Operating Manual for Milling Machine of AUCTECH controller V2.0

Table of Contents

I. Operating Instructions

1 controller Interface

1.1 Panel interface

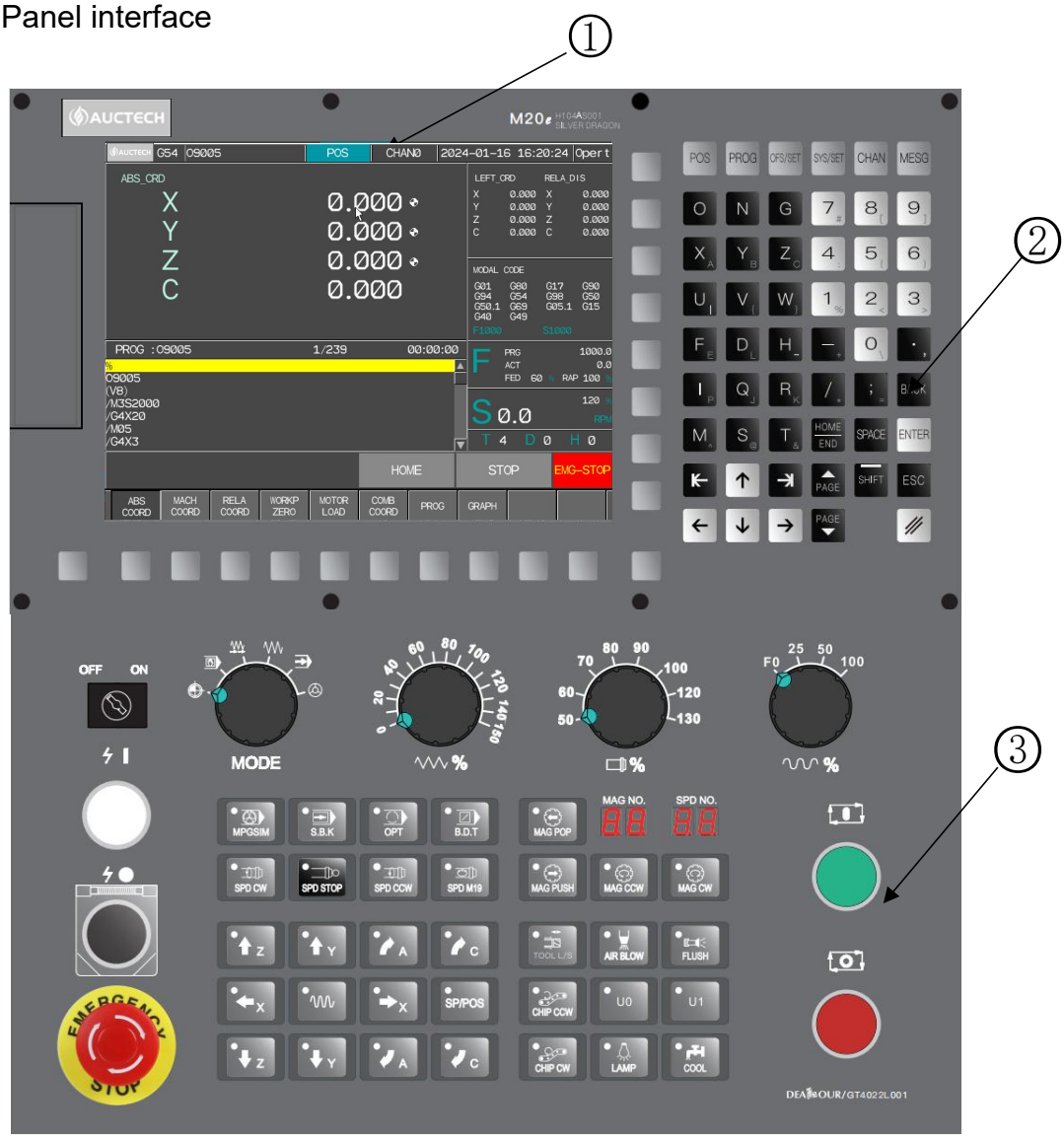


Fig. 1- 1 controller operation panel

1: Display of controller main screen

2: Character key program input panel and function shortcut keys

3: Function area of operating and control panel

1.2 Software interface

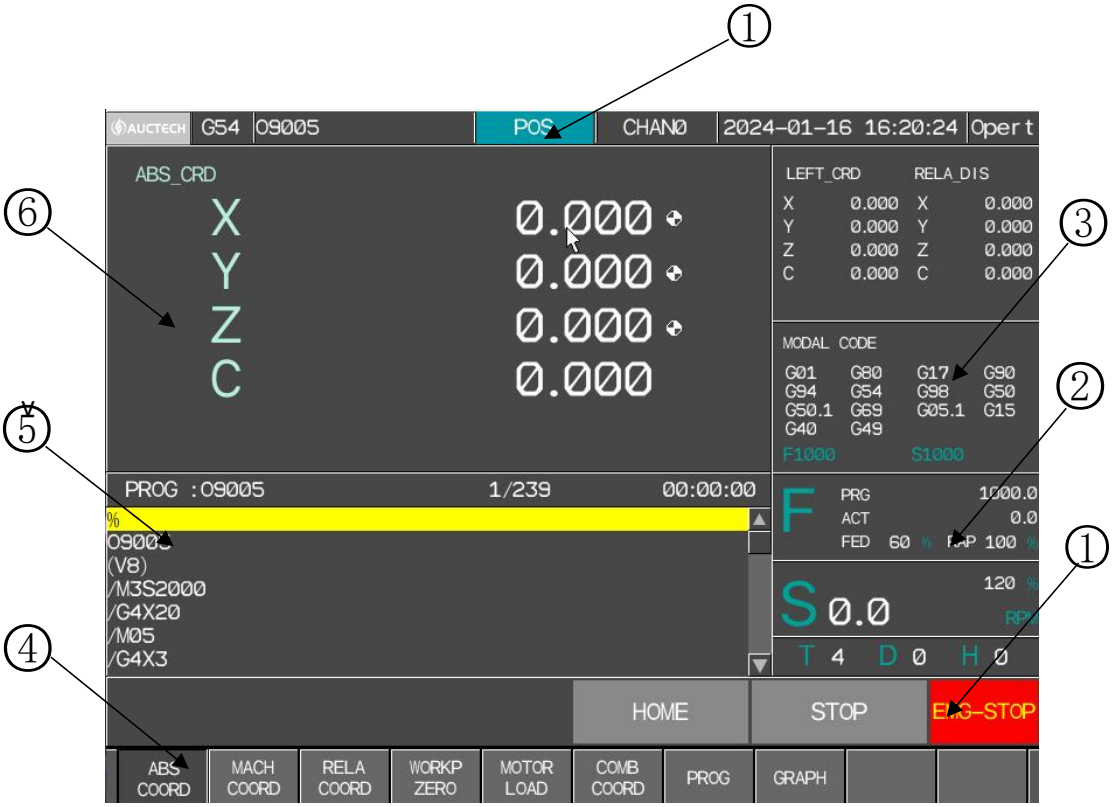


Fig. 1- 2 CNC controller interface

1: Status bar
2: Process parameters
3: Modal commands
4: Menu operations
5: Program monitoring
6: Coordinate display

■ Status bar

The status bar shows the controller operating status, from left to right, they are the controller logo icon, operating mode, program operating status, controller operating status and controller time.

● Working mode

The controller consists of five operating modes: automatic, MDI, manual, incremental and return to the reference point. This area shows the current operating mode of the controller.

- Program running status

The program includes three running states: stop, run and feed hold. This area shows the current state of the program.

- Controller operating status

When there are no alarms in the controller, the display shows "operating normally".

When the controller is alarmed, it is displayed as different alarm texts depending on the alarm.

When more than one of the following alarms occur at the same time, the controller operating status is displayed in the priority order of reset, overtravel, emergency stop and alarm.

- Controller reset

When the reset button is pressed, the controller performs a reset.

The controller operating status displays as "reset".

- Overtravel

When the table is pressed against the upper limit switch, an overtravel alarm appears on the controller.
The controller operating status shows as "overtravel".

- Emergency stop

When the emergency stop switch is pressed, an emergency stop alarm appears on the controller.
The controller operating status displays as "emergency stop".

- Other alarms

When an alarm other than reset, overtravel or emergency stop occurs, the controller operation status displays as "alarm".

■ Coordinate display area

This area shows the current position of the machine tool. The menu switches to display user coordinates, machine tool coordinates, relative coordinates, remaining feed, workpiece zero point, following error, etc.

■ Program monitoring area

This area displays the program name, program run line number, machining time and program code of the currently running program.

- Process parameters area

  This area shows the process parameters currently used for machining, such as feed rate, spindle speed, tool compensation, magnification adjustment, etc.

- Modal command area

  This area displays the currently valid modal G commands, F codes, and S codes.

- Menu operation area

  This area displays the menu buttons available for operation under the current screen.

1.3 Mode of operation

The controller comprises five modes of operation.

- Automatic

In automatic mode you can run machining programs, execute MTS commands, etc.
The auto mode is switched by the **< Auto>** key on the operation panel.

- MDI

In MDI mode you can run single-section programs, execute MTS instructions, etc.
The MDI mode is switched by the **< MDI>** key on the operation panel.

- Manual

In manual mode you can perform operations such as moving the feed axis and rotating the spindle manually.
The manual mode is switched by **< Manual>** key on the operation panel.

- Incremental

In the incremental mode you can use hand crank control of the machine tool movement, manual incremental movement of the feed axis and other operations.
The incremental mode is switched by the **< incremental>** key on the operation panel.

- Return to reference point

In the return to reference point mode, zeroing operation can be performed for each feed axis.
The return to reference point mode is switched by < **Return to reference point**> key on the operation panel.

2 Login path for controller function

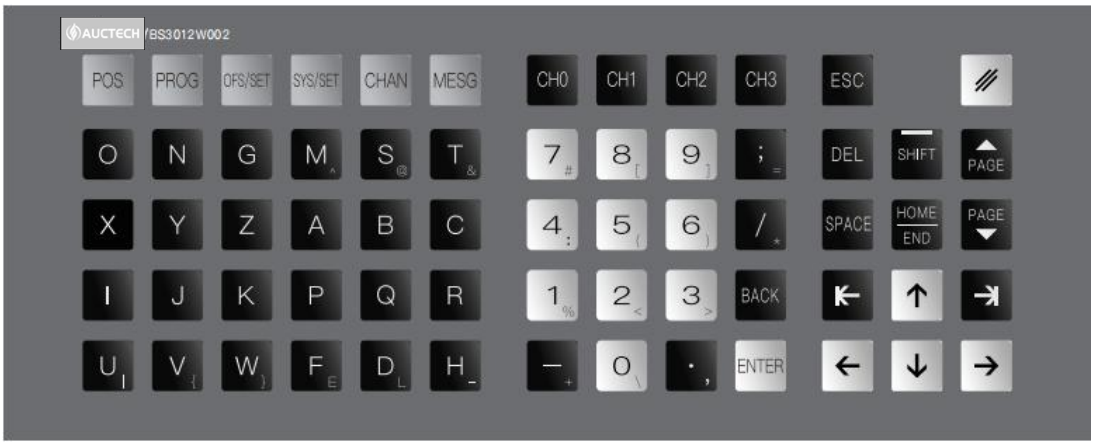


Fig. 2-1 Shortcut panel input screen

POS-Position page
PROG-Program page
OFS/SET - User settings
SYS/SET - controller setting
CHAN - Channel switching
MESG - Diagnostic function

2.1 List of key POS position functions

2.1.1 Functional area under the POS button

- Absolute coordinates
- Mechanical coordinates
- Relative coordinates
- Workpiece zero point
- Motor load
- Integrated coordinates
- Programs
- Graphical simulation

- Hand crank interruption
- Following error

Fig. 2-2 Functional area under the POS button

2.2 List of key PROG program functions

2.2.1 Functional area under the PROG button

- Program Editing
- Archive Management
- Program check
- Save to load

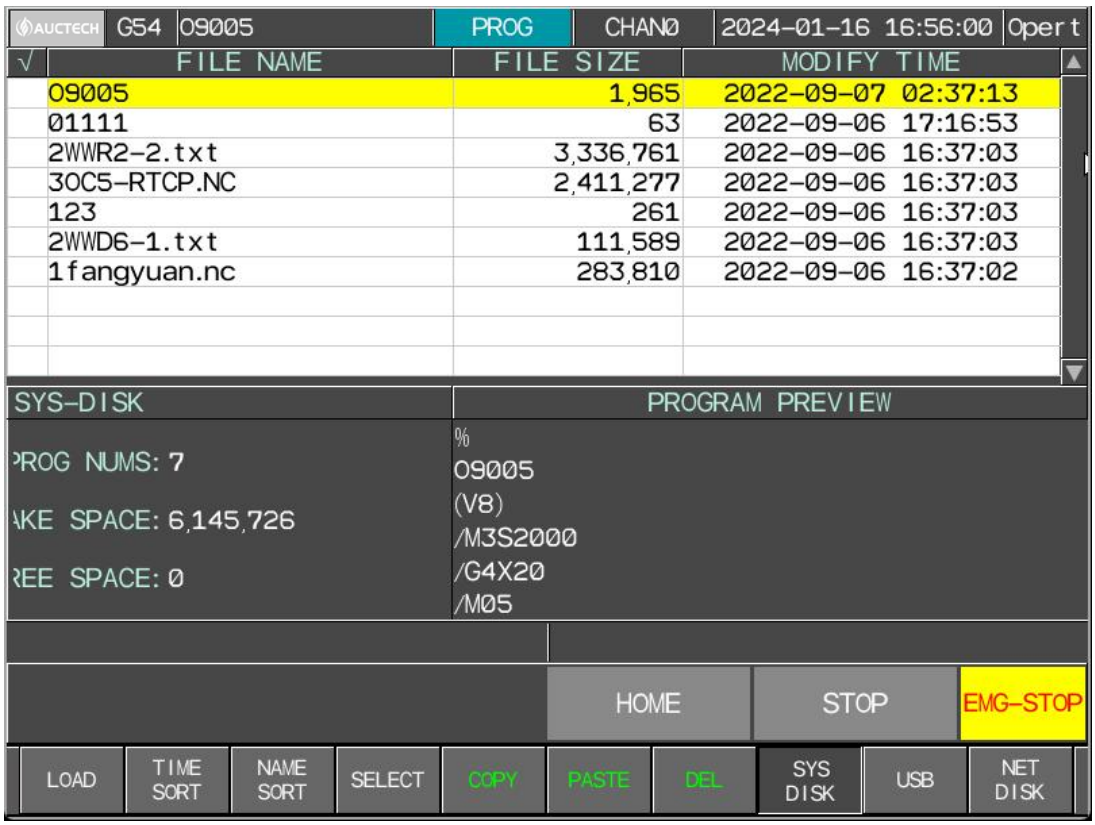


Fig. 2-3 Functional area under the -PROG button

- controller Disk/U-Disk

PROG program button is pressed for several times in succession to switch between program editing screens and function button areas

2.3 List of key OFS/SET user functions

2.3.1 Function area under the OFS/SET button

- Tool compensation
- Coordinate controller
- Variables
- Machining statistics
- Tool measurement

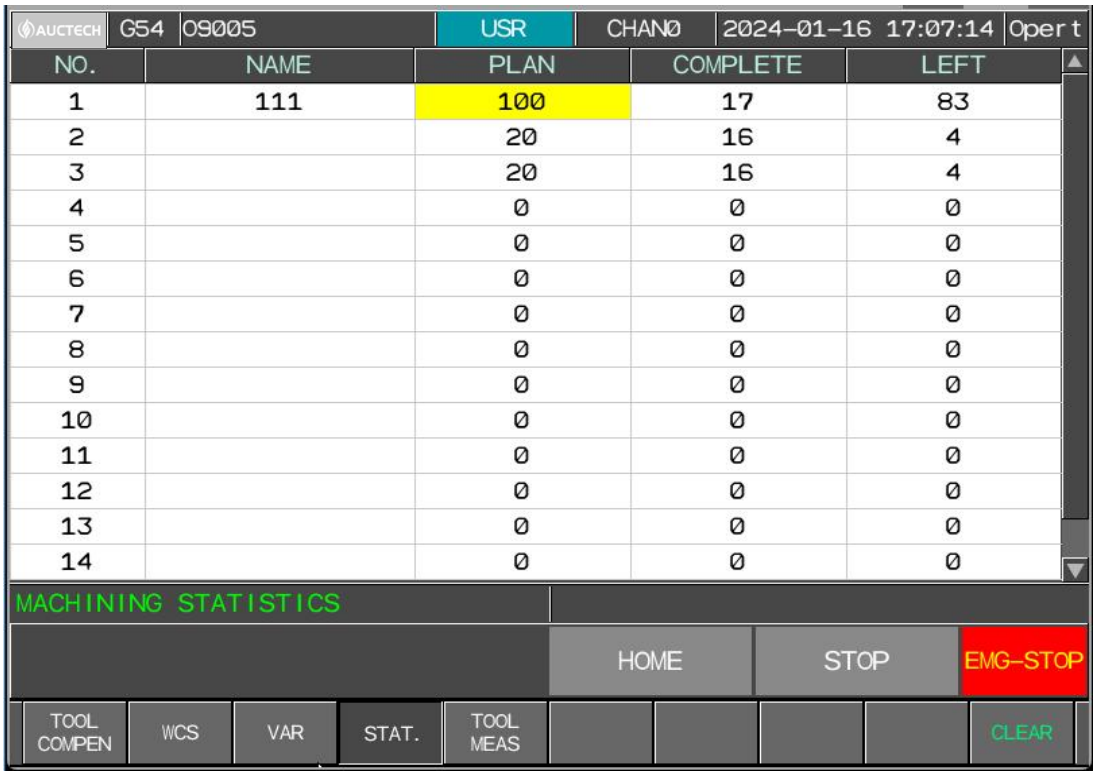


Fig. 2 - 4 Functional area under the - OFS/SET button

2.4 List of key SYS/SET setting functions

2.4.1 Function area under the SYE/SET button

- Parameter settings
- M Code
- User Alarms
- PLC News
- Tool magazine table
- Access management
- controller Information
- RTCP calibration
- Touch screen calibration

Fig. 2-5 Function area under the -SYS/SET button

2.5 List of key MESG diagnostic functions

2.5.1 Function area under the MESG button

- Alarm messages
- Alarm log
- Servo diagnosis
- controller PLC

Fig. 2-6 Functional area under the - MESG button

2.6 List of key CHAN channel functions

2.6.1 CHAN button for fast multi-channel switching

3 Manual operation of the controller

3.1 Manual return to reference point

3.1.1 Ways for returning to the reference point

There are three ways for returning to the reference point
(selected by the axis parameters).

■ Positive-negative-positive way

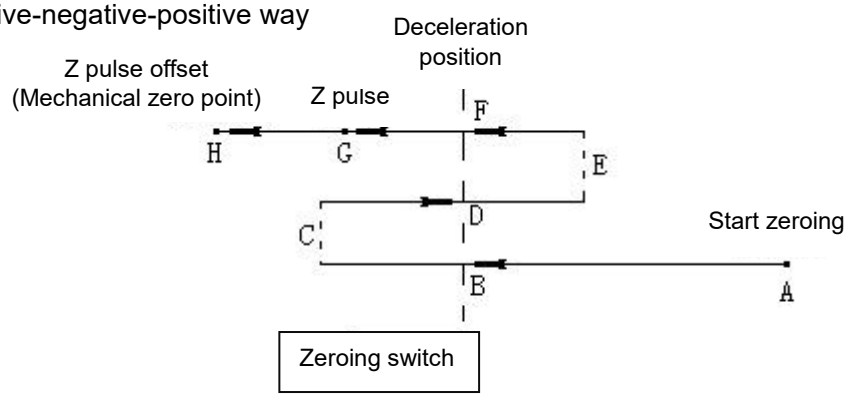


Diagram 2- 1 Positive-Negative-Positive Zeroing Process

● High-speed positioning stage (A→B)

The process from the start of the zeroing until the zeroing switch is pressed is the high-speed positioning stage.
High speed positioning stage uses zeroing rapid traverse speed (set by axis parameters).

● Transition stage (B→F)

After pressing on the zeroing switch at high speed positioning, return to break off the zeroing switch and then move forward to press on the zeroing switch, this process is the transition stage.

The transition stage uses a zeroing intermediate speed (set by the axis parameters).

- Z pulse search stage (F→G)

Once the transition stage is complete, the search for shaft encoder Z pulses begins.
The Z pulse search stage uses the Z pulse search speed (set by the axis parameters).

- Z pulse position offset (G → H)

The mechanical zero point can be at the Z pulse position or offset relative to the Z pulse position (the offset is set by the axis parameters).
This process uses the Z pulse position offset speed (set by the axis parameters).

■ Positive-negative-negative way

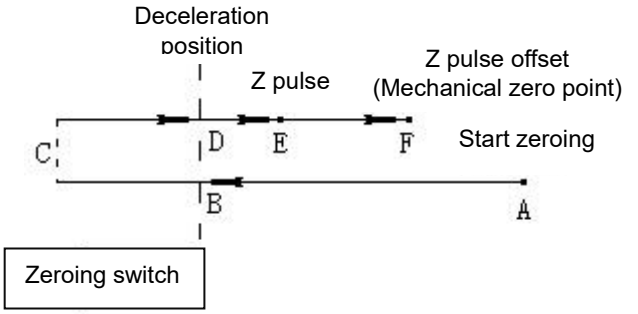


Diagram 2-2 Positive-negative-negative zeroing process

- High-speed positioning stage (A→B)
- Transition stage (B → D)
- Z pulse search stage (D→E)
- Z pulse position offset (E → F)

■ Positive-positive-positive way

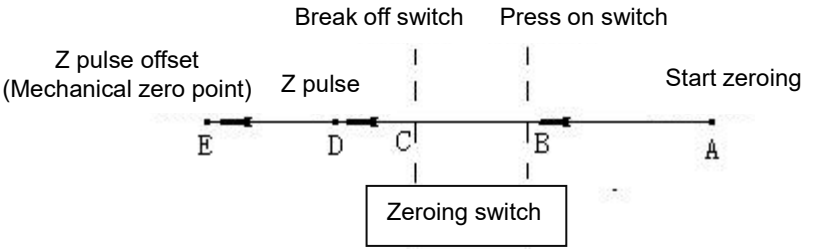


Diagram 2-3 Positive-positive Zeroing process

- High-speed positioning stage (A→B)
- Transition stage (B→C)
- Z pulse search stage (C→D)
- Z pulse position offset (D → E)

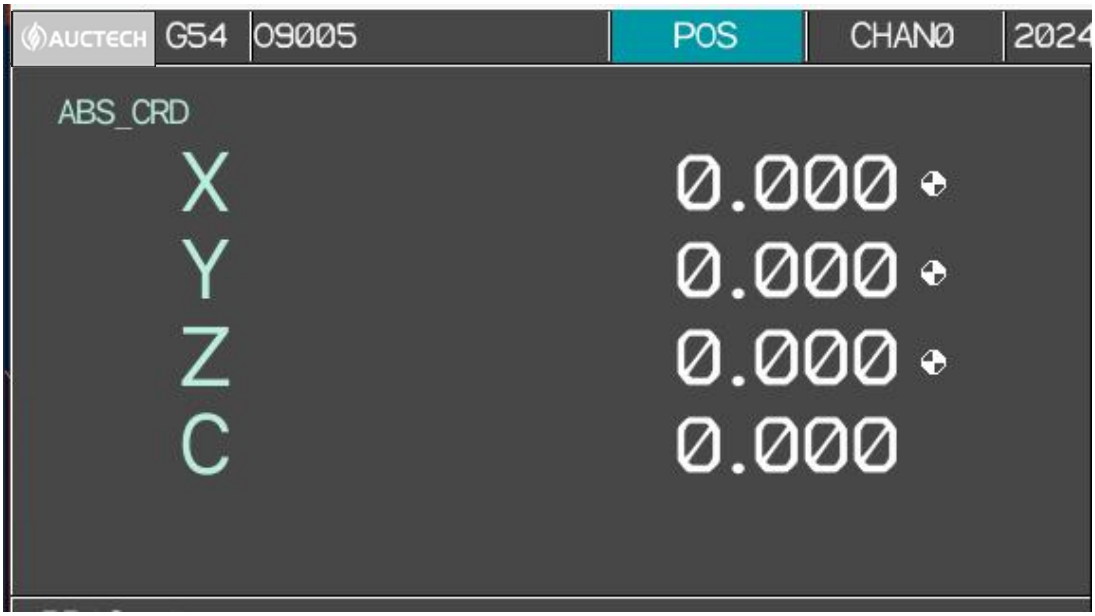
7.3.1 The direction for returning to the reference point

The direction for returning to the reference point is set by the machine tool manufacturer, see the machine tool's operating instructions.

