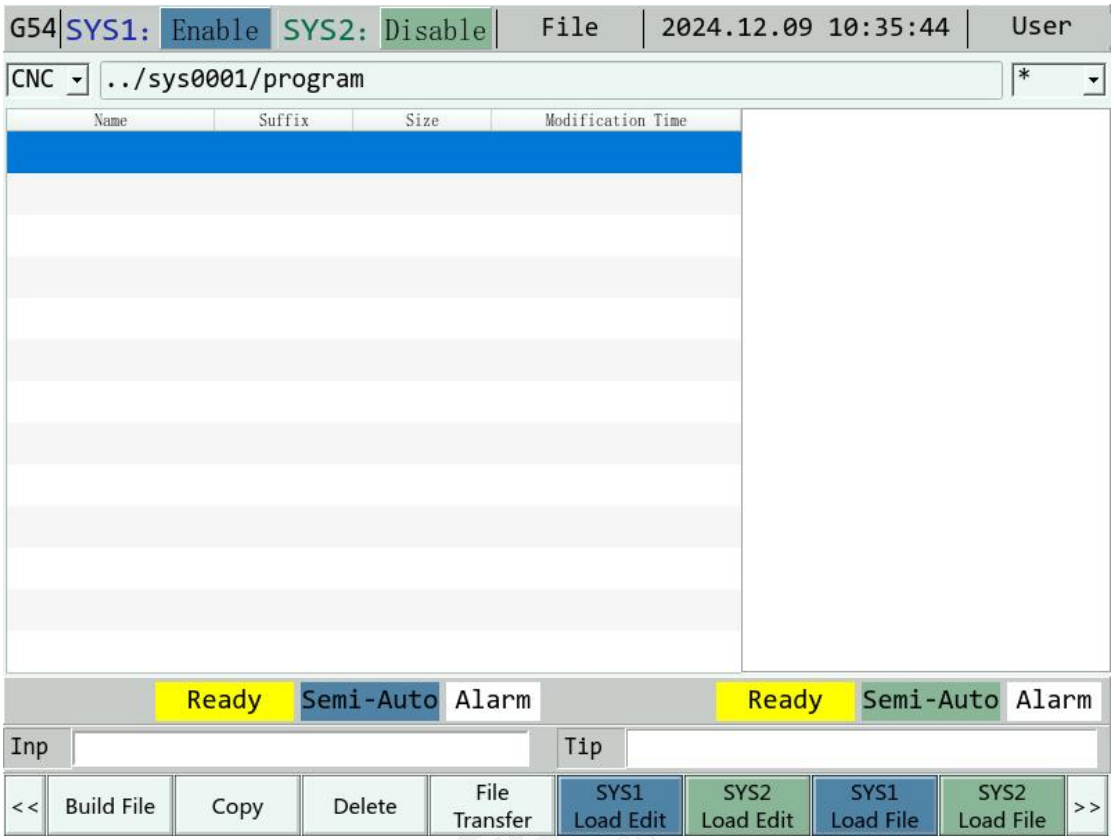
- **Operation Path**

From the "Program Editing" page, press [>>] → [>>] → F6 [Chinese Input], F7 [English Uppercase], or F8 [English Lowercase].

- **Description**

Switches between input methods in the program editing page, allowing the input of Chinese characters for comments, making it easier for users to understand the program.

1.6 Program Selection



- **Operation Path**

1. Path 1: Use the system panel shortcut button [Edit/Program Selection] to switch to the "Program Selection" page.
2. Path 2: From the "Program Editing" page, press F4 [File Management] to go to the "Program Selection" page.

- **Description**

Used for managing processing program files, including creating, copying, deleting, and transferring files.

- **Operation Instructions**

1、Use the arrow keys (↑, ↓) to move the cursor and select a program file.

2、Use the [≪] and [≫] keys to scroll the page up and down.

3、Use the system panel buttons F5 [Load to Editor 1], F6 [Load to Editor 2], F7 [Load SYS1 Program], or F8 [Load SYS2 Program] to load the program file into the editor or to load it into the corresponding channel for processing.

1.6.1 Create New File

- **Operation Path**

From the "Program Selection" page, press F1 [Create New File].

- **Description**

Used to create a new processing program file, defaulting to a CNC file.

- **Operation Instructions**

1、Press the [Create New File] button, a dialog box will pop up prompting for the file name.

2、Press F1 [Confirm] to complete the file creation.

- **Notes**

If the file name entered does not have a file extension, the default extension ".CNC" will be added.

1.6.2 Copy File

- **Operation Path**

From the "Program Selection" page, press F2 [Copy File].

- **Description**

Used to copy and rename the current file under focus, effectively backing up the program file.

- **Operation Instructions**

1、Press the [Copy File] button, a dialog box will appear asking for the new file name. After entering the name, the file will be copied.

2、If the new file name is the same as an existing file, a prompt will ask if you want to overwrite the file.

3、If the newly created file does not have a file extension, it will default to a ".CNC" file.

1.6.3 Delete File

- **Operation Path**

From the "Program Selection" page, press F3 [Delete].

- **Description**

This button is used to delete a program document.

- **Operation Description**

1、Use the arrow keys (↑, ↓) to move the cursor and select the program file to delete. Press F3 [Delete], a confirmation dialog will appear.

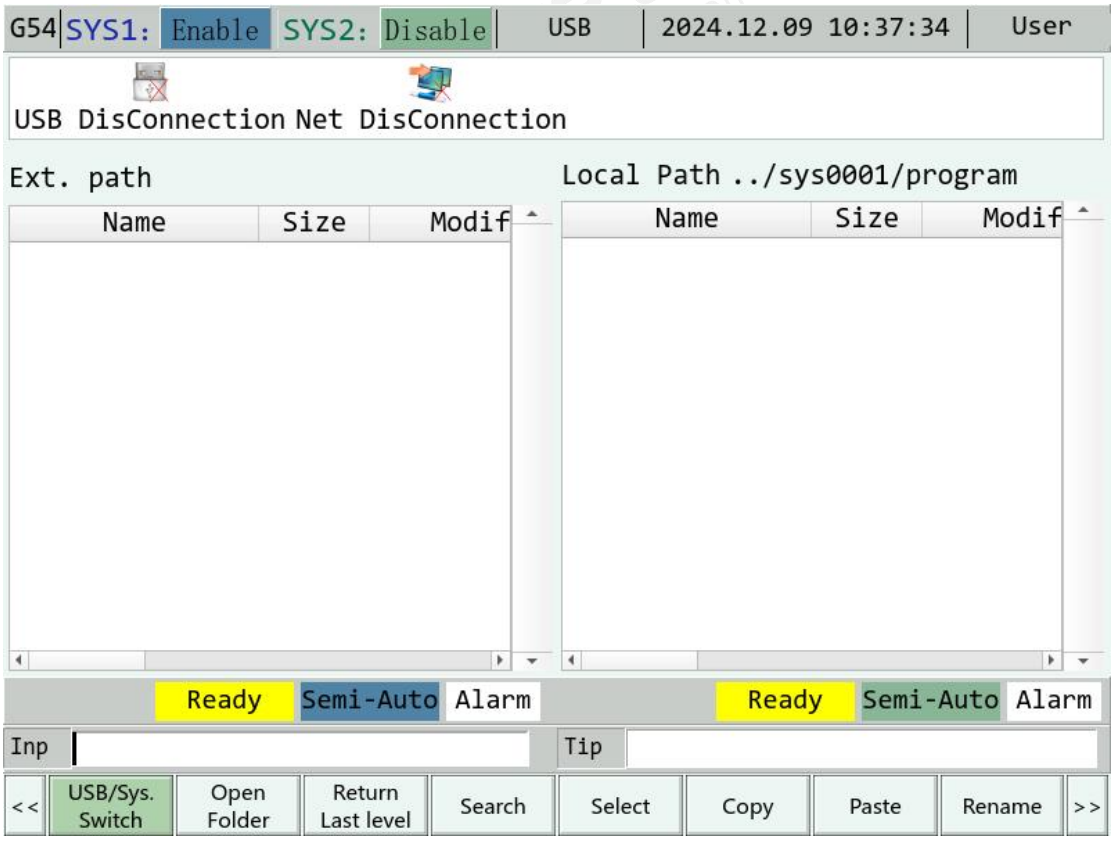
2、Press F1 [Confirm Delete] to delete the file and automatically close the confirmation dialog.

3、Press F2 [Cancel] to cancel the deletion.

- **Notes**

You cannot delete a file that is currently being edited or loaded for processing. The deletion will fail if the selected file is in use.

1.6.4 File Transfer



- **Operation Path**

From the "Program Selection" page, press F4 [File Transfer].

- **Description**

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

1.6.4.1 USB/System File Toggle

- **Operation Path**

From the "File Transfer" page, press F1 [USB/System File Toggle].

- **Description**

Switches the focus between the controller's file selector and the external device's file selector. Cursor movement is only possible in the file selector currently in focus.

1.6.4.2 Open Folder

- **Operation Path**

From the "File Transfer" page, press F2 [Open Folder].

- **Description**

Opens the folder currently selected by the cursor.

1.6.4.3 Go Back

- **Operation Path**

From the "File Transfer" page, press F3 [Go Back].

- **Description**

Navigates to the parent directory of the current folder.

1.6.4.4 Search

- **Operation Path**

From the "File Transfer" page, press F2 [Search].

- **Description**

Used to locate files.

- **Operation Instructions**

1、Press the [Search] button to open a dialog box and input the file name.

2、Use F1 [Find Next] or F2 [Find Previous] to search.

3、Press F3 [Cancel] to abandon the search.

1.6.4.5 Select

- **Operation Path**

From the "File Transfer" page, press F4 [Select].

- **Description**

Selects the file currently under the cursor, enabling operations like Copy and Paste.

1.6.4.6 Copy

- **Operation Path**

From the "File Transfer" page, press F5 [Copy].

- **Description**

Copies the selected file.

1.6.4.7 Paste

- **Operation Path**

From the "File Transfer" page, press F7 [Paste].

- **Description**

Pastes the copied file.

1.6.4.8 Rename

- **Operation Path**

From the "File Transfer" page, press F7 [Rename].

- **Description**

Renames the selected file.

1.6.4.9 Create Folder

- **Operation Path**

From the "File Transfer" page, press [>>] → F1 [Create Folder].

- **Description**

Creates a new folder in the current directory.

1.6.4.10 Switch External Device

- **Operation Path**

From the "File Transfer" page, press [>>] → F2 [Switch External Device].

- **Description**

Switches between external devices, such as USB drives and computers.

1.6.4.11 Select External Device

- **Operation Path**

From the "File Transfer" page, press [>>] → F3 [Select Device].

- **Description**

Used to select an external device.

1.6.4.12 Select All

- **Operation Path**

From the "File Transfer" page, press [>>] → F4 [Select All].

- **Description**

Selects all files in the current directory.

1.6.4.13 Deselect All

- **Operation Path**

From the "File Transfer" page, press [>>] → F5 [Deselect All].

- **Description**

Deselects all files in the current directory.

1.6.4.14 Delete

- **Operation Path**

From the "File Transfer" page, press [>>] → F6 [Delete].

- **Description**

Deletes all selected files in the current directory.

1.6.5 Edit

- **Operation Path**

From the "Program Selection" page, press F5 [Load to Editor 1] or F6 [Load to Editor 2].

- **Description**

Used for editing processing programs. Refer to 1.5 [Program Editing] for details.

1.6.6Load for Processing

- **Operation Path**

From the "Program Selection" page, press F7 [Load SYS1 Program] or F8 [Load SYS2 Program].

- **Description**

Loads the program file under the cursor into the designated channel for processing and switches to the corresponding "Processing Monitor" page.

- **Notes**

If the program is currently running, this operation is invalid, and an error will be displayed.

1.6.7 Create Folder

- **Operation Path**

From the "Program Selection" page, press [>>] → F1 [Create Folder].

- **Description**

Creates a new folder in the current directory.

- **Operation Instructions**

1、Press F1 [Create Folder], and a dialog box will appear.

2、Enter the folder name and press F1 [Confirm] to complete the creation.

3、Press F2 [Cancel] to abort and close the dialog box.

1.6.8 Go Back

- **Operation Path**

From the "Program Selection" page, press [>>] → F2 [Go Back].

- **Description**

Navigates to the parent directory of the current folder.

1.6.9 Delete Folder

- **Operation Path**

From the "Program Selection" page, press [>>] → F3 [Delete Folder].

- **Description**

Deletes the folder currently under the cursor.

1.6.10 Rename

- **Operation Path**

From the "Program Selection" page, press F4 [Rename].

- **Description**

Renames the selected processing program file.

- **Operation Instructions**

1、Use the arrow keys [↑】【↓] to move the cursor to the program file to be renamed, then press F4 [Rename].A dialog box will appear; input the new file name.

2、Press F1 [Confirm] to complete the renaming and close the input dialog box.

3、Press F2 [Cancel] to cancel renaming and close the dialog box.

1.6.11 Search

- **Operation Path**

From the "Program Selection" page, press [>>] → F7 [Search].

- **Description**

Searches for a program file and moves the cursor to the line where the file is located.

1.6.12 Macro Program Files/Processing Program Files

- **Operation Path**

From the "Program Selection" page, press [>>] → F8 [Macro Program Files].

- **Description**

1、Pressing F8 [Macro Program Files] enters the macro program file page, where the F8 key changes to Processing Program Files.

2、On the macro program file page, you can view macro program files (with the .MACRO extension) stored in the system's memory. New files are created in CNC format (.CNC).

3、Pressing Processing Program Files displays the processing program files (with the .CNC extension) stored in the system's memory. The F8 key changes back to Macro Program Files.

4、By default, macro program files cannot be modified, as doing so may cause functional errors.

1.7 Tool Compensation/Settings

- **Operation Path**

1、Path 1: Use the system panel shortcut key [Tool Compensation/Settings] to switch to the "Wear Compensation" page.

2、Path 2: From the "Machine Position" page, press F3 [Offset/Settings] to switch to the "Wear Compensation" page.

- **Description**

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

- **Operation Instructions**

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

2、Use [⩓】【⩔] to scroll the page up and down.

3、Use [Home] or [End] to quickly move the cursor to the beginning or end of the row.

4、In the tool compensation table, entering values directly in the input field represents setting the length compensation/wear compensation values. Entering the corresponding axis name along with the value represents tool setting operations.

5、Use [SYS1 Length Compensation], [SYS2 Length Compensation], [SYS1 Wear Compensation], or [SYS2 Wear Compensation] to switch between channels 1 and 2 for length and wear compensation.

6、Use [Workpiece Shift] or [Work Coordinate System] to jump to the coordinate system interface.

7、Use [Absolute Input] or [Incremental Input] to select the type of input values.

1.7.1 Tool Setting

- **Description**

This function is used to set the tool's tool-tip dimensions. The actual tool-tip size is calculated as the tool-tip radius plus the tool wear radius.

- **Parameter Descriptions**

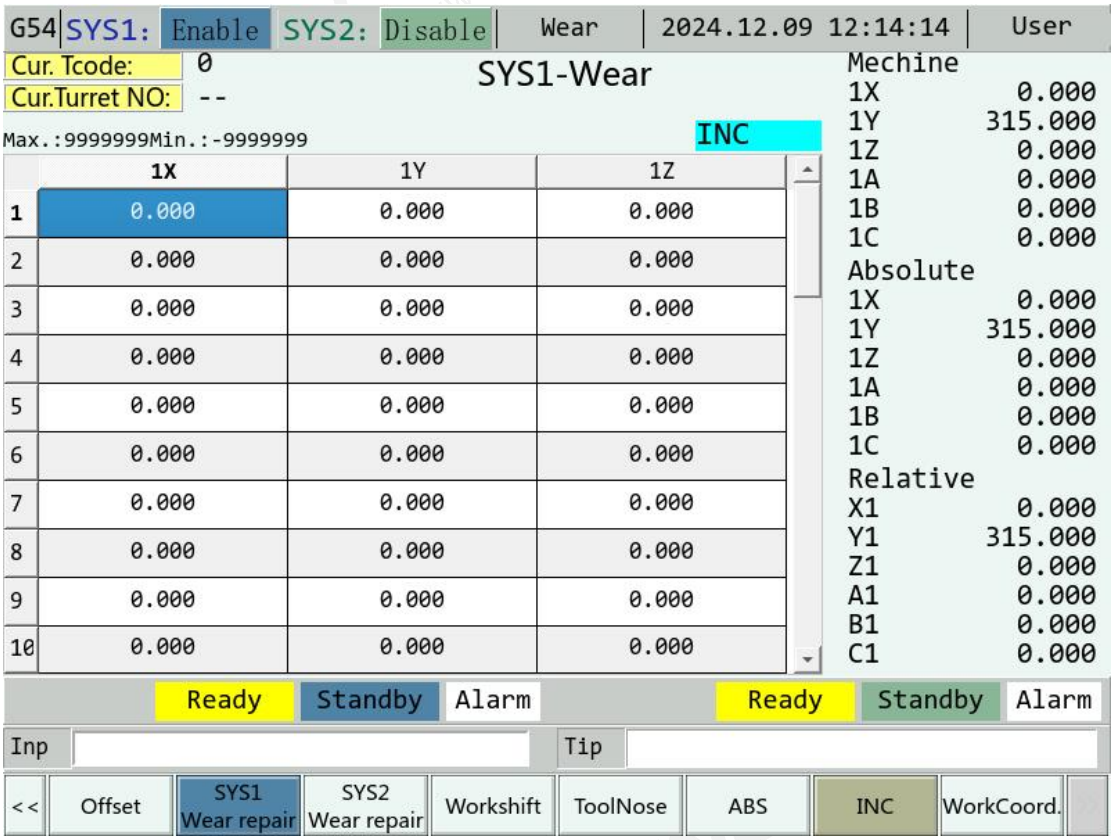
1、Tool-tip Direction: Defines the direction of the tool-tip during processing.

2、Tool-tip Radius: The G41/G42 tool-tip radius compensation.

3、Tool-tip Wear: The adjustment for the smaller tool-tip radius during G41/G42 compensation.

4、Tool-tip Directions: Eight tool-tip directions are provided by Yida, which can be selected according to the actual processing situation. Please refer to the Yida Programming Manual - G41/G42 for details.

1.7.2 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: From the "Machine Position" page → F3 [Offset/Setting] → "Wear Compensation" page.

- **Description**

1、Set the tool wear compensation value.

2、Tool compensation = Tool length + Tool wear.

- **Parameter Description**

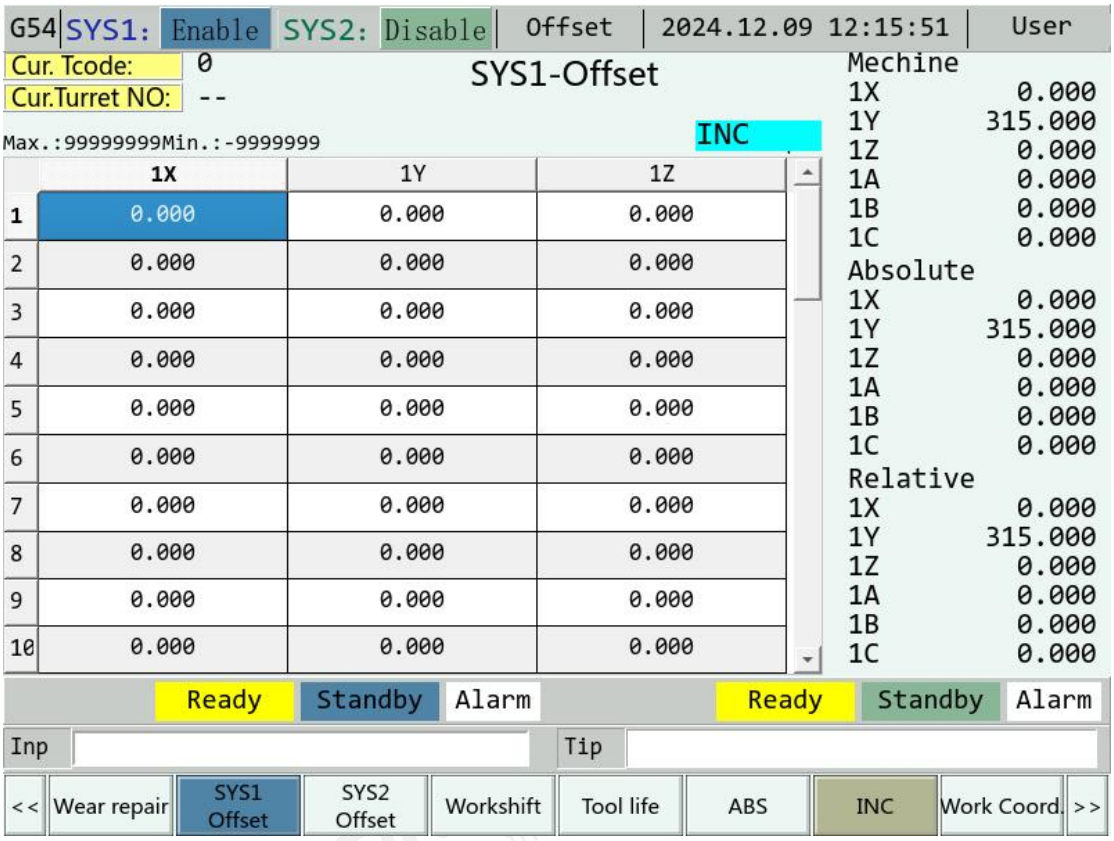
Set the maximum and minimum values for wear compensation.

- **Notes**

1、When setting the tool length compensation, the corresponding axis wear compensation value will be automatically reset to zero.

2、If the wear compensation value is modified during machining and has not yet been executed to the corresponding T-code, the modified value will take effect immediately. If the T-code has already been executed, the modified value will take effect the next time the program executes the T-code.

1.7.3 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: From the "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.

- **Notes**

1、When setting the tool length compensation, the corresponding axis wear compensation value will be automatically reset to zero.

2、Length compensation values cannot be modified during machining.

1.7.4 SYS1

- **Operation Path**

From the "Wear Compensation" or "Length Compensation" page → F2 [SYS1].

- **Description**

Press this key to switch the length compensation/wear compensation for Channel 1 (SYS1).

1.7.5 SYS2

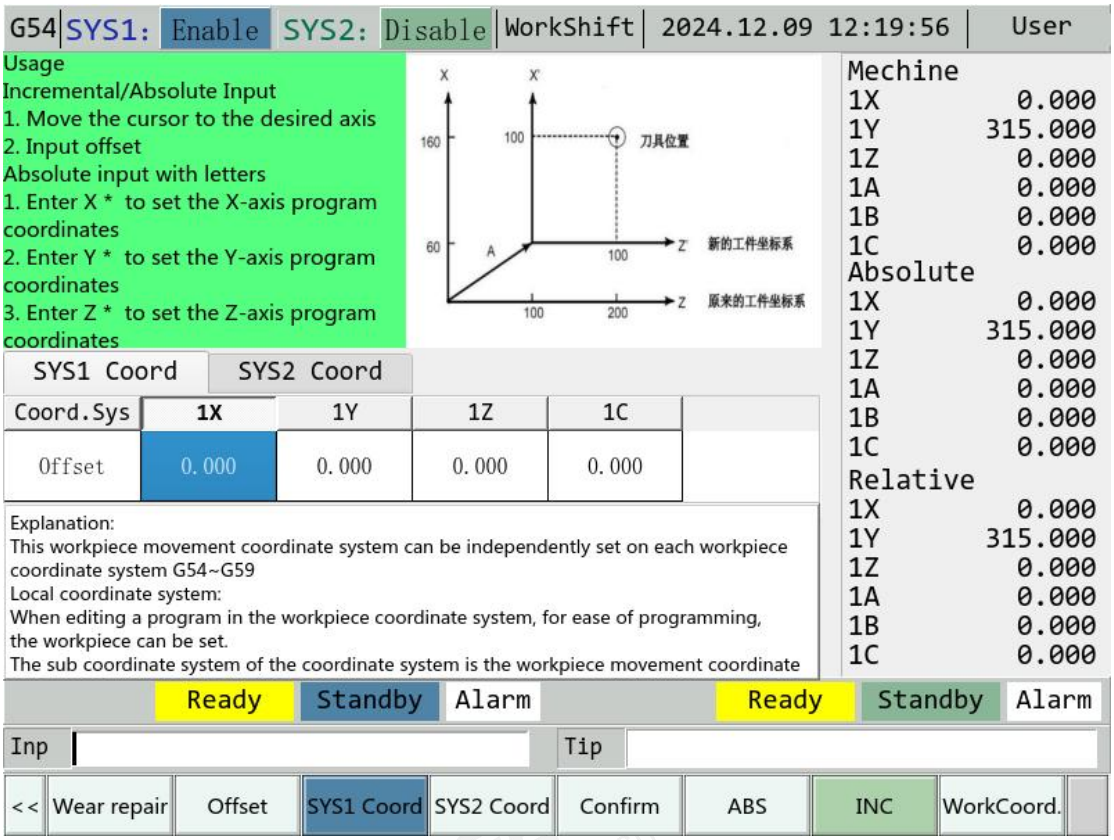
- **Operation Path**

From the "Wear Compensation" or "Length Compensation" page → F3 [SYS2].

- **Description**

Press this key to switch the length compensation/wear compensation for Channel 2 (SYS2).

1.7.6 Workpiece Shift



- **Operation Path**

「Wear Compensation」or 「Length Compensation」page → F4【Workpiece Shift】.

- **Description**

1、Press this button to open the 「Workpiece Shift」 page.

2、Workpiece shift is an external offset coordinate system that allows global offsets of programmed coordinates.

1.7.7Absolute Input

- **Operation Path**

「Wear Compensation」or 「Length Compensation」page → F6【=Input (Absolute Input)】.

- **Description**

This button sets the input method for the table's input area. When the button turns green, 「=Input (Absolute Input)」indicates that the input is an absolute value.

1.7.8 Incremental Input

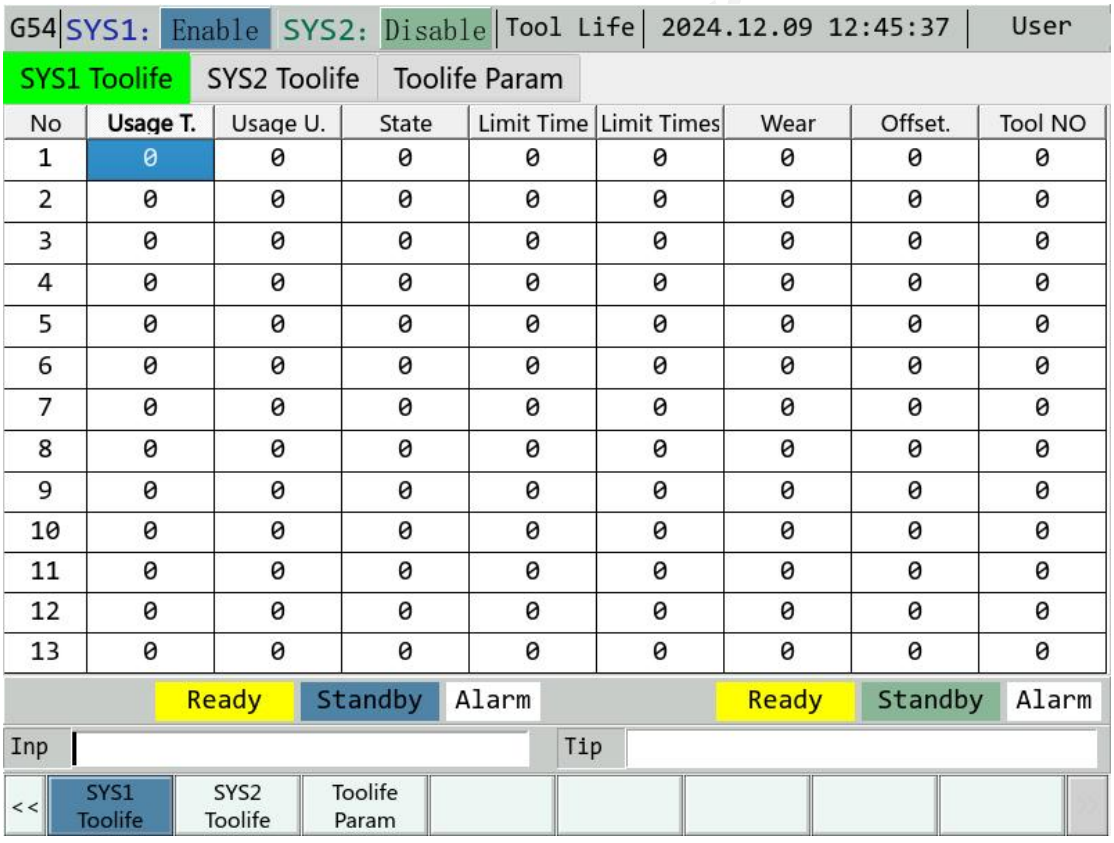
- **Operation Path**

「Wear Compensation」or 「Length Compensation」page → F7【+Input (Incremental Input)】.

- **Description**

This button sets the input method for the table's input area. When the button turns green, 「+Input (Incremental Input)」indicates that the entered value is incremental, adding to the existing value.

1.7.9 Tool Life Management



- **Operation Path**

「Machine Position」page → F3【Offset/Setup】 → F5 to enter the 「Tool Life」page.

- **Description**

This function manages tools, including tool management, group management, and tool parameter settings.

1.7.9.1 Tool Management

- **Operation Path**

「Tool Life」page → F1【Tool Management】.

- **Description**

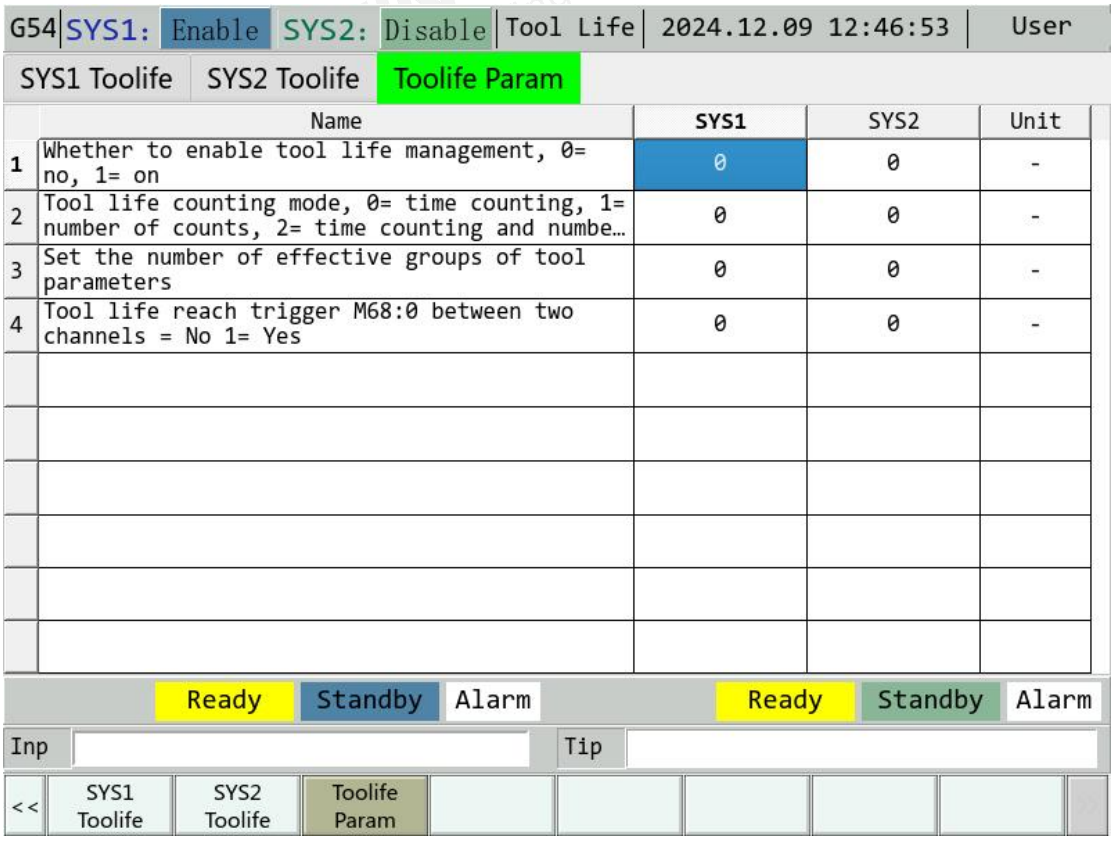
1、Manages tool details, such as current usage time, usage count, tool life status, time limit settings, usage limit settings, wear compensation numbers, length compensation numbers, and tool number assignments.

2、Use the direction keys 【↑】【↓】【←】【→】 to move the cursor.

3、Use 【⩓】【⩔】 to scroll pages.

4、Use 【Home】【End】 to quickly move the cursor to the beginning or end of a row.

1.7.9.2 Tool Parameter Settings



- **Operation Path**

「Tool Life」page → F3【Tool Parameters】.

- **Description**

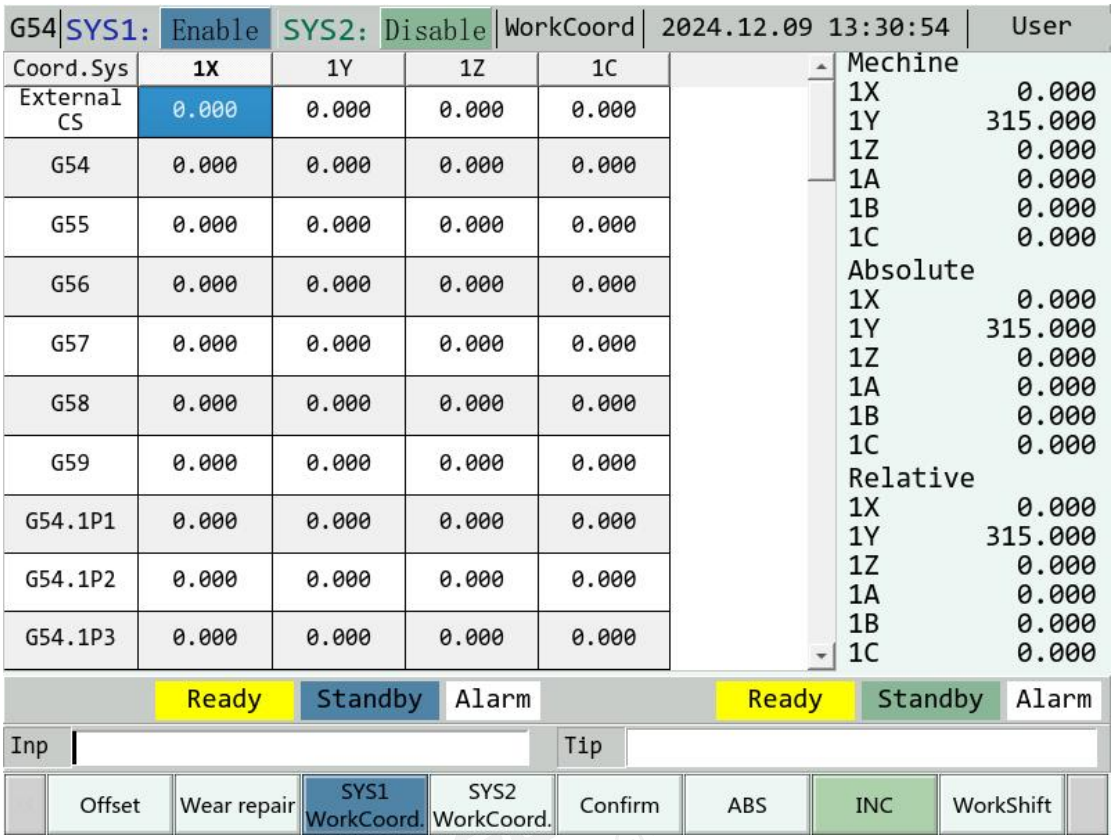
1、Sets tool parameters.

2、Use the direction keys 【↑】【↓】 to move the cursor.

3、Use 【≪】【≫】 to scroll pages.

4、Use 【Home】【End】 to quickly move the cursor to the beginning or end of a row.

1.7.10 Workpiece Coordinate System



- **Operation Path**

「Wear Compensation」or 「Length Compensation」page → F8【Workpiece Coordinate System】.

- **Description**

1、This button switches to the 「Workpiece Coordinate System」 page for setting workpiece coordinates.

2、If G54.1P1-G54.1P48/G54-G59 is not specified 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.

- **Operating Instructions:**

1、Use the direction keys 【↑】【↓】【←】【→】 to move the cursor.

2、Use 【≪】【≫】 to scroll pages.

3、Use 【Home】【End】 to quickly move the cursor to the beginning or end of a row.

4、Input values directly or input the axis name along with the value for tool setting within the workpiece coordinate system.

- **Notes:**

If not used for global offset purposes, re-zero the tool after setting the workpiece coordinate system.

1.7.10.1 Length Compensation

- **Operation Path**

「Workpiece Coordinate System」page → F1【Length Compensation】 → 「Length Compensation」page.

- **Description**

Refer to 1.7.3 "Length Compensation" for details.

1.7.10.2 Wear Compensation

- **Operation Path**

「Workpiece Coordinate System」page → F2【Wear Compensation】 → 「Wear Compensation」page.

- **Description**

Refer to 1.7.2 "Wear Compensation" for details.

1.7.11 Copy Current Row

- **Operation Path**

「Length Compensation」page → 【>>】 → F1【Copy Current Row】.

- **Description**

1、Copies the values in the tool length compensation list.

2、Use the direction keys 【↑】【↓】【←】【→】 to move the cursor.

3、Use 【≪】【≫】 to scroll pages.

4、Use 【Home】【End】 to quickly move the cursor to the beginning or end of a row.

1.7.12 Paste to Current Row

- **Operation Path**

「Length Compensation」page → 【>>】 → F2【Paste to Current Row】.

- **Description**

1、Pastes the copied data to the corresponding option, e.g., data copied from the X-axis option must be pasted into the X-axis option.

2、Use the direction keys 【↑】【↓】【←】【→】 to move the cursor.

3、Use 【≪】【≫】 to scroll pages.

4、Use 【Home】【End】 to quickly move the cursor to the beginning or end of a row.

1.7.13 Clear Current Row

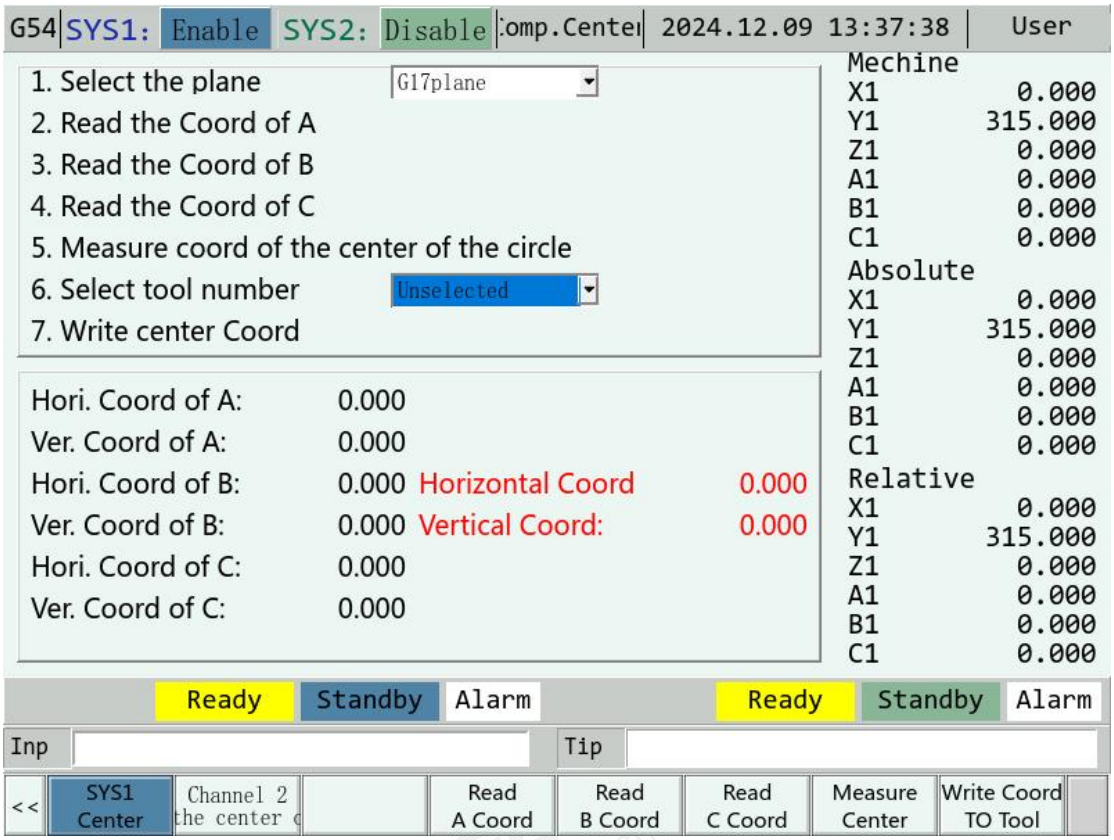
- **Operation Path**

「Length Compensation」page → 【>>】 → F3【Clear Current Row】.

- **Description**

Clears the values in the row where the cursor is currently located.

1.7.14 Three-Point Centering



- **Operation Path**

「Length Compensation」page → 【>>】 → F6【Three-Point Centering】.

- **Description**

Quickly obtain the center coordinates of a workpiece circle, commonly used for centering operations during workpiece alignment.

1.8Machining Monitoring

- **Operation Path**

Use the system panel shortcut key 【Monitoring】 to switch the page to 「Machining Monitoring」.

- **Description**

This page provides essential information needed for monitoring during machining. By default, dual-channel machining monitoring is displayed, but single-channel monitoring can also be selected.

1.8.1 Screen Description

1.8.1.1 Machine Control Area

This area displays current machine information:

1、Current channel program coordinates.

Remaining distance of the current channel.

Current channel feed rate.

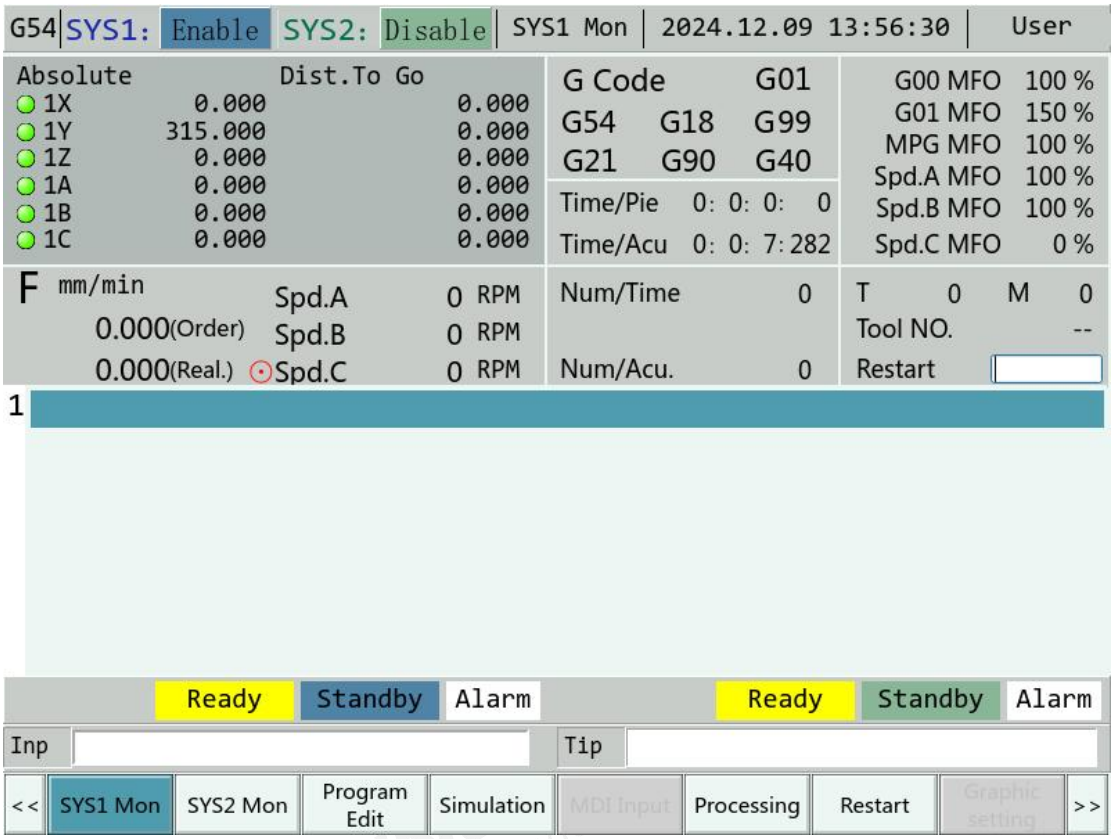
Current channel spindle speed.

1.8.1.2 Program Code Monitoring Area

This area displays the program content being executed on the current channel.

A blue cursor indicates the specific block currently being executed in the program.

1.8.1.3 Machining Information Display Area



- **Description**

1、This area overlaps with the 「Machining Information Settings」 area.

2、Use the F6【Machining Information Settings】 key to toggle the display.

- **Screen Description:**

1、G-Code Status

- Displays the G-codes currently being executed in the system for the active channel.

2、Current Work Time

- Shows the machining time for the current workpiece.
- The time resets when the program restarts.

3、Cumulative Work Time

- Displays the total machining time from the first program execution to the present.
- Can be reset using the second group of keys, specifically the 【Clear Cumulative Time】 key.

4、Override Rates

- G00 Rate Override.
- G01 Rate Override.
- MPG Rate Override.
- Spindle Speed Override.

5、Cumulative Count

- Displays the total number of workpieces machined on the machine.
- This value is not reset automatically; it can be manually reset in the Machining Information Settings under "Cumulative Count".

6、Current Count

- Displays the number of workpieces machined during the current session.
- This count is reset when the machining program is executed for the first time in a session.

7、Machining Tool Data

- T-Code (4-digit display):
- The higher two digits represent the current tool number being used.
- The lower two digits represent the current tool offset number.
- Displays the tool number currently loaded in the turret.

8、M-Code

- Displays the M-code currently being executed.

9、Restart Block

- Allows setting the starting block for restarting.

- ❖ N: Specifies the starting block as line N.
- ❖ N=0: Starts from the first line.
- ❖ If N exceeds the maximum program line count, the system triggers an alarm.

1.8.1.4 Processing Settings Display Area

- **Description**

1、This area overlaps with the "Processing Information Area."

2、Use the Processing Information Settings button to toggle between displays.

- **Screen Description**

1、Displays the current processing time and cumulative processing time.

2、Feed rate settings (Note: Currently displayed only, cannot be modified)

- ❖ Set the processing rate.

3、Spindle speed settings (Note: Requires login for operational permission)

- ❖ Set the spindle speed.

4、Cumulative completion (Note: Requires login for operational permission)

- ❖ Set the total number of workpieces processed by the machine.
- ❖ The system does not automatically reset; manual resetting is required.

5、Current completion (Note: Requires login for operational permission)

- ❖ Set the number of workpieces currently being processed.
- ❖ The workpiece count is reset when the processing program is changed.
- ❖ Works in conjunction with M codes, where M15 increments the count by 1 and M16 resets the count. The processing pauses when the desired workpiece count is reached.

6、Planned completion (Note: Requires login for operational permission)

- ❖ Set the upper limit of the workpieces to be processed.
- ❖ A prompt will appear when the required workpiece count is reached, and the machine will pause processing.

1.8.1.5 Graphical Simulation Display Area

❖ **Description**

1、This area displays the actual tool movement path of the current processing program.

2、Use the F4 [Graphical Simulation Display] button to toggle the display.

1.8.2 Load Program for Editing

- **Operation Path**

Each channel's "Processing Monitoring" page → F3 [Load Program for Editing]

- **Description**

Load the program file currently being executed on the channel into the code editor and switch to the "Program Editing" page.

- **Note**

If this button is pressed during program execution, the screen will switch to the "Program Editing" page, but the program file cannot be edited.

1.8.3 Graphical Simulation Display

- **Operation Path**

Each channel's "Processing Monitoring" page → F4 [Graphical Simulation Display]

- **Description**

1、Used to toggle the display of the graphical simulator.

2、The graphical simulator must be displayed in order to adjust the graphical settings.

3、For detailed information, refer to 1.7.8 "Graphical Adjustment."

1.8.4 MDI Input

- **Operation Path**

"Processing Monitoring" page → F5 [MDI Input]

- **Description**

Edit the MDI program to be executed.

- **Operation Description**

1、Switch the mode to "MDI" mode. The F3 [MDI Input] button is only activated in MDI mode.

2、Press the MDI Input button to open the editing dialog.

3、After editing the program in the dialog, select to load it into Channel 1 or Channel 2. The system will load the edited program into the processing storage area.

4、Press the Start button to execute the MDI program.

- **Note**

This button is only functional in "MDI" mode. The auxiliary panel MDI button can switch the mode to MDI.

1.8.5 Monitoring Information Toggle

- **Operation Path**

"Processing Monitoring" page → F3 [Monitoring Information Toggle]

- **Description**

Toggle between the display of basic processing information and axis multiplication factors.

1.8.6 Numerical State Monitoring

- **Operation Path**

"Processing Monitoring" page → F7 [Numerical State Monitoring]

- **Description**

Open the system IO monitoring interface.

1.8.7 Restart

- **Operation Path**

Each channel's "Processing Monitoring" page → F7 [Restart]

"Program Editing" page → 【>>】→ F7 [Restart from Current Line]

- **Description**

1、This function allows you to specify a certain program block as the restart block, so that processing can be resumed from that specific block.

2、Switch to "Automatic Execution" mode. In the "Processing Information" area of each channel's "Processing Monitoring" page, input the restart block number and press the Start button. Processing will resume from the specified block.

3、Switch to "Automatic Execution" mode, move the cursor in the editing page to the desired restart position, press F7 [Restart from Current Line], and the page will automatically switch to the monitoring page. Press the Start button to resume processing from the specified block.

1.8.8 Graphic Adjustment

- **Operation Path**

"Processing Monitoring" page → F4 [Graphic Adjustment]

- **Description**

Allows you to perform graphical simulation of the program currently being executed on the channel.

1.8.8.1 Plane Selection

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F1 [Plane Selection]

- **Description**

Allows switching the plane for the graphic simulation display.

1.8.8.2 Zoom In (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F2 [Zoom In]

- **Description**

Allows zooming in on the simulated trajectory of the program.

1.8.8.3 Zoom Out (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F3 [Zoom Out]

- **Description**

Allows zooming out the simulated trajectory of the program.

1.8.8.4 Fit to Screen

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F4 [Fit to Screen]

- **Description**

Adjusts the graphic simulation trajectory to fit the screen, ensuring that the entire trajectory is visible.

1.8.8.5 Clear

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F5 [Clear]

- **Description**

Clears the simulated trajectory from the screen.

1.8.8.6 Window Adjustment (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment]

- **Description**

Allows you to move the graphic simulator's window.

1.8.8.6.1 Move Window Left (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F1 [Move Window Left]

- **Description**

Moves the window of the graphic simulator to the left.

1.8.8.6.2 Move Window Right (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F2 [Move Window Right]

- **Description**

Moves the window of the graphic simulator to the right.