3.1.3 Operation for returning to reference point

■ Operating steps

- Press the <**Return to reference point**> key on the operation panel to switch to return to reference point mode.
- Press the corresponding axis selection key on the operation panel (whether the axis is zeroed positively or negatively is set by the machine tool manufacturer, see the machine tool's operating instructions);
- The axis starts to zeroing;
- Zeroing completed: When the feed axis stops and the reference point icon appears on the corresponding axis in the controller interface, the zeroing for that axis is completed.



Zeroing completed

■ Termination of returning to the reference point

The zeroing can be cancelled at any time during the return to the reference point by pressing the < **Feed Hold**> key.

■ Cautions

- The position of the table and the tool should be adjusted before returning to the reference point to avoid movement interference during the return to the reference point.

- The return to the reference point is achieved by the PLC program prepared by the machine tool manufacturer and the relevant operation is governed by the machine tool manual.

3.2 Manual inching feed

A inching feed is the process in which the corresponding axis starts to move while the axis selection key is pressed, until the axis decelerates and stops when the key is released.

3.2.1 Operating steps

- Switching the controller to manual mode.
- Press the corresponding axis selection key and the axis starts to move.
- Release the axis selection key and the axis stops moving.

3.2.2 Movement speed

The manual movement speed is divided into two gears: low speed and high speed, which are switched by the <**Quick**> key on the operation panel.

- **Low speed gear**

When the indicator light for the <**Quick**> key on the operation panel is off, the manual movement speed is in low speed, which is set by the axis parameters (see the machine tool's operating instructions).

- **High-speed gear**

When the indicator for the <**Quick**> key on the operation panel is on, the manual movement speed is in high speed, which is set by the axis parameters (see the machine tool operating instructions).

3.3 Manual incremental feeds

In incremental mode, when one of the single step length selection keys < ×1>, < ×10>, < ×100> or < ×1000> on the operation panel is on, the manual incremental feed mode is used, otherwise the handwheel control mode is used.

3.3.1 Operating steps

- Switching the controller to incremental mode.
- Select the appropriate movement step and ensure that the indicator light for the corresponding key is on.
- By pressing the axis selection key, the axis starts to move and stops automatically after moving a specified step length.

3.3.2 Definition of step length

- **×1**: 0.001mm
- **×10**: 0.01mm
- **×100**: 0.1mm
- **×1000**: 1mm

3.4 Handwheel operation

3.4.1 Parameter setting in relation to handwheel control

NO.	NAME	VALUE	AUT	ACT
0000040	HAND WHEEL REVERSE	1 ▼	M	NOW
0000041	HAND WHEEL SMOOTH COEF	30	M	RST
0000045	HAND-WHEEL SELECT	0	M	RST

■ Hand crank smoothing factor

The larger the hand crank smoothing factor, the smoother the shaft movement and the more pronounced the motion delay will be when the hand crank stops.

The hand cranking smoothing factor is set by controller parameter 0000041.

■ Hand crank direction setting

When the direction of the handwheel-controlled table movement is incorrect, the direction of the handwheel-controlled table movement can be changed by controller parameter 000040.

3.4.2 Operating steps for the handwheel control

- Press the <**incremental**> key on the operation panel to switch the controller to incremental mode (confirmation: the <**incremental**> key indicator light is on and the CNC controller operating mode and status bar show "handwheel/stop" status).
- Toggle the axis selection band switch on the handwheel so that the switch points to the axis to be operated (when the switch points OFF, the handwheel is off and no axis can be operated).
- Select the appropriate handwheel magnification by toggling the handwheel magnification switch.
- Rotate the handwheel to move the selected shaft.

4 Automatic-run for machining

In the automatic modem, you can run machining programs and

execute MDI commands.

Two types of machining programs can be run in automatic mode.

- controller program: the controller program is stored on the program memory area that comes with the CNC controller.
- U-disk programs: U-disk programs are stored on some mobile storage devices with USB interface, such as USB flash drives, mobile hard drives, etc..

Any machining program must first be loaded into memory before it can run. controller programs and U-disk programs can be loaded into memory by the CNC controller and run directly.

4.1 Selecting a machining program

7.3.1 Go to the program selection screen

How to access the program selection screen: [**Programs**] → [Program Management] → [controller Disk/USB stick].

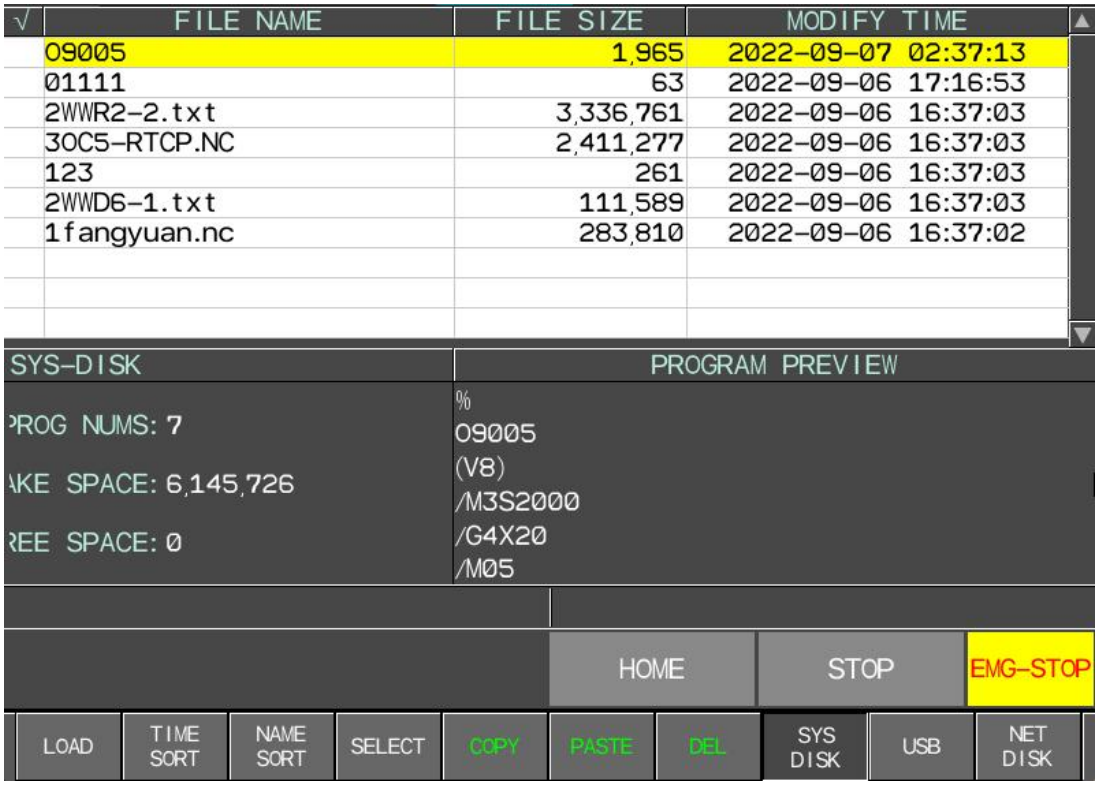


Fig. 4 - 1 Program selection screen

The menus that can be operated under the program selection screen window include:

[**Load]**: loads the selected program into memory.
[**Sort by time**]: The program is sorted by the time it was modified.
[**Sort by name**]: The program is sorted by its name.
[**Program Management**]: Performs operations such as copying, pasting and deleting programs.
[**controller Disk**]: Displays a list of programs on the controller's internal memory.
[**USB stick**]: Displays the list of programs on the USB stick.
[Program name change]: Update the program name online.
[Program breakpoint]: Saves the machining breakpoint and resumes machining at the breakpoint.
[Specify row]: Specify the row to start machining.
[Specify Target]: Specify the target to start machining e.g. N-code or T-code etc.
[Find]: Find the program code and the target instruction.
[Program check]: Check that whether the program is correct or not

Fig. 4 - 2 Program editing interface

4.1.2 Program Preview

On the program selection screen, when a program is selected with the cursor, you can preview the selected program on the program preview screen at the bottom right.

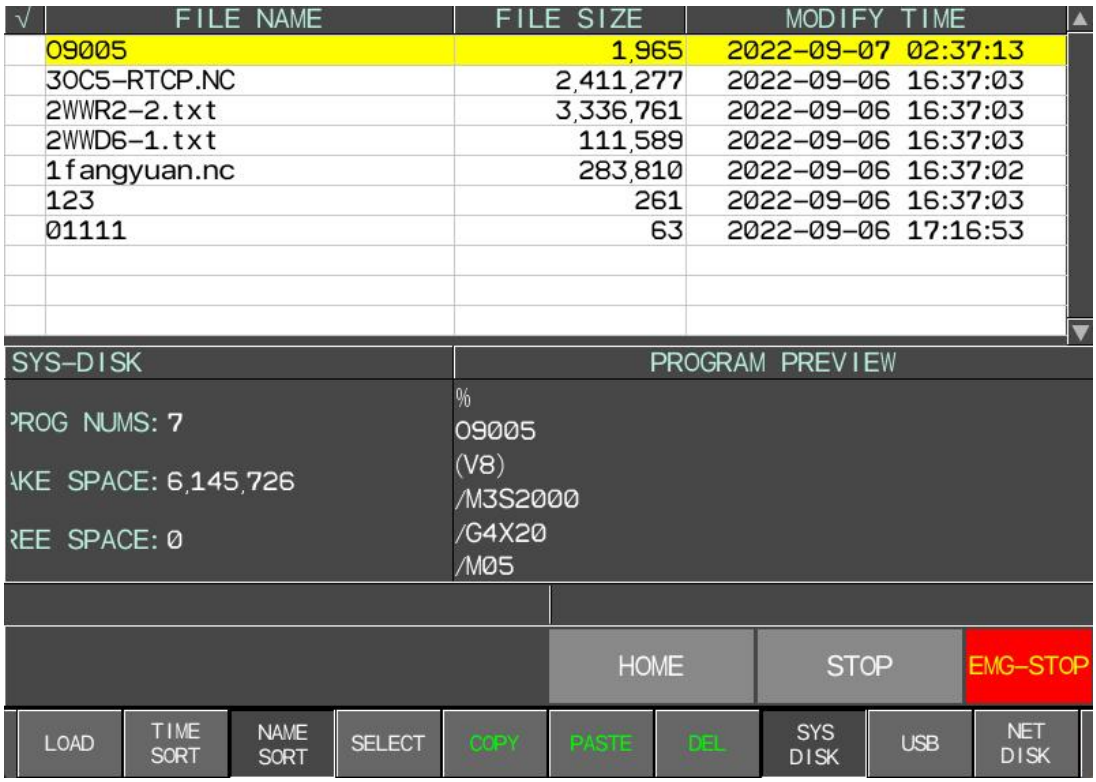


Fig. 4 - 3 Program preview screen

4.1.3 Loading controller programs

To load a controller program, proceed as follows.

- Switching to the program selection screen.
- When entering the program selection screen for the first time, the controller displays a list of controller programs by default, or you can switch to the controller memory by pressing the menu [controller Disk].
- Use the up and down arrow keys [↑], [↓] or the [PageUp], [PageDown] keys to select the program to be loaded.
- Press menu [Load] to load the selected program into memory.

4.1.4 Loading a program in USB stick

The steps for loading the program in USB stick are described as follows:

- Switching to the program selection screen.
- Press menu [**USB stick**] to switch to the USB stick program list.
- Use the up and down arrow keys [↑], [↓] or the [**PageUp**], [**PageDown**] keys to select the program to be loaded.
- Press menu [**Load**] to load the selected program into memory.

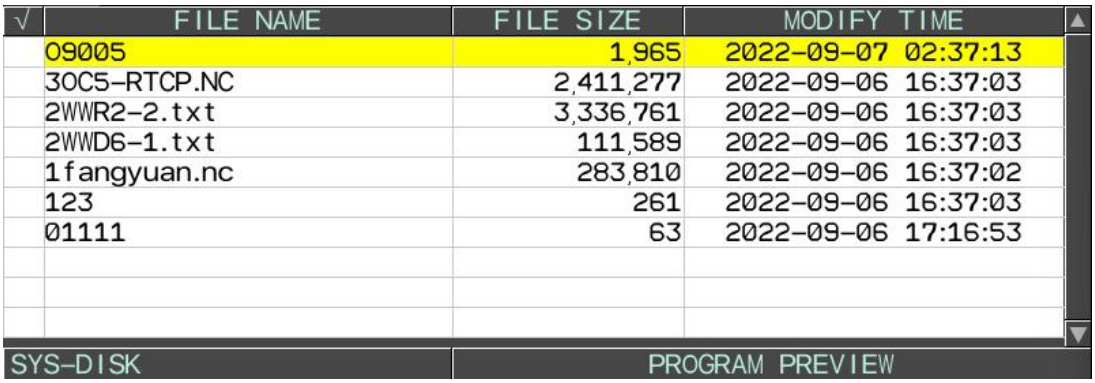


Fig. 4 - 4 Program load list

4.2 Program start/pause/termination

4.2.1 Program start

When the machining program is ready to be loaded into memory, the program can be started by pressing the <**Cycle Start**> key on the operation panel.

After the program has started, the **<Cycle Start**> key indicator light is on and the CNC controller interface status bar shows "running".

4.2.2 Program pause

The program can be paused at any time during operation by pressing the < **Feed Hold**> key on the operation panel.

When the program is paused, the < **Feed Hold**> key indicator light is on and the CNC controller interface status bar shows the "Feed Hold" status.

When the program is paused, it does not exit the program machining status, you can start the program again at any time by pressing the < **Cycle start**> key.

When the program is paused, the controller can be switched to manual or handwheel mode to move the axes manually; when switched back to automatic mode again to continue running the program, the controller will automatically return to the program breakpoint. Therefore, it should be ensured that there is no motion interference during the return to the program breakpoint.

Note: < **Feed Hold**> is not valid when executing the rigid tapping command (G74/G84).

4.3 Program breakpoint

During the machining process, for some reason, the process stops midway, the segment the program is executed as well as the position of the machine tool, modal information, etc. when machining is stopped is called the program's breakpoint information

The program breakpoint saving and restoring function is used when it is necessary to restart machining from the program breakpoint.

4.3.1 Program Breakpoint Window

To enter the program breakpoint window: [**Program**] → page up → [**Program breakpoint]**

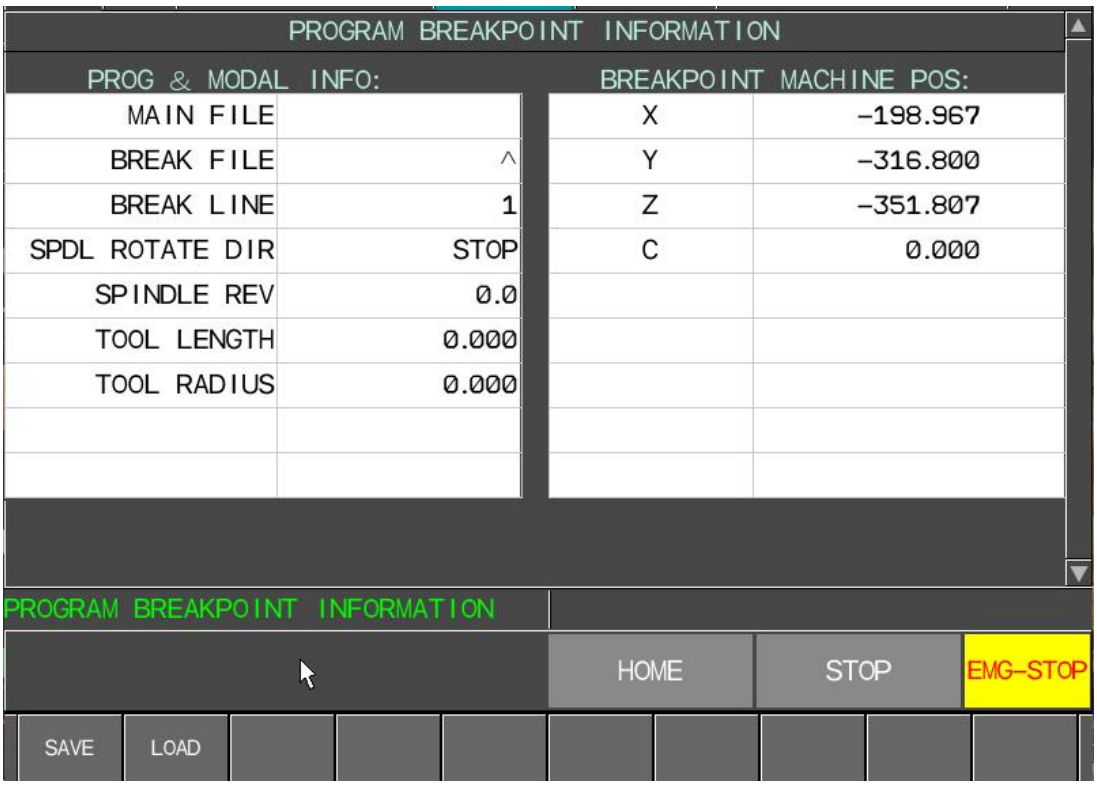


Fig. 4 - 5 Program breakpoint screen

4.3.2 Breakpoint Save

The breakpoint data can be saved when the program is in the feed-hold state; in the non-feed-hold state, program breakpoints cannot be saved.

In the feed-hold state, switch to the program breakpoint window (as shown above) and press the menu [**Save breakpoint**] to complete the saving of the program breakpoint.

4.3.3 Breakpoint Recovery

The program breakpoint can be resumed when the program is stopped (no program running).

To access the program breakpoint window: [Program] → [Program Breakpoint]

The program breakpoint management window is shown inFigure 3- 6 shown.

To restore a saved breakpoint, do the following.

- Access to the breakpoint management window.
- Press the controller menu [**Resume Breakpoint**], if the breakpoint is valid, the controller will automatically load the breakpoint program and perform the breakpoint recovery.

4.3.4 Cautions

- When resuming a program breakpoint, the program-related auxiliary functions need to be switched on manually (e.g. spindle rotation, cooling, etc.).
- If the machining parameters of the breakpoint program (e.g. tool length, tool radius, zero point of the workpiece coordinates, etc.) have changed, the breakpoint file stored before the parameter change can no longer be used for machining recovery, otherwise it will result in a machining error.

4.4 Program debugging functions

4.4.1 Single segment running

Before or at any time during the automatic running of the machining program, press the **<Single Segment>** key on the operation panel, when the key indicator light is on, the machining program is in the single segment running state, i.e. the execution of a program segment is automatically suspended, at this time the cycle start light goes out and the feed hold light is on, press the **<Cycle Start>** key again to start the execution of the next segment.

When the **< Single Segment>** key indicator is off, the program runs continuously, i.e. no pause is executed between program segments until the end of the program.

Pressing the <**Single Stage**> key repeatedly switches between single stage operation and continuous operation status.

4.4.2 Jumping Paragraphs

When the key indicator light is on, the segment skipping function is activated, i.e. segments preceded by the segment skipping symbol "\" will be skipped (not executed) during the execution of the program.

< When the "\" key indicator goes out, the jump function is switched off and the program segment with the jump symbol "\" is executed during the machining process.

Pressing the < Jump> key repeatedly toggles the Jump function on and off.

4.4.3 Program verification

For newly written machining programs, the program check function can be used to quickly verify that the programming instructions are correct.

Access: Shortcut PROG program pressed twice -> Program check

For newly written machining programs, the graphical function can be used to quickly verify that the programming track is correct.

Login path: POS position-> Graphics

Fig. 4 - 6 Simulation of the graphical track of the program

4.4.4 Machine tool locks

Before running the program automatically (automatic non-running state), press the **<Machine tool Lock>** key, when the key indicator light is on, it means that the program is in a simulated machining state. This means that the program runs normally, the coordinates are refreshed normally and the toolpath is displayed normally, but the actual position of each axis of the machine tool remains unchanged.

By executing the machining program in machine tool lock, the user can check the correctness of the program by observing the running status of the program from the display of the toolpath and coordinate changes.

Pressing the **<Machine tool lock>** key repeatedly switches the machine tool lock on and off when the automatic mode is programmed to stop.

Difference between machine tool lock and program verification.

- The operating speed of the machine tool lock is the programmed speed.
- Program verification draws tool paths as fast as possible to quickly verify the correctness of the programmed track.

4.4.5 Z-axis lock

Before automatically running the machining program (automatic non-running state), press the **<Z-axis lock>** key, when the key indicator light is on, it means that the machining program is in Z-axis locked state. This means that the program runs normally, the coordinates are refreshed normally and the tool path is displayed normally, but the actual position of the machine tool Z-axis remains unchanged (other axes move normally).

Press the **<Z-axis lock>** key repeatedly to toggle the Z-axis lock on and off when the automatic mode is programmed to stop.

4.4.6 Empty run

Before running the program automatically or at any time during the machining process, press the **<Null run>** key on the operation panel, when the key indicator light is on, it means that the null run is valid, i.e. the feed speed specified by the F command in the program is not valid during the program run, the command speed of the program run is set by the channel parameters, the feed magnification is also valid.

<Empty run> When the key indicator lamp is off, the empty run is invalid and the program run speed is specified by the F command.

Pressing the **<Run empty>** key repeatedly toggles the run empty function on and off.

4.5 Handwheel interruption

■ Overview

When the handwheel interrupt function is switched on, the handwheel rotation can be superimposed on the current axis movement in real time by rotating the handwheel during program operation, this superimposition being achieved by offsetting the zero point of the workpiece coordinate controller.

The amount of handwheel interruption shifts the workpiece coordinate controller and the local coordinate controller. Thus, although the handwheel interruption moves the machine tool, the coordinate values in the workpiece and local coordinate controllers remain unchanged.

Handwheel interruptions will offset all workpiece coordinate controllers.

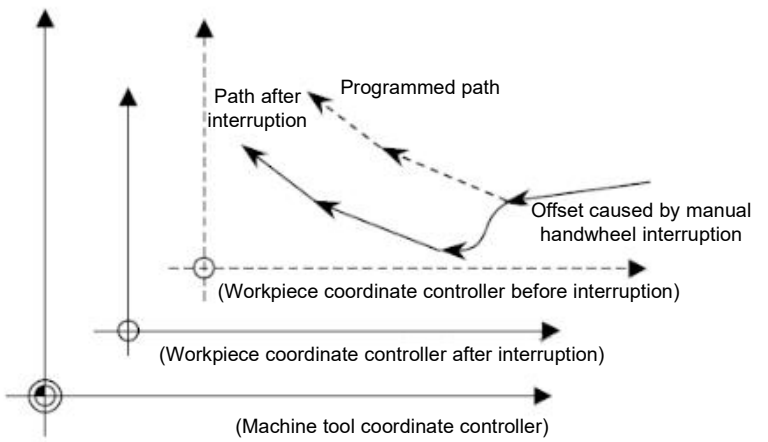


Fig. 4 - 7 Handwheel interruption

■ View handwheel interruptions

To check the amount of handwheel interruption: [**Position**] → [**Handwheel Interruption**].

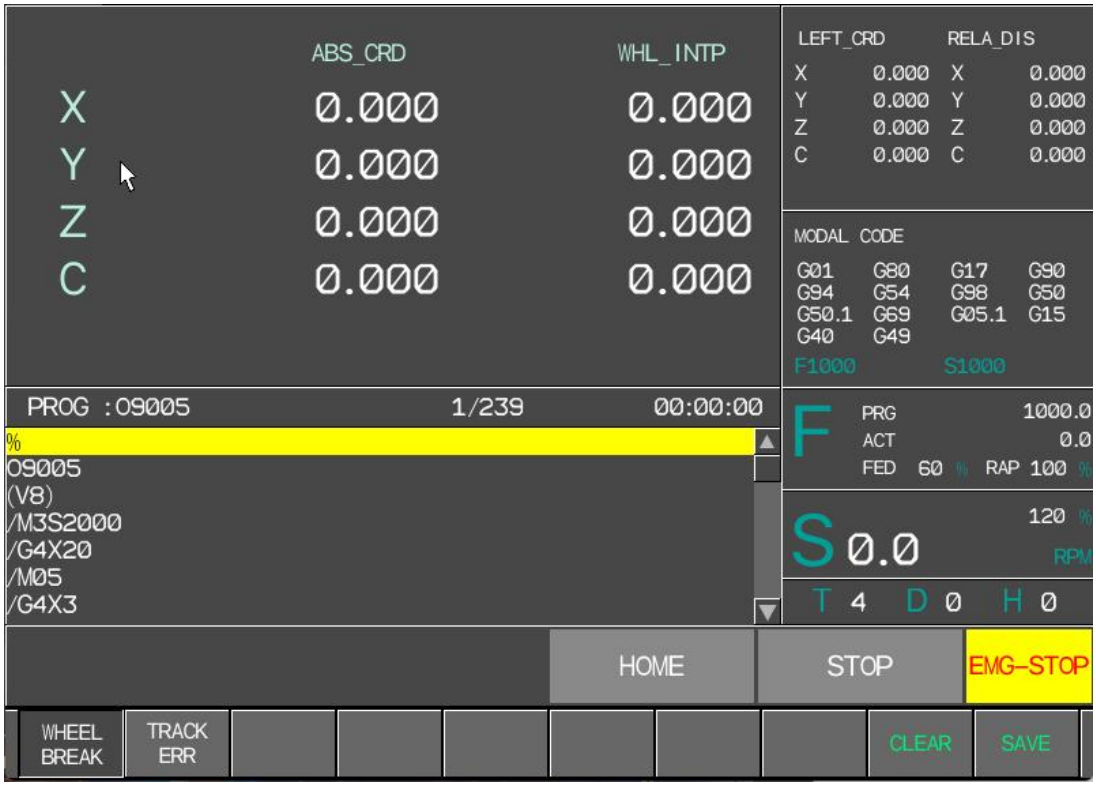


Fig. 4 - 8 Handwheel interruption diagnosis screen

■ Clear handwheel interrupt volume

To clear the handwheel interrupt amount: **[Position] → [Handwheel Interrupt] → [Clear]**

The handwheel interrupt amount can only be cleared in the program stop state.

In addition, the manual return to the reference point automatically clears the handwheel interrupt amount.

For other removal methods refer to the instructions provided by the machine tool manufacturer.

■ Save handwheel interruptions

How to save the handwheel interrupt amount: **[Position]** → **[Handwheel interrupt]** → **[Save].**

After executing a handwheel interrupt, the offset superimposed on the zero point of the workpiece is temporary and will be lost after a power failure. In order to retain this offset even after a power failure, the handwheel interrupt needs to be saved.

The handwheel interrupt volume is saved by superimposing it directly onto the external zero offset.

The handwheel interrupt amount can only be saved during program stop or feed hold.

For other storage methods refer to the instructions provided by the machine tool manufacturer.

4.6 MDI program operation

In the MDI method it is possible to enter a program segment manually and run the program without saving it.

The format of the program instructions entered by the MDI is the same as that of the automatically run machining program.

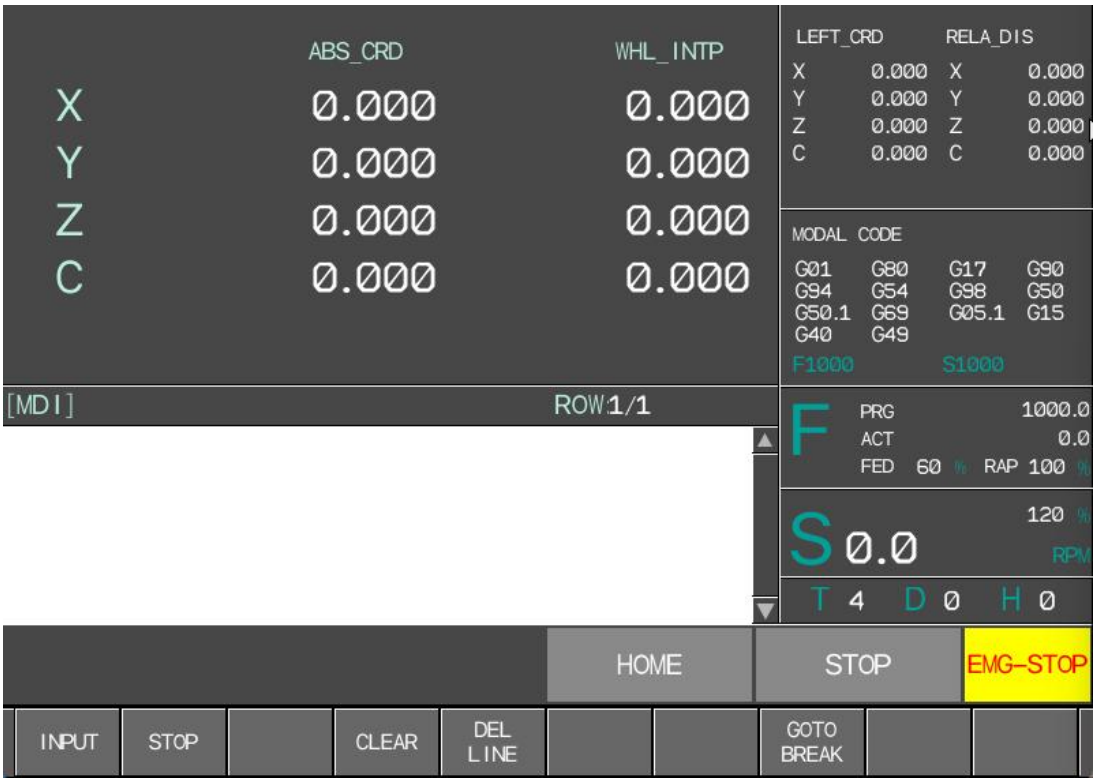


Fig. 4 - 9 MDI screen

4.6.1 MDI program execution steps

- Switching the controller to automatic mode.
- Press the [**MDI**] key to switch to the MDI program input window, which allows the input of MDI programs when the cursor is flashing.
- Enter the MDI program command and press the [Entry] menu when you have finished entering.
- Switching the controller to automatic mode.

- Press the **< Cycle start>** key to start the execution of the MDI program.

Once the MDI program has been entered, it must be confirmed by pressing the **[Entry]** menu to enter the command buffer and wait for execution; otherwise, it will not be executed even if the **< Cycle Start>** command is pressed.

4.6.2 MDI program pause/resumption/termination

(a) After the MDI program has been suspended, the execution status of the instruction and the data buffer are still preserved, so that the instruction can be resumed.

When the MDI program is terminated, the execution status of the instruction is cleared, the data buffer is emptied and the instruction cannot be recovered, only restarted.

- Suspend command operation

The MDI program can be suspended at any time during execution by pressing the **< Feed Hold>** key on the operation panel.

- Resume command operation

When the MDI program is suspended, the execution of the current instruction can be resumed by pressing the **< Cycle Start>** key.

- Terminate command run
  - (a) After the command has been paused, press the [**Stop**] menu to terminate running the MDI program.
  - Press the [**RESET**] key to terminate running the MDI Program.

4.6.3 MDI Operational settings

The user parameter 0000200 sets whether the MDI program is cleared after MDI execution is complete.

PARA INDEX	NO.	NAME	VALUE	AUT	ACT
ALL	0000200	CLEAR CODE WHILE MDI FINISH?	0	UO	NOW
SYS CFG	0000205	ALARM CLEAR MODE	0	UO	NOW
I/O	0000210	GRAPH COORD SYSTEM	0	UO	NOW
HAND WHEEL	0000211	ZOOM RATIO OF XY	1.0	UO	NOW
COMMU	0000212	ZOOM RATIO OF YZ	1.0	UO	NOW
SHARE DISK	0000213	ZOOM RATIO OF ZX	1.0	UO	NOW
DISPLAY	0000214	ZOOM RATIO OF XYZ	1.0	UO	NOW
USR PARA	0000215	CLEAR GARPH WHILE CYCLE START?	1	UO	NOW
	0000216	DISPLAY VELOCITY CURVE?	1	UO	NOW

Fig. 4 - 10 MDI setting parameters

5 Program management

5.1 Classification of program documents

- controller program: A program stored on the internal memory of the controller.
- External programs: programs stored on an external USB stick or removable hard disk.

The type of program is determined by the position and form in which it is stored, and when the position or form is changed, the type to which the program belongs will change accordingly.

For example, when an external program is copied into the controller's internal memory, the program type changes from external to controller program.

5.2 Editing of program files

Modification of program files requires the input of user administrator and above privileges.

The program file must be loaded before it can be modified.

5.2.1 Go to the program edit window

The top of the program edit window displays information about the program, including

- Program name: the name of the file of the currently open program.
- Program lock icon: when the lock icon is locked, the program cannot be modified (e.g. program is running, no modification rights, etc.).
- Program save icon: this icon is displayed when a program modification has not been saved; it disappears after saving.
- line: the line of the program in which the cursor is currently located and the total number of lines of the program.
- column: the column in which the cursor is currently located.
- Find: Find the code instruction needed to search

■ Edit an existing program file

- Press the [**Program**] key to switch to the program screen.
- Press the [**Select program**] menu to access the program selection surface.
- Move the cursor to select the Program to be modified.
- Press the [**Load**] menu to load the selected program into memory.
- Press the [**Edit**] menu to enter the program edit window.

■ Create a new program file

- Press the [**Program**] key to switch to the program screen.
- Press the [**New**] menu and enter the name of the new program file in the input box that pops up.
- Once the file name has been entered and confirmed by the near [**ENTER**] key, you will be taken directly to the program editing window.

The program name of a new program cannot be renamed with a program name that already exists in memory, otherwise the controller reports an error.

The program name 9000~9999 is used for the extended program name, if the new program name is between 9000~9999, the controller will indicate that the new program is an extended program, and the new extended program will be stored inside the CNC controller.

5.2.2 Save file

Saving a program text means that the file is written to the program memory for permanent storage. Once the file has been saved, it will not be lost even if the controller is powered down.

If the program opened is a program on the controller's internal memory (i.e. controller program) or an extension program, the program is stored in the internal memory when it is saved; if the program opened is a program on external memory (i.e. external program), the program is stored in the external memory (USB stick) when it is saved.

Press the **[Save Load]** menu under the program editing window to save the program file.

■ Cautions

After a program has been modified in the program edit window, the controller program buffer is updated to the modified contents in real time, at which point a cycle is pressed to start and the modified program is run, even if the program has not been saved.

To discard the current modification, the program can be reloaded without saving it.

5.2.3 Save file as

Save File As is also saving a file, the difference is that the file name is not changed when you "Save File", it is saved under the original file name. "Save File As" saves the file as a new file, but the original file remains.

If the file has been modified after it has been opened, Save File As will only store the changes in the new file, leaving the original content unchanged.

■ File Save As Program.

- Press the controller menu **[Save As]** in the program editing screen to bring up the program name input box.

Fig. 5 - 1 Save program as

- Enter the new file name in the input box that pops up, confirm that it is correct and press [**ENTER**] to start saving the file separately.

5.2.4 Find /continue searching

The Find operation searches for a specified string in an open program file.

Find does not distinguish between upper and lower case letters, i.e. the letters are the same in upper and lower case.

The find operation searches backwards from the current cursor position to the end of the program.

■ Find

- Press the [**Find**] menu under the program editing screen to bring up the input box as shown.

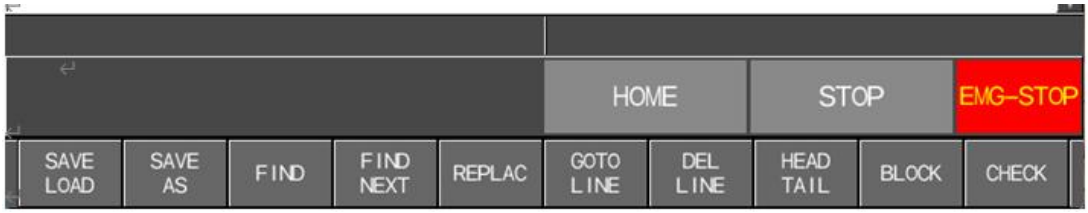


Fig. 5 - 1 Program search input box

- Enter the string to be searched for in the input box.
- Press **[ENTER]** after confirming that the input is correct and the controller starts to look backwards from the current cursor position to find the entered string.
- When a given string is found, the controller places the current cursor after the string and displays the string as selected; otherwise, if the entire file is searched and the given string is not found, the search fails.

■ Keep looking

After a search operation has been performed, the search string entered by the user will be saved. If you want to continue searching for the same string, you can use the **[Continue Search]** menu instead of re-entering the search string, which will be searched backwards from the current cursor position.

5.2.5 Replacements

The Replace operation finds the specified string in a file and replaces it with a new string.

The replace operation replaces all strings after the cursor position (strings before the cursor position are not replaced).

The replace operation is not case-sensitive.

■ Operating steps

Positioning the cursor at the start of the replacement search.

- Press the controller menu [**Replace**] under the full-screen editing screen to bring up the input box shown as **"Error! Reference source not found."**;

Figure 4-4 Replace input box

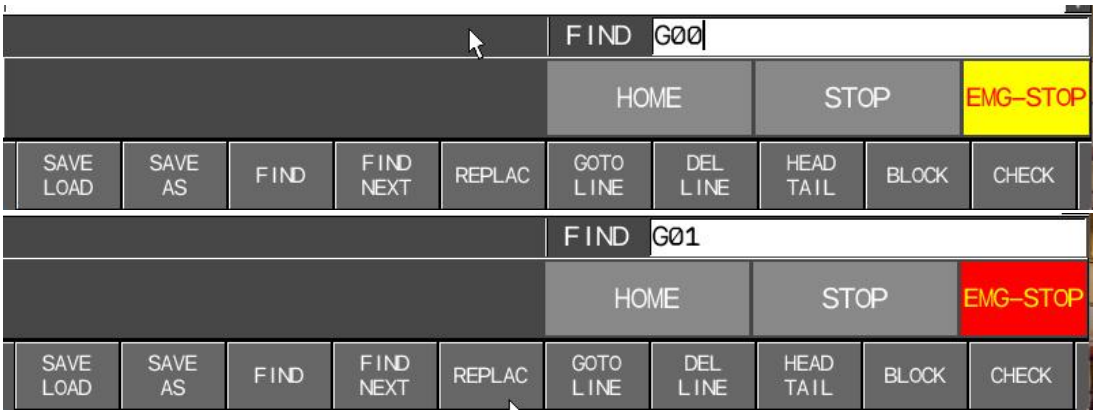


Fig. 5 - 2 Program replacement input box

- Enter the object of the search (the string being replaced) and the new string in the input box that pops up.
- After confirming that the input is correct press the **[ENTER]** key and the controller starts replacing the given string after the current cursor position.

5.2.6 Program Block

A program block is a continuous block of string units in a program.

The position of a program block is determined by the block head (the start position of the block) and the block tail (the end position of the block).

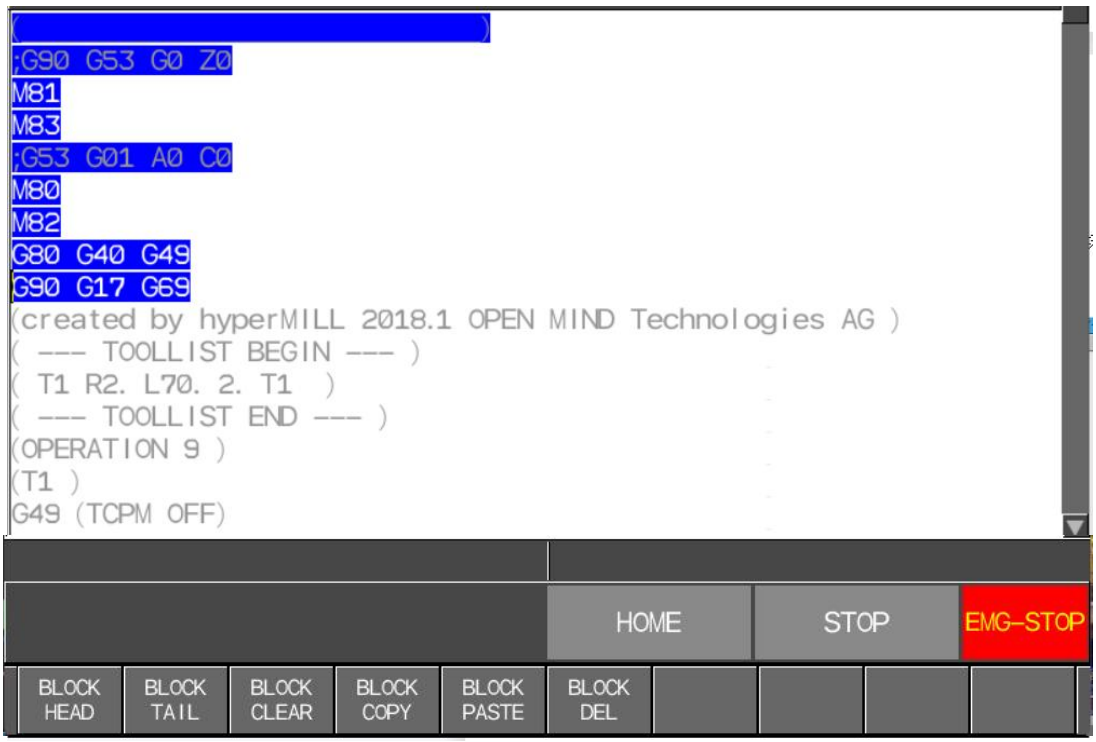


Fig. 5 - 3 program blocks

■ Program block operation menu

- [**define block head**]

Defines the current cursor position as the starting position of the block.

- [**Define end of block**]

Defines the current cursor position as the end position of the block.

- **[block cleared]**

Clears an already defined block, but not the paste buffer.

- **[block copy]**

Copy the defined block to the paste buffer. Once copied the block can be inserted into the file at the position specified using the paste operation.

- **[block paste]**

Insert the block from the paste buffer to the position of the cursor.

The block paste operation does not clear the contents of the paste buffer, so it is possible to use the block paste operation to insert the same block into different places in the file in succession.

5.2.7 Grammar check

The syntax check allows you to check for programming errors in a program without running the program.

The syntax check only checks for syntax errors in the program, it does not check for errors in the programming track.

It is recommended that a syntax check is carried out before running a new machining program to avoid syntax errors that could cause a stoppage in the middle of the process.

Fig. 5 - 4 Program check

5.2.8 Management of program documentation

Operations related to program file management include.

- Deletion of program files
- Copying of program files
- Renaming of program files
- Sorting of program files
- Selection of program files

5.2.9 Program management interface

To access the program management screen: **[Programs]** → **[Program Management].**

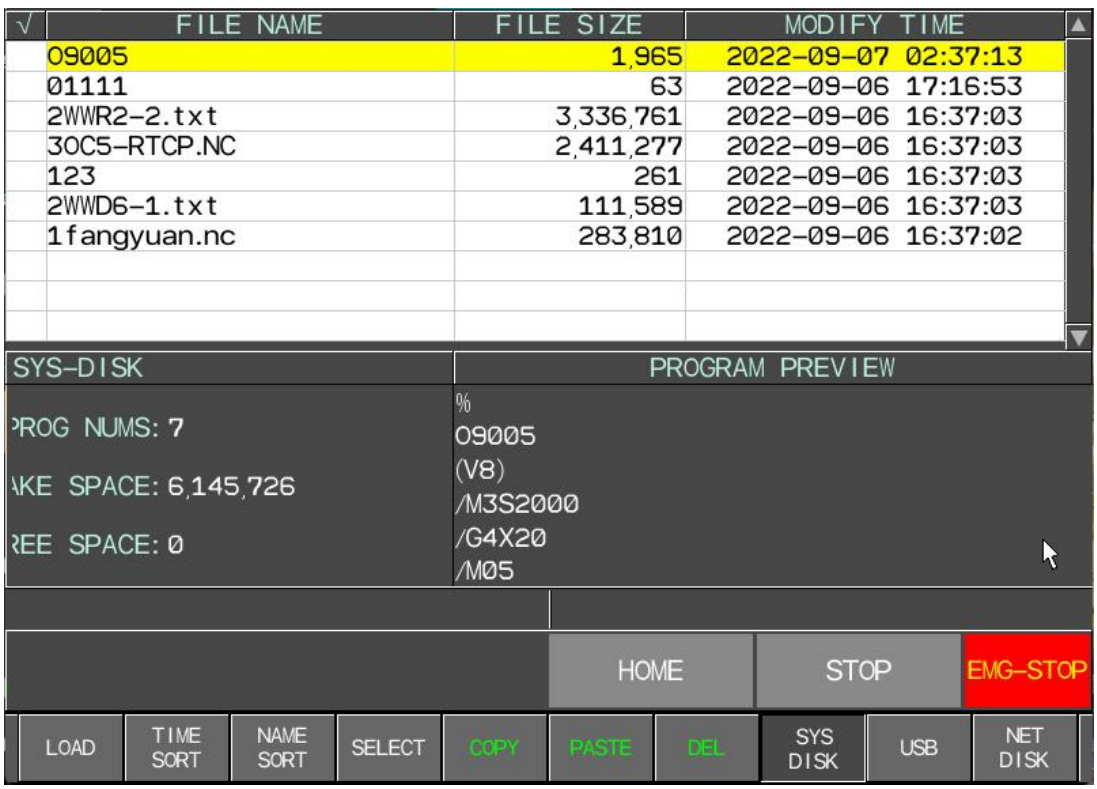


Fig. 5 - 5 Program archive management

5.2.10 Deletion of program files

- Switching the controller to the program management interface.
- Select the memory to be operated using the **[controller Disk]** and **[USB Drive]** menus.
- Move the cursor to select the program file to be deleted and press the [**Selected**] menu to make the file selected (a tick in front of the program name indicates that the file has been selected); press the [**Selected**] menu continuously to toggle between selected and unselected states.
- Repeat the previous step if you want to delete more than one file; if you want to delete all files, you can simply press the [**Select All**] menu.
- Press the menu [**Delete**] and a confirmation message will pop up as shown.