1.8.8.6.3 Move Window Up (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F3 [Move Window Up]

- **Description**

Moves the window of the graphic simulator upwards.

1.8.8.6.4 Move Window Down (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F4 [Move Window Down]

- **Description**

Moves the window of the graphic simulator downwards.

1.8.8.6.5 Zoom In (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F5 [Zoom In]

- **Description**

This function is the same as 1.7.8.2 [Zoom In].

1.8.8.6.6 Zoom Out (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F6 [Window Adjustment] → F6 [Zoom Out]

- **Description**

This function is the same as 1.7.8.3 [Zoom Out].

1.8.8.7Contour Mode Toggle (Not Supported)

- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → F7 [Contour Mode Toggle]

- **Description**

This function switches between "Automatic Contouring" and "Single Block Contouring" modes.

1.8.8.8 Contour Overall Settings

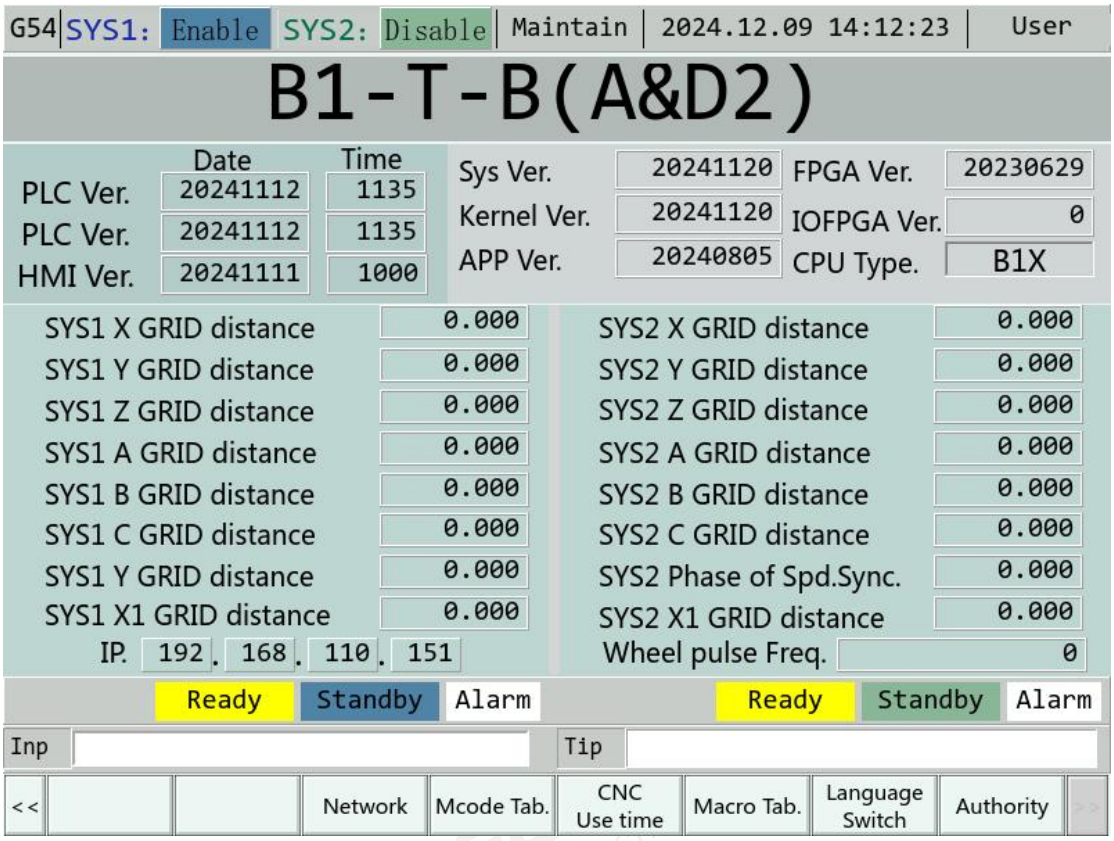
- **Operation Path**

"Processing Monitoring" page → F8 [Graphic Adjustment] → [>>] → F1 [Contour Overall Settings]

- **Description**

This function allows you to configure the contouring settings, including the color of the contour, layer color, and background color.

1.9 Help/Alerts



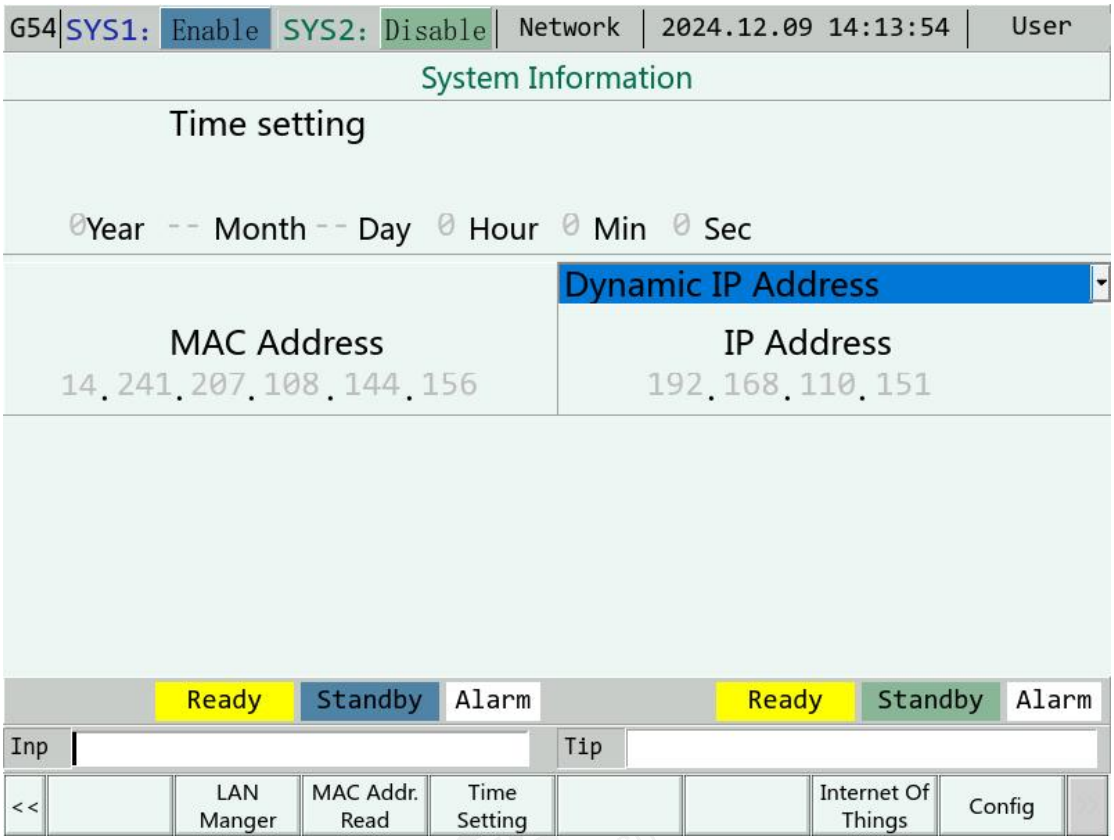
- **Operation Path**

"Machine Position" page → F5 [Maintenance]

- **Description**

This page displays system help information, such as the system software version, network IP address, etc.

1.9.1 Network Settings



- **Operation Path**

"Maintenance" page → F3 [Network Settings]

- **Description**

This function is used for controller IP settings, physical address monitoring, time settings, and more.

1.9.1.1 Network Management

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F2 [Network Management]

- **Description**

his function is used to read and set the controller's IP address.

1.9.1.1.1 Read IP

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F2 [Network Management] → F1 [Read IP]

- **Description**

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

1.9.1.1.2 Set IP

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F2 [Network Management] → F2 [Set IP]

- **Description**

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

- **Operation Description**

1、To set the IP address, enter the desired IP address in the IP address input box.

2、Press [Set IP], and the controller's IP address will be set to the entered value.

1.9.1.2 Read Physical Address

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F3 [Read Physical Address]

- **Description**

This function is used to read the controller's network card physical address, which is unique.

1.9.1.3 Time Settings

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F4 [Time Settings]

- **Description**

This function is used to set the system display time.

1.9.1.3.1 Set Time

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F4 [Time Settings] → F1 [Set Time]

- **Description**

This function is used to set the system time.

1.9.1.4 WIFI Module

- **Operation Path**

"Help" page → F3 [Network Settings] → F6 [WIFI]

- **Description**

This function is used to connect to an external network.

1.9.1.5 Internet of Things (IoT)

- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F7 [IoT]

- **Description**

This function is used to connect to an external network.

1.9.1.6 Configuration File

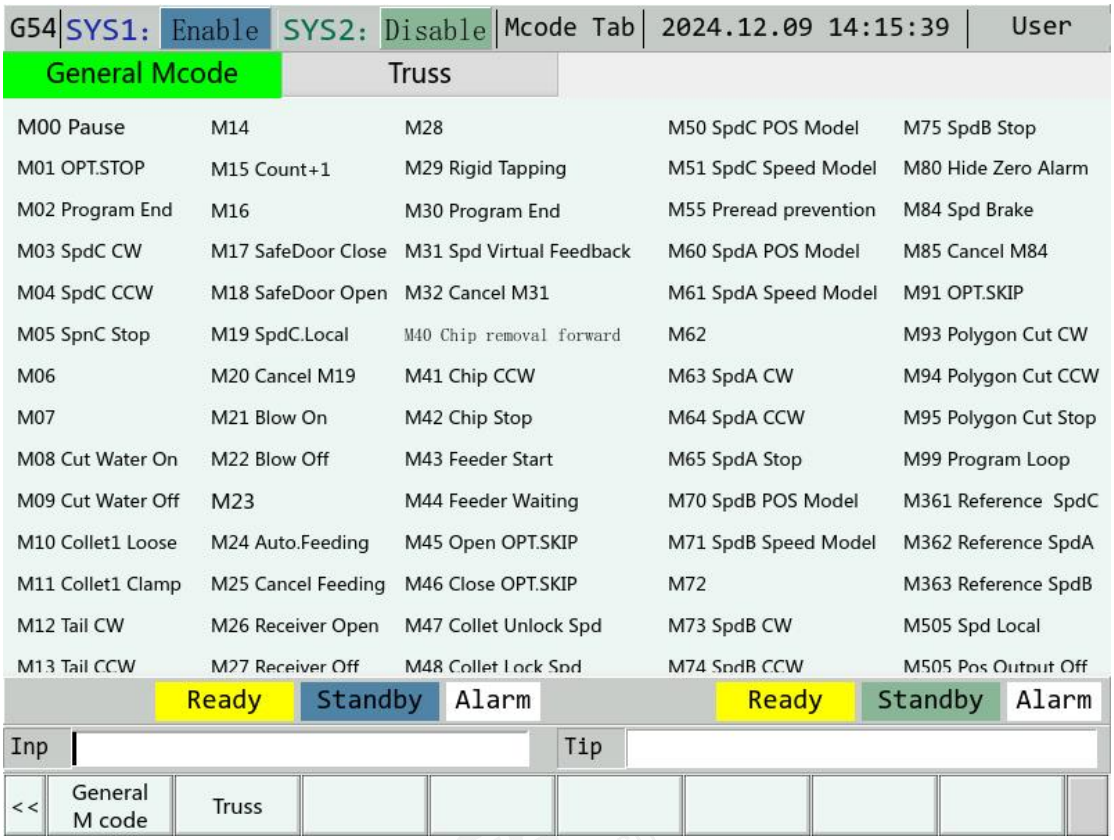
- **Operation Path**

"Maintenance" page → F3 [Network Settings] → F8 [Configuration File]

- **Description**

This function is reserved for future use.

1.9.2 M Code Table



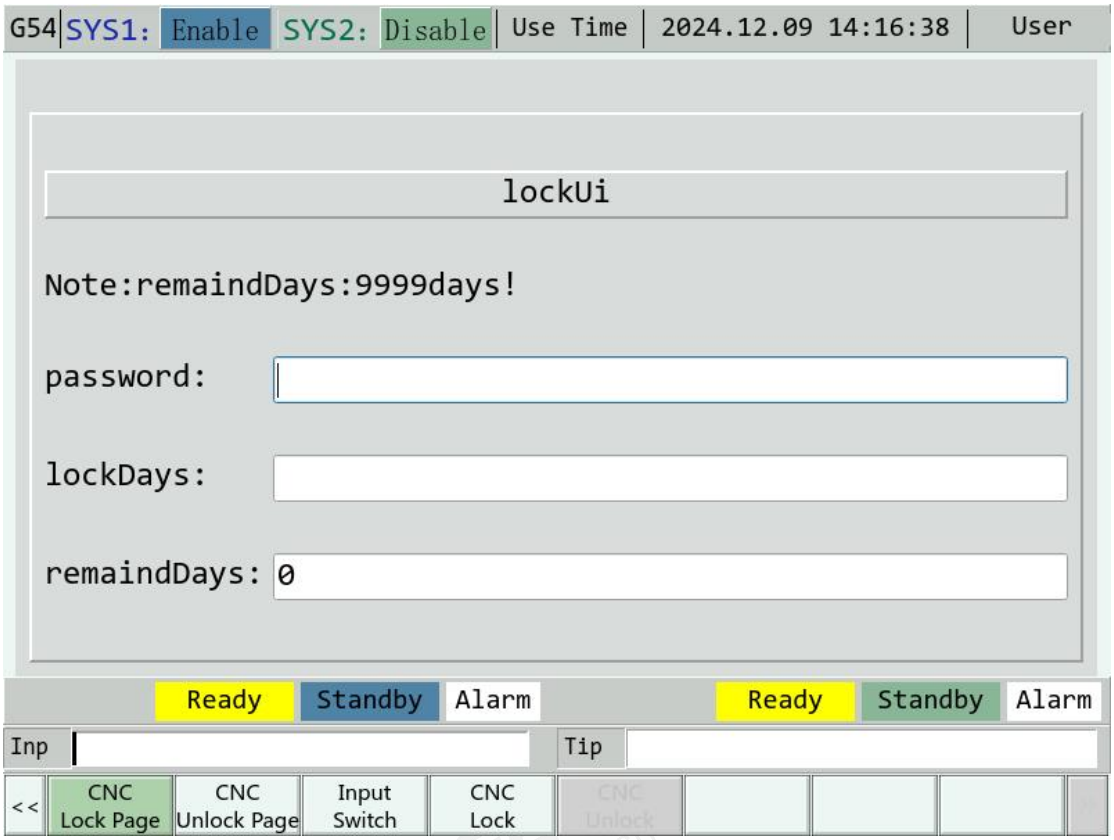
- **Operation Path**

"Maintenance" page → F4 [M Code Table]

- **Description**

This function is used to query commonly used auxiliary M codes for system programming and robotic arm programming.

1.9.3 Controller Lock/Unlock



- **Operation Path**

"Maintenance" page → F5 [Controller Lock/Unlock]

- **Description**

This function is used to display the controller usage period management interface. For detailed instructions, refer to the unlock machine operation document.

1.9.3.1 Lock Machine Page

- **Description**

Used to lock the system's usage period.

- **Operation Description**

On the lock machine page, enter the user password, lock days, and remaining reminder days. Then, use the Lock Machine button to complete the system's usage period lock.

1.9.3.2 Unlock Page

- **Description**

Used to unlock the system's usage period or to relock the usage period.

- **Operation Description**

On the unlock page, input the controller serial number into the unlock software. Based on the unlock days (unlock software setting for complete unlock or limited usage days), generate the corresponding unlock code. Enter the generated unlock code into the controller input field along with the remaining reminder days, and then press the Unlock button to complete the unlock or relock.

1.9.3.3 Switch Input Fields

- **Description**

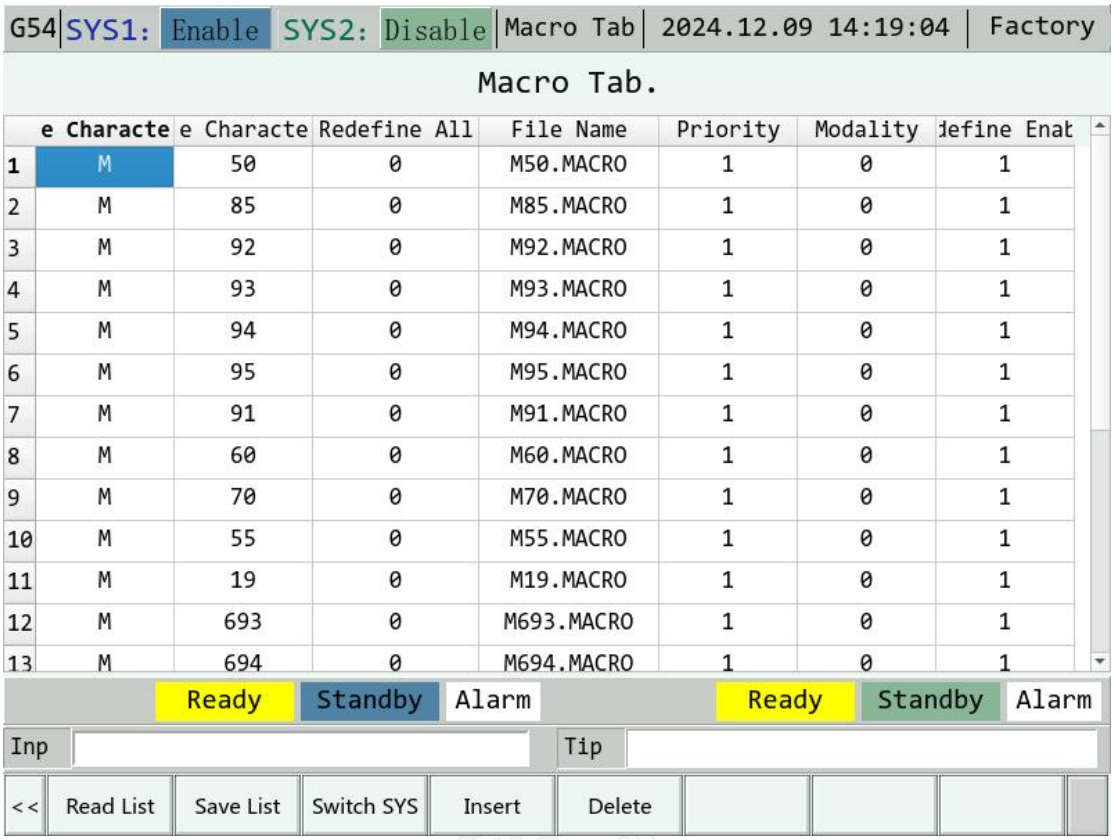
Used for switching the focus of input fields on the lock machine interface.

1.9.3.4 Lock/Unlock

- **Description**

Used in conjunction with the controller's lock and unlock functions.

1.9.4 Custom Jump



- **Operation Path**

"Maintenance" page → [>>] → F2 [MACRO]

- **Description**

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

- **Operation Description**

1、Set the required jump parameters on the configuration table, including the letter, value, jump type, target file name, priority, whether it is modal, and whether to use the jump.

2、Press F2 [Save Chain] to automatically generate the configuration file.

- **Note**

设 After configuring the table and saving it, a system reboot is required for the changes to take effect.。

1.9.5 Language Switch

- **Operation Path**

"Maintenance" page → F7 [Language Switch]

- **Description**

This function is used to switch the language display on the screen. Currently, it supports Simplified Chinese and English.

1.9.6 Permission Management

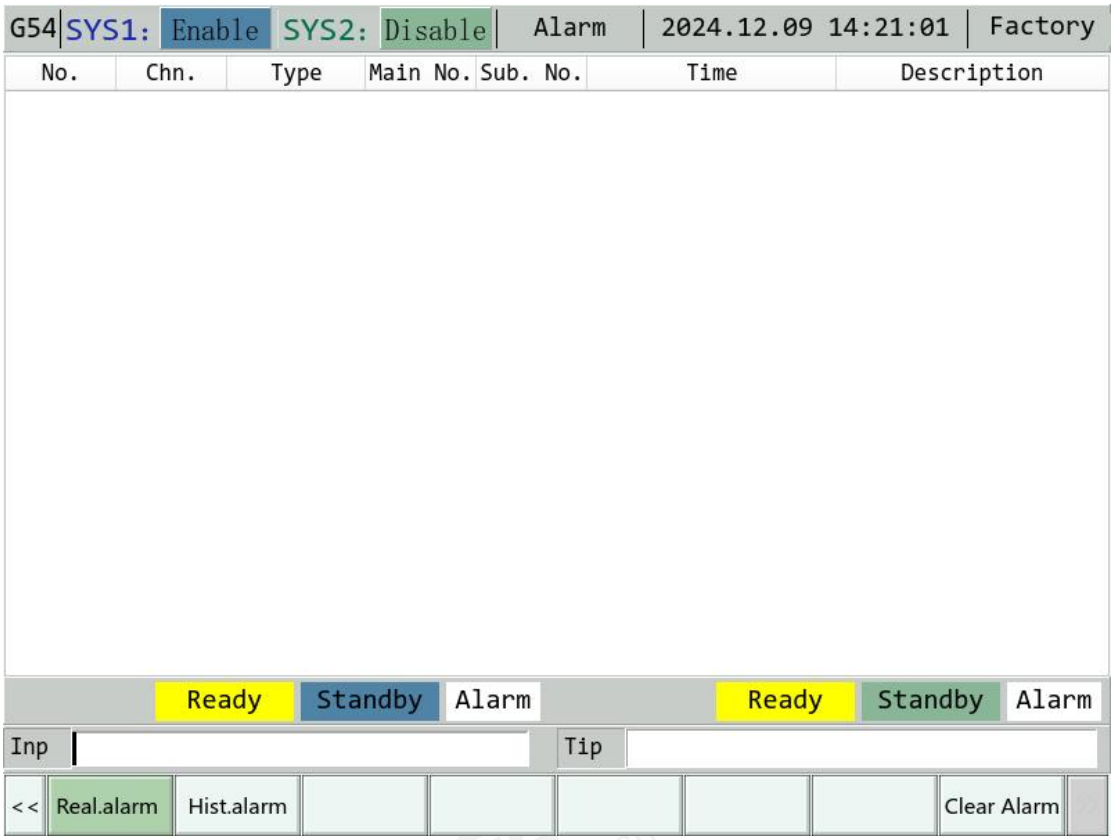
- **Operation Path**

"Maintenance" page → F8 [Permission Management]

- **Description**

This function is used to display the permission management interface. For detailed description, refer to the fourth section on Permission Management.

1.9.7 Alerts



- **Operation Path**

Method 1: Use the system panel shortcut key [Help/Alerts] to switch to the "Alert Monitoring" page.

Method 2: "Maintenance" page → F8 [Alerts]

- **Description**

This page is used to view the controller's current alerts and historical alerts.

1.9.7.1 Current Alerts

- **Operation Path**

1、"Alert Monitoring" page → F1 [Current Alerts]

2、Use the arrow keys [↑] and [↓] to move the cursor.

3、Use the arrow keys [←] and [→] to switch between public channel and channel 1 alert displays.

- **Description**

This function is used to view the alerts currently occurring on the controller.

1.9.7.2 Historical Alerts

- **Operation Path**

"Alert Monitoring" page → F2 [Historical Alerts]

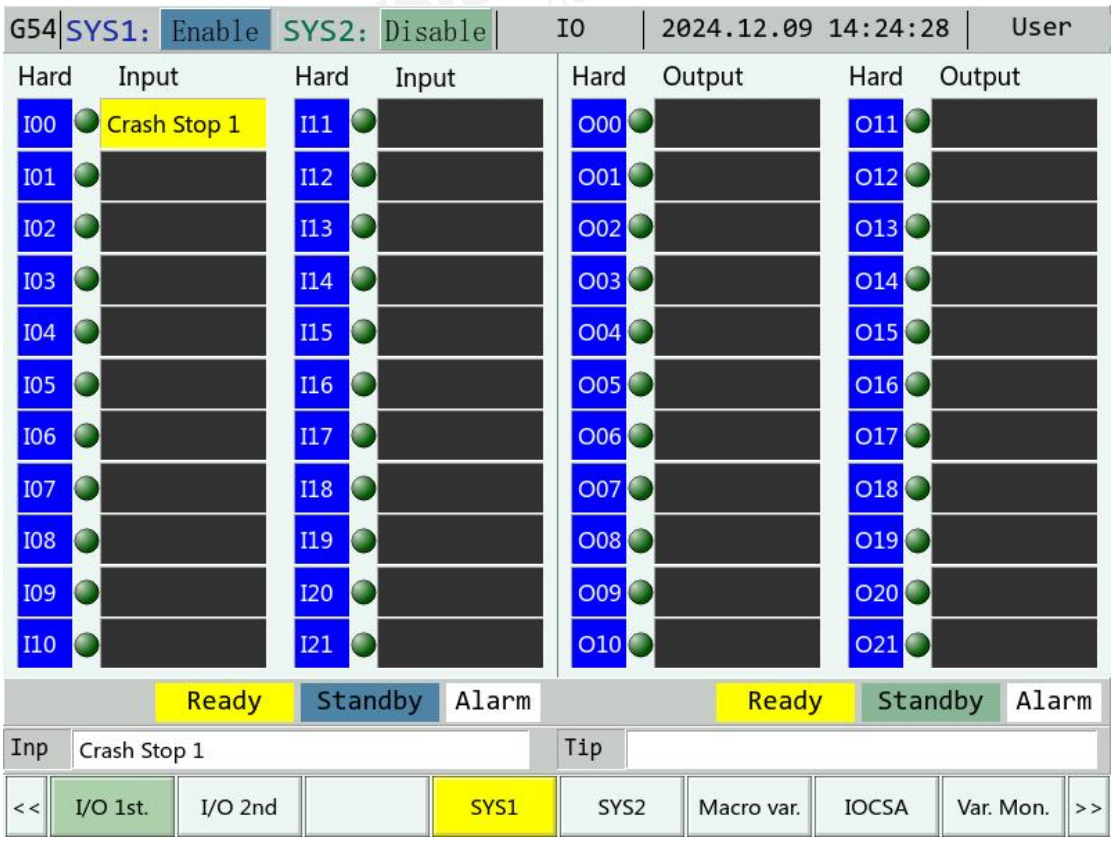
- **Description**

1、This function is used to view the 200 most recent alerts that occurred on the controller, listed in chronological order from top to bottom, with the most recent alert displayed at the top.