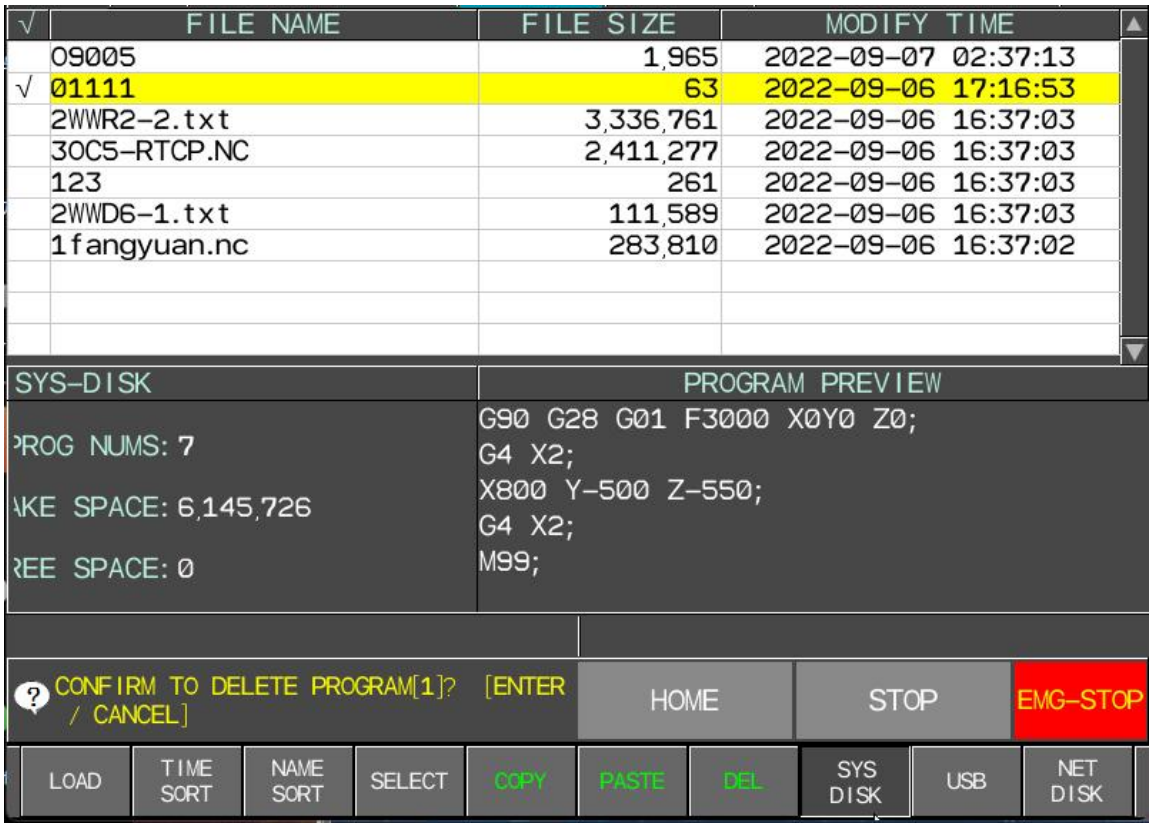


Fig. 5 - 5 Confirmation of program deletion

- Press the [**ENTER**] key to complete the file deletion operation after confirming that there are no errors; or press the [**CANCEL**] key to cancel the file deletion operation.

5.2.11 Copying of program files

- Switching the controller to the program management interface.
- Use the **[controller disk]** and **[USB stick]** menus to select the memory where the program to be copied is located.
- Move the cursor to select the program file to be deleted and press the [**Selected**] menu to make the file selected (a tick in front of the program name indicates that the file has been selected); press the [**Selected**] menu continuously to toggle between selected and unselected states.
- Repeat the previous step if you want to copy more than one file; if you want to copy all files, you can simply press the [**Select All**] menu.
- When the selection of the program is complete, press the [**Copy**] menu and the controller generates a list of copies of the program.
- Select the target memory for program copying using the **[controller disk]** and **[USB stick]** menus.
- Press the [Paste] menu to start the copy process.

5.2.12 Sorting of program files

File sorting is the process of sorting the program files in the current program list window according to certain rules.

- [**name sorting**]

Sort by the order of the ASCII code of the program file name.

Positive order: the file name with the smaller ASCII value comes first.

Reverse order: the file name with the larger ASCII value comes first.

Press the **[Name Sort]** menu repeatedly to toggle the program names in positive or reverse order.

- [**Chronological**]

Sort by the time of modification of the program file.

Positive order: documents with recent modifications are listed first.

Reverse order: files that have been modified farther back in time are listed first.

Press the **[Time Sort]** menu repeatedly to switch between positive or negative time sorting.

6 Graphical display

The graphical display is where the tool path is drawn on the screen in the form of a graphical simulation.

Check the correctness of the program by observing whether the track graph matches the design path.

6.1 Access to the graphical interface

To access the graphical interface: **[Position]** → **[Graph].**

In the displayed graph of the machining track, the red line is the track for rapid positioning (G00) and the green line is the track during tool feed (G01/G02/G03).

The graph can display four views XY, YZ, ZX, XYZ, which are switched via the **[View Switch]** menu.

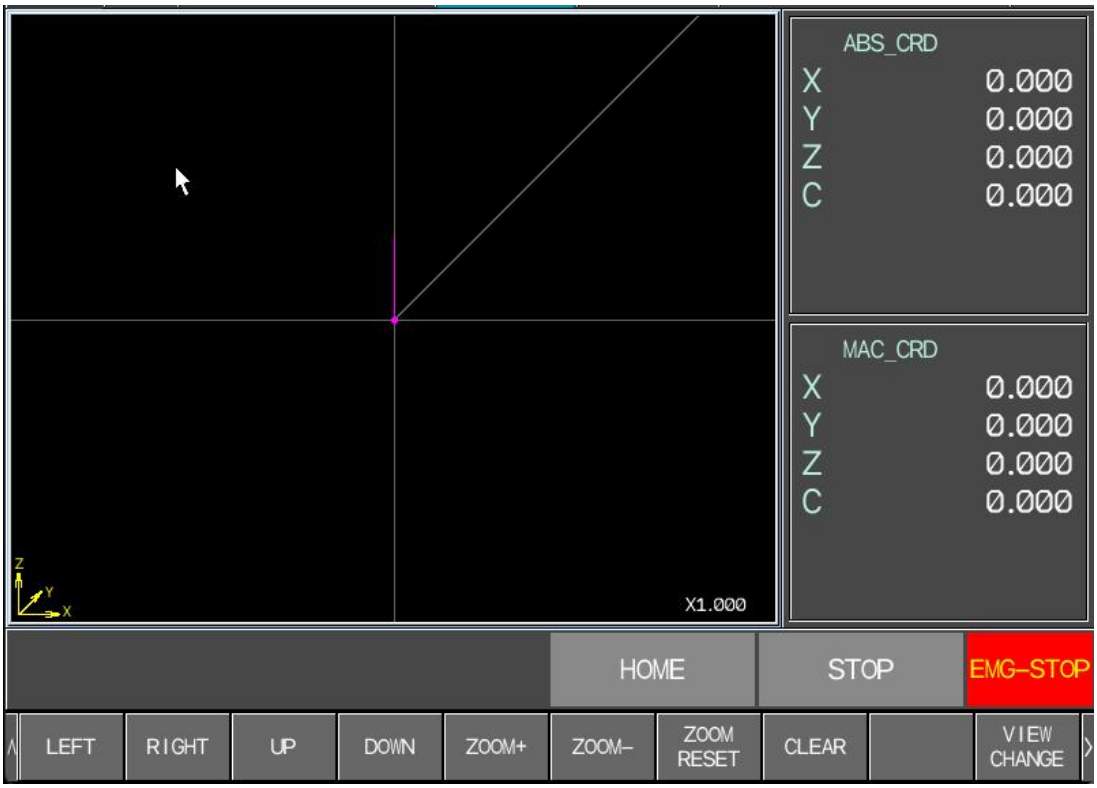


Fig. 6 - 1 Graphical simulation

6.2 Graphical operations

- [**Shift Left**]: Moves the graph in the window to the left. If the simulation is outside the right border of the graphics window, you can press the [Shift **Left**] menu to move the graphic to the left, moving the part that is outside the right border out of the window to make it visible.
- [**Shift Right**]: moves the window graphics to the right.
- [**Move Up**]: moves the window graphic upwards.
- [**Move Down**]: moves the window graphic downwards.
- [**Zoom**]: Zoom in to show the zoom factor of the graph.
- [**Zoom out**]: reduces the zoom factor of the displayed graph.
- [**Restore**]: Restores the scaling factor of the displayed graphics to 1.
- [**Clear**]: Clears the display graphics.

6.3 Graphics settings

Graphical parameter setting, requires input user operator and above permissions.

■ Graph shows the coordinate controller

The coordinate controller for the graphic display can be set with the user parameter 0000210.

- 0000210=0: Display of local coordinate controller trajectories
- 0000210=1: Display of the workpiece coordinate controller track
- 0000210=2: Display of the machine tool coordinate controller track

■ Graphical scaling factor

The zoom factor for the graphs displayed in each view is set by the user parameters 0000211 to 0000214.

The scaling factor can be modified directly by the user parameters or set in the graphical interface using the **[Zoom In]** and **[Zoom Out]** menus.

■ Cycle start clear graphics

Whether the cycle starts to clear the graph is set by the user parameter 0000215.

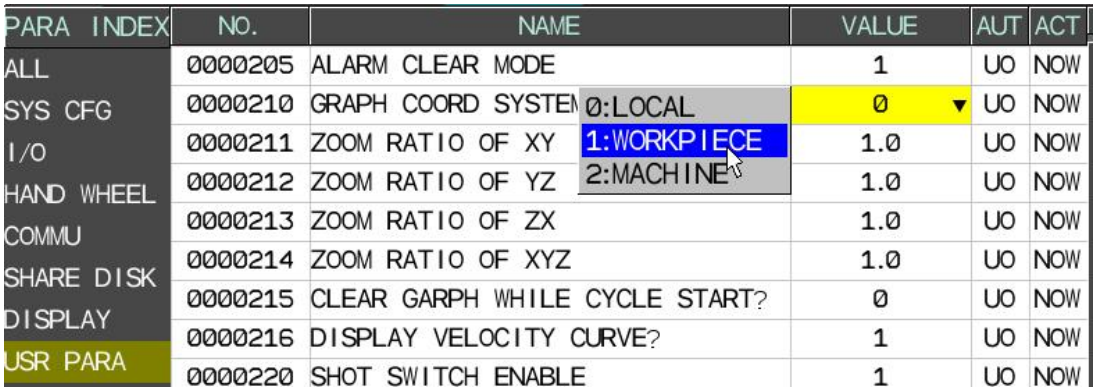


Fig. 6 - 2 Graph-related parameter settings

7 Data input

7.1 Tool compensation

7.1.1 Overview

Tool compensation parameters include.

- Tool length

Refers to the length deviation of the tool in relation to the reference tool.

Usually there is a reference tool in the tool magazine and the length of the reference tool is set to 0. The length of the other tools in the tool magazine is the length deviation from the reference tool.

- Tool radius

Refers to the radius of the cutting part of the tool.

- Tool wear

This parameter is used as a correction when the controller performs tool length or radius compensation.

Tool length compensation value = Tool length set value - Tool length wear

Tool radius compensation value = Tool radius setting - Tool radius wear

7.1.2 Tool compensation interface

To access the tool compensation screen: **[User]** → **[Tool Compensation]** or press the **[User]** key shortcut twice in succession

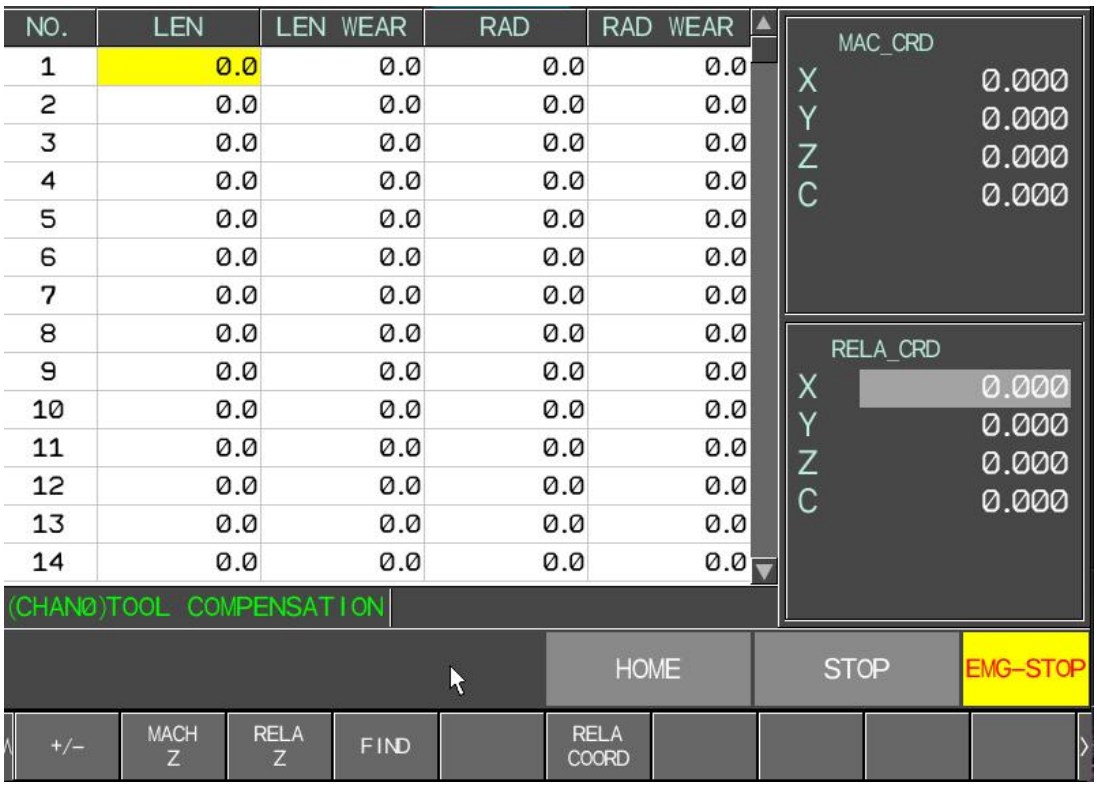


Fig. 7 - 1 Tool compensation screen

7.1.3 Method of setting tool compensation parameters

■ Method of modifying tool parameters

- Access to the tool compensation screen.
- Press the up, down, left and right keys [↑], [↓], [←], [→] or [**PageUp**], [**PageDown**] of the editing keyboard to locate the tool parameter to be set.
- Press the keyboard [**Enter**] key or enter the tool parameters directly to bring up the input box.
- Enter the new parameter value in the pop-up input box and

confirm the input by pressing the [**Enter**] key on the keyboard when you are finished.

■ Quick input menu

- **[Enter machine tool Z]:** Inputs the current machine tool Z coordinate value to the tool length parameter.
- **[Enter relative Z]:** Inputs the current relative Z coordinate value to the tool length parameter.
- **[Add or Subtract]:** Incremental addition or subtraction of the current data.
- **[Find]:** Quickly locate the column with the tool number to be modified.

7.2 Coordinate controller setting up and tool setting measurements

7.2.1 Coordinate controller overview

The coordinate controllers used in the controller include the machine tool coordinate controller, the workpiece coordinate controller and the local coordinate controller, the relationship between the coordinate controllers is shown in the diagram below.

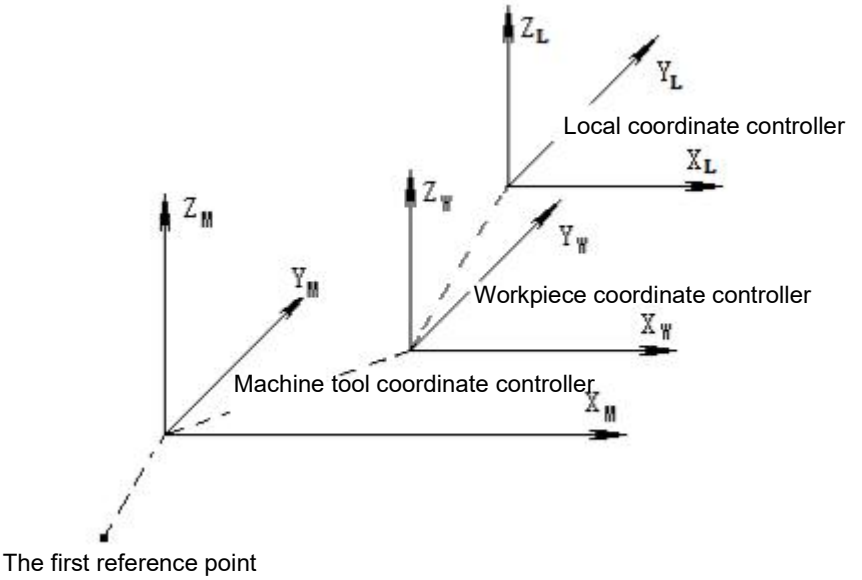


Fig. 7-2 Schematic diagram of the coordinates

■ Reference points

The reference point is a fixed position on the machine tool.

The controller can be set up with a total of six reference points, the first of which is the datum of the machine tool's coordinate controller.

The position of the reference point is set by the axis parameter (where XX is the physical axis number).

- 2XX0090: Reference point confirmation error value
- 2XX0091: First reference point
- 2XX0092: Second reference point
- 2XX0093: Third reference point
- 2XX0094: Fourth reference point
- 2XX0095: Fifth reference point
- 2XX0096: sixth reference point

■ Machine tool coordinate controller

The machine tool coordinate controller is the reference coordinate controller for machine tool control and also for the workpiece coordinate controller.

When the controller is powered up, the machine tool coordinate controller is established by performing a manual return to the reference point. Once the machine tool coordinate controller has been established it remains unchanged until the power is removed.

With absolute motors, the machine tool coordinate controller is not lost even after a power failure and there is no need to manually return to the reference point after re-powering.

The relationship between the machine tool coordinate controller and the position of each reference point is shown in Figure 7-2.

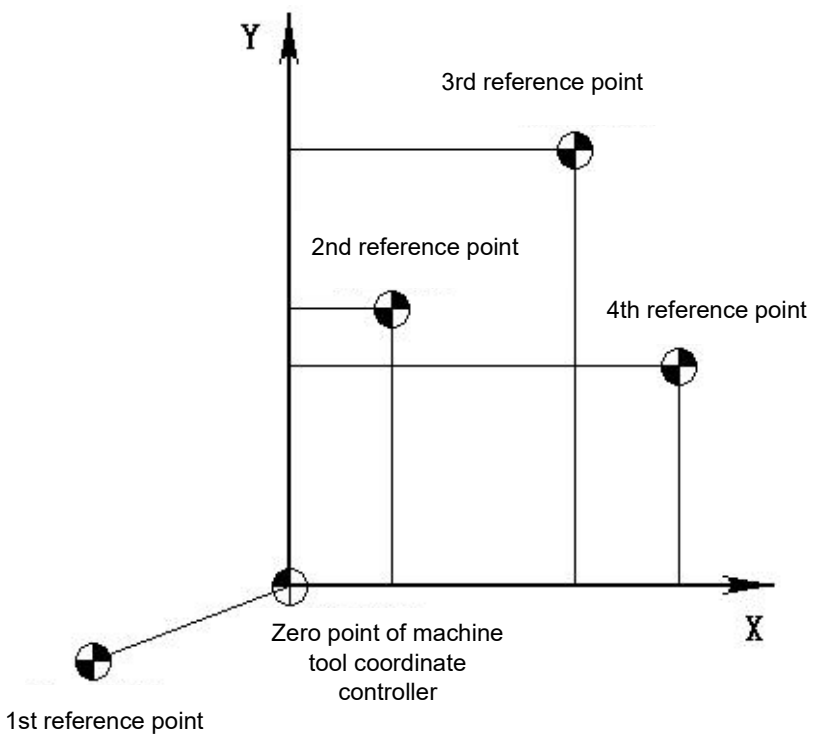


Fig. 7-3 Position relationship between the machine tool coordinate controller and the reference points

■ Workpiece coordinate controller

The workpiece coordinate controller is the coordinate controller established on the machine toold part and is the datum of the local coordinate controller.

The controller can be set up with six standard workpiece co-ordinate controllers (G54 to G59) and 93 additional workpiece co-ordinate controllers (G54.1P1 to P93).

The zero point of the workpiece coordinate controller is set by a parameter, the set value of which is the offset of the zero point of the workpiece coordinate controller relative to the zero point of the machine tool coordinate controller.

■ Local coordinate controller

A local co-ordinate controller is a controller of co-ordinates established at a local position on a machine tool part, also known as a programming co-ordinate controller.

The local co-ordinate controller is established with respect to the workpiece co-ordinate controller, usually with a zero offset, i.e. the local co-ordinate controller and the workpiece co-ordinate controller coincide, when the workpiece co-ordinate controller is also the programmer co-ordinate controller.

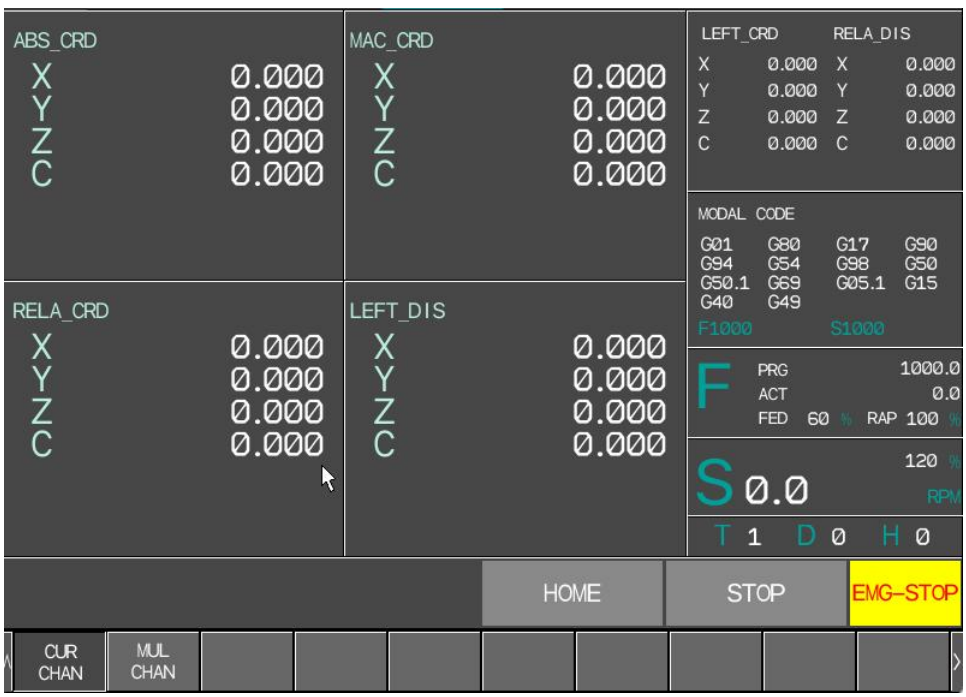


Fig. 7 - 4 controller integrated coordinate controller screen

7.2.2 Workpiece coordinate controller settings

To access the workpiece coordinate controller: **[User]** → **[UP].**

→ **[WCS].**

Fig. 7 - 5 Workpiece coordinate controller setting screen

■ Method of setting the workpiece coordinate controller

- Access to the workpiece coordinate controller setting interface.
- Use the page turn keys [**PageUp**], [**PageDown**] to select the workpiece coordinate controller (external zero offset, G54 to G59, additional coordinate controller).
- After selecting the workpiece coordinate controller, use the up and down arrow keys [↑], [↓] to select the coordinate axis to be set.
- With the axes selected, press the [**ENTER**] key or enter the coordinate values directly to bring up the input box.
- Enter the new value of the zero offset of the workpiece coordinates in the input box.
- After making sure that the input is correct, press the [**ENTER**] key to confirm the input.

■ Workpiece coordinate controller related operations

● [**current position**]

The current position of the coordinate axes is set as the zero position of the workpiece and the machine tool coordinates of the current position are automatically entered into the coordinate controller parameters.