2、Use the arrow keys [↑] and [↓] to move the cursor.

3、Use the arrow keys [←] and [→] to switch between public channel and channel 1 alert displays.

1.10 Status Monitoring



- **Operation Path**

"Machine Location" page → F6 [Status Monitoring]

- **Description**

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

1.10.1 Monitoring 1

- **Operation Path**

"Machine Position" page → F6 [State Monitoring] → F1 [IO Monitoring 1]

- **Description**

Used for monitoring the status of IO points on the controller's IO status page.

1.10.2 IO Monitoring 2

- **Operation Path**

"Machine Position" page → F6 [State Monitoring] → F2 [IO Monitoring 2]

- **Description**

Used for monitoring the status of additional IO points on the controller's IO status page.

1.10.3 SYS1/SYS2

- **Operation Path**

"Machine Position" page → F6 [State Monitoring] → F4 [SYS1] / F5 [SYS2]。

- **Description**

Used for switching between the current monitored channels.

1.10.4 Macro Variables

- **Operation Path**

"Machine Position" page → F6 [State Monitoring] → F6 [Macro Variables]

- **Description**

Used for monitoring and setting the macro variables of the controller.

1.10.5 IOCSA

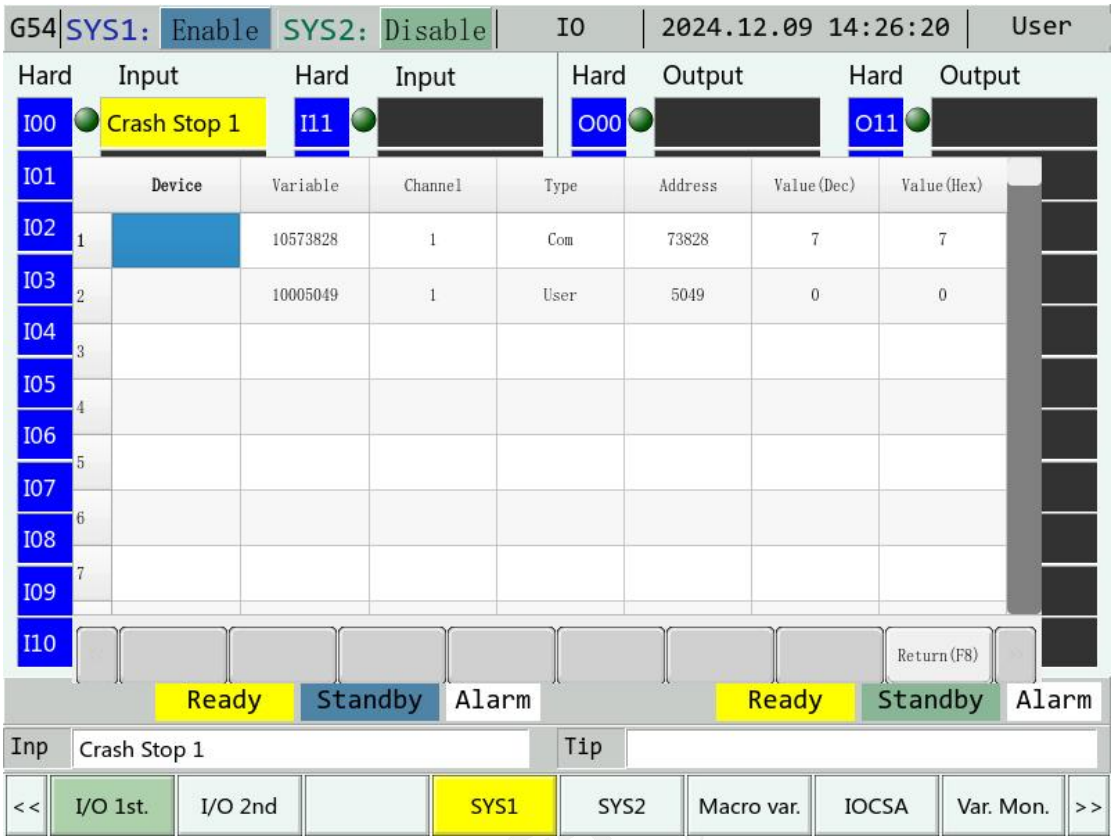
- **Operation Path**

  "Machine Position" page → F6 [State Monitoring] → F7 [IOCSA]

- **Description**

  Used for monitoring the status of the controller's software IOCSA.

1.10.6 Variable Monitoring



- **Operation Path**

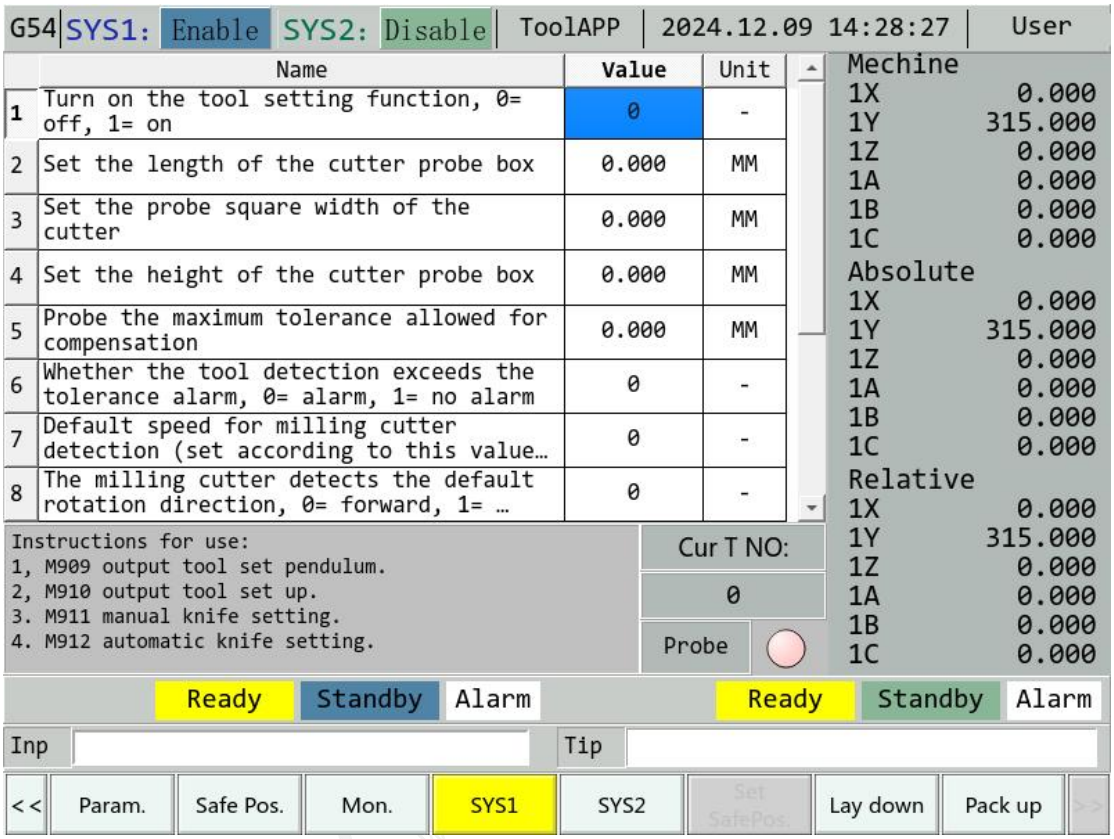
"Machine Position" page → F6 [Status Monitoring] → F8 [Variable Monitoring].

- **Description**

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

1.11 Extended Functions

1.11.1 Tool Setter



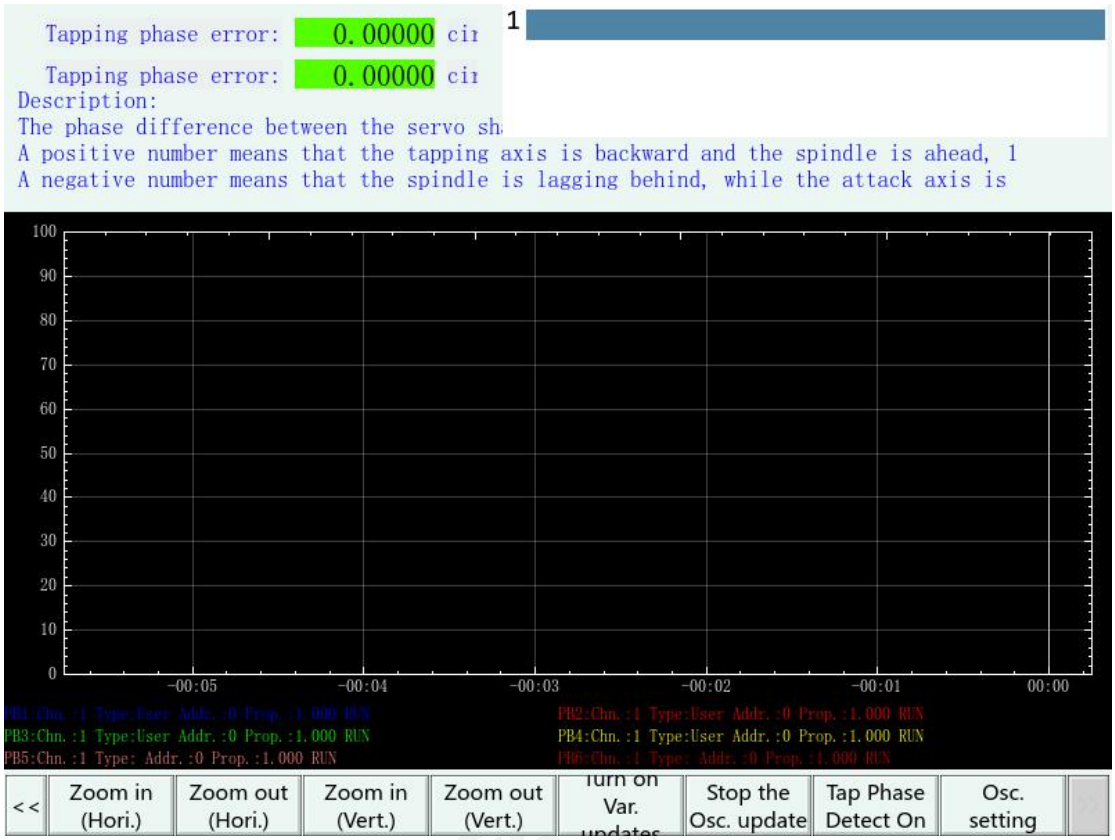
- **Operation Path**

"Machine Position" page → F7 [Extended Functions] → F1 [Tool Setter].

- **Description**

This feature enables the use of the tool setter for automatic tool calibration, simplifying the tool setting process. For detailed steps, refer to the Tool Setter operation documentation.

1.11.2 Variable Oscilloscope



- **Operation Path**

"Machine Position" page → F7 [Extended Functions] → F2 [Variable Oscilloscope].

- **Description**

This feature is used to monitor variable changes and the phase errors of tapping axes.

1.11.2.1 Horizontal Zoom In

- **Horizontal Zoom In**

[Variable Oscilloscope] page → F1 [Horizontal Zoom In].

- **Description**

Magnifies the data units along the horizontal axis of the oscilloscope.

1.11.2.2 Horizontal Zoom Out

- **Operation Path**

[Variable Oscilloscope] page → F2 [Horizontal Zoom Out].

- **Description**

Shrinks the data units along the horizontal axis of the oscilloscope.

1.11.2.3 Vertical Zoom In

- **Operation Path**

[Variable Oscilloscope] page → F3 [Vertical Zoom In].

- **Description**

Magnifies the data units along the vertical axis of the oscilloscope.

1.11.2.4 Vertical Zoom Out

- **Operation Path**

[Variable Oscilloscope] page → F4 [Vertical Zoom Out].

- **Description**

Shrinks the data units along the vertical axis of the oscilloscope.

1.11.2.5 Enable Variable Updates

- **Operation Path**

[Variable Oscilloscope] page → F5 [Enable Variable Updates].

- **Description**

Activates continuous updates for variables, keeping the oscilloscope waveforms refreshed in real-time.

1.11.2.6 Pause Oscilloscope Updates

- **Operation Path**

[Variable Oscilloscope] page → F6 [Pause Oscilloscope Updates].

- **Description**

Stops oscilloscope updates, allowing detailed observation of waveforms and data at a specific moment.

1.11.2.7 Start Tapping Phase Error Detection

- **Operation Path**

[Variable Oscilloscope] page → F7 [Start Tapping Phase Error Detection].

- **Description**

This feature is used to detect phase errors for each axis during tapping operations.

1.11.2.8 Oscilloscope Settings

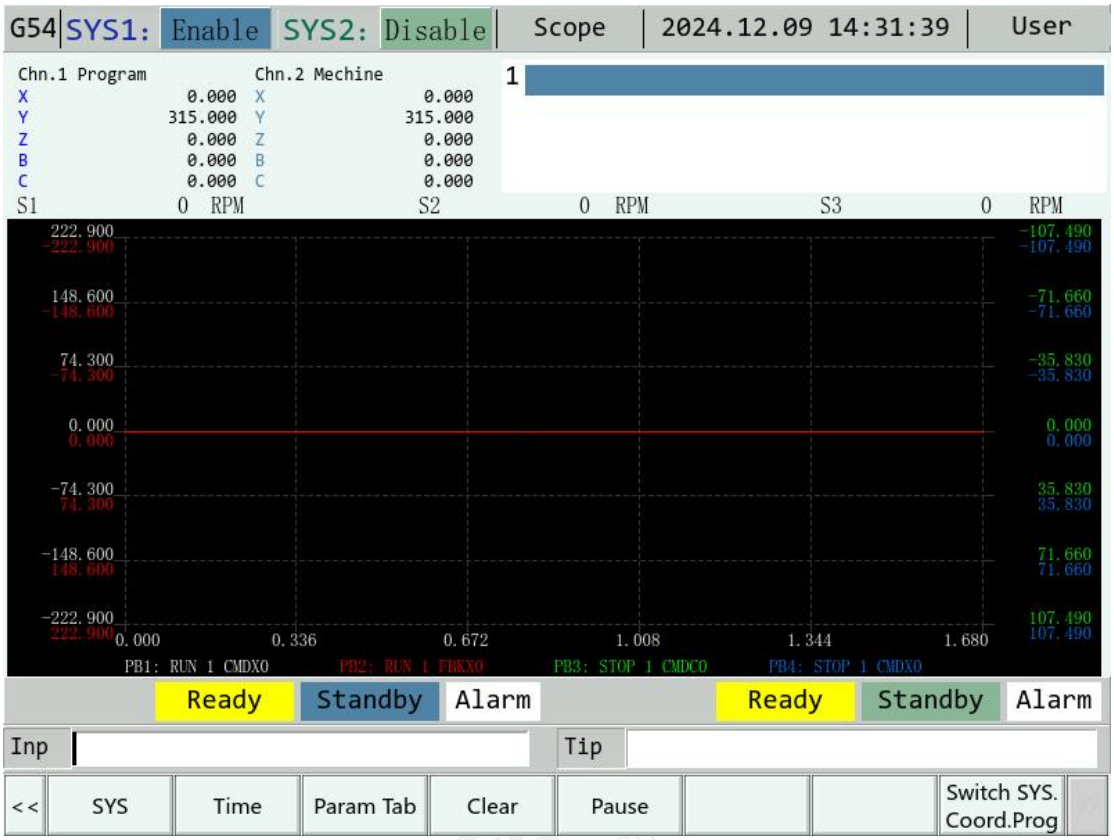
- **Operation Path**

[Variable Oscilloscope] page → F8 [Oscilloscope Settings].

- **Description**

This feature allows configuration of the oscilloscope interface display, data settings, and channel settings.

1.11.3 Axis Oscilloscope



- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope]

- **Description**

This feature monitors the command and feedback waveforms of each axis.

1.11.3.1 Channels

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel]

- **Description**

Adjusts the position and display mode of the waveforms for the four oscilloscope channels.

1.11.3.1.1 PB1

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [PB1]

- **Description**

Allows adjustments to the position and display settings of Channel 1's waveform. The methods for adjusting Channels 2, 3, and 4 are identical and not repeated here.

1.11.3.1.2 PB1 Up

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F1 [PB1 Up]

- **Description**

Moves the waveform of Channel 1 upward.

1.11.3.1.3 PB1 Down

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F2 [PB1 Down]

- **Description**

Moves the waveform of Channel 1 downward.

1.11.3.1.4 PB1 Zoom In

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F3 [PB1 Zoom In]

- **Description**

Vertically enlarges the waveform of Channel 1.

1.11.3.1.5 PB1 Zoom Out

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F4 [PB1 Zoom Out]

- **Description**

This function vertically shrinks the waveform of Channel 1.

1.11.3.1.6 PB1 Reset Position

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F5 [PB1 Reset Position]

- **Description**

Adjusts the waveform of Channel 1 to its standard position.

1.11.3.1.7 PB1 Standard Ratio

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F6 [PB1 Standard Ratio]

- **Description**

Sets the waveform size of Channel 1 to a standard ratio.

1.11.3.1.8 PB1 Show/Hide

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel]

→ F1 [PB1] → F7 [Show/Hide]

- **Description**

Toggles the display of the waveform for Channel 1.

1.11.3.1.9 PB1 Invert/Non-Invert

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F1 [Channel] → F1 [PB1] → F8 [Invert/Non-Invert]

- Description

Switches between inverted and non-inverted waveform displays for Channel 1.

1.11.3.2 Time Adjustment

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F2 [Time]

- **Description**

Adjusts the time span of the waveform displayed in the current oscilloscope window for horizontal scaling.

1.11.3.2.1 Horizontal Zoom In

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F2 [Time] → F1 [Horizontal Zoom In]

- **Description**

Horizontally magnifies the waveform display.

1.11.3.2.2 Horizontal Zoom Out

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F2 [Time] → F2 [Horizontal Zoom Out]

- **Description**

Horizontally shrinks the waveform display.

1.11.3.2.3 Standard Ratio

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F2 [Time] → F3 [Standard Ratio]

- **Description**

Adjusts the waveform's horizontal scale to the standard ratio.

1.11.3.3 Overall Settings

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F3 [Overall Settings]

- **Description**

Configures the channels, axes, and pulse types for all four oscilloscope channels. The configuration method for Channel 1 is detailed below, and the process is identical for other channels.

- **Procedure**

Click [Overall Settings] to open the settings table.

1、Enter the following for Channel 1:

- Channel: 1
- Axis: X

- Type: Command

2、Axis options: X, Y, Z, A, B, C, X1–X5, Y1–Y5, Z1–Z5, A1–A5, B1–B5, C1–C5.

3、Type options: Command, Feedback, or Voltage.

4、Click [Apply] to save the settings.

5、Channel 1's waveform is now set to display the Command pulse for Axis X.

6、Additionally, you can configure the following:

- Show/Hide grid.
- Display probe indicators.
- Layout (horizontal or vertical).
- Invert waveforms.
- Linear waveform display.
- Waveform ratio.
- Zero position and colors.

1.11.3.4 Clear

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F4 [Clear]

- **Description**

This function clears all waveforms displayed in the oscilloscope window.

1.11.3.5 Pause Oscilloscope

- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F5 [Pause]

- **Description**

Pauses waveform updates, freezing the current display for easier observation and analysis.

1.11.3.6 Switch Channel

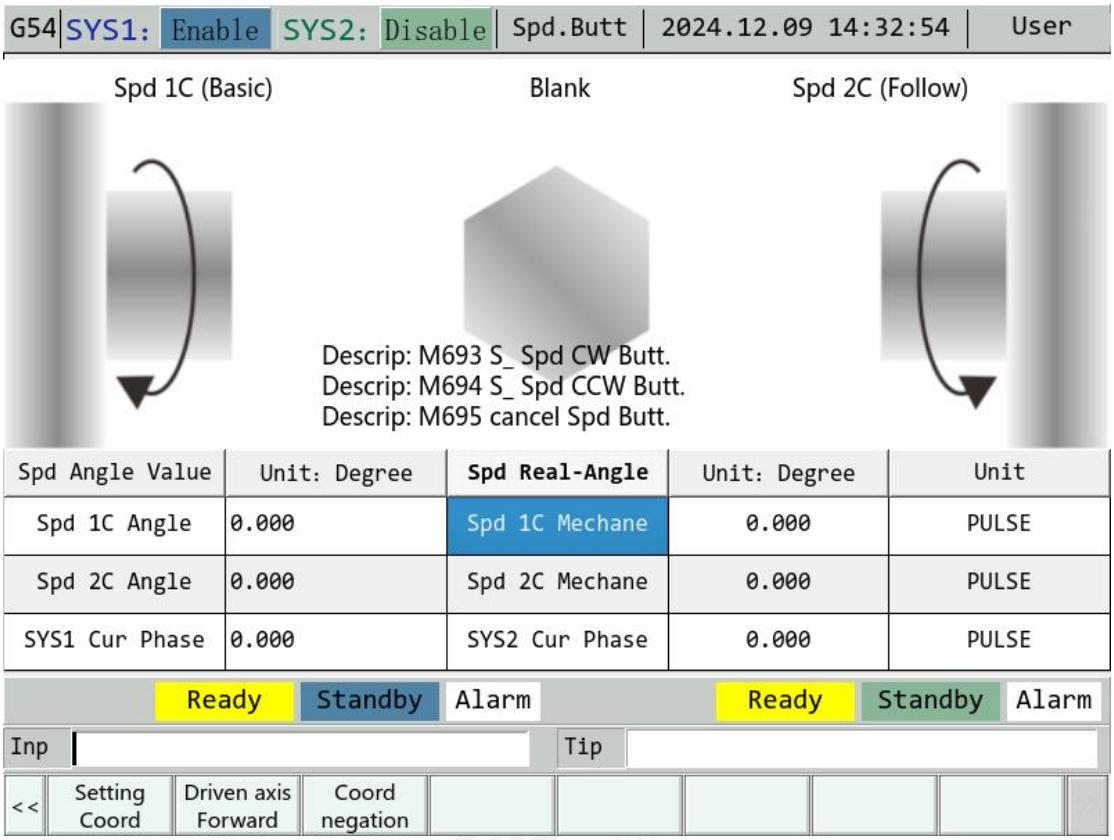
- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F3 [Axis Oscilloscope] → F8 [Switch Channel Coordinates & Program]

- **Description**

This function toggles between displaying channel program coordinates, machine coordinates, and the currently executing program.

1.11.4 Spindle Docking



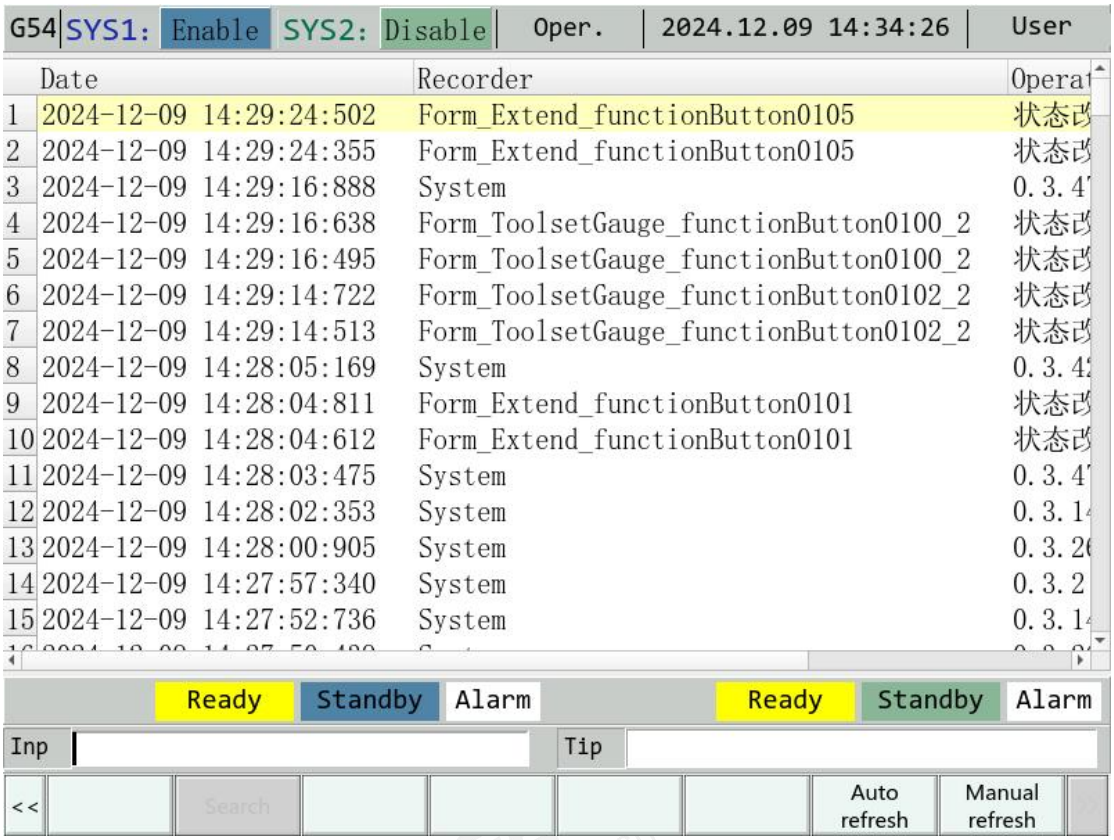
- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F4 [Spindle Docking]

- **Description**

This function enables spindle docking, which is used for dual-spindle workpiece processing. For detailed procedures, refer to the spindle docking operation manual.