This menu is used in conjunction with the axis selection cursor, i.e. only the axis selected by the axis selection cursor is operated on.

For example, if the cursor is displayed on the Y-axis of the G56, then when the [**Current Position**] menu is operated, the machine tool coordinate value of the position where the Y-axis is located is used as the Y-axis offset value of the G56 and is automatically entered into the Y-axis parameters of the G56.

● [**Offset**]

Offset the Workpiece zero point position of the currently selected axis of the cursor by a distance.

After the user has entered the offset distance, the controller automatically calculates the new zero position and stores it in the corresponding axis zero parameter.

Fill in a positive value to shift in the positive direction, fill in a negative value to shift in the negative direction.

● **[Measurement]**

The measurement page contains two methods of finding the coordinates of the workpiece origin of the machine toold workpiece, namely four-point centering and three-point centering.

● **[find]**

Queries the basic coordinate controller, additional coordinate controllers and external zero offsets.

7.2.3 Workpiece coordinate controller measurement

By collecting a number of specific coordinate points on the workpiece, the controller can automatically calculate the position of the workpiece coordinate controller.

■ Four-point sub-central measurement

The centre of the part is the midpoint of the two measuring points as the zero position of the workpiece.

Two coordinate points on the workpiece need to be measured in the score.

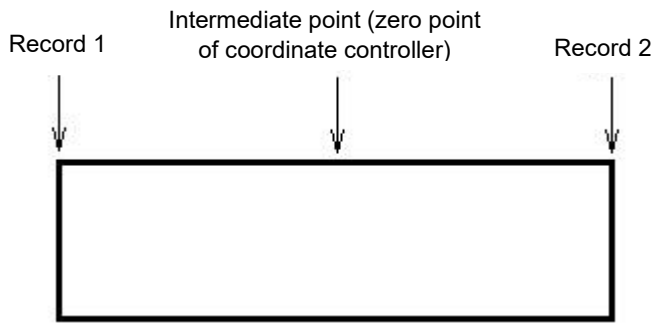


Fig. 7 - 6 Principle of the centering measurement

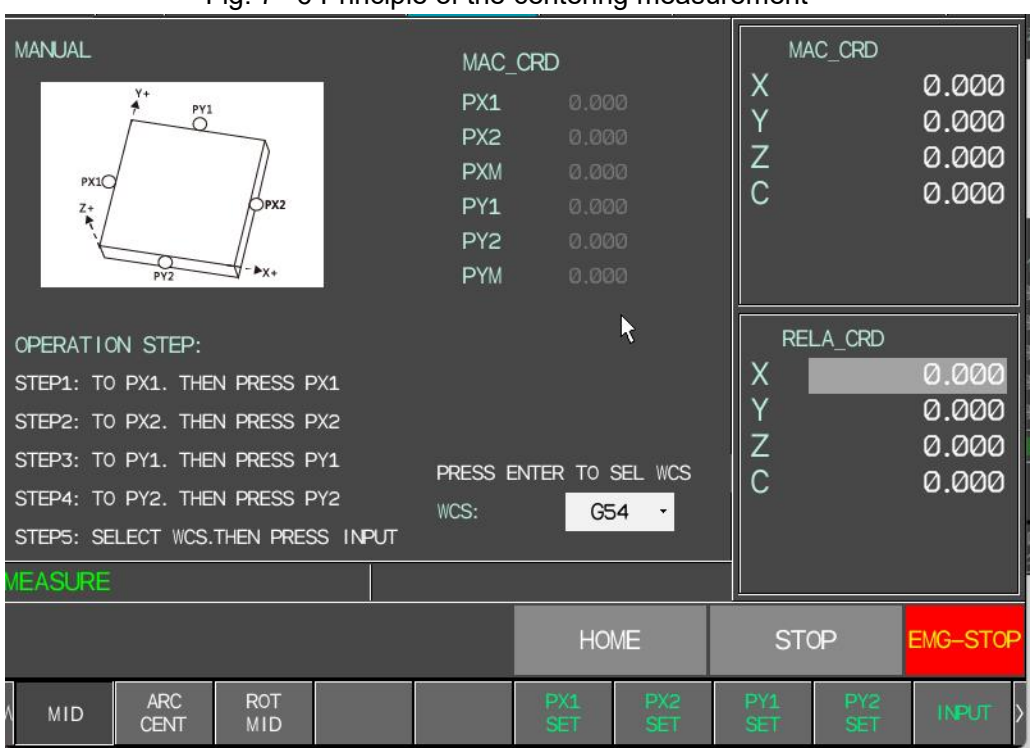


Fig. 7 - 7 Four-point split measurement screen

Measurement method:

- Bringing the tool to point PX1 and pressing the PX1 setting.
- Bringing the tool to point PX2 and pressing the PX2 setting.
- Bring the tool to the PY1 point and press the PY1 setting.
- Bring the tool to the PY2 point and press the PY2 setting.
- Press the **[Enter]** pull-down menu and select the workpiece coordinate controller to be entered
- The values measured by pressing **[Auxiliary point setting]** are calculated and automatically entered into the corresponding coordinate controller

■ Three point centred circle measurement

If the zero point of the workpiece is at the centre of a circle on the workpiece, the workpiece coordinate controller can be measured using the circle centre measurement function.

Round centre measurement involves measuring any three points on the workpiece circle.

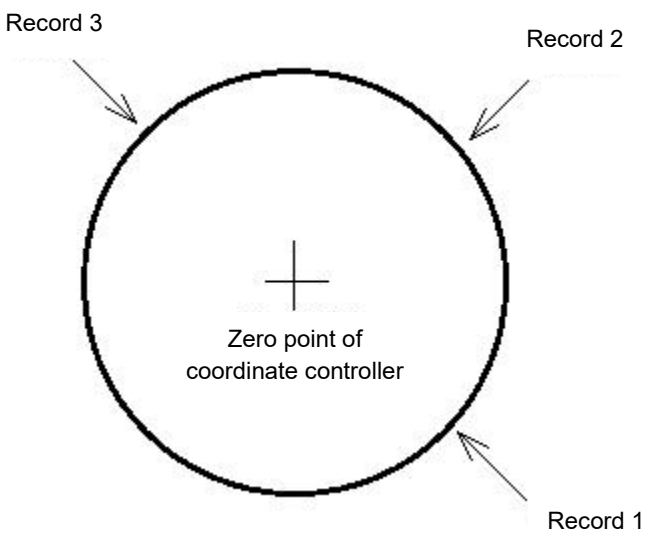


Fig. 7 - 8 Principle of three-point centering measurement

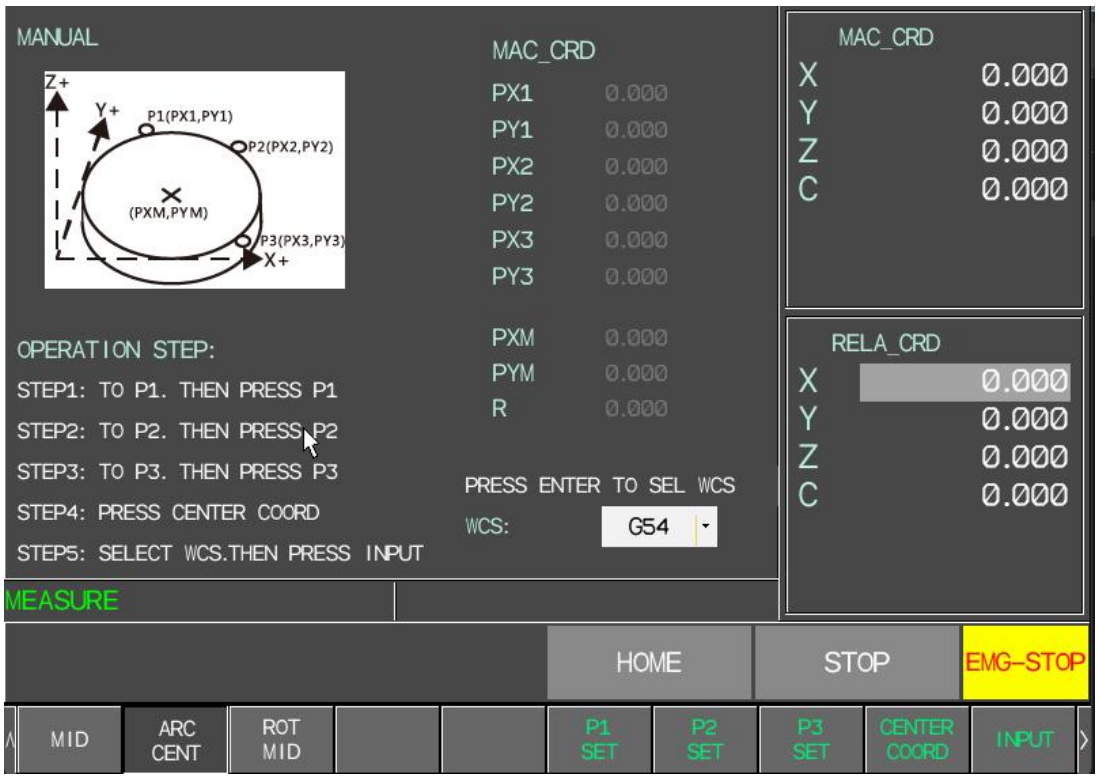


Fig. 7 - 9 Three-point centering measurement screen

Measurement method:

- Move the tool to the position of point P1 on the workpiece circle and record the coordinates of this point according to the P1 setting.
- Move the tool to the position of point P2 on the workpiece circle and record the coordinates of this point according to the P2 setting.
- Move the tool to the position of point P3 on the workpiece circle and record the coordinates of this point according to the P3 setting.
- Press the centre coordinates button
- Press the **[Enter]** pull-down menu and select the workpiece coordinate controller to be entered
- The values measured by pressing **[Auxiliary point setting] are** calculated and automatically entered into the corresponding coordinate controller

7.2.4 Automatic tool setting and measuring

Automatic tool setting login method: User settings -> Tool measurement

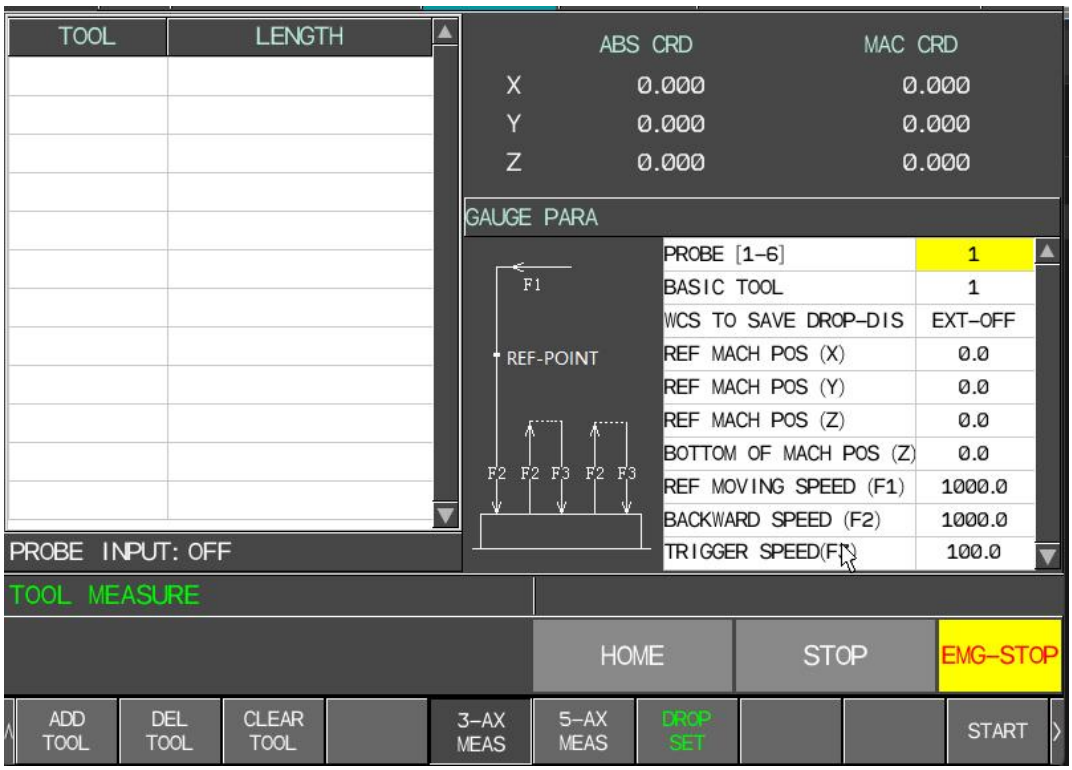


Fig. 7 - 10 Automatic tool setting measurement screen

■ Measurement parameter setting

- Probe selection: 1-6 with six sets of probes available, default probe 1 used, different probe controllers have different input signal positions
- Gauge machine tool coordinates: the machine tool coordinates of the tool setting instrument, the spindle tool will first move quickly before the measurement, the Z axis begins to measure
- Z-axis maximum measurement distance: Z-axis measurement down the maximum distance, more than the distance, the alarm, did not touch the tool setting instrument
- External offset reference: the reference tool is selected, generally using the No. 1 tool as the reference tool and measuring the difference in distance between the tool setting gauge and the surface of the workpiece.

- Reference point positioning speed F1; speed of the Z-axis reaching the reference point
- Measuring speed F2: Measuring speed at the reference point to the tool setting gauge
- Trigger speed F3: Second downward measurement speed

■ Measurement methods

- Set up the measuring parameters, move the measuring tool to a safe Z-axis position directly above the measuring instrument and record the current mechanical coordinates to fill in the mechanical coordinates of the measuring instrument
- Select the reference tool number, by default tool number 1 is selected as the reference tool. The controller will prompt for the start and end tool numbers when adding a tool, both set to 1
- The controller selects the automatic mode and presses start measurement, tool number 1 will automatically measure the tool setting gauge 3 times and then automatically calculates the average of the three test values to fill in the tool complement value of the tool number to be used
- The offset reference is determined, the automatic measurement of the reference tool No. 1 is finished, the tool is brought manually to the workpiece surface, the offset reference button is pressed and the deviation value is filled into the external offset Z
- The controller starts measuring in automatic mode, adds the tool and automatically adjusts and measures the tool according to the tool number added by the user, all at once.
- Finally, all measurement data is automatically filled in the tool complement, check that there are no abnormalities in the data and press the save result button, the measurement is complete.

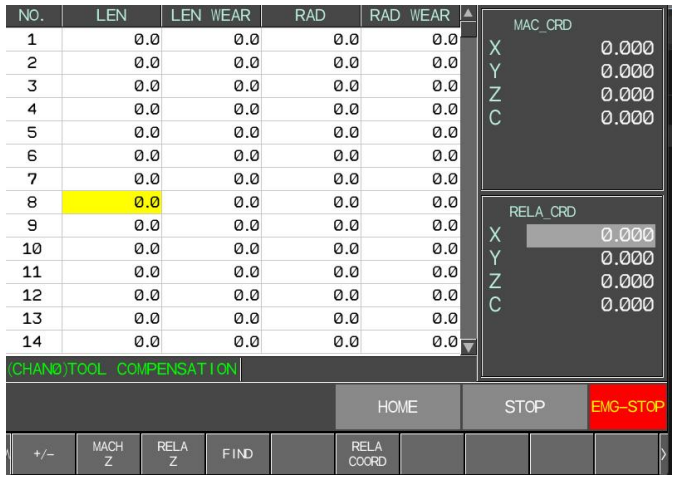


Fig. 7 - 11 Tool compensation screen

7.2 Controller parameters settings

7.3.1 Parameter authority hierarchy

The controller parameters are divided into three levels according to the level of authority.

- Controller manufacturer level

The parameters at this level are core parameters related to the operation of the controller. Modification of such parameters may cause abnormalities in the operation of the controller and require the highest level of authority for their modification.

- Machine tool manufacturer level

The parameters at this level belong to the machine tool configuration parameters and are modified by the machine tool manufacturer.

- User level

The parameters at this level are the various parameters used by the end user. The user level parameters can be further divided into two levels of access: user administrator and user operator.

In the parameter setting screen, the permissions for controller manufacturer-level parameters are displayed as 'S'; machine tool manufacturer-level parameters are displayed as 'M'; and user-level parameters are displayed as 'UM' or 'UO'.

Details of permissions can be found in the section "Permissions Management".

The table corresponding to the modification of the three levels of authority and the three levels of parameters is as follows.

Parameters / Permissions	Controller manufacturers Parameters	Machine tool manufacturers Parameters	User parameters
Controller manufacturer's rights	√	√	√
Machine tool manufacturer's competence	×	√	√
User rights	×	×	×

'√' indicates that the authority of the row is able to modify the parameters of the column

'x' means that the authority of the row is not able to modify the parameters of the column

7.3.2 Parameter settings

To access the parameter setting screen: **[SETTING]** → **[PARAMETERS]**

The process of setting the parameters.

- Enter the parameters to modify the required permissions.
- Access to the parameter setting window.
- Use the keyboard < ↑> and < ↓> keys or the **< PageUp>** and **< PageDown>** keys to locate the parameter item to be modified, or use the search method to locate the parameter number to be modified directly.
- Press the [**ENTER**] key after selecting the parameter item or enter the parameter content directly to bring up the input box.
- Enter the new parameter value in the pop-up input box, confirm that it is correct and press [**ENTER**] again to confirm the entry.

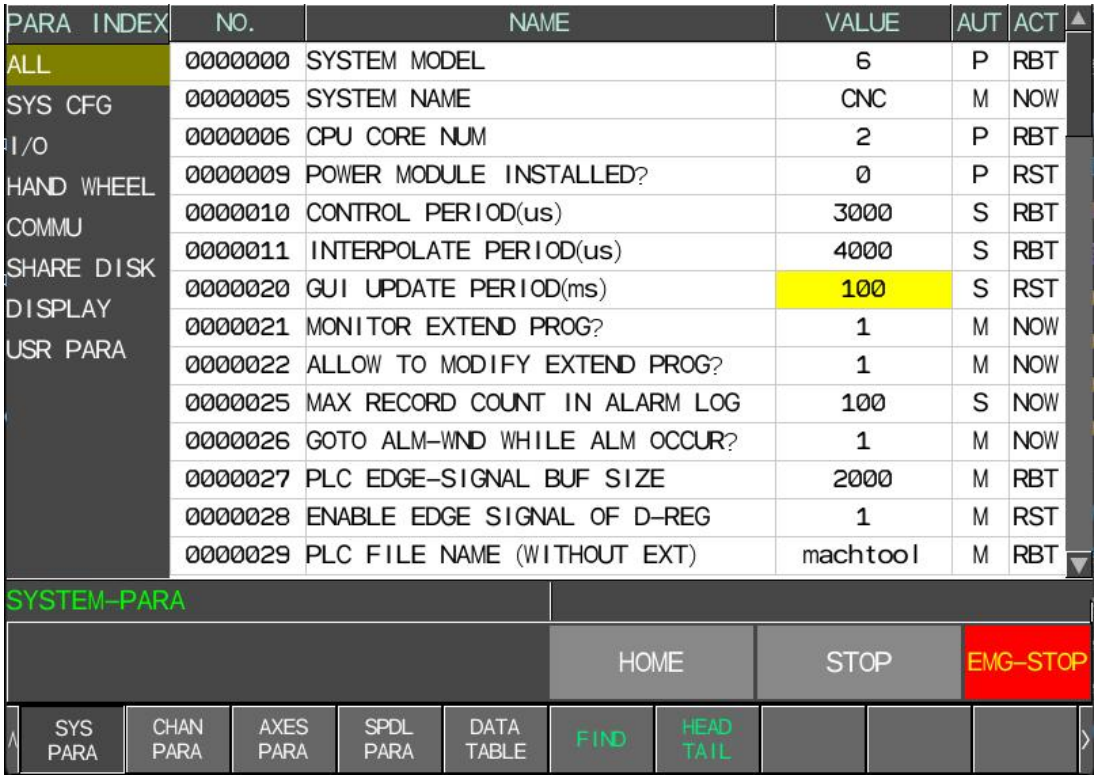


Fig. 7 - 12 Parameter setting screen

7.4 Tool magazine configuration

7.4.1 Configuring tool storage data storage space

The data related to the tool magazine is stored in the D registers and the storage space for each data is configured by D1000 to D1003.

- D1000: Tool magazine capacity, i.e. the maximum number of tools to be loaded in the tool magazine.
- D1001: Offset address of the tool library table in the D register.
- D1002: Offset address of the tool set type table in the D register.
- D1003: Offset address of the tool type table in the D register.

7.4.2 Configuring the tool magazine table

To enter the tool magazine configuration screen: [**Setting**] → [**Tool magazine table**]

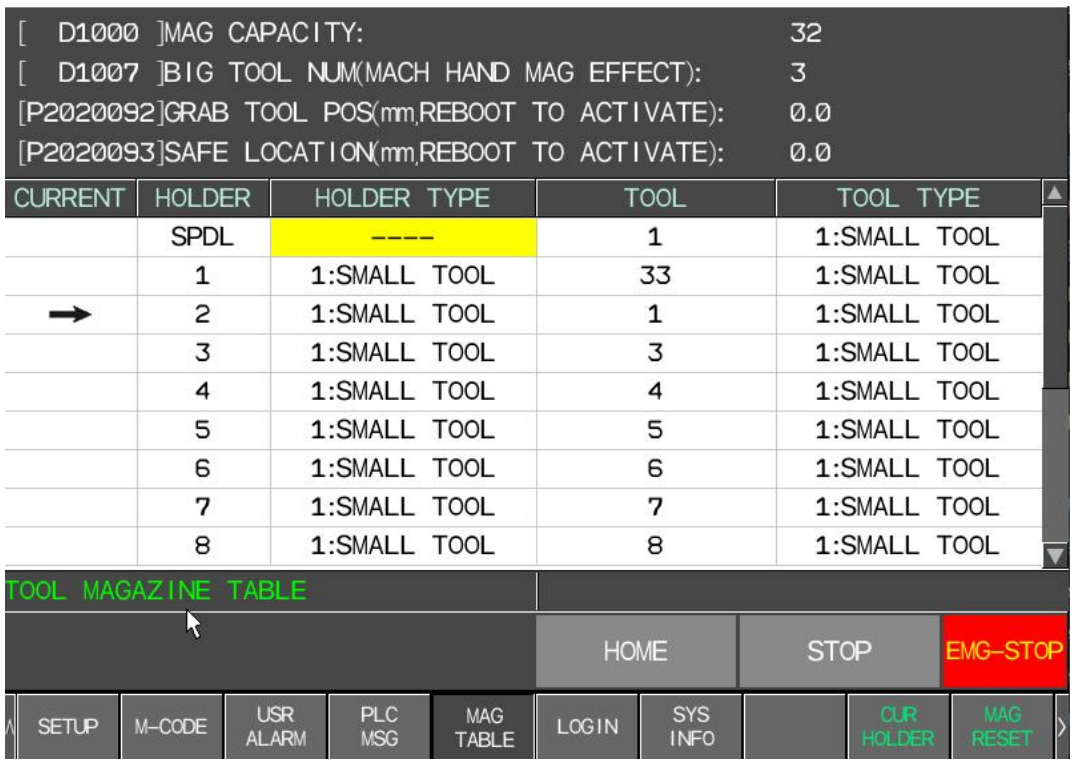


Fig. 7 - 13 Tool magazine table setting screen

■ Menu of operations related to the tool magazine table

● [**current tool set**]

Set the row where the cursor is located to the current tool set (i.e. the tool set at the tool change position).

● [**Reset**]

Reset tool magazine table, reset data includes.

- All tool numbers in order
- All sheath types revert to pocket knives
- All tool types back to small knives
- Set set no. 1 to the current cutter set
- Empty spindle tool number (i.e. spindle is tool-less)

Note: Please communicate directly with the machine tool manufacturer or our technical staff in detail for the commissioning of different models of tool magazines.