1.11.5 Operation Log



- **Operation Path**

[Machine Location] → F7 [Extended Functions] → F5 [Operation Log]

- **Description**

Records user operation steps. This feature is useful for troubleshooting or reproducing issues during after-sales service by tracing improper actions that caused the problem.

1.12 Parameter Settings

- **Operation Path**

[Machine Location] → F8 [Parameter Settings] (Opens the parameter directory by default.)

- **Description**

This function allows the configuration of controller axis control parameters and user-defined parameters.

- **Operation Instructions**

1、Use the arrow keys [↑], [↓], [←], [→] to navigate the cursor.

2、Use [⩓] and [⩔] to scroll pages up and down.

3、Press the [Enter] key to input values.

- **Note**

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

1.12.1 Previous Item

- **Operation Path**

[Machine Location] → F8 [Parameter Settings] → F1 [Previous Item]

Description

- **Description**

This key allows you to switch to the previous parameter item.

1.12.2 Next Item

- **Operation Path**

[Machine Location] → F8 [Parameter Settings] → F2 [Next Item]

- **Description**

This key allows you to switch to the next parameter item.

1.12.3 Open Directory

- Operation Path

[Machine Location] → F8 [Parameter Settings] → F3 [Open Directory]

- Description

This key opens the directory page to select the corresponding parameter for modification.

1.12.4 Search by Number

- Operation Path

[Machine Location] → F8 [Parameter Settings] → F5 [Search by Number]

- Description

This key searches the parameter number in the current parameter table and jumps to the corresponding line.

1.12.5 Parameter Overview

- Operation Path

[Machine Location] → F8 [Parameter Settings]

- Description

This key opens the system's parameter directory for quick navigation to specific parameter tables.

1.12.5.1 Parameter Overview Table

- **Description**

1. Displays the system’s complete list of functional parameters.
2. Use the arrow keys [↑], [↓], [←], [→] to navigate the cursor.
3. Use the [Enter] key to jump to the selected parameter table.

1.12.5.2 Axis Parameters

- **Description**

1. Displays all axis-related functional parameters and acceleration/deceleration parameters.
2. Use the arrow keys [↑], [↓], [←], [→] to navigate the cursor.
3. Use the [Enter] key to jump to the selected parameter table.

1.12.5.3 Process Parameters

- **Description**

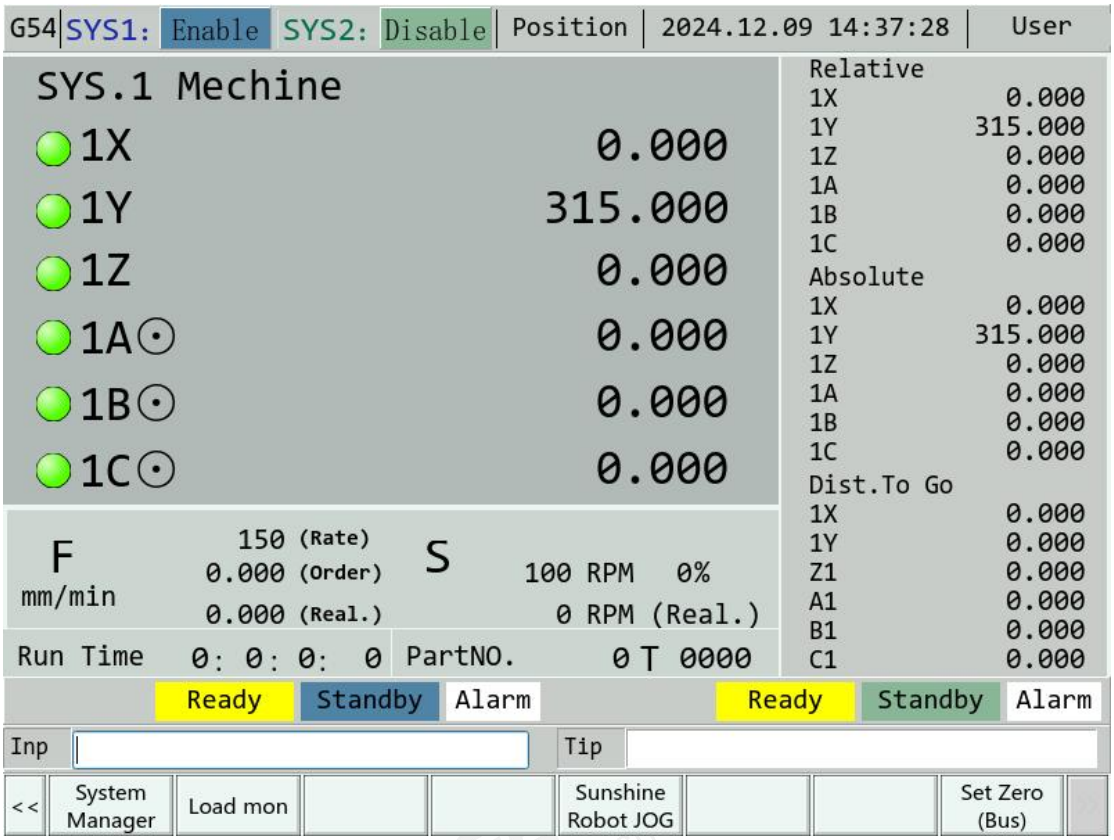
1. Displays the system's G-code comprehensive parameters and common drilling, tapping, and threading parameters.
2. Use the arrow keys [↑], [↓], [←], [→] to navigate the cursor.
3. Use the [Enter] key to jump to the selected parameter table.

1.12.5.4 IO/Mechanism Parameters

- **Description**

1. Displays IO-related and mechanism-related parameters, such as turret, tailstock, spindle brake, lubrication oil, etc.
2. Use the arrow keys [↑], [↓], [←], [→] to navigate the cursor.
3. Use the [Enter] key to jump to the selected parameter table.

1.13 System Management



- **Operation Path**

[Machine Location] → [>>] → F1 [System Management]

- **Description**

This feature is used for managing backup packages, data backup, data restoration, and restoring factory settings.

- **Note**

To access the system management page, a password is required. The default password is "112233".

1.13.1 Backup Packages

- **Operation Path**

[Machine Location] → [>>] → F1 [System Management] → F1 [Backup Packages]

- **Description**

This feature is used to manage the controller's backup packages. It allows you to add,

delete, edit comments, and copy the backup packages to a USB drive. For detailed descriptions, please refer to the "Controller Backup" document.

1.13.2 Data Backup

- **Operation Path**

[Machine Location] → [>>] → F1 [System Management] → F2 [Data Backup]

- **Description**

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

1.13.3 Data Restoration

- **Operation Path**

[Machine Location] → [>>] → F1 [System Management] → F3 [Data Restoration]

- **Description**

This feature is used to restore the backed-up controller data to the system, effectively updating the system's data. For detailed descriptions, please refer to the "Controller Backup" document.

1.13.4 Restore Factory Settings

- **Operation Path**

[Machine Location] → [>>] → F1 [System Management] → F8 [Restore Factory Settings]

- **Description**

This feature is used to clear the controller's data and restore the factory settings.

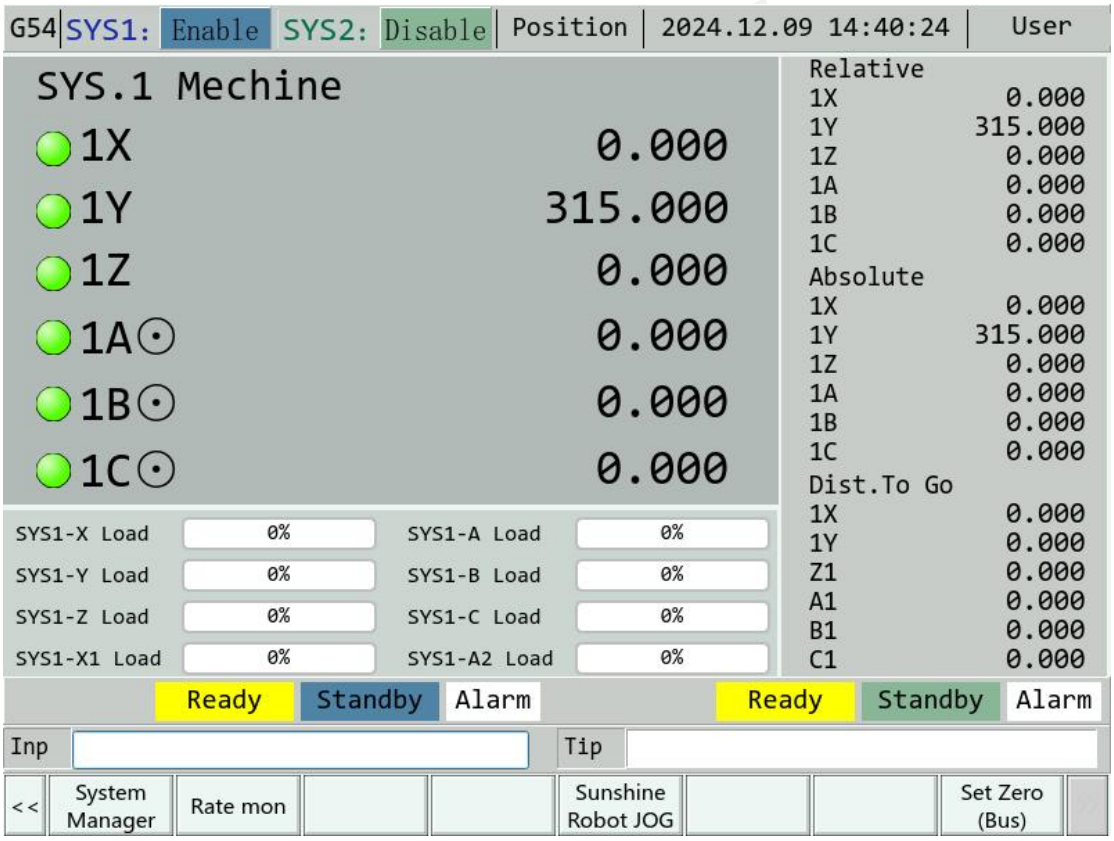
1.14 Load Monitoring/Feed Rate Monitoring

- **Operation Path**

[Machine Location] → [>>] → F2 [Load Monitoring/Feed Rate Monitoring]

- **Description**

This feature is used to toggle the display interface. Pressing this key once will display the load rate of each axis. Pressing it again will show the feed rate (F), spindle speed (S), machining time, part count, and tool number (T).



1.15 Bus Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F5 [Bus Axis Zero Setting]

- **Description**

This function is used to set the machine zero point for bus axes.

1.15.1 X Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F1 [X Axis Zero Setting]

- **Description**

Set the zero point for the X axis.

1.15.2 Y Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F2 [Y Axis Zero Setting]

- **Description**

Set the zero point for the Y axis.

1.15.3 Z Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F3 [Z Axis Zero Setting]

- **Description**

Set the zero point for the Z axis.

1.15.4 A Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F4 [A Axis Zero Setting]

- **Description**

Set the zero point for the A axis.

1.15.5 B Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F5 [B Axis Zero Setting]

- **Description**

Set the zero point for the B axis.

1.15.6 C Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F6 [C Axis Zero Setting]

- **Description**

Set the zero point for the C axis.

1.15.7 X1 Axis Zero Setting

- **Operation Path**

[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F7 [X1 Axis Zero Setting]

- **Description**

Set the zero point for the X1 axis.

1.15.8 Switch Channel

- **Operation Path**

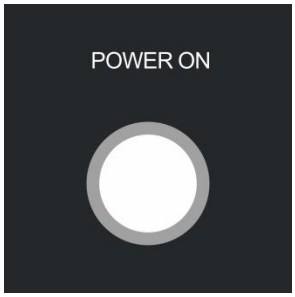
[Machine Location] → [>>] → F8 [Bus Axis Zero Setting] → F8 [Switch Channel]

- **Description**

Switch to the channel where the bus axis absolute zero point needs to be set.

3. Mechanical Operation Panel Description

2.1 Panel Operation Buttons

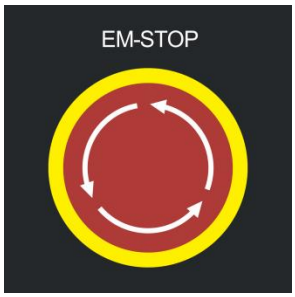


- **Power On**

Turns on the controller power, enabling machine operation.

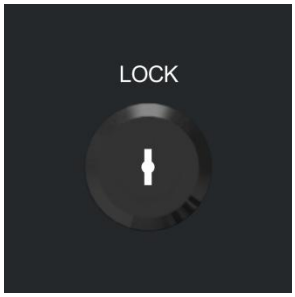
- **Power Off**

Turns off the controller power, disabling machine operation.



- **Emergency Stop**

Press this button to immediately stop machine motion in case of safety concerns or operational failures. The controller triggers an alarm, and the IO system adjusts controls accordingly.



- **Program Lock**

Activates program lock functionality, preventing program editing.

- **Automatic Mode**

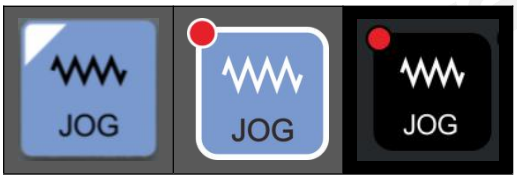
In this mode, pressing the start button automatically executes the machining program.

- **Home Mode**

This mode moves all axes back to their preset zero positions or mechanical zero points.

- **MDI Mode**

This mode is used to execute on-the-fly programs without the need to load machining files.



- **Manual Mode (JOG)**

This mode manually drives the selected axis. Ensure the desired axis is selected before operation.

- **Operation Method Mode**

This mode uses the MPG to drive the selected axis. Ensure the desired axis is selected before operation.

- **Inching Mode**

This mode allows incremental movement of the selected axis. Ensure the desired axis is selected before operation.

- **Program Simulation**

This function allows the MPG to simulate program execution in Automatic or MDI mode, facilitating verification of machining paths.

- **Single-Step Execution**

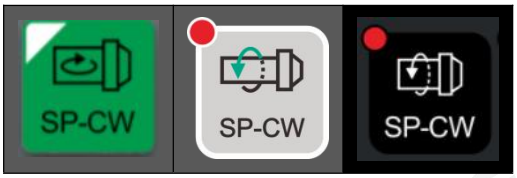
This function enables step-by-step execution of the machining program in Automatic or MDI mode.

- **Skip Function**

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

- **Optional Stop (M01)**

This function pauses the program at blocks containing the M01 command during execution in Automatic or MDI mode.



- **Spindle Forward**

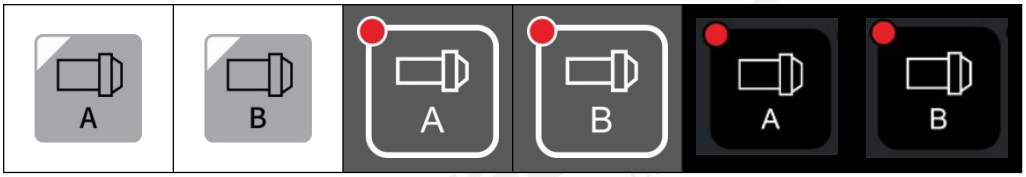
Activates the spindle in the forward direction.

- **Spindle Reverse**

Activates the spindle in the reverse direction.

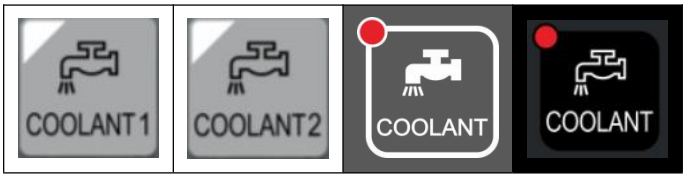
- **Spindle Stop**

Stops the spindle.



- **Spindle A/B**

Controls forward rotation of spindles A and B.



- **Coolant**

Manually toggles the coolant on or off.

- **Lubrication**

Triggers a single lubrication cycle manually.

- **Tool Magazine Forward/Reverse**

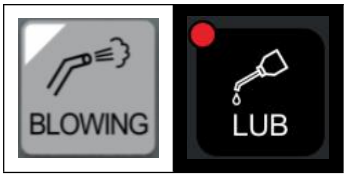
Controls forward or reverse rotation of the tool magazine.

- **Chuck**

Controls the chuck to open or close.

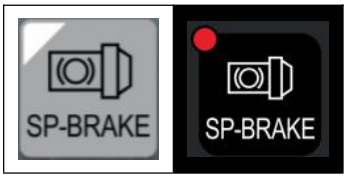
- **Work Light**

Controls the work light.



- **Air Blow**

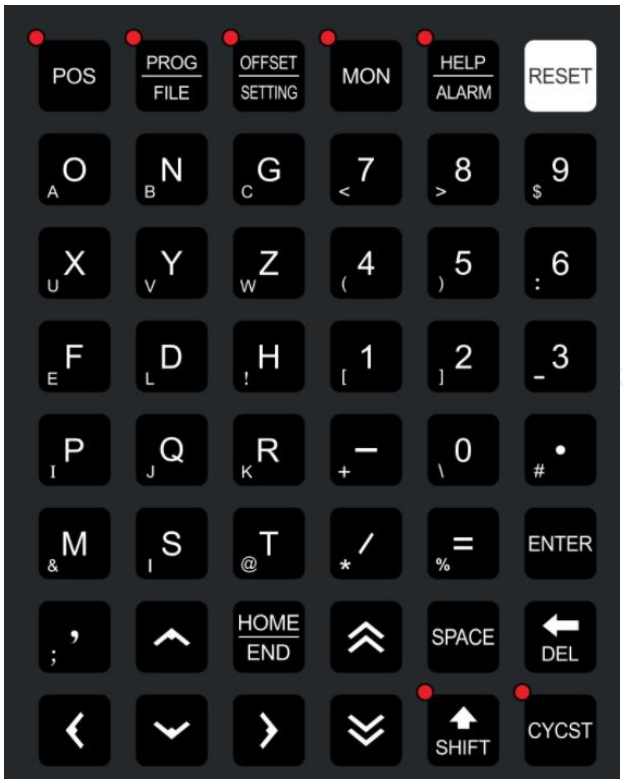
Controls the air blow system.



- **Spindle Brake**

Controls the spindle brake.

2.2System Text Keys Description



- Position: Switches to the machine position page.
- Edit/Program Select: Toggles between the program editing page and the program selection page.
- Tool Offset/Settings: Switches between the wear offset page and the length offset page.
- Monitoring: Switches to the machining monitoring page.
- Help/Alarm: Toggles between the help page and the alarm monitoring page.
- A~Z Keys: Alphabet keys (26 in total).

- 0~9 Keys: Numerical input keys.
- Input: Used for entering letters, numbers, symbols, etc.
- Delete: Backspace key to delete characters.
- Space: Inserts a space.
- Reset: Commonly used to terminate a program, activate parameter changes, or clear alarms after parameter modifications.
- Start of Line/End of Line: Moves the cursor to the beginning or end of the line.
- Switch: Enables input of the subscript characters located in the lower-left corner of number and letter keys when pressed (indicated by a light).
- Start: Initiates program execution.
- Other Common Keys: Commonly used keys not individually detailed.

3.Operation

3.1 System Status

The system displays different status indications based on its current state. The operating status of each channel is displayed separately. Below are the conditions for triggering each status.

3.1.1 Not Ready

The system imposes different restrictions based on various alarms.

- **Trigger Conditions:**

1. There is an active alarm in the system.
2. The axis has not returned to the origin.

3.1.2 Ready

The system is ready for operation.

- **Trigger Conditions:**

1. The system alarm has been cleared, and the axis has returned to the origin.
2. When the system is in a "Processing" or "Pause" state, pressing the Reset button will switch the system status to "Ready."

3.1.3 Processing

The system is executing a program for machining.

- **Trigger Conditions:**

When the system is in the "Ready" state and begins executing the machining program, the system status will change to "Processing."

3.1.4 Pause

The system pauses the machining program during its execution.

- **Trigger Conditions**

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

- **Note**

During the pause state, the spindle can still operate normally.

3.2Alarm Status

Indicates whether the system currently has any active alarms.

3.2.1 Flashing Alarm

- **Trigger Conditions**

The alarm flashes when there is any active warning or error in the system.

3.2.2 Non-flashing Alarm

- **Trigger Conditions**

The alarm is not flashing when there are no active warnings or errors in the system.

3.3 Machine Preparation

3.3.1 Manual Functions

The controller provides four manual control functions for the axis, which are as follows.

3.3.1.1 Manual Continuous Feed

- **Description**

1、Drives the axis to move continuously in one direction.

2、Multiple axes can move simultaneously.

3、Allows multiple channels to drive axis movements simultaneously.

- **Operating Conditions**

1、The system has not triggered an emergency stop or serious axis errors.

2、The operation mode is set to "Manual Mode."

- **Operation Method**

1、Select the axis keys "X↑, X↓, Y↑, Y↓, Z→, Z←" to control the corresponding axis continuous movement.

2、The feed rate and acceleration/deceleration time can be set through parameters.

3、The feed rate is limited by the G00 maximum feed rate.

3.3.1.2 Manual Rapid Feed

- **Description**

1、Drives the axis to move continuously at a G00 rapid feed rate in one direction.

2、Multiple axes can move simultaneously.

3、Allows multiple channels to drive axis movements simultaneously.

- **Operating Conditions**

The system has not triggered an emergency stop or serious axis errors.

All axes have returned to their home positions.

The operation mode is set to "Manual Mode."

- **Operation Method:**

1、Press and hold the [Rapid] button to activate the function (the button light will turn on).

2、Select the axis keys "X↑, X↓, Y↑, Y↓, Z→, Z←" to control the corresponding axis rapid movement.

3、The feed rate is adjusted by the G00 maximum feed rate.

4、Acceleration/deceleration time is set through parameters.

3.3.1.3 Incremental Feed

- **Description**

1. Drives the axis to move a fixed distance in one direction.
2. Multiple axes can move simultaneously.
3. Allows multiple channels to drive axis movements simultaneously.

- **Operating Conditions**

The system has not triggered an emergency stop or serious axis errors.

The operation mode is set to "Incremental Mode."

- **Operation Method**

1、Select the axis keys "X↑, X↓, Y↑, Y↓, Z→, Z←" to control the corresponding axis movement.

2、Each press of the key triggers one movement.

3、The movement distance for each press is set as follows:

- ❖ X1: 0.001 mm per movement
- ❖ X10: 0.010 mm per movement
- ❖ X100: 0.100 mm per movement
- ❖ This setting is shared with the MPG multiplier selection.

3.3.1.4 Operation Method Feed

❖ **Description**

Drives the axis to move continuously in one direction.

❖ **Operating Conditions:**

1、The system has not triggered an emergency stop or serious axis errors.

2、The operation mode is set to "MPG Mode."

3、Only one channel can be selected for axis movement at a time. Multiple channels cannot be selected for axis movement.

● **Operation Method**

1、Switch the axis on the MPG or select the axis from the auxiliary panel to decide which axis to control with the MPG.

2、Rotate the MPG to drive the corresponding axis. The direction of MPG rotation determines the direction of axis movement.

3、The movement distance per click of the MPG is set as follows:

❖ X1: 0.001 mm per rotation

❖ X10: 0.010 mm per rotation

❖ X100: 0.100 mm per rotation

❖ This movement multiplier is shared with the incremental feed distance setting.

3.3.2 Program Processing

3.3.2.1 Automatic Execution

❖ **Description**

Executes the "Processing Program" automatically.

❖ **Operating Conditions**

1. The system is in a no-alarm state.
2. The operation mode is set to "Automatic Mode."

❖ **Operation Method**

1、First, press the [Auto] button. The button light will turn on to indicate activation.

2、Press the [Start] button, and the system will execute the current processing program.

3、The system status will change from "Ready" to "Processing."

4、Once the processing program ends, the system status will switch back to "Ready."

5、If the program uses the M99 loop or the fully automatic mode, after the current program ends, the system will trigger the start again and continue the program execution.

3.3.2.2 Single Block Execution "MDI"

- **Description**

Executes "MDI" (Manual Data Input) processing.

- **Operating Conditions**

1、The system has not triggered an emergency stop, axis errors, or other critical alarms.

2、The operation mode is set to "MDI Mode."

- **Operation Method**

1、First, press the [MDI] button. The button light will turn on to indicate activation.

2、The system will execute the current single block program content.

3、The system status will change from "Ready" to "MDI."

4、Once the program ends, the system status will switch back to "Ready."

5、If the M99 loop is used, the system will continue executing the program.

3.3.3 Origin Mode

Processing coordinates are set based on the mechanical origin as a reference. Therefore, after powering on the controller, a return-to-origin operation is required to confirm the mechanical origin. When using a bus absolute value encoder, the return-to-origin operation is not necessary.

- **Operating Conditions:**

1、The system has not triggered an emergency stop, axis errors, or other critical alarms.

2、The operation mode is set to "Origin Mode."

- **Operation Method**

1、Press the axis keys "X↑, X↓, Y↑, Y↓, Z→, Z←" to move the corresponding axis to its origin position.

2、Once the axis completes the return-to-origin operation and stops, the system will reset the mechanical coordinate for that axis to zero.

3、The return-to-origin method, direction, and speed can be configured through parameters.

- **Notes**

he software limit is invalid during the return-to-origin process for all axes.

3.4 Tool Preparation

While editing the processing program, the relationship between the tool tip and the workpiece is described, without considering the actual tool position or the length differences between different tools. Therefore, tool offset settings need to be properly measured and entered into the controller before processing.

The system provides the following 4 types of offset methods: Tool Length Offset, Tool Wear Offset, Tool Tip Compensation, and Workpiece Coordinate Offset.

- Tool Length Offset：

Tool number is enabled by T-code, and the tool number corresponds to the offset number.

- Tool Wear Offset:

Tool wear compensation is enabled by T-code, often used for tool wear, thermal expansion and contraction, or fine adjustments to tool length calibration.

- Tool Tip Radius Compensation：

Tool number is enabled by T-code, and tool tip radius compensation is activated using G41/G42 in the program.

- Workpiece Coordinate System Offset

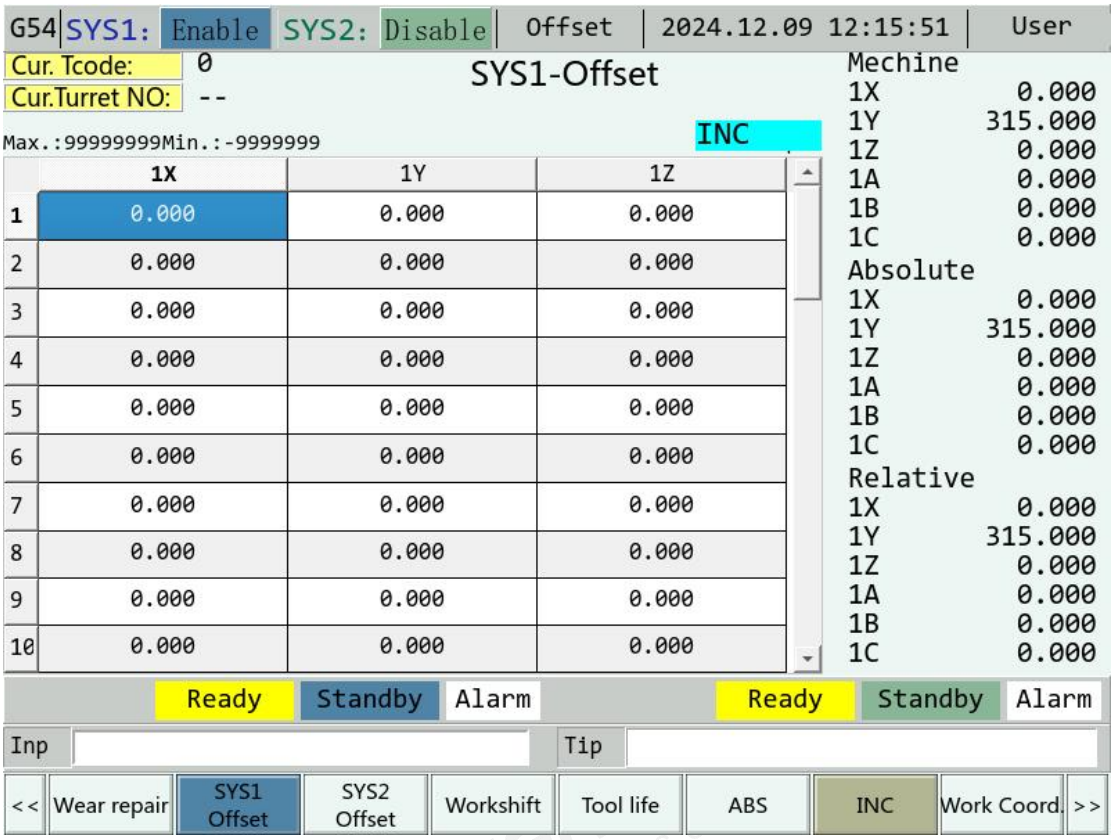
G54.1P1-G54.1P48, G54-G59 coordinate system settings, and workpiece shift coordinate system settings.

3.4.1 Tool Length Offset

While editing the processing program, the relationship between the tool tip and the workpiece is described, without considering the actual tool position or the length differences between different tools. Therefore, each tool's length offset needs to be carefully measured and entered into the controller before processing.

- **Operation Description**

Press the shortcut button [Tool Offset/Setting] twice to enter the Length Offset page, or on the "Machine Position" page, press [Offset/Setting] and then press [Length Offset] to enter the Tool Length Offset page.



1. Use Arrow Keys (↑, ↓, ←, →) to move the cursor to the desired value position.
2. Three Input Methods:

❖ Absolute Input (= key): The button turns green, the cursor moves to the corresponding position, and you directly input the value. This is generally used for absolute values, such as tool radius compensation or tool length offset.

❖ Incremental Input (+ key): The button turns green, the cursor moves to the

corresponding position, and you input the value to be added. The position value will then be the sum of the current value and the input value. This is generally used for tool radius wear compensation or tool wear correction.

- ❖ Teach-in Input (X***, Z***, etc.): You can input values using the axis names like X, Z, Y, or X1, X2, etc., where "***" represents the taught value. This method is used for teaching tool length offset (this process is also called tool setting).
- ❖ Use the [SYS1] and [SYS2] buttons to switch between tool length offset or wear compensation pages for different channels.

3、Compensation Calculation Formula

- ❖ Actual G41/G42 Compensation: (Tool Radius Compensation + Tool Radius Wear Compensation)
- ❖ Actual Tool Compensation: (Tool Length Compensation + Tool Wear Compensation)
- ❖ When using teach-in input, the cursor does not need to move to the correct axis position. It only needs to move to the correct tool number, and the controller will automatically fill in the values based on the taught axis name.
- ❖ When teach-in input is completed, the corresponding wear compensation value will be cleared. For example, if the tool length for Tool No. 1 along the X-axis is taught, the wear compensation value for Tool No. 1 along the X-axis will be reset to zero.

3.4.2 Tool Length Measurement

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

- **Before measurement, confirm the following:**

1、Whether the external offsets are correct.

2、Whether the workpiece coordinate system offset is correct.

3、Whether the workpiece move coordinate system offset is correct.

3.4.2.1 Z-Axis Tool Length Measurement

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

- **Operation Steps**

1、Move the cursor to the tool number whose compensation value you wish to modify, e.g., Tool No. 1.

2、Use manual operation to move the tool until the tool tip touches the Z-axis work zero point, which is the origin of the Z-axis program coordinate. Then, use the teach-in input "Z0" to set the current position as the Z-axis work zero point.

- **Z-Axis Cutting Measurement Input**

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

- **Operation Example**

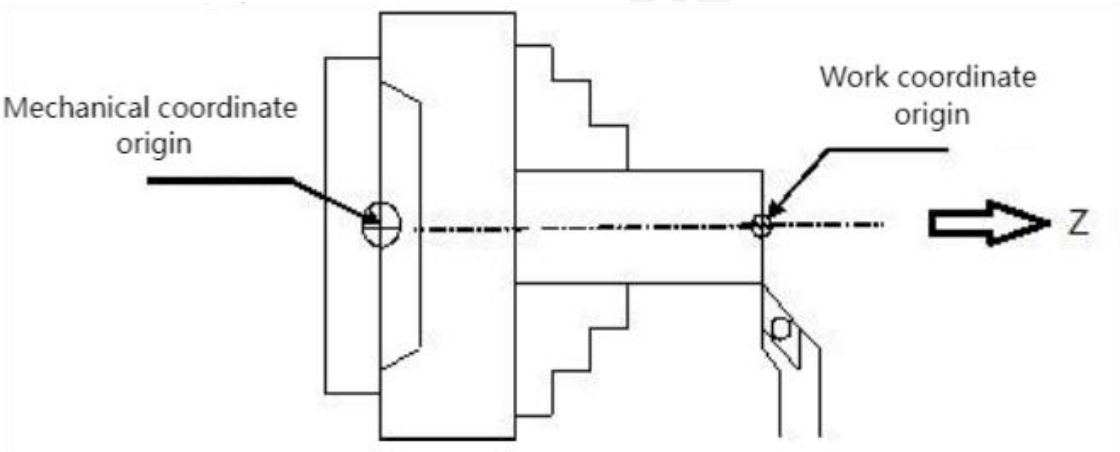
1、Clamp a round bar in the spindle.

2、Manually move the tool tip above the workpiece surface and confirm that cutting along the X-axis direction can reach the workpiece.

3、Rotate the spindle in the forward direction and manually cut along the X-axis.

4、Retract the tool along the X-axis, but do not move the Z-axis.

5、Use the teach-in input "Z0", and the cut surface will be set as the Z-axis work zero point.



- **Description**

When the teach-in input is completed, the corresponding wear compensation value will be cleared. For example, if the Z-axis tool length is set using the teach-in input for Tool No. 1, the wear compensation value for Tool No. 1's Z-axis 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 to assist in the measurement.

- **Operation Steps**

1、Press the shortcut key 【Tool Offset/Setting】 twice to enter the length offset page, or on the "Machine Position" page, press 【Offset/Setting】, then press 【Length Offset】 to enter the tool length offset page.

2、Move the cursor to the tool number whose compensation amount needs to be modified, for example, Tool No. 1.

3、Manually move the tool until the tool tip touches the X-axis work zero point, i.e., the origin of the X-axis program coordinates. Use teach-in input "X0" to set the current work zero point position.

- **Operation Example**

1、The spindle holds a round bar with a diameter of 10mm.

2、Manually move the tool until the tool tip is less than 5mm away from the center of the workpiece, ensuring that the tool can cut the workpiece when moved along the Z-axis.

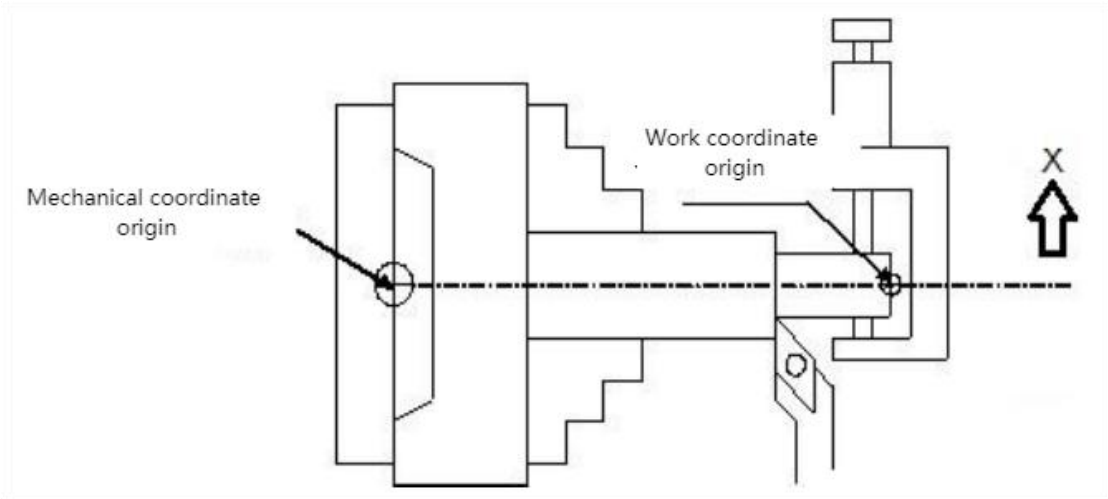
3、The spindle rotates forward, and the tool is manually moved along the Z-axis direction to perform turning.

4、Retract the tool along the Z-axis direction, without moving the X-axis.

5、Use a caliper to measure the diameter "D" of the cut workpiece.

6、Use teach-in input to enter this measurement value, "X+/-D". For example, if the workpiece diameter is measured to be 9mm, input "X9." If using the X radius axis, input "X4.5."

7、The positive/negative sign is determined by the machine's front or rear tool post. If the tool tip is located at X+ (tool tip is on the centerline), the sign is "+"; if the tool tip is at X-, the sign is "-".



- Description

When the teach-in input is completed, the corresponding wear compensation value will be cleared. For example, if the X-axis tool length is set using teach-in input for Tool No. 1, the wear compensation for Tool No. 1's X-axis will be cleared to zero.

3.4.3Tool Wear Compensation

1、Tool Wear Setting is commonly used for adjusting tool wear, thermal expansion and contraction, or fine-tuning tool length calibration. After the tool length is set on a lathe, the tool length value is not randomly modified, but the tool wear setting is used to fine-tune cutting dimensions.

2、After setting the tool length via teach-in input, the tool wear is automatically set to zero. Based on trial cutting results and measured dimensions, if there is any error, tool wear compensation is used to adjust the tool length.

3、Actual Tool Length = Tool Length + Tool Wear.

- **Operation Description**

1、Press the shortcut key【Tool Offset/Setting】to enter the wear compensation page, or on the "Machine Position" page, press【Offset/Setting】to enter the tool wear compensation

page.

2、Use the page-up/down keys (≯, ≮) or arrow keys (↑, ↓, ←, →) to move the cursor to the position where you want to change the value.

- **Operation Method**

1、Press【= Input (Absolute Input)】key to make the button turn green. The cursor moves to the corresponding position, and the value can be directly input.

2、Press【+ Input (Incremental Input)】key to make the button turn green. The cursor moves to the corresponding position, and the value to be added is entered. The new value will be the sum of the current value and the input value.

3、Positive/Negative Symbol +/-: The input value can be designated with the symbol “+/-”, which determines the direction of the compensation.

- ❖ If the tool tip is to be moved in the positive direction, input the plus sign “+”.
- ❖ If the tool tip is to be moved in the negative direction, input the minus sign “-”.

- **Example:**

If the turning result is 10μm larger than programmed, the tool tip will be adjusted to X-, and then input "-0.010" into the tool wear value. This will add "-10μm" to the current tool wear value, and during the next cutting, the tool tip’s cutting path will shift 10μm in the X-negative direction.

- **Parameters:**

The maximum value for tool wear compensation can be set via parameters to prevent errors during input that could cause collisions.

3.4.4 Tool Tip Radius Compensation

Because the tool tip is rounded, only the precise turning tool length is measured. The tool tip radius is used to compensate for tool tip dimension errors.

- **Operation Description:**

1、Press the shortcut key【Tool Offset/Setting】to enter the tool tip radius wear compensation/compensation page, or on the "Machine Position" page, press

【Offset/Setting】to enter the tool tip radius wear compensation page, then press【Length Compensation】to switch to the tool tip radius compensation page.

2、Use the page-up/down keys (≯, ≮) or arrow keys (↑, ↓, ←, →) to move the cursor to the position where you want to change the value.

3、There are three types of tool tip data：

- ❖ Tool Tip Radius: The radius of the tool tip.
- ❖ Tool Tip Radius Wear: Tool tip radius wear, where the actual tool tip size = Tool Tip Radius + Tool Tip Radius Wear.
- ❖ Tool Tip Direction: The tool tip direction, with eight directions to choose from, based on the shape of the tool tip (please refer to the "Yida Lathe Programming Manual" for more details on G41/G42). Use G41/G42 to activate the tool tip compensation.

3.5 Program Preparation and Execution

This section introduces how to specify the machining program for processing and how to conduct machining tests.

3.5.1 Specify Machining File

- **Operating Conditions:**

This applies to all modes except "MDI" mode.

- **Operation Steps:**

1、pecify the current program being edited as the machining program.

- ❖ Switch to the "Program Edit" page.
- ❖ Click【Load Machining】to switch to the "Machining Monitoring" page.
- ❖ The program from the "Program Edit" page will be specified as the machining program.

2、Specify a program file from the "File Management" as the machining program.

- ❖ Switch to the "Program Selection" page.
- ❖ Move the cursor to the program you want to load for machining.

❖ Click【Load Machining】to switch to the "Machining Monitoring" page.

❖ The program from the "Program Selection" page will be specified as the machining program.

- **Verification:**

You can confirm whether the machining program has been successfully specified through the following:

❖ The program name displayed at the top of the screen.

❖ The program content displayed in the "Machining Monitoring" page.

3.5.2 Graphic Simulation

The system provides a convenient program content simulation function. After editing the program, it is easy to simulate the machining path of the program. This function also includes a program check feature that helps users quickly detect syntax errors or unreasonable actions in the machining program. It is recommended to check the program content using this function after editing.

- **Operation Steps:**

1、In the "Program Selection" page, choose the program to be edited.

2、After editing the program, click【Graphic Simulation Display】->【Graphic Adjustment】->【Static Tracing】on the "Machining Monitoring" page.

3、Scan the program content.

4、Once the scan is complete, the system will begin graphic simulation based on the program content, continuing until the entire program is simulated.

- **Description**

1、In the simulation graphic:

❖ Solid lines represent cutting paths.

❖ Dotted lines represent movement paths.

2、If there are any syntax errors or issues in the program during the scan, the system will trigger an alarm and display the error line number.

3.5.3 Test Machining

3.5.3.1Program Prediction (MPG Simulation)

This section explains how to use the MPG simulation for machining tests.

- **Operating Conditions:**

Available in "MDI" and "Automatic Execution" modes.

- **Operation Steps:**

Execute the program machining.

Press the【Program Prediction】key on the control panel.

Turn the MPG to simulate machining.

Turn the MPG clockwise to execute the program from the current line to the next.

Turn the MPG counterclockwise to execute the program from the current line to the previous line. This feature is also known as "MPG Rewind.

- **Verification:**

1. You can confirm if handwheel simulation has been successfully enabled through the following:
2. The【Program Prediction】light on the auxiliary panel is on.
3. During machining, once the handwheel simulation is activated, the machine immediately slows down to 0, until the handwheel is moved or the 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:**

Available in "MDI" and "Automatic Execution" modes.

- **Operation Steps:**

1、Execute the program machining.

Press the【Single Block Execution】key on the control panel.

After completing the current single block, the system will decelerate and stop at 0, entering the single block stop state.

Press the【Start】key to resume machining. After completing the next single block, the system will again enter the single block stop state.

3.5.4 Machining Monitoring

This section explains how to manage the workpiece count.

- **Operating Conditions:**

None.

- **Description**

1、Total Workpieces:

The total number of workpieces processed by the machine is accumulated.

2、Required Workpieces:

The required number of workpieces specified in a single machining program. When the program runs with M99 for continuous machining, the machine will pause when the specified workpiece count is reached, and a prompt will be issued indicating that the required workpiece count has been reached.

3、Workpiece Count:

❖ When the program runs with M99 for continuous machining, the workpiece count will keep accumulating.

❖ Workpiece count reset (cleared to 0) occurs under the following conditions:

① The required workpiece count is reached.

② The machining program is switched.

③ The required workpiece count is modified, and the new required count is less than the current workpiece count.

④ M16 is executed.

3.6 System Alarm Handling

To avoid errors that could compromise personnel or machine safety, the system or PLC has been configured with numerous protective measures. When these protective conditions are triggered, the system will issue warnings or alarms to alert the user. This section explains how to check and resolve alarms when they occur.

3.6.1 Emergency Stop

When a machine malfunction or unexpected action occurs that could jeopardize personnel or machine safety, pressing the emergency stop button will immediately stop the machine’s operation. Once the emergency stop button is pressed, it will be locked. While the specifications may vary depending on the manufacturer, usually rotating the button will unlock it. This button can interrupt the machine’s operation, and the issue must be resolved before it can be unlocked.

3.6.2 Alarm Display

Alarms are divided into current alarms and historical alarms. The operation to view these alarms is described in section 1.8.8 "Alarms."

3.6.2.1 Current Alarms

1. The current alarm status of the system.
2. When an alarm occurs, the controller will pop up an alarm window displaying the alarm content.
3. Pressing the back button 【《】 will cancel the popup.
4. If the alarm has not been cleared, pressing 【Reset】 will pop up the alarm window again. Note that for some alarm types, pressing reset may clear the alarm.
5. Switching to the "Alarms" page will automatically display the current alarms.

3.6.2.2 Historical Alarms

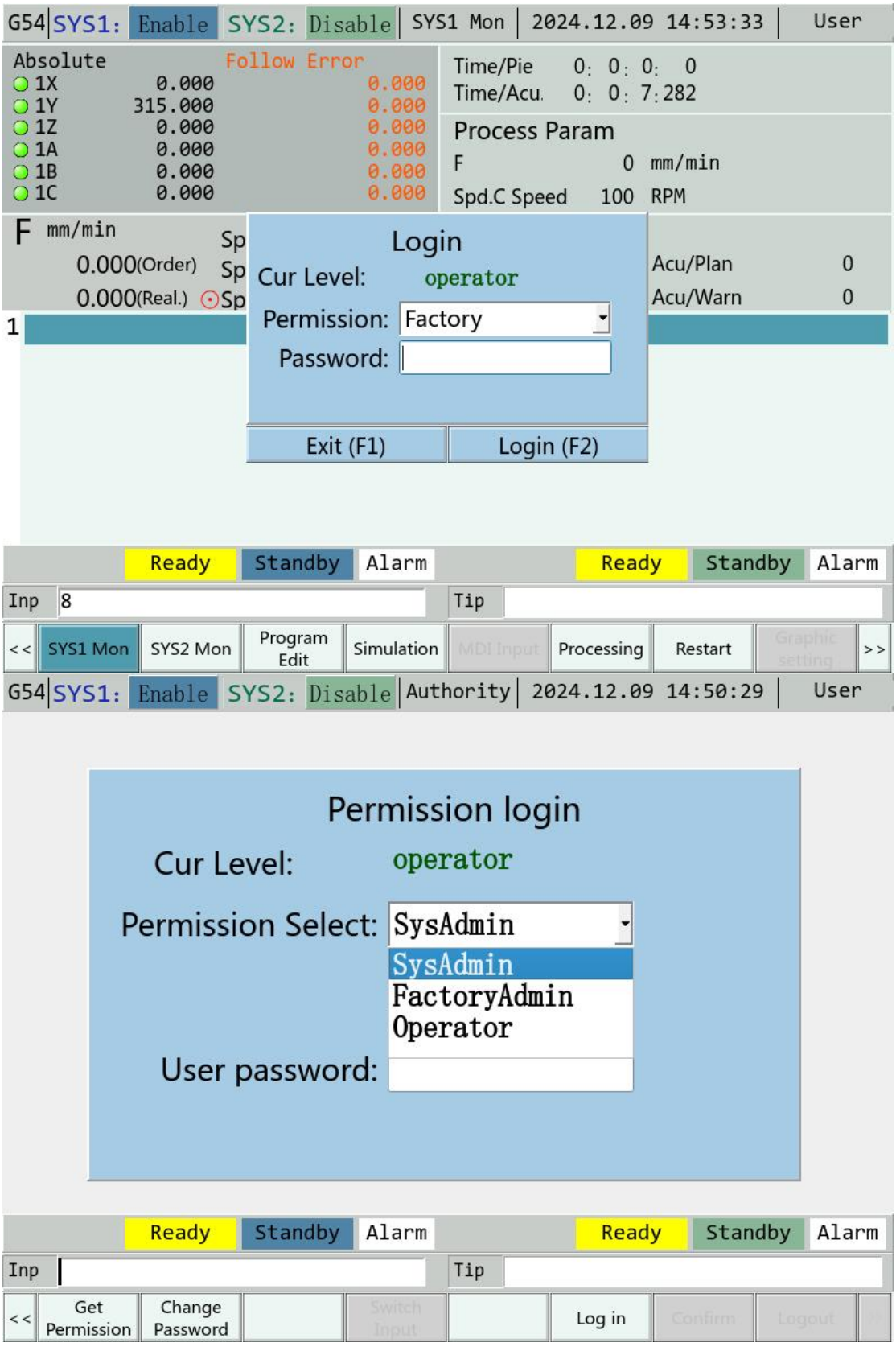
1. Alarms that have occurred in the system previously can be reviewed to help identify potential causes at the time of the alarm.
2. Switching to the "Alarms" page and clicking "Historical Alarms" will display past alarms.
3. When multiple alarms appear, they will be listed in order of occurrence, with the most recent alarms listed at the top.

4.Permission Management

4.1 Parameter and System Protection

This is used to protect special system functions and operations, making them accessible only to authorized personnel with the correct password. This prevents accidental misuse of system functions that could lead to system malfunctions.

4.1.1 Password Login



- **Operation Path**

1、Path 1: "Machine Position" page → F8 [Parameter Settings] → Enter values in the setting field, the system will automatically check the current parameter permissions and pop up the permission interface → F1 [Login].

Path 2: "Machine Position" page → F5 [Maintenance] → F8 [Permission Management] → F1 [Permission Access Interface].

- **Description**

This function is used to enter the password to obtain operation permissions.