7.5 M-code extensions

7.5.1 M Code Overview

When the auxiliary function M code is executed, the controller outputs a given signal (high or low) to the specified I/O address, which can be used by the user to control various switches on the machine tool.

The following M-codes are handled internally by the controller and do not need to be extended by the user for definition.

- M00: Program suspended.
- M01: Program selection stop.
- M02: End of program, program stays on that line.
- M30: End of program.
- M98: Subroutine call.
- M99: Subroutine return.

The controller can be extended with up to 500 M-codes.

For M-codes processed internally by the controller, the user can also expand the definition.

7.5.2 M Code Properties

- **M Code**

  M code values, e.g. M618, M620.

- **Function description**

  Describe the function achieved by this M code.

- **Output signal**

  Specifies the address of the register that is output when the M code is executed. This register can be used in the PLC program for the relevant control.

- **Output values**

  Sets the status of the write to the output register when the M code is executed.

  - Output low
  - Output high

  This attribute is not valid if the M code does not set the output signal.

- **Waiting for a signal**

  Refers to the completion acknowledgement signal after the execution of the M code.

  If the M code is set with a wait signal, the program will be suspended after the M code is executed until the wait signal changes to the specified state.

  If no wait signal is set, the pause time of the program after the execution of the M code is completed is specified by the delay attribute of the M code.

- **Waiting value**

  Sets the status value of the wait register after the execution of the M code.

  - Wait value of 0. When the wait register is low, M code execution is complete.
  - Waiting value of 1. When the waiting register is high, M code execution is complete.

  This attribute is not valid if the M code does not set a wait signal.

- **Time delay**

  Set the time (in milliseconds) for the program to pause after the execution of the M code, and for the program to continue down the line when the pause time is reached.

  When a wait signal is set, the pause time starts when the wait signal arrives.

- **Calling Programs**

  Specifies the name of the extension to be called when the M code is executed.

7.5.3 M code execution process

- Internal controller machining (for internal M-codes only).
- Write the output register, if no output register is set, this step is skipped.
- Waiting register completion acknowledgement, if no waiting register is set, this step is skipped.
- delay, if the delay is set to 0, this step is skipped.
- Calling subroutines, if no calling Program is set, this step is skipped.
- M code execution is complete.

7.5.4 M code extension Definition

To access the **M-Code** extension screen: **[Settings]** → **[M-Code]**

A setting item with "-" in the window indicates that the parameter is not valid (no setting is required).

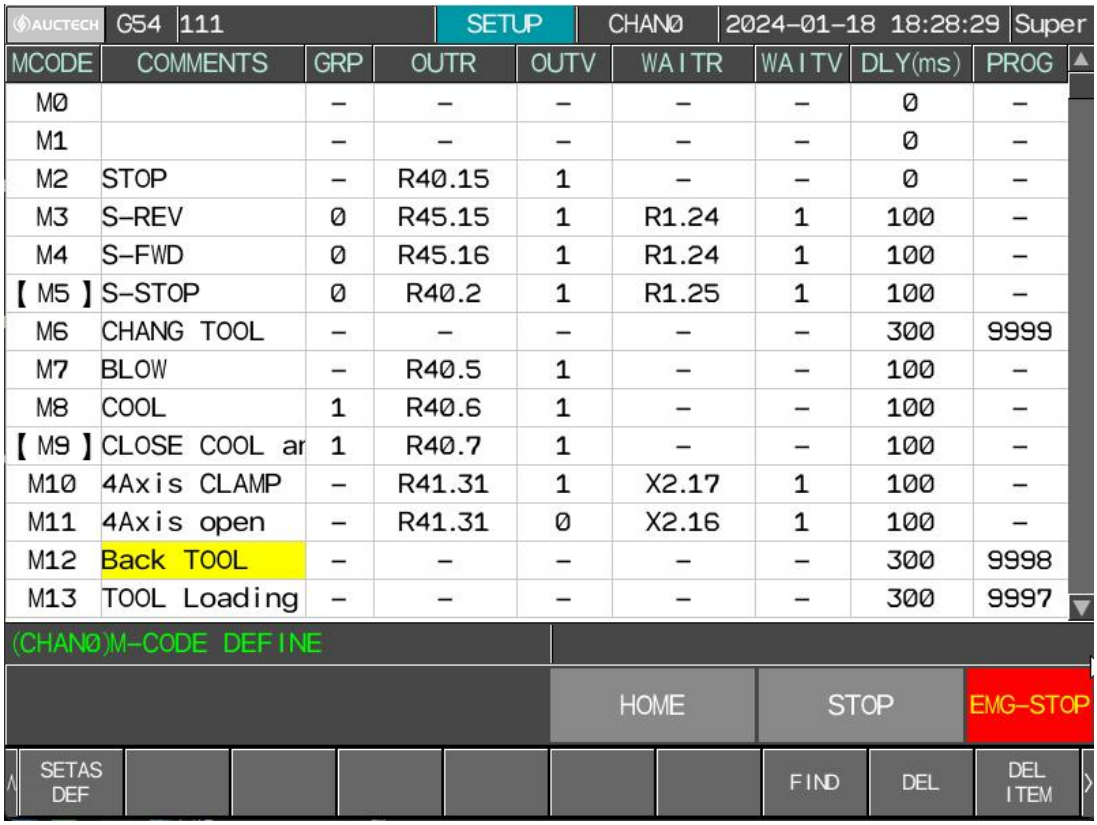


Fig. 7 - 14 M code setting screen

7.6 Alarm prompts

7.6.1 Alarm Categories

■ Controller internal alarms

This part of the alarm is defined internally by the CNC controller and cannot be changed by the user.

The internal alarm address range of the controller is A0.0 to A59.31.

■ User Alarms

User extended alarms are defined by the user and have an address range of A60.0 to A69.31, for a total of 320 points.

■ PLC News

PLC message prompts are user defined and 200 message numbers are provided for customer use.

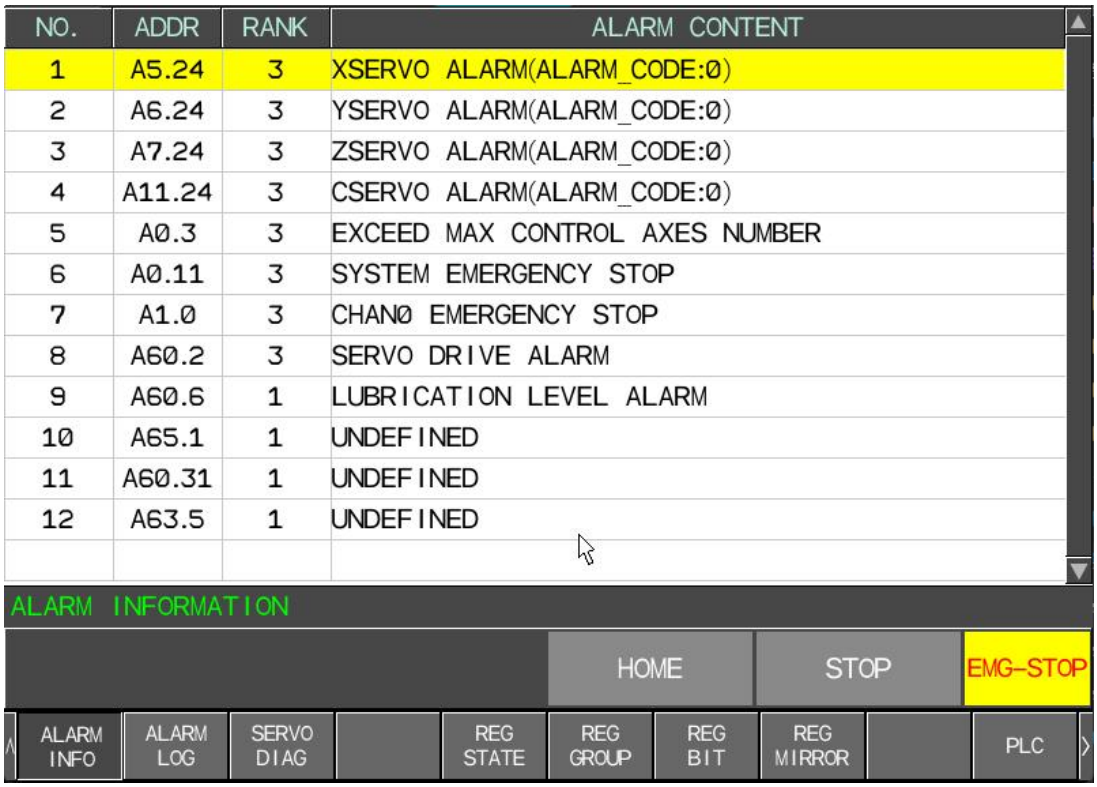


Fig. 7 - 15 Alarm display screen

7.6.2 Alarm properties

■ Address

Refers to the address of the A register corresponding to the alarm.

■ Channel Mask

Set alarm enable on each channel.
If the corresponding channel mask bit is 1, the alarm is valid for that channel, otherwise it is invalid.

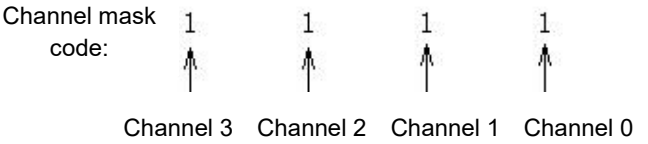


Figure 6- 14 Channel mask

■ Level

Alarm level	controller restrictions
1	• Display of alarm messages • Does not affect controller operation or program running
2	• Display of alarm messages • Cannot run programs • Allows manual operation
3	• Display of alarm messages • Cannot run programs • Cannot be operated manually

■ Alarm content

The alarm text displayed in the controller interface when an alarm occurs.

■ Keywords

Similar to the alarm content, except that the keywords are more concise and facilitate the display of comments in the PLC program.

7.6.3 User Alarms

To access the **user** alarm extension screen: **[Settings]** → **[User Alarm].**

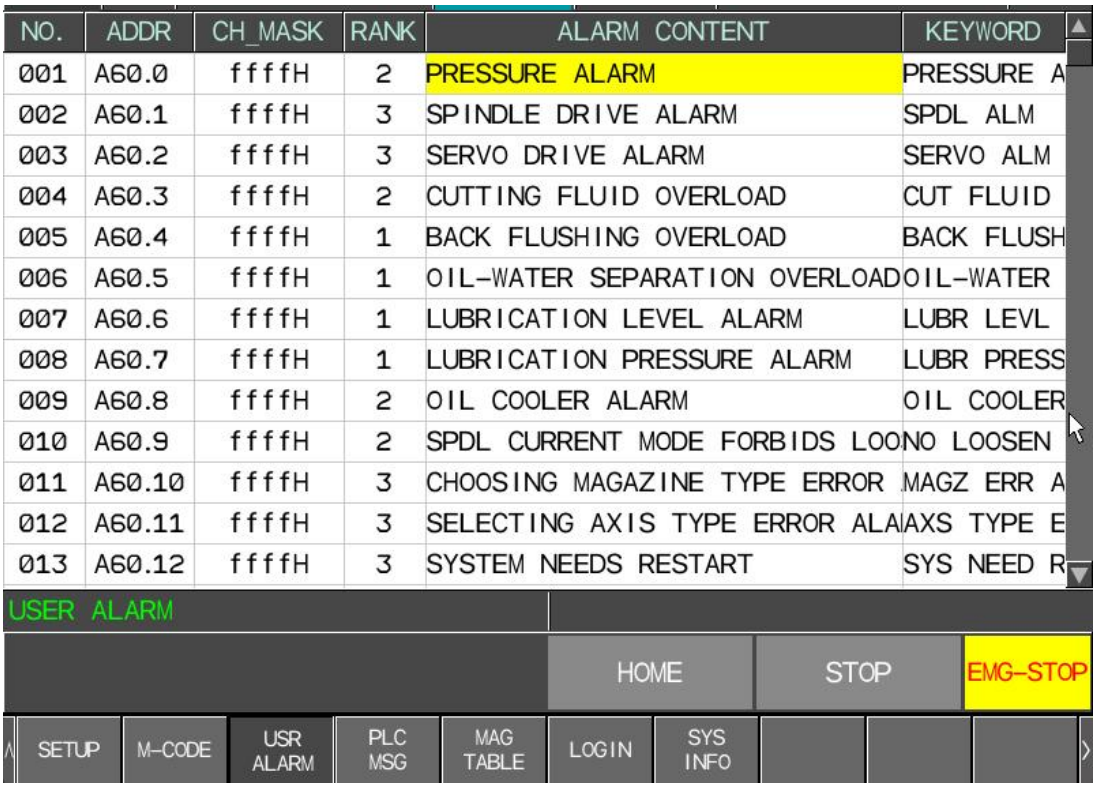


Fig. 7 - 16 User alarm content setting screen

■ Steps for setting up user alarm extension definitions

- Use the keyboard up, down, left and right arrow keys [↑], [↓], [←], [→] or [PageUp], [PageDown] to select the alarm item to be set.
- Press the [**ENTER**] key after selecting the alarm item or enter the setting value directly to bring up the input box.
- Enter the setting data in the input box.
- After confirming that the input is correct, press the [**ENTER**] key to confirm the input.

7.6.4 PLC Message Prompt

To access the PLC message screen: **[Settings]** → **[PLC Message].**

Fig. 7 - 17 PLC message content setting screen

■ Program for setting up PLC prompt extension definitions

- Use the keyboard up, down, left and right arrow keys [↑], [↓], [←], [→] or [PageUp], [PageDown] to select the message item to be set.
- Press the [**ENTER**] key after selecting the good news item or enter the setting value directly to bring up the input box.
- Enter the setting data and the prompt in the input box.
- After confirming that the input is correct, press the [**ENTER**] key to confirm the input.

7.7 Access management

7.7.1 Overview

Permissions are used to divide and restrict controller functions.

Users can only use functions that are equivalent to their privileges. Users with higher privileges can use the functions of lower privileges, but users with lower privileges cannot use the functions of higher privileges.

The controller is divided into four levels of authority, from highest to lowest, as follows

- Controller manufacturer level access (**initial password 2388**)
- Machine tool manufacturer level access (**initial password 990**)
- User administrator level access (**initial password 400**)
- User operator level access (**initial password 111**)

7.7.2 Permissions management

How to access the permission management screen: [**Settings**] → [**Permission Management]**

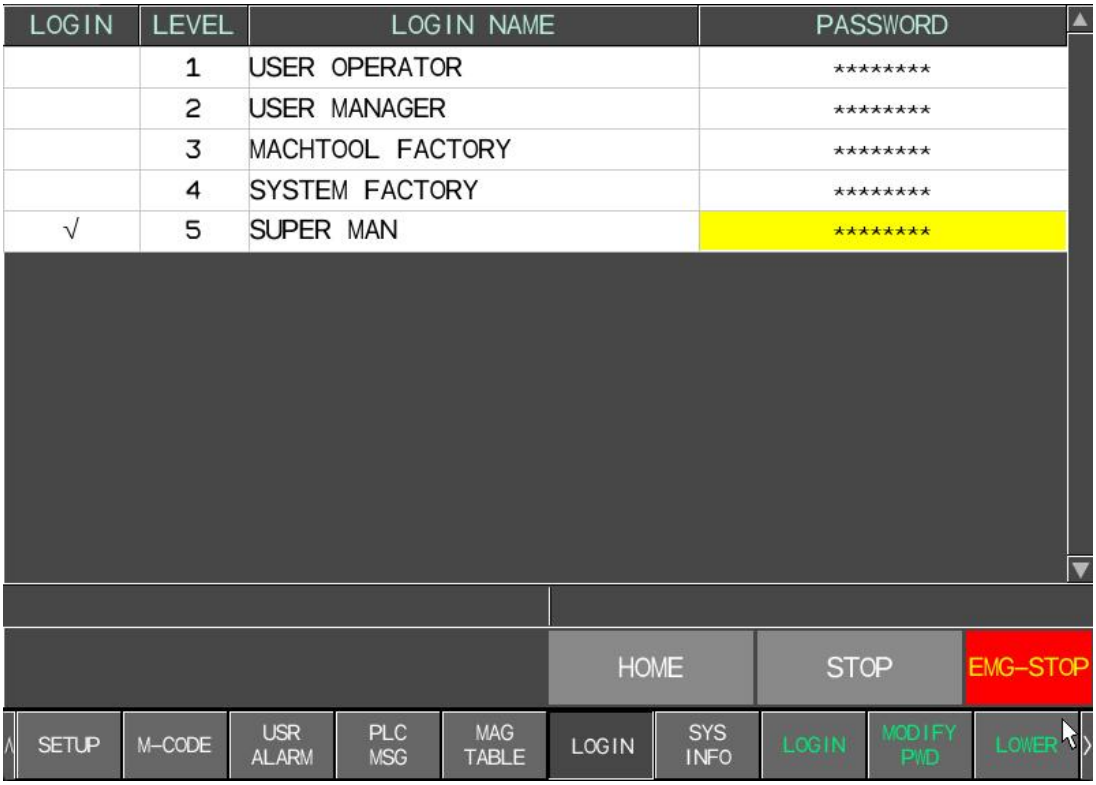


Fig. 7 - 18 Permissions login screen

■ Permissions Login

- Access to the permissions management interface.
- Move the cursor and select the name of the authority to be logged in.
- Press the [ENTER] key to bring up the password entry box.
- Enter the access code in the input box and confirm by pressing [ENTER].
- If the password is entered correctly and the login is successful, a 'tick' is displayed in front of the corresponding permission name, otherwise a password error is indicated.

■ Change permission password

- Access to the permissions management interface.
- Move the cursor and select the name of the authority whose password needs to be changed.

- Press the [**ENTER**] key to bring up the old password entry box.
- Enter the old password in the input box and press [**ENTER**] to confirm. If entered correctly, a new password entry box pops up.
- Enter the new password in the input box and confirm by pressing [**ENTER**], the new password needs to be entered twice.
- If the old password is entered successfully and the new password entered twice is the same, the password change is successful.

■ Permissions downgrade

Permissions downgrade allows downgrading from higher to lower level permissions.

Each time you press the [**downgrade**] menu, the privileges are reduced by one level until you log out.

7.8 Variable viewing and modification

The value of a variable can either be modified using code in a macro program or by viewing and manually modifying the value of the variable in a window.

To access the variable modification window: **[User]** → **[Variables]**

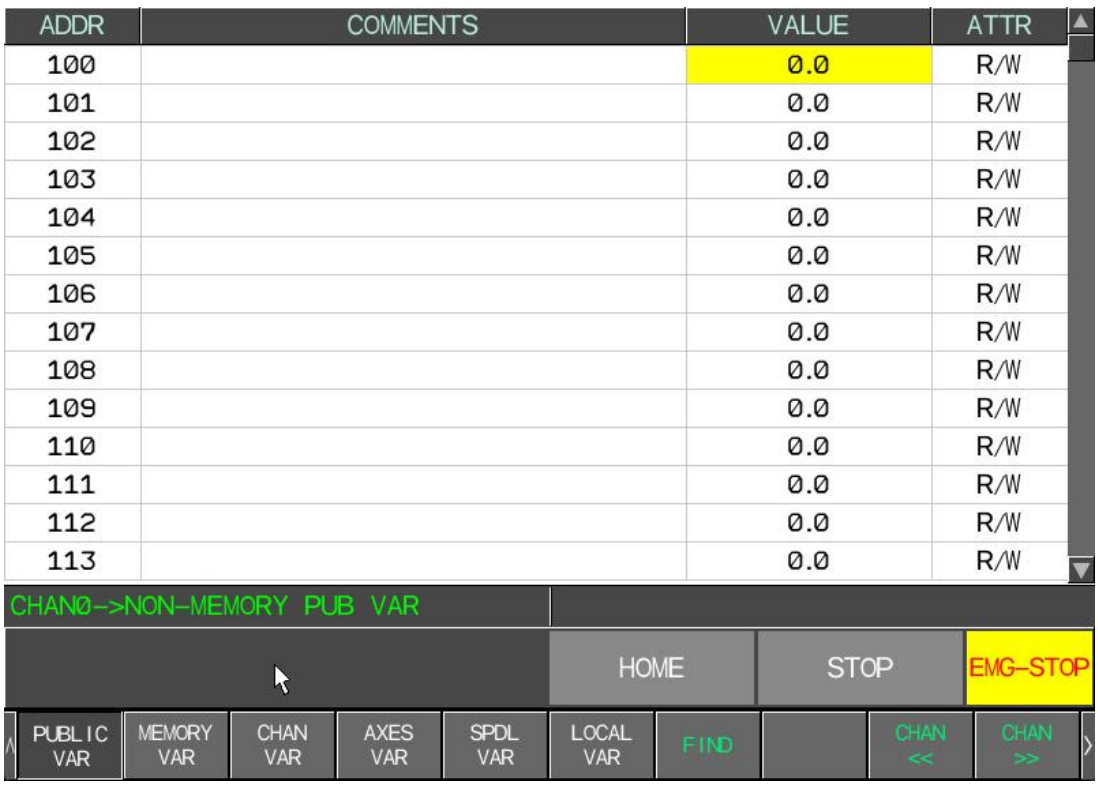


Fig. 7 - 19 Controller variables screen

Variables are divided into local variables, public variables, memory variables and controller variables (including channel variables, axis variables and spindle variables), where local variables, public variables and memory variables can be modified and other variables are read-only and cannot be modified.

7.9 Machining count

The statistics function is a function for counting the number of pieces machine toold on a workpiece.

To access the statistics window: **[User]** → **[Machining statistics]**.

Use of workpiece machining counters.

1. controller provides 20 different workpiece counting command codes and records
2. M60\M61\M62... And so on from the first to the twentieth set of counts
3. When the machining program reads the corresponding M code, the count of completed machining is added by 1. When the total number of pieces planned to be machine toold is reached, the controller issues a completion prompt and pauses.

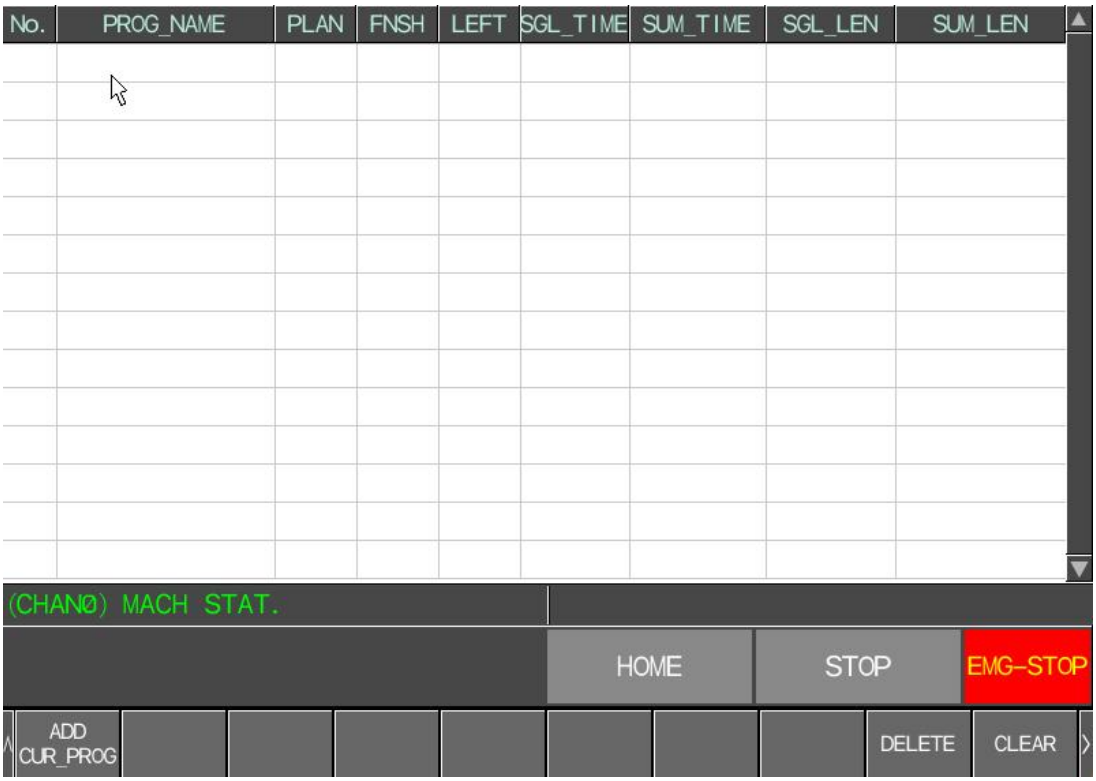


Fig. 7 - 20 Machining count statistics screen

7.10 controller Information

7.10.1 Data export

The data export function allows you to export controller-related data to a USB stick.