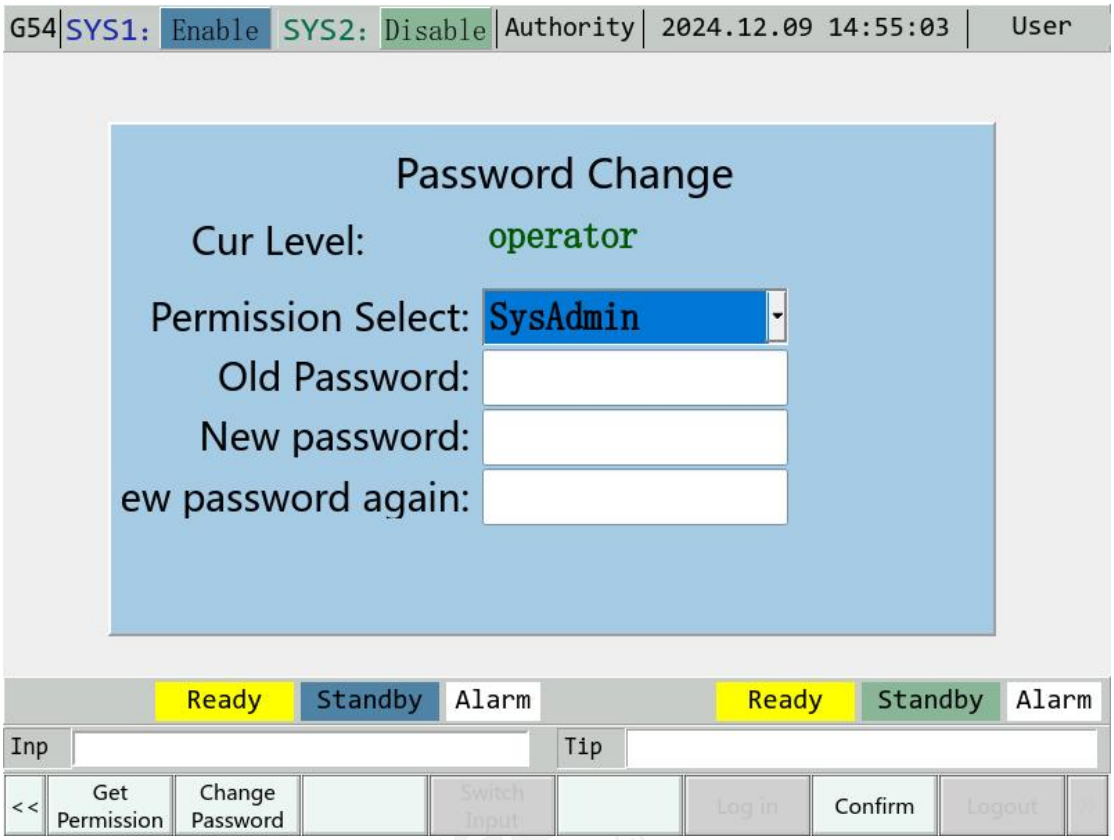
- **Operation Description**

1. Use the system panel's numeric keypad to enter the password.
2. Use the [∧][∨] keys to move to the "Permission Selection" and press the [Enter] key to confirm the operation permissions.

- **Note**

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

4.1.2 Password Modification



- **Operation Path**

"Machine Position" page → F5 [Maintenance] → F8 [Permission Management] → F2 [Password Modification Interface].

- **Description**

This function is used to modify the permission password.

- **Operation Description**

1. Use the system panel's numeric keypad to enter the original password and the new password.
2. Press F7 [Confirm Modify Password] to confirm the password change.

- **Note**

The original password and the new password cannot be the same. If they are, the system will display the message "The new password cannot be the same as the original password."

4.1.2.1 Move Down

- **Operation Path**

"Password Modification Interface" page → Press 【∨】 key.

- **Description**

Used to switch the input field and move the focus down.

4.1.2.2 Move Up

- **Operation Path**

"Password Modification Interface" page → Press 【∧】 key.

- **Description**

Used to switch the input field and move the focus up.

4.1.2.3 Confirm Password Modification

- **Operation Path**

"Password Modification Interface" page → F7 [Confirm Modify Password].。

- **Description**

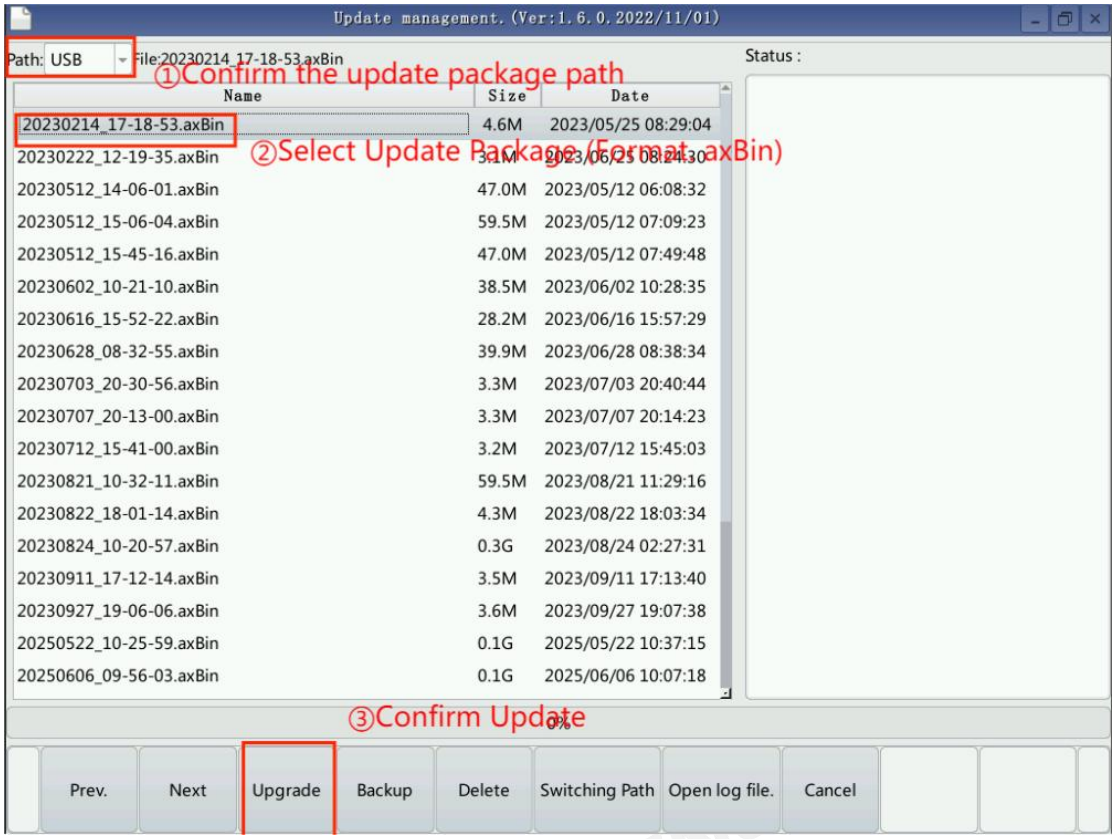
Confirms the password modification.

5. Appendix

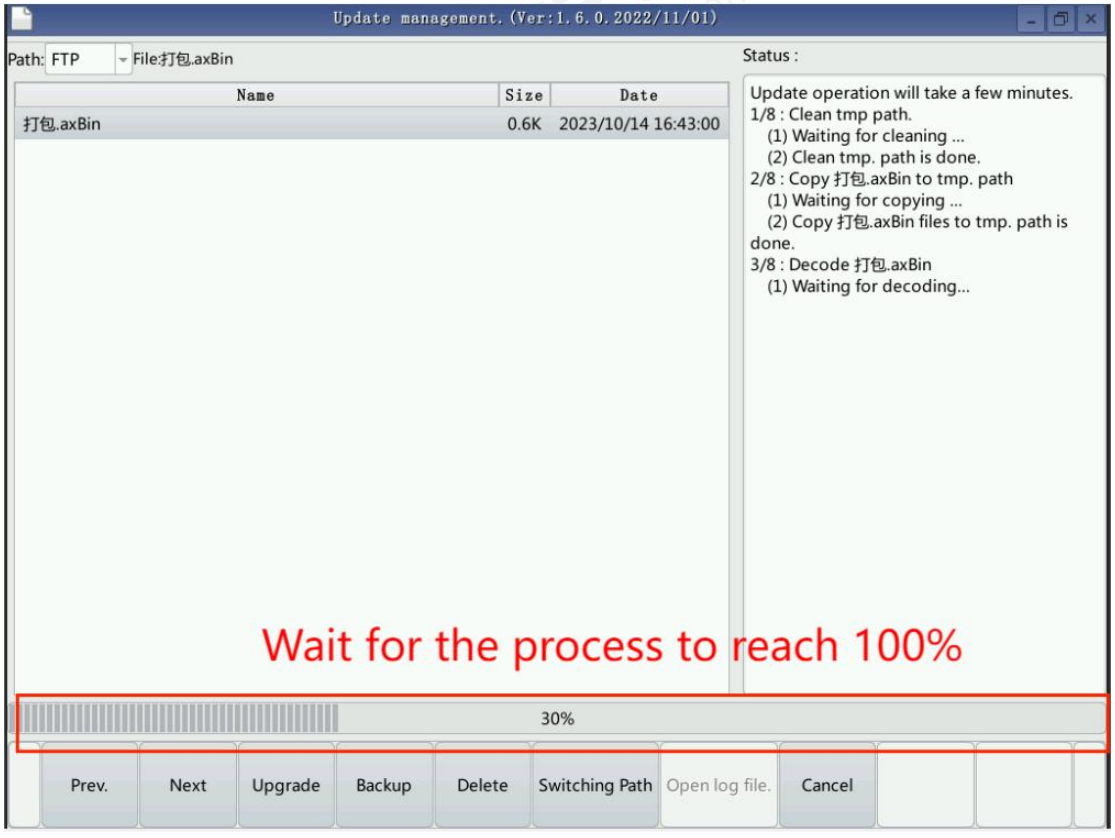
5.1 Software Upgrade/Backup Operation Steps

5.1.1 Software Upgrade Steps

Step 1: Transfer the update package to the USB drive and insert the USB drive into the CNC. Wait for the system to display the update interface.

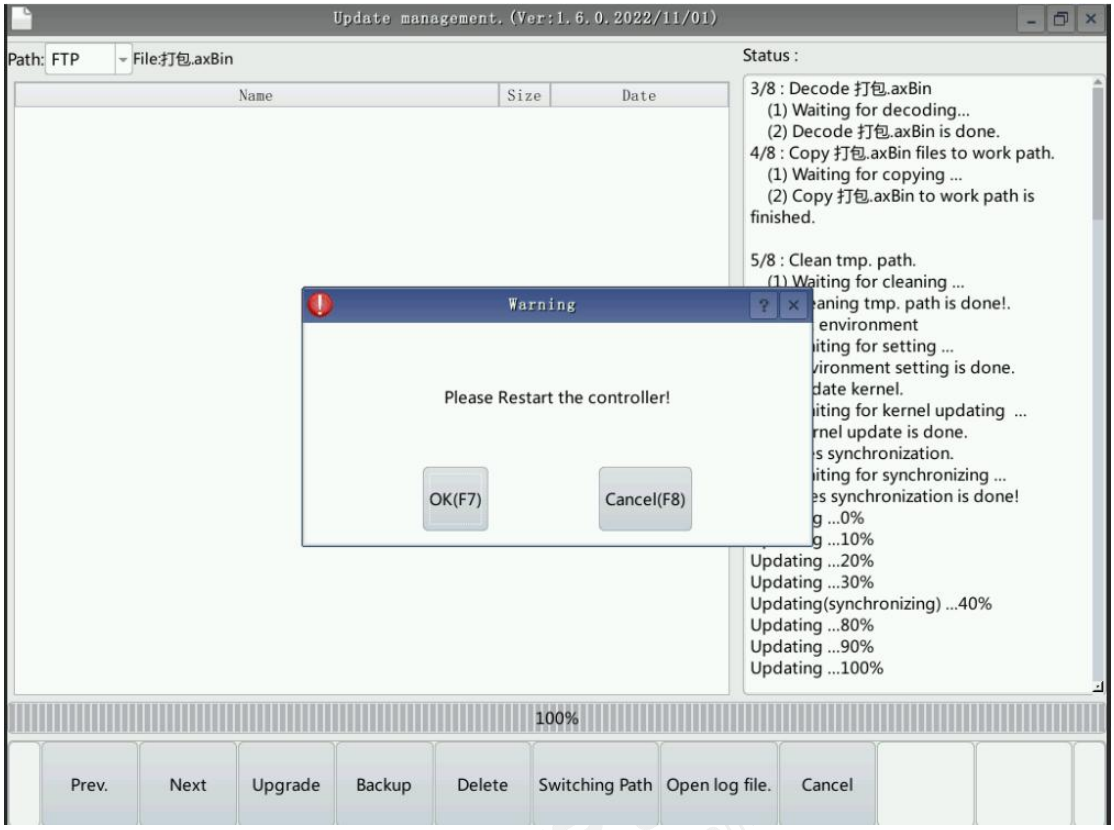


Step 2: During the update process.



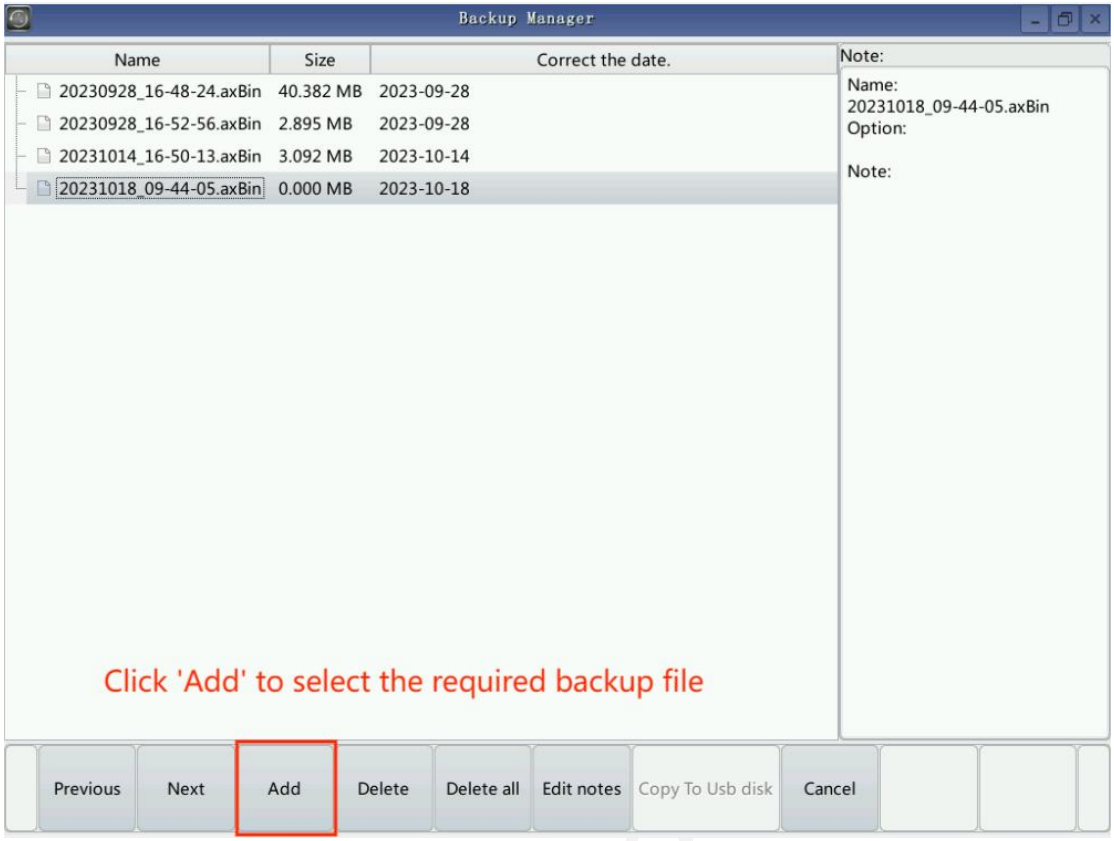
Step 3: After the update is complete: When the update progress reaches 100%, the message "Please restart the system!" will appear. After restarting the system, the update

will be finished.

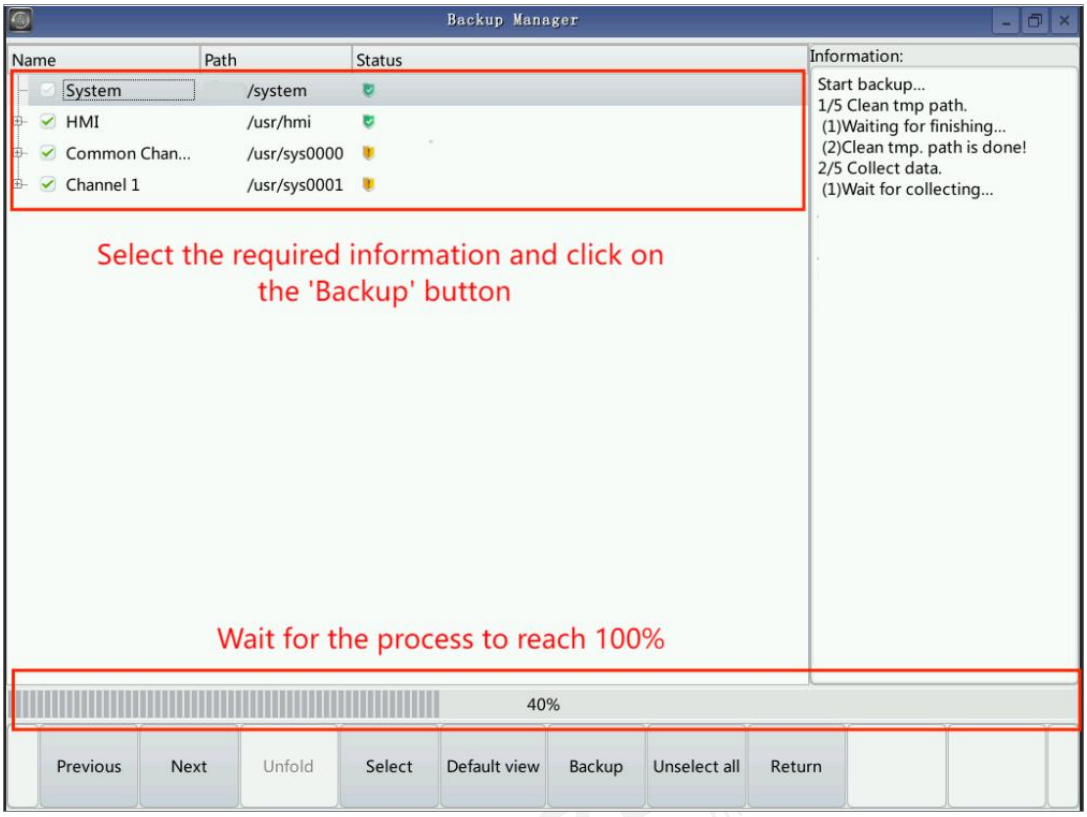


5.1.2 Software Backup Steps

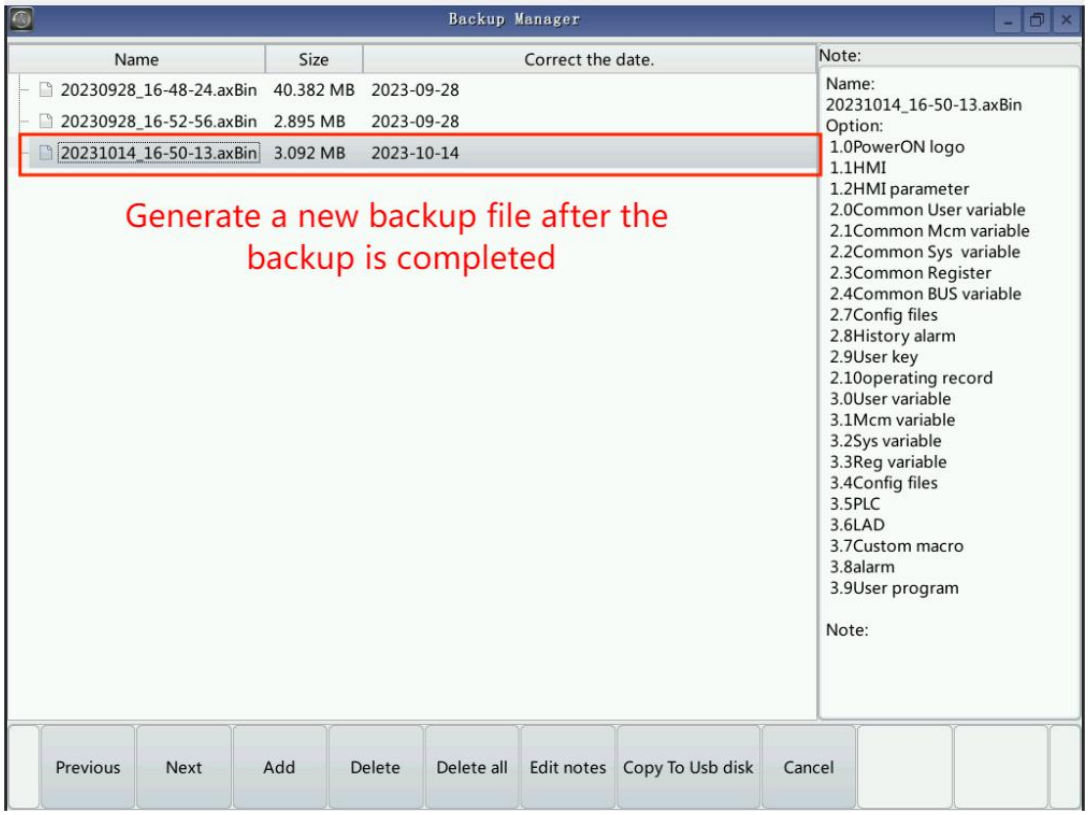
Step 1: Click the "Backup" button to enter the backup interface.



Step 2: Select the backup file and perform the backup operation.



Step 3: After the backup is complete, a new data file will be generated.



6.Lathe Wiring Description

6.1 System Installation Environmental Requirements

The B-series controller must be installed and used within the following environmental conditions. Using the controller outside these parameters may result in abnormal operation

Surrounding Environmental Conditions	**Operating Temperature**	0℃ to 45℃
	Storage or Transportation Temperature	-20℃ to 55℃
Humidity	**Normal Conditions**	Relative humidity less than 80% RH
	Short-term Conditions	Maximum 95% RH
Vibration Limits	**During Operation**	Maximum of 0.075 mm at 5 Hz frequency
Noise	**During Operation**	Maximum voltage pulse of 2000V/0.1 x 10^-6 seconds every 0.01 seconds
Temperature Rate of Change		Maximum 1.1°C per minute
Other		For environments with dust, cutting fluid, and organic solvents, please contact the manufacturer.

6.2 Controller Box Design Considerations

- The controller and auxiliary panel boxes must be sealed, preventing dust from entering.
- The internal temperature of the box should not exceed the surrounding environmental temperature by more than 10°C.
- The cable entry and exit points must be sealed.
- To avoid noise interference, the distance between cables, each unit, and AC power sources should be at least 100mm. If there are magnetic fields, the distance must be at least 500mm.
- For servo driver installation, refer to the servo operation manual.

6.3 Internal Temperature Design of the Box

- The internal temperature of the box should not rise more than 10°C above the surrounding environment temperature. When designing the box, the primary factors to consider are heat sources and heat dissipation area. While customers may have limited control over heat sources, the heat dissipation area is an important factor. The allowable temperature rise inside the box can be estimated using the following formula
- With cooling fan: Allowable temperature rise = 1°C per 6W per 1m²
- Without cooling fan: Allowable temperature rise = 1°C per 4W per 1m²
- The rise refers to the internal temperature increase when the box has a heat source of 6W (with cooling fan) or 4W (without cooling fan) and a heat dissipation area of 1m².
- Example 1 (with cooling fan):

❖ Heat dissipation area = 2m².

❖ Allowable internal temperature rise = 10°C.

❖ Maximum allowable heat source = 6W × 2 × 10 = 120W.

❖ If the heat source inside the box exceeds 120W, additional cooling devices, such as cooling fins, must be added.

- Example 2 (without cooling fan):

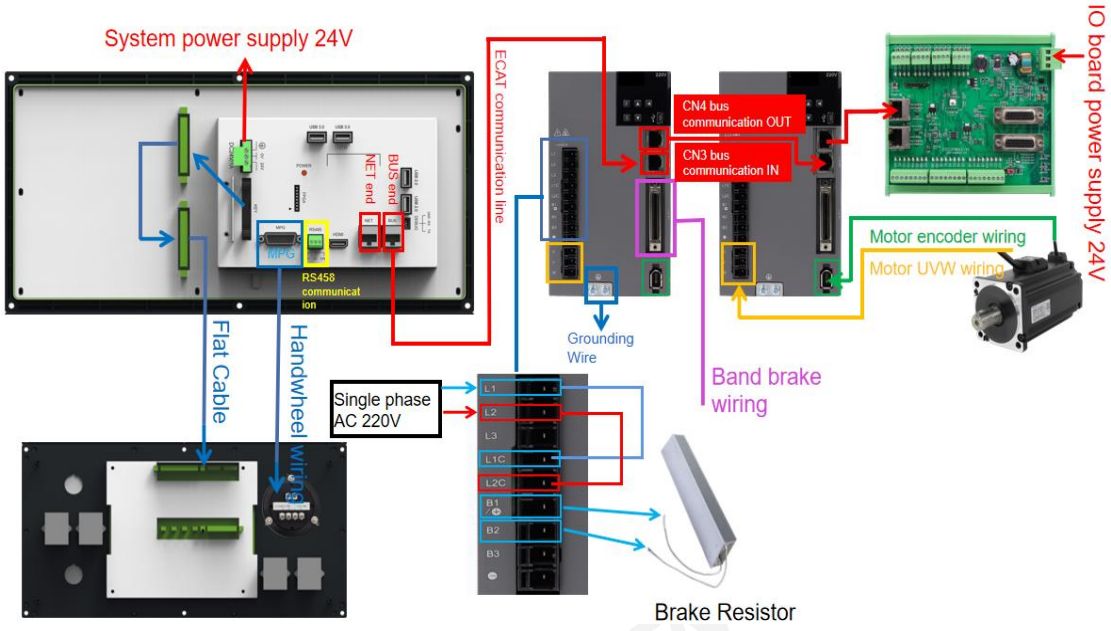
❖ Heat dissipation area = 2m².

❖ Allowable internal temperature rise = 10°C.

❖ Maximum allowable heat source = 4W × 2 × 10 = 80W.

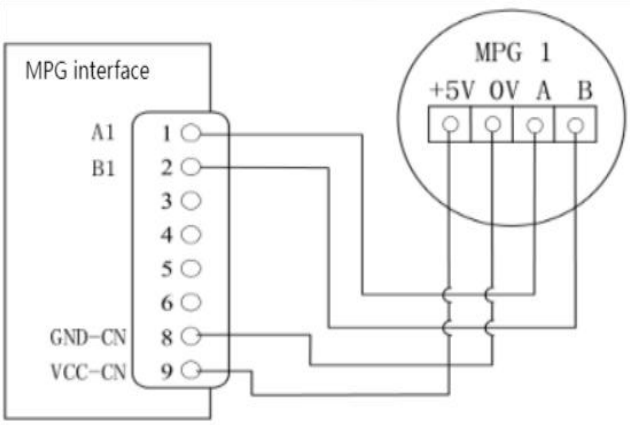
❖ If the heat source inside the box exceeds 80W, additional cooling devices, such as fans or cooling fins, must be added.

6.4System Wiring

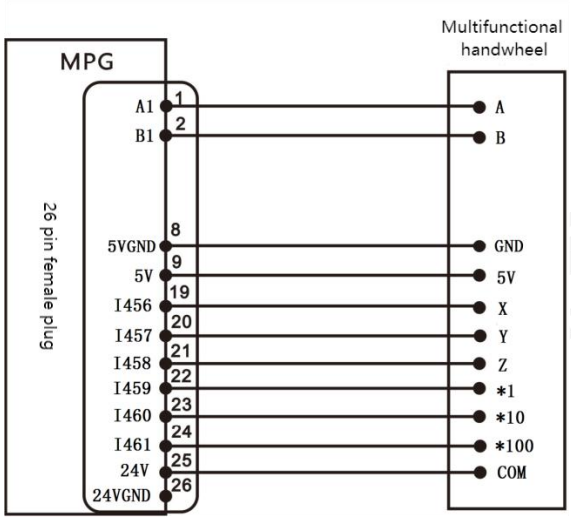


6.5 MPG Wiring

6.5.1 Standard MPG Wiring



6.5.2 6-Axis Multifunctional MPG Wiring

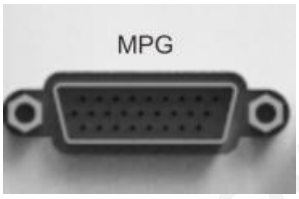


Note:Verify if the multifunctional MPG COM common terminal is powered by 24V.

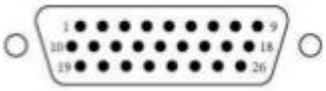
6.5.3 8-Axis Multifunctional MPG Wiring

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 Operation Method Port Pin Definition Diagram



● **Operation Method**



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	CAND+	CAND-
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 Definition

● **P1-P6 Servo Axial**

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 is grounded for 5V/VCMD/ADC**

*** Pin 7 VCMD is for a -10V to +10V analog signal**

● **Operation Method**

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	CAND+	CAND-
PIN	**19**	**20**	**21**	**22**	**23**	**24**	**25**	**26**	
Definitio	I456	I457	I458	I459	I460	I461	24V	24VG	

n								ND	

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

*** Pin 26 24VGND-CN is grounded for I-point/G31**

- **Spindle voltage command control - paired with the 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 grounded for 5V/VCMD/ADC**

*** Pin 7 VCMD is for a 0V to +10V analog signal**

- **Spindle voltage command control - paired with the servo driver**

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 is grounded for 5V/VCMD/ADC *Pin 7: VCMD for analog signal in the range of -10V to 10V									

● Spindle pulse command control - paired with the servo driver - 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 is grounded for 5V/VCMD/ADC**									

● Spindle pulse command control - paired with the servo driver - 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**	
Defini	24VG	24V	Enabl	Alarm	CCW-	CCW	CW-	CW+	

tion	ND		e			+			
*** Pin 8 GND is grounded for 5V/VCMD/ADC**									

注 Note:

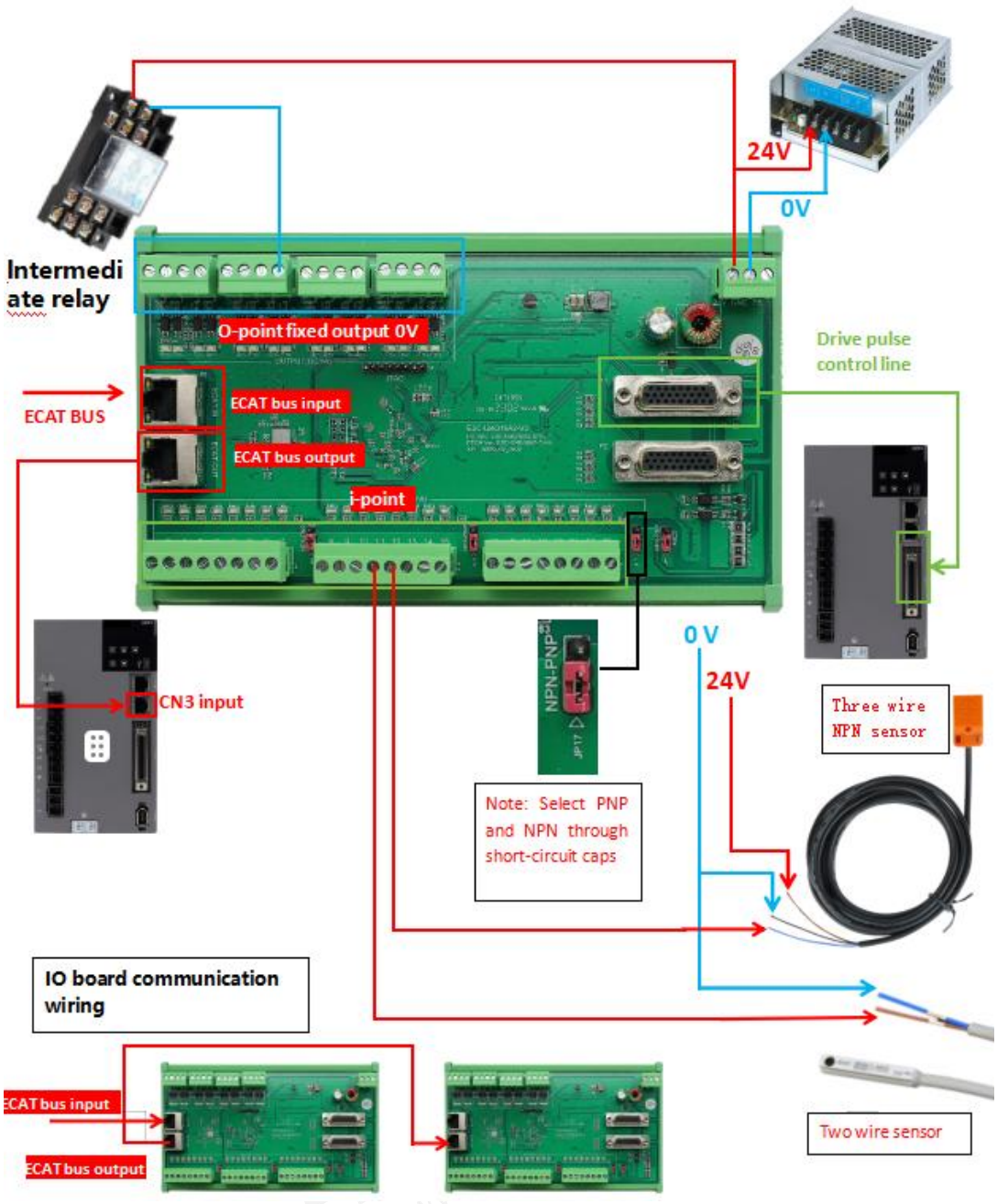
The pin definitions for each servo axis are the same except for the IO points. The pins marked with a "*" in the above table represent IO points. Pin 8 is for 5V power ground, voltage command ground, etc.; Pin 26 is for interface IO and 24V GND. It is prohibited to connect the GND pins (Pin 8 and Pin 26) together.

The IO point definitions for each axis are as follows:：

Pin position	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 Diagram



6.7.2 I/O Points Function Definition

I Points Definition		O Points Definition	
I0	Emergency Stop	O0	Cutting Fluid
I1	Foot Switch	O1	Lubricating Oil
I2	External Start	O2	Collet Loosen
I3	External Pause	O3	Collet Tighten
I4	Oil Injection Machine Abnormality	O4	Lighting
I5	Hydraulic Abnormality	O5	Red Light
I6	Insufficient Lubrication Pressure	O6	Green Light
I7	X-Axis Home Position	O7	Yellow Light
I8	X-Axis Positive Limit	O8	Hydraulic Station Start
I9	X-Axis Negative Limit	O9	Spindle Brake Tighten
I10	Y-Axis Home Position	O10	X-Axis Brake
I11	Y-Axis Positive Limit	O11	Y-Axis Brake
I12	Y-Axis Negative Limit	O12	Spindle Torque Reduction
I13	Z-Axis Home Position	O13	Blow Air On
I14	Z-Axis Positive Limit	O14	Z-Axis Brake
I15	Z-Axis Negative Limit	O15	Spindle Stop Positioning
I16	Cutting Fluid Abnormality (Reserved)	O16	Spindle Brake Loosen
I17	Reserved	O17	Reserved
I18	Reserved	O18	Tailstock Advance
I19	Safety Door Signal	O19	Tailstock Buffering
I20	Feeding Machine Feeds to Position	O20	Tailstock Retreat
I21	Feeding Machine Shortage	O21	Reserved
I22	Feeding Machine Alarm	O22	Part Catcher Extend
I23	Spindle Home Position Signal 1	O23	Safety Door Closed
I24	Spindle Home Position Signal 2	O24	Safety Door Open

I25	Tailstock Advance Knob	O25	Chuck 2 Tightened
I26	Tailstock Retreat Knob	O26	Chuck 2 Released
I27	Spindle Brake/Chuck Tightened Position Signal	O27	Feeding Machine Start
I28	Spindle Brake Release Position Signal	O28	Reserved
I29	Feeding Machine Material Change Completed	O29	Reserved

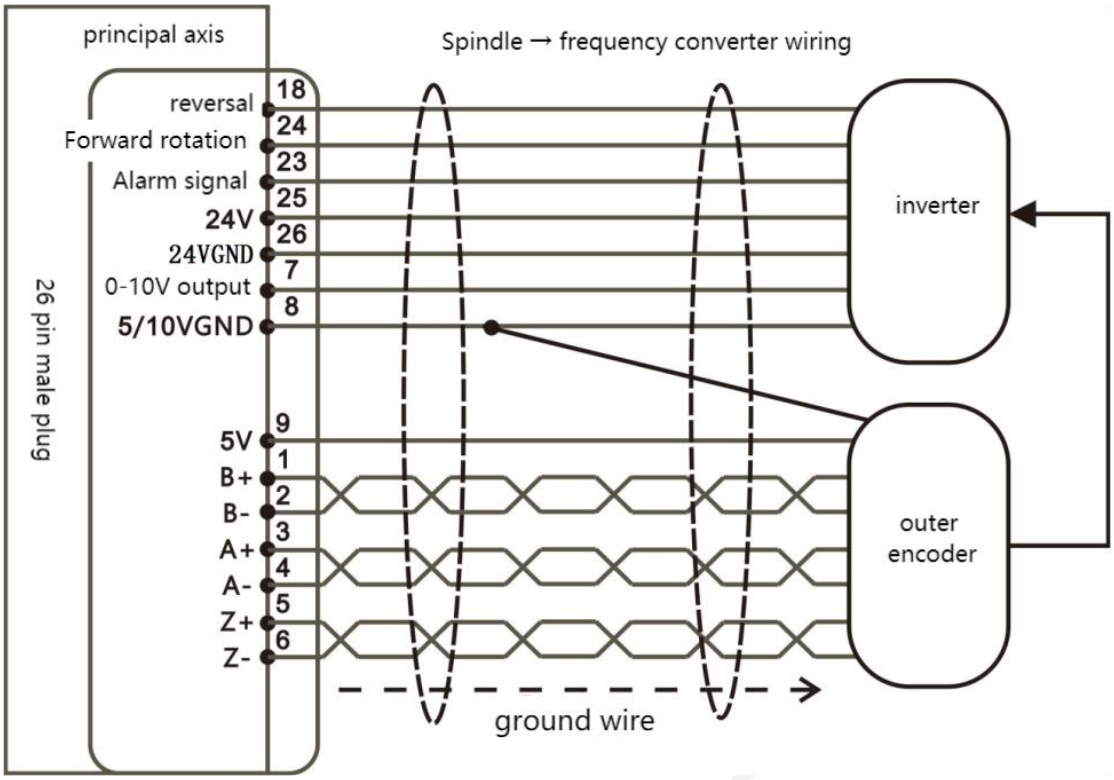
6.8 Spindle Control Wiring

Note: The following spindle control wiring methods are all PNP.

6.8.1 Voltage Command Type

A. With Inverter

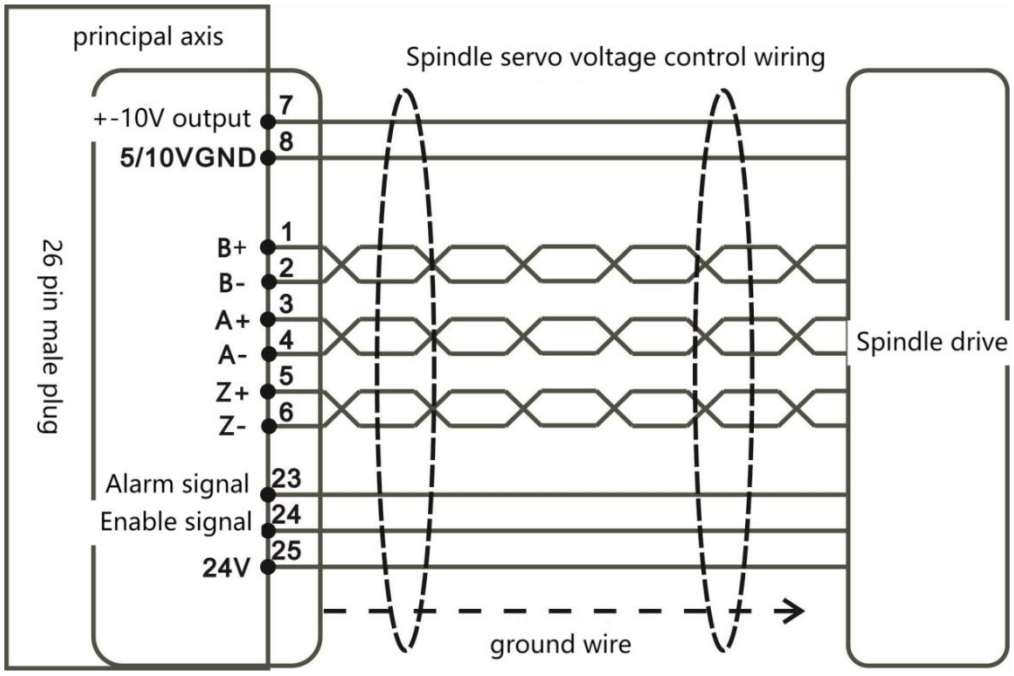
Voltage Command Control - Open-Loop Wiring：



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

B. With Servo Driver

Spindle Voltage Command Control:

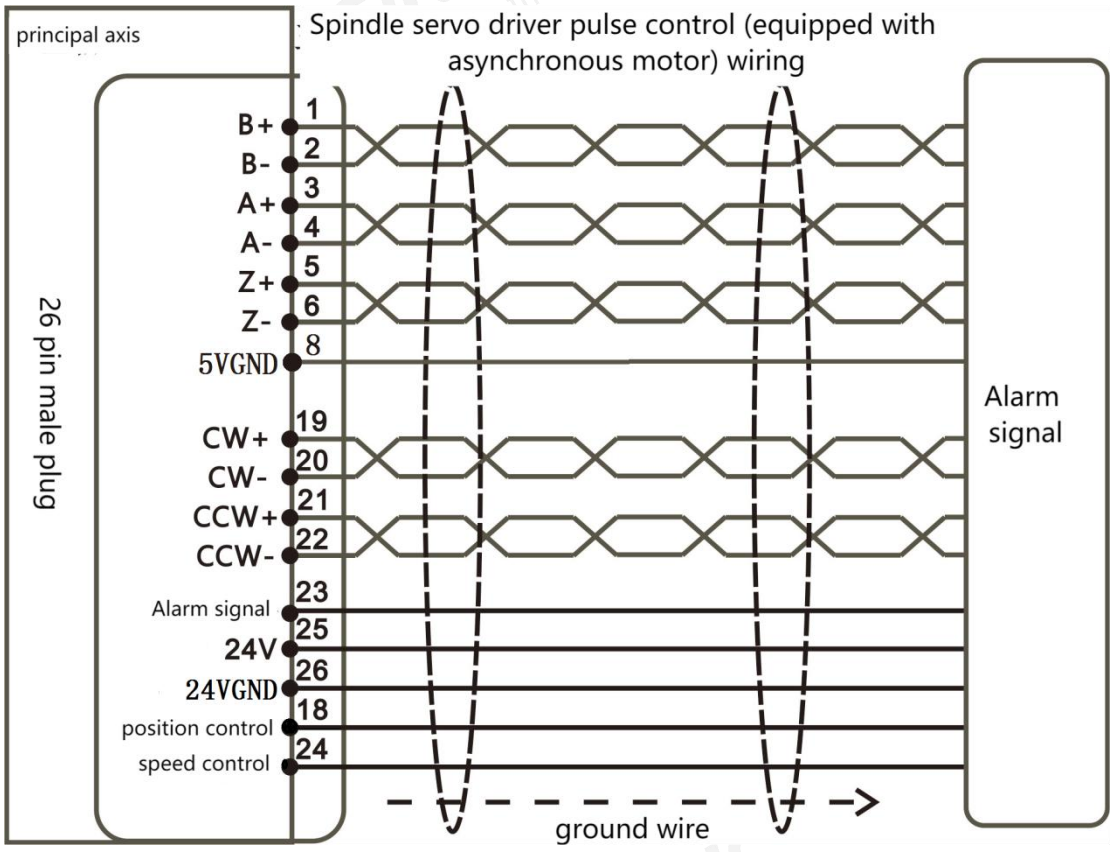


Note: For detailed pin definitions of the driver, please refer to the corresponding driver manual. The system axis connector is a 26-pin female header, so the spindle wiring plug must be a 26-pin male header.

6.8.2 Pulse Command Type

A. With Servo Driver - Asynchronous Motor (e.g., Ultra-Synchronous Spindle)

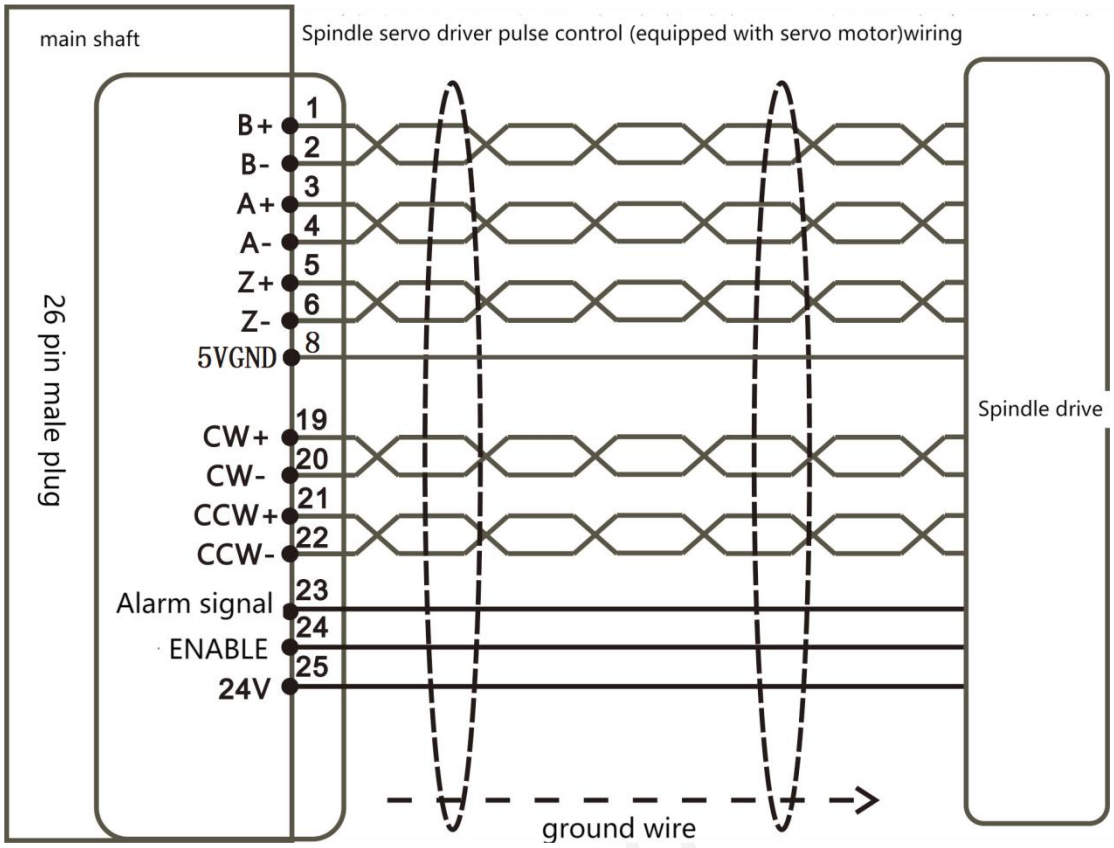
Spindle (servo) position mode and speed mode wiring:



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

B. With Servo Driver - Servo Motor (e.g., Yaskawa Spindle)

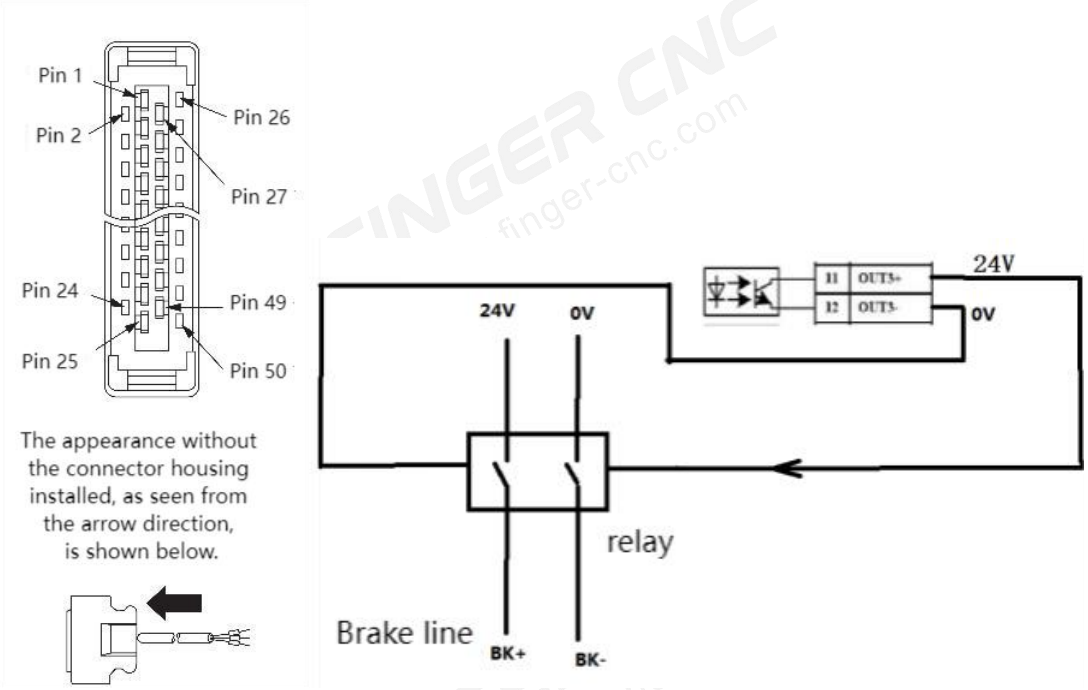
Spindle (servo) pulse command control:



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

6.9 The wiring diagram for the drive motor brake.

6.9.1 I7 Drive brake wiring



i7 Drive

Parameters: Pn50F=X2XX, third digit is 2, brake pins are 11 and 12.
Pn50F=X6XX, 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 driver cannot be set to the same value, otherwise, the brake output signal will be invalid.

6.10 Wiring Precautions

- Wiring Terminals: When wiring the machine, ensure that the wire ends are properly terminated or soldered.
- Wiring Accuracy: If non-standard servo cables (not Yida standard) are used, make sure to measure the correctness of all connection pins before powering up. Incorrect wiring can lead to abnormal controller command outputs and potentially cause controller malfunction.

- External 24V DC Power Supply: For the external 24V DC power supply used in wiring, make sure it has safety certifications and protective functions to avoid faults caused by wiring errors. (Recommended specifications: must meet EN60950 & UL1950 requirements).
- MECHATROLINK-III: If using the MECHATROLINK-III function, to ensure smooth network communication and avoid noise interference, it is recommended to use network cables of CAT5e or CAT6 specifications.
- Grounding Wire Description:

A. The size of the grounding wire should follow the technical standards of electrical equipment. The shorter the grounding wire, the better.

B. The controller's grounding wire should not share the same ground as high current loads such as welding machines or high-power motors. They must be grounded separately to avoid interference or damage to the system.

7.Disclaimer

This document provides information regarding FINGER CNC series products. It does not grant any intellectual property rights or licenses, whether explicitly or implicitly, and does not grant any rights through prohibition of speech or other means. Apart from the responsibilities stated in the product's sales terms and conditions, our company assumes no other liability. Furthermore, our company makes no express or implied warranty regarding the sale and/or use of the product, including but not limited to its suitability for specific purposes, merchantability, or any infringement of patents, copyrights, or other intellectual property rights. Unauthorized use without written permission from our company constitutes infringement, and the company will take legal action against infringers as per applicable laws. Our company reserves the right to modify product specifications and descriptions at any time without prior notice.

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：201,No. 8, Chengding Street, Zhongcun Street, Panyu District, Guangzhou City, Guangdong Province

Official Website

Official Wechat