Data export requires the appropriate permissions to be entered and a USB stick needs to be inserted before exporting.

To access the data export screen: **[Settings]** → **[controller Information]** → **[Data Export]**

■ Operation menu

- [select]

Set the data where the cursor is currently located to the selected state ('√' is displayed in front of the data). Press the [Select] menu repeatedly to switch between selected and unselected states.

- [select all]

Set all data to the selected state. Pressing the [Select All] menu repeatedly toggles the selected and unselected status of all data.

- [Export] Exports the selected data to a USB stick.

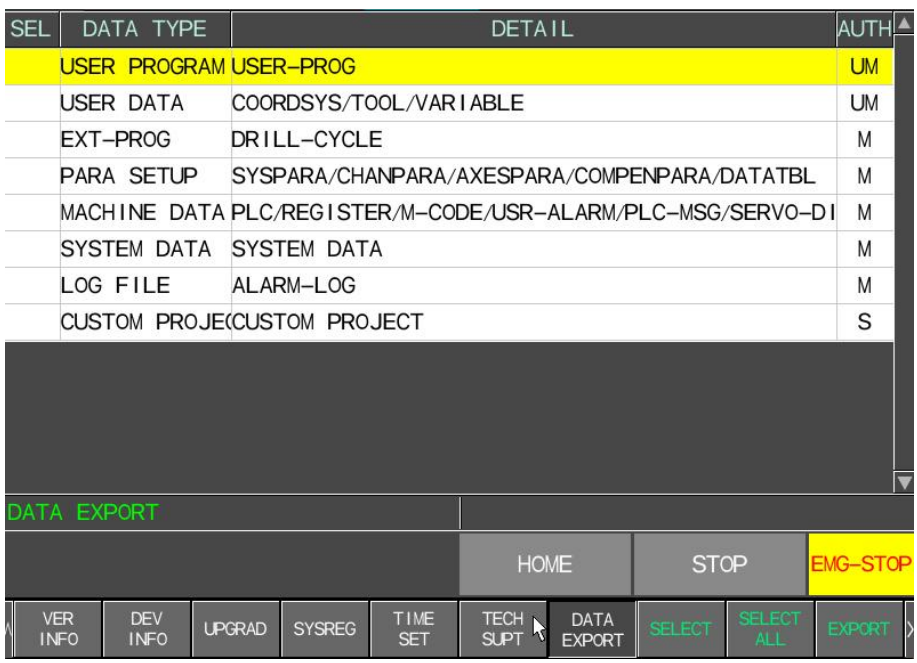


Fig. 7 - 21 Data export screen

7.10.2 controller time setting

How to access the data export screen for the **controller time**: **[Settings] → [controller letter message] → [time setting]**

To access the data export screen **with the term registration code**: **[Settings] → [Settings [controller Information] → [controller Registration] →[REG CODE]**

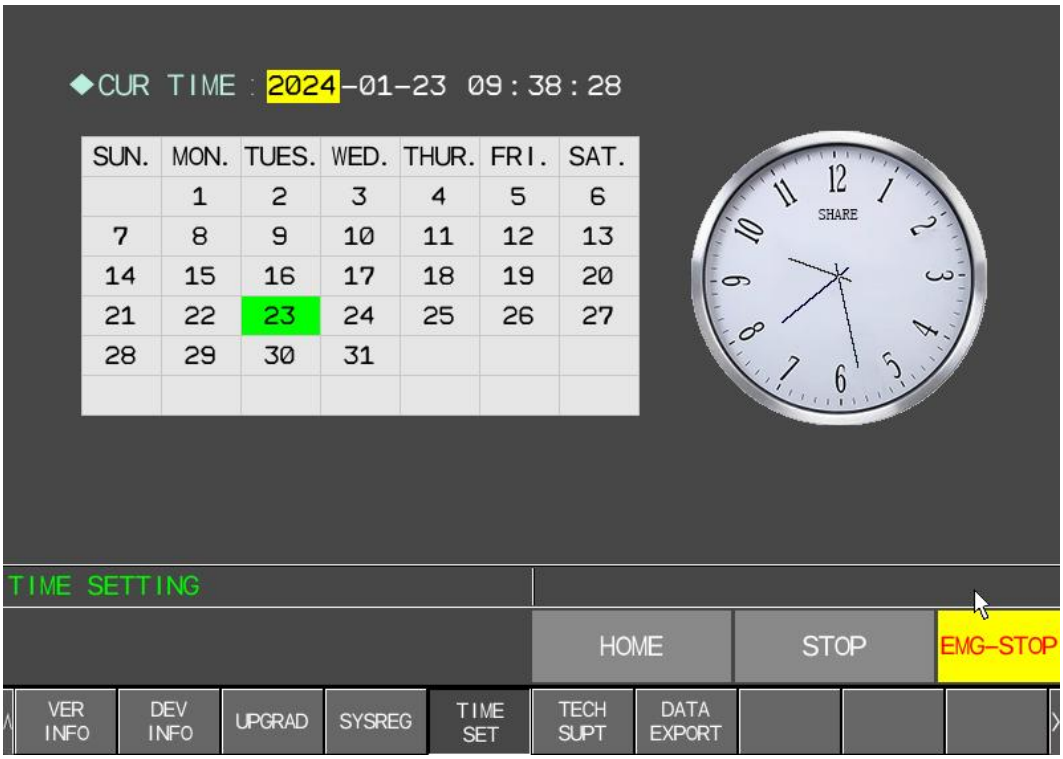


Fig. 7 - 22 Time setting screen

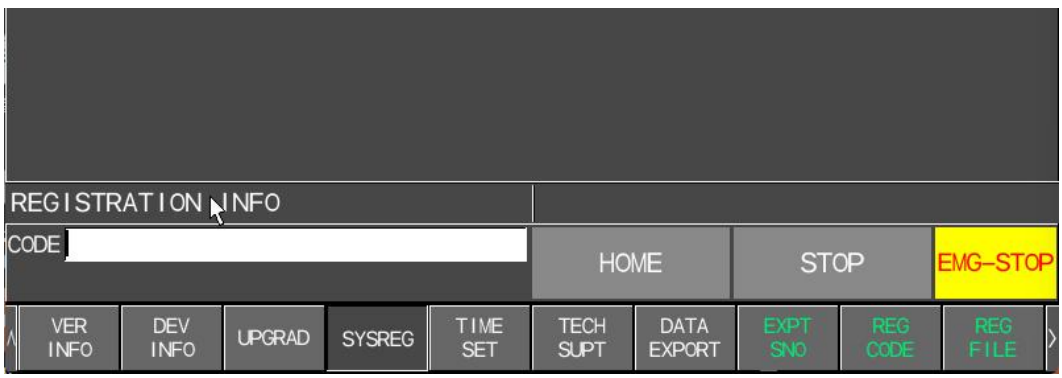


Fig. 7 - 23 Duration of use registration code

8 PLC Programming

The programming methods for the ladder language are described in the PLC Programming Manual. This chapter does not describe the programming methods, but only the use of the programming environment (programming interface) in detail.

8.1 PLC online programming and diagnostics

8.1.1 PLC programming interface

To access the PLC programming screen: **[Diagnostics] → [PLC Ladder].**

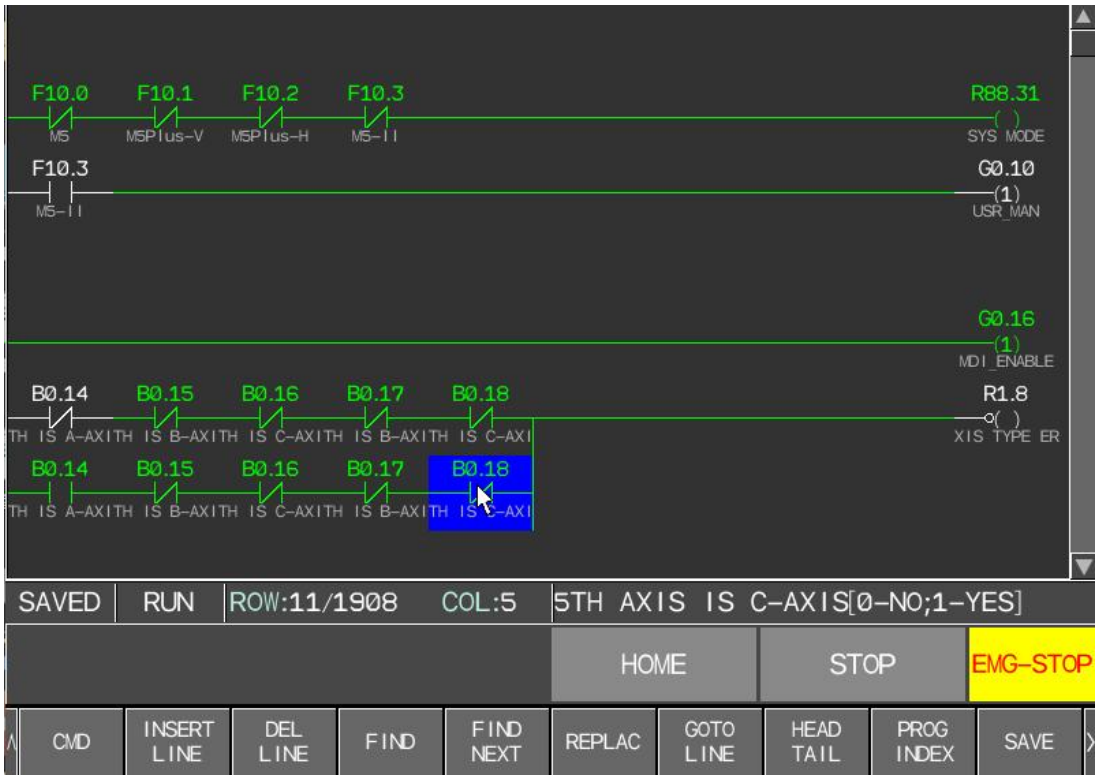


Figure 8- 1 PLC programming interface

8.1.2 Ladder program composition

The ladder program is divided into an arithmetic area and an output area.

The operation area includes the registers or functional instructions involved in the ladder operation, the registers in the operation area can be either input signals or output signals. the PLC can only read the registers in the operation area.

The output area specifies the output register for the result of the operation. The registers in the output area can only be writable registers to which the PLC performs write operations.

PLC ladder program consists of normally open contact, normally closed contact, horizontal lead line, vertical lead line, output coil, normally open contact rising edge, normally open contact falling edge, normally closed contact rising edge, normally closed contact falling edge, set output, reset output and function instruction.

8.1.3 Programming menu Description

■ Basic command menu

- Normally open contacts
- Normally closed contacts
- Output coils
- Output coil (take reverse output)
- Horizontal lead-in lines
- Vertical lead-in lines
- Rising along
- Descending along
- Reset output
- Reset output

Fig. 8-2 Function bar for basic commands

■ Function command menu

After selecting the **[Function Command]** menu under the **[Command]** menu, a list of function command menus pops up in the upper left corner of the PLC interface.

Select the function instruction by moving the cursor up, down, left and right arrow keys [↑], [↓], [←], [→] on the function instruction menu. After the cursor has selected the functional instruction, press [**ENTER**] to insert the selected functional instruction at the position of the cursor in the PLC programming window.

Press the [**CANCEL**] key to close the function command menu.

For details of how to use the functional instructions, please refer to the PLC Programming Manual.

Fig. 8-3 Functional instructions

■ Ladder diagram editing menu

● **[insert line]**

Inserts a new line in front of the line where the current cursor is located.

After a row insertion operation, the current cursor becomes a blank row and the rows following the cursor are moved down one row in turn.

Figure 8-4 Insert row

● **[delete line]**

Deletes the line where the cursor is currently located.

After a delete row operation, the row in which the current cursor is located is deleted and the rows following the cursor are moved up one row in turn.

Figure 8-5 Delete line

If a functional instruction exists on the line where the cursor is located, the function of deleting the line differs depending on the position of the cursor.

- The [Delete Line] operation deletes the entire line of program occupied by the function instruction if the cursor is hovering over the function instruction.

- If the cursor rests on a basic instruction, [Delete Line] deletes only the section of the basic instruction where the cursor is located (no functional instructions are deleted).

● **[locate line]**

Moves the cursor to the specified line position.

● **[Find]/[Continue to find]**

The lookup function allows you to find a component with a specified name in a ladder diagram; the lookup function is not case sensitive.

The search function is a fuzzy search, i.e. it starts from the head of the component name. For example, if the search target is "Y3.1", you can find "Y3.1", "Y3.10", "Y3.15 "etc.

To find a component precisely, enter the full component name.

Once you have found a target, you can press the [**Continue Search**] menu to go backwards to find the next target without having to enter the component name when continuing the search.

● **[Reserved]**

The ladder diagram is written to memory and saved so that it is not lost even after a power failure.

When editing changes are made to the ladder program, they are done on the PLC backup program. Therefore, the controller is still running the original PLC program during the editing process.

After the ladder diagram has been saved, the controller starts running the new PLC program.

● **[block manipulation]**

The block operation copies an area of the ladder into the paste buffer for pasting or cutting.

Menus related to block operations.

- [**Define block header**]: defines the line where the cursor is located as the start line of the block.
- [**Define end of block**]: defines the line where the cursor is located as the end line of the block.
- [**Block Copy**]: Copies the defined block to the cursor position.
- [**Block Cut**]: Copies the defined block to the cursor position, while deleting the block in its original position.

- **[Disabled]**

The masked parts of the PLC program become greyed out and are not executed.

To use the mask function, you must move the cursor to the output node on the right and press the [**Mask**] menu key.

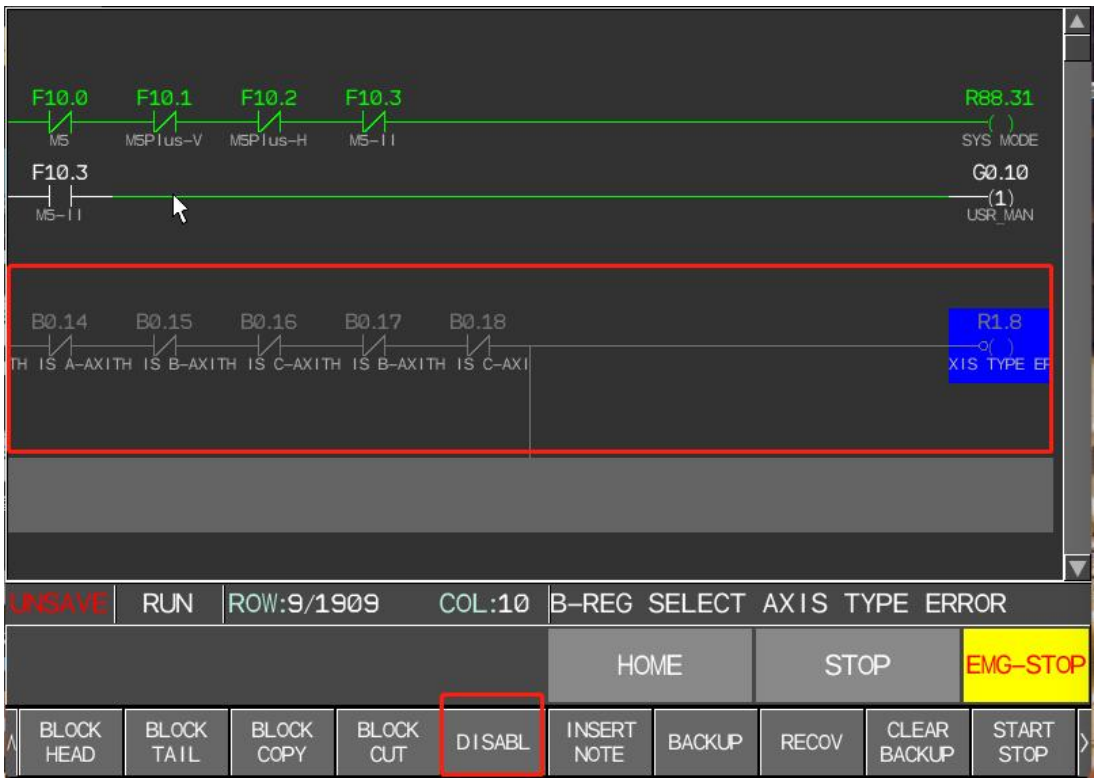


Fig. 8-6 Masking function

- **[Replace]**

The replace function replaces the component specified in the ladder diagram with a new component of another type.

- **[Backup]/[Restore]**

Backup means that a copy of the PLC program is made and stored in the backup memory.

Recovery means restoring the PLC backup program stored in the backup memory to the controller running PLC program.

Figure 8-7 Restoring the ladder diagram

8.1.4 PLC online diagnostics

The running status of the PLC program can be observed through the ladder interface during operation.

A green node indicates that the node is on (value of 1).

A white node indicates that the node is disconnected (value of 0).

8.1.5 Ladder programming example

Normally open and normally closed contacts and output coils can be used to perform operations such as with, or and without.

For example, to implement the operation Y02.0 = (X02.0 & (/X02.5)) | (/X03.0) (where '/' means to take the inverse), the ladder is programmed as follows.

- Position the cursor at the start of the programming position, tap on the menu (normally open contacts), enter "X02.0" in the component input box, enter to confirm, component X02.0 appears at the current cursor position.
- Move the cursor one position to the right, tap Menu (normally closed contact), enter the address number "X02.5" in the component input box, enter to confirm, component X02.5 appears at the current cursor position.
- Position the cursor at the beginning of the next line, tap Menu (normally closed contact), enter the address "X03.0" in the component input box, enter to confirm, component X03.0 appears at the current cursor position.
- Move the cursor one place to the right and tap on the menu (horizontal guide line) to draw a horizontal guide line at the current cursor position.
- Move the cursor up one position and tap on the menu (Vertical guide lines) to draw a vertical guide line at the bottom right of the current cursor.
- Click on the menu (Output Coil), enter the address "Y02.0" in the component input box and enter to confirm, the output coil and the necessary horizontal lead wires will be generated automatically.

Fig. 8-8 Example of a ladder diagram

8. 2 Register monitoring

8.2.1 Register Overview

The registers are the computing units of the PLC. By monitoring the status of the registers, you can understand the operation status of the PLC.

The main types of registers are as follows.

- Register X: Signal input from the machine tool to the PLC.
- Register Y: PLC output to the machine tool.
- Register F: Signal input from the CNC controller to the PLC.
- Register G: PLC output to the CNC controller.
- Register K: Signal input from the operation panel to the PLC.
- Register L: PLC output to the operation panel.
- Register R: PLC intermediate register.
- Register A: Alarm signal.
- Register B: Power failure save register.

8.2.2 Register access fields

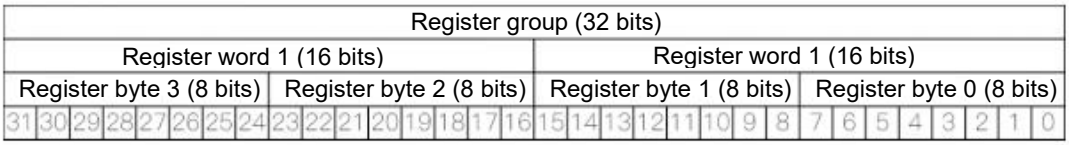


Figure 8- 9 Register access fields

All registers are 32-bit, and each register can be divided by access field into.

- Register Group

The entire 32-bit register.

The group is accessed as register name + group number, e.g. F120, i.e. accessing the F register of group 120.

- Register Word

Each register can be divided into two words of 16 bits each.

The word is accessed as register name + 'W' + group number + '. ' + font size, e.g. FW120.0, i.e. to access the 0th word of the 120th group F register.

Font size range 0 to 1.

- Register bytes

Each register can be divided into 4 bytes of 8 bits each.

Bytes are accessed as register name + 'B' + group number + '. ' + byte number, e.g. FB120.0, i.e. accessing the 0th byte of the 120th group F register.

Byte number range 0 to 3.

- Register Bits

32 bits per register.

Bits are accessed as register name + group number + '. ' + bit number, e.g. F120.0, i.e. accessing bit 0 of the 120th group F register.

Bit number range 0 to 31.

8.2.3 Register Status

To access the register status monitor screen: **[Diagnostics]** → **[Register Status]**

The register status allows monitoring of the status of each bit of the register and is distinguished by different colours.

- Red: the status of this bit is 1
- Grey: the status of this bit is 0

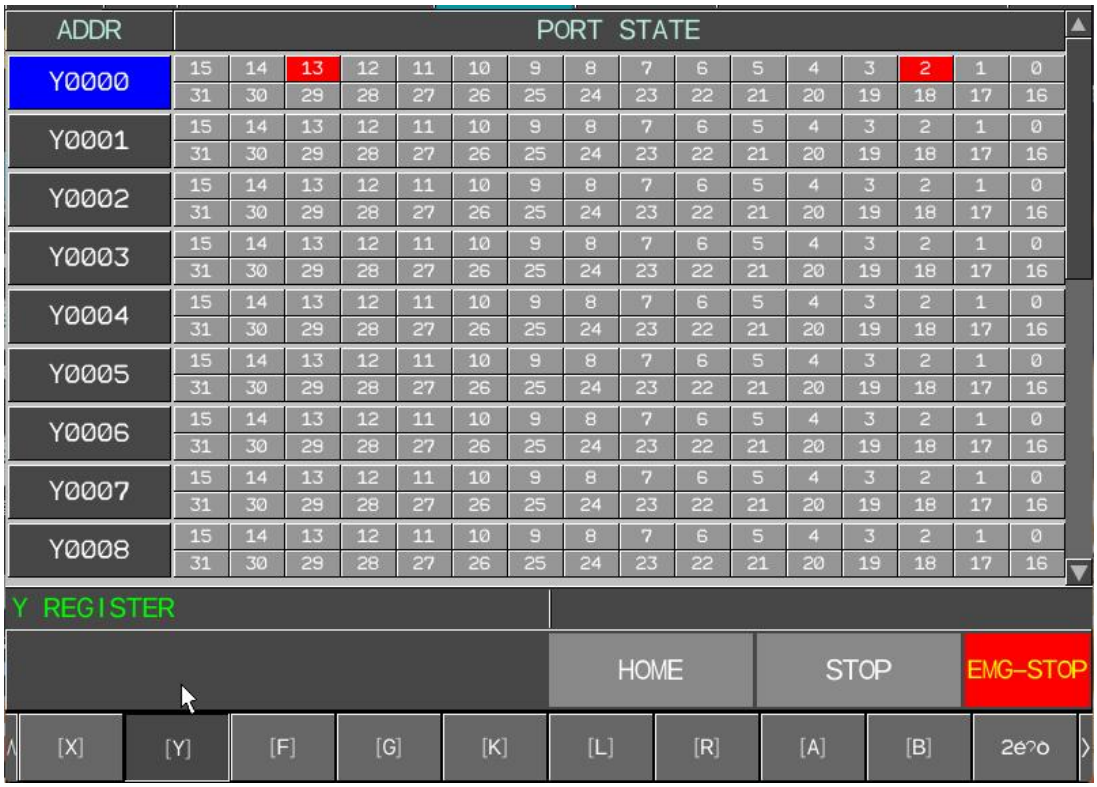


Figure 8- 10 Register Status

8.2.4 Register Group

To access the register group monitoring screen: **[Diagnostics] → [Register Group]**

The register group is used to monitor the value of each set of registers, which is displayed as a decimal integer.

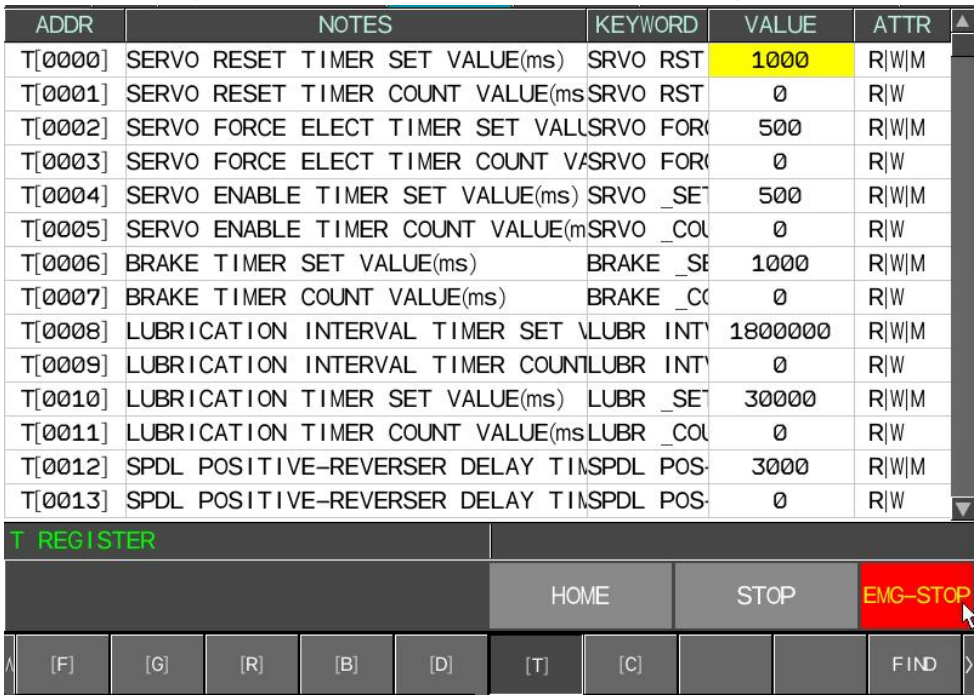


Figure 8- 11 Register group

Each set of registers has five displays.

- Address: sequential address of the register set.
- Note: A description of the function of this group of registers.
- Keywords: keywords are similar to comments, only more concise to facilitate display in the PLC program.
- Value: the data currently stored in this group of registers.
- Attributes: type of register read/write and storage type.
  - R: Readable register
  - W: Writable register
  - M: Memory type register (power failure save)

8.2.5 Register Bits

To access the register bit monitoring screen: **[Diagnostics]** → **[Register Bit]**

Register bits are similar to register status in that they both display the status of each bit of the register. The difference is that the register bits can display more detailed information about each bit.

Each register bit also displays five items, the same as the register group.

ADDR	NOTES	KEYWORD	STATE	ATTR
X0.0	PRESSURE MEASURE	PSSR MEAS	OFF	R
X0.1	LUBR LEVEL ALARM	LUBR LEVL	OFF	R
X0.2	LUBR PRESSURE ALARM	LUBR PSSR	OFF	R
X0.3	CUT FLUID OVRD	CUT FLUD O	OFF	R
X0.4	BACK FLUSH OVRD	BACK FLSH	OFF	R
X0.5	TOOL MAGAZINE MOTOR OVERLOAD	MAGZ MOTOR	OFF	R
X0.6	TOOL ARM MOTOR OVERLOAD	TL ARM MOT	OFF	R
X0.7	PANEL EMERGENCY STOP	PANEL STOP	OFF	R
X0.8	SPDL TIGHTEN SIGNAL	SPDL TIGHT	OFF	R
X0.9	SPDL LOOSEN SIGNAL	SPDL LOOSE	OFF	R
X0.10	SPDL LOOSEN BUTTUN	SPDL LOOSE	OFF	R
X0.11	TOOL HEAD COUNT	TL-HEAD CO	OFF	R
X0.12	TOOL HEAD ADVANCE ARRIVAL	TL-HEAD AD	OFF	R
X0.13	TOOL HEAD BACK ARRIVAL	TL-HEAD BA	OFF	R

Figure 8- 12 Register Bits

Note: Comments on register bits are displayed directly on the PLC ladder diagram. In the PLC program interface, the keyword of the bit is displayed below the register bit and its full comment content can be displayed in the bottom right corner.

8. 3 PLC messages

8.3.1 Overview

Use the PLC function command MSG to display a specified message on the controller screen.

PLC messages can be entered in the controller interface and the PLC program simply specifies the corresponding message number.

After the controller displays the PLC message, you can press any key to clear it.

8.3.2 PLC message set

The definition of the PLC message can be entered in the PLC message screen.

To access the PLC message settings screen: **[Settings]** → **[PLC Message].**

The left-hand side of the PLC message setting window shows the message number, which is called by the PLC function instruction MSG; the right-hand side shows the PLC message content.

Figure 8-13 PLC Message Setting Window

9 Cycle function of rotary axis

■ Overview

The rotary axis cycle function prevents coordinate overflow of the rotary axis.

When the cyclic function of the rotary axis is switched on, for incrementally programmed instructions, the tool moves the distance specified in the instruction; for absolute value instructions, the controller does the following.

- Transformation of the programmed command coordinates to 0 to 360°.
- If no shortest path is set, the direction of movement of the tool is determined according to the relationship between the converted target position and the current position: if the target position is smaller than the current position, it moves in the negative direction; if the target position is larger than the current position, it moves in the positive direction.
- If the shortest path has been set, the tool moves in the direction of the shortest path to the target position.

■ Settings

The rotary axis cycle function is set by the axis parameters (where XX is the physical axis number).

- 2XX0180: coordinate cycle enable, on when set to 1, off when set to 0
- 2XX0181: Coordinate cycle width.
- 2XX0182: Short path selection enable, on when set to 1, off when set to 0.

Short path selection is only valid when the cycle function is switched on.

■ Example

Assume that the A axis is the axis of rotation and that the momentum is 360° per transfer.

Turn on the cycle function for the rotary axis and set it to select the shortest path. When the following program is executed, the A-axis moves as shown below.

G90 A0.	Actual movement	After the end of the move Absolute coordinate values
N1 G90 A-150.0;	-150	210
N2 G90 A540.0;	-30	180
N3 G90 A-620.0;	-80	100
N4 G91 A380.0;	+380	120
N5 G91 A-840.0;	-840	0

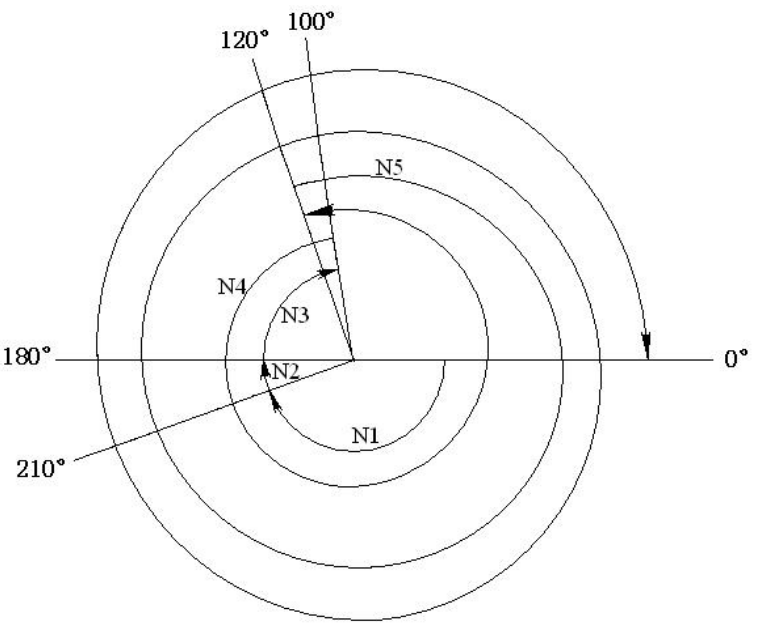


Fig. 9- 1 Example of a rotary axis cycle function

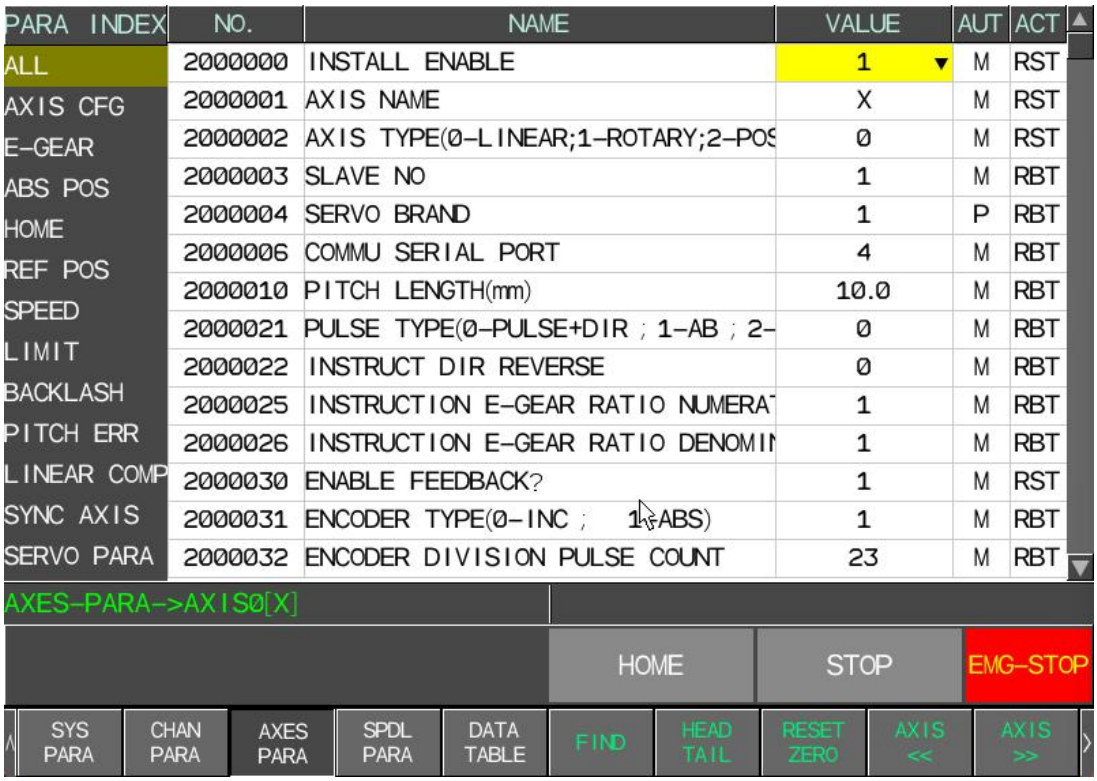


Fig. 9-2 Rotary axis cycle controller setting screen

10 Compensation settings

10.1 Backlash compensation

There is usually a certain amount of clearance between the screw and the table, which can affect the positioning accuracy of the machine tool.

The backlash compensation values for the machine tool are measured by the machine tool shop and the corresponding compensation parameters are set in the controller.

The compensation of the backlash is set by the axis parameters (where XX is the physical axis number).

- 2XX0400: Backlash compensation enable
- 2XX0401: Feed backlash (μm)
- 2XX0402: Quick-shift backlash (microns)
- 2XX0403: Backlash compensation step (μm)

■ Note

After modifying the backlash compensation parameters, it is necessary to revert to the reference point for the change to take effect.

Fig. 10- 1 Backlash parameter setting screen

10.2 Pitch error compensation

Usually there is a certain pitch error in the manufacture of the screw, which affects the positioning accuracy of the machine tool.

The compensation of pitch errors is carried out in segments, by first dividing the travel range of the table into segments and then setting the compensation value at each segment node.

■ Parameter settings

Pitch error compensation is set by the axis parameters (XX in the parameter number is the physical axis number).

- Pitch error compensation enabled
  - 2XX0410: Pitch error compensation enable
  - 2XX0412: Pitch error cyclic compensation enable
- Pitch error compensation types
  - 2XX0411: Pitch error compensation type (0 - unidirectional pitch complement; 1 - bidirectional pitch complement)
- Starting point for pitch error compensation
  - 2XX0413: Starting point for pitch error compensation (machine tool coordinate position)
- Pitch error compensation spacing
  - 2XX0414: Pitch error compensation pitch

If the compensation spacing is greater than 0, compensation is carried out in segments in the positive direction from the starting point of compensation.

If the compensation spacing is less than 0, compensation is made in segments in the negative direction from the start of the compensation.

When the compensation spacing is equal to 0, no compensation is applied.

- Pitch error compensation points
  - 2XX0417: Pitch error compensation points
- Pitch error compensation data storage address

The pitch error compensation data is stored in the data sheet and the offset address is set by the following parameters.

- 2XX0415: Pitch error compensation data offset address (direction 1)
- 2XX0416: Pitch error compensation data offset address (direction 2)

■ Note After modifying the pitch error compensation parameters, you need to go return to the reference point to make it effective.

■ Pitch error compensation example 1: unidirectional pitch compensation / positive stroke

Assuming a compensation starting point of 0 for the X-axis, a travel of 550 mm and a compensation pitch of 50 mm, the parameters are set as follows.

- 2000413: The starting point for X-axis pitch error compensation is set to 0.
- 2000414: The X-axis pitch error compensation pitch is set to 50 (compensation in the positive direction from the compensation starting point).
- 2000411: X-axis pitch error compensation type set to 0.
- 2000417: X-axis pitch error compensation points set to 11.
- 2000415: The pitch error compensation data offset address (direction 1) is set to 0, i.e. the starting address of the compensation data in the data table is 4000000.

The measured X-axis pitch error data and compensation settings are listed as follows:

Direction of movement	Command position (mm)	Actual position (mm)	Compensation table settings (micron)
Positive direction	0	0	4000000 =0
	50	49.996	4000001 = 4
	100	100.003	4000002 = -3
	150	150.005	4000003 = -5
	200	200.015	4000004 = -15
	250	249.998	4000005 = 2
	300	300.007	4000006 = -7
	350	350.012	4000007 = -12
	400	400.025	4000008 = -25
	450	450.10	4000009 = -10
	500	499.991	4000010 = 9
	550	549.985	4000011 = 15

■ Pitch error compensation example 2: unidirectional pitch compensation / negative travel

Assuming a compensation starting point of 0 for the X-axis, a travel of -550 mm and a compensation pitch of 50 mm, the parameters are set as follows.

- 2000413: The starting point for X-axis pitch error compensation is set to 0.
- 2000414: The X-axis pitch error compensation pitch is set to -50 (compensation in the negative direction from the starting point of compensation).
- 2000411: X-axis pitch error compensation type set to 0.
- 2000417: pitch error compensation points set to 11.
- 2000415: The pitch error compensation data offset address (direction 1) is set to 0, i.e. the starting address of the compensation data in the data table is 4000000.

The measured X-axis pitch error data and compensation settings are listed as follows:

Direction of movement	Command position (mm)	Actual position (mm)	Compensation table settings (micron)
Negative	0	0	4000000 = 0
	-50	-49.996	4000001= -4
	-100	-100.003	4000002 = 3
	-150	-150.005	4000003 = 5
	-200	-200.015	4000004 = 15
	-250	-249.998	4000005 = -2
	-300	-300.007	4000006 = 7
	-350	-350.012	4000007 = 12
	-400	-400.025	4000008 = 25
	-450	-450.10	4000009 = 10
	-500	-499.991	4000010 = -9
	-550	-549.985	4000011 = -15

■ Pitch error compensation example 3: Bi-directional pitch compensation / positive stroke

Assuming that the starting point for compensation in the X-axis is reference point 0, the travel is 550 mm and the compensation pitch is 50 mm, the parameters are set as follows.

- 2000413: The starting point for X-axis pitch error compensation is set to 0.
- 2000414: X-axis pitch error compensation pitch set to 50.
- 2000411: X-axis pitch error compensation type set to 1.
- 2000417: The number of pitch error compensation points is set to 11.
- 2000415: the direction 1 compensation data offset address is set to 0.
- 2000416: The 2nd direction compensation data offset address is set to 100.

The measured X-axis pitch error data and compensation settings are listed as follows:

Direction of movement	Command position	Physical position	Compensation table setting (micron)
Positive	0	0	4000000 = 0
	50	49.996	4000001 = 4
	100	100.003	4000002 = -3
	150	150.005	4000003 = -5
	200	200.015	4000004 = -15
	250	249.998	4000005 = 2
	300	300.007	4000006 = -7
	350	350.012	4000007 = -12
	400	400.025	4000008 = -25
	450	450.10	4000009 = -10
	500	499.991	4000010= 9
	550	549.985	4000011= 15
Negative	0	0.002	4000100 = -2
	50	50.004	4000101 = -4
	100	99.995	4000102 = 5
	150	149.992	4000103 = 8
	200	200.006	4000104 = -6
	250	250.010	4000105= -10
	300	300.003	4000106 = -3
	350	349.998	4000107 = 2
	400	399.993	4000108 = 7
	450	450.005	4000109 = -5
	500	500.012	4000110 = -12
	550	549.985	4000111 = 15

■ Pitch error compensation example 4: Bi-directional pitch compensation / negative travel

Assuming a compensation starting point of reference point 0 for the X-axis and a travel of -550 mm, with measurements taken at 50 mm intervals, the parameters are set as follows.

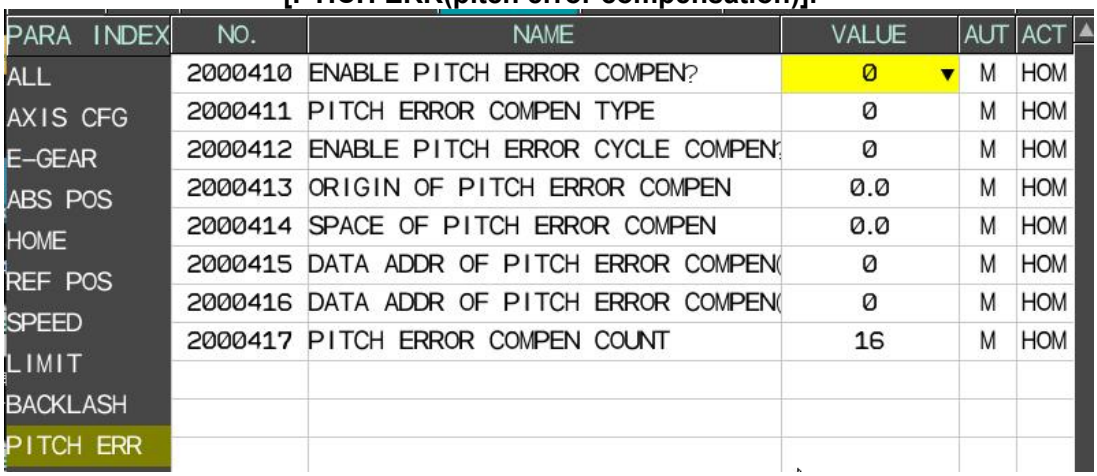
- 2000413: The starting point for X-axis pitch error compensation is set to 0.
- 2000414: X-axis pitch error compensation pitch set to -50.
- 2000411: X-axis pitch error compensation type set to 1.
- 2000417: pitch error compensation points set to 11.
- 2000415: the direction 1 compensation data offset address is set to 0.
- 2000416: The 2nd direction compensation data offset address is set to 100.

The measured X-axis pitch error data and compensation settings are listed as follows:

Direction of movement	Command position	Physical position	Compensation table setting (microns)
Negative	0	0	4000000 = 0
	-50	-49.996	4000001 = -4
	-100	-100.003	4000002 = 3
	-150	-150.005	4000003 = 5
	-200	-200.015	4000004 = 15
	-250	-249.998	4000005 = -2
	-300	-300.007	4000006 = 7
	-350	-350.012	4000007 = 12
	-400	-400.025	4000008 = 25
	-450	-450.10	4000009 = 10
	-500	-499.991	4000010= -9
	-550	-549.985	4000011= -15
Positive	0	0.002	4000100 = -2
	-50	-50.004	4000101 = 4
	-100	-99.995	4000102 = -5
	-150	-149.992	4000103 = -8
	-200	-200.006	4000104 = 6
	-250	-250.010	4000105= 10
	-300	-300.003	4000106 = 3
	-350	-349.998	4000107 = -2
	-400	-399.993	4000108 = -7
	-450	-450.005	4000109 = 5
	-500	-500.012	4000110 = 12
	-550	-549.985	4000111 = -15

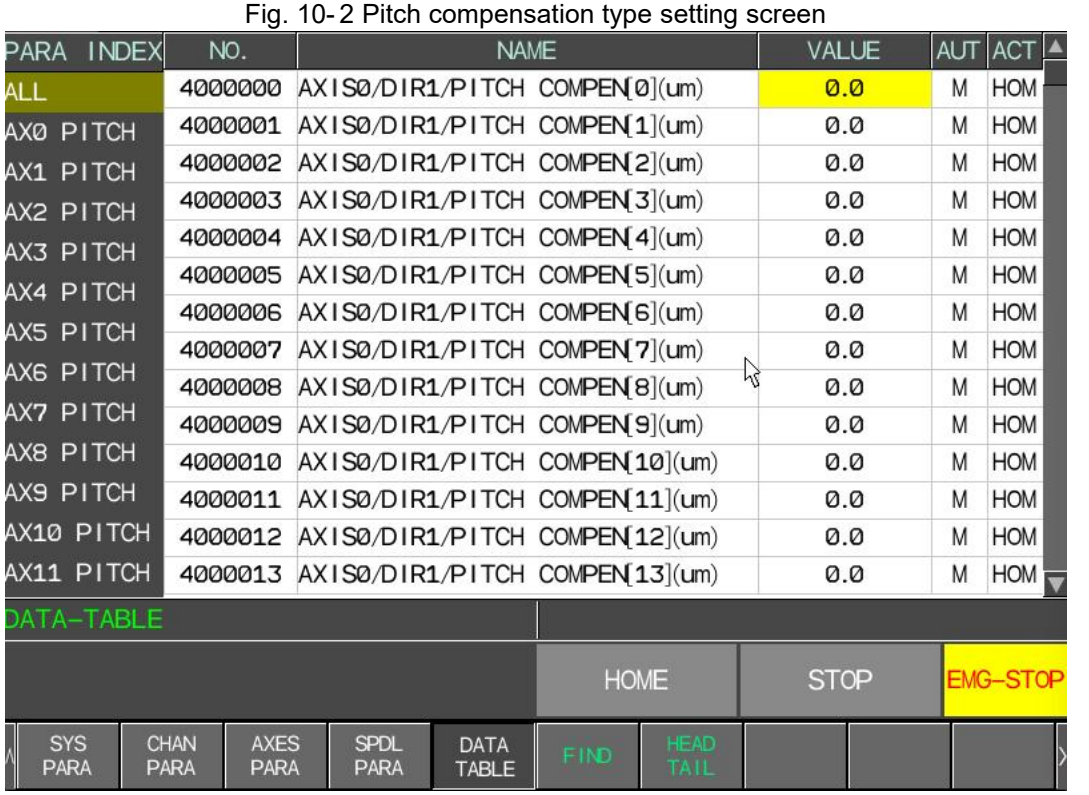
■ Input of pitch compensation values

Once the pitch compensation values have been measured, fill in the controller path: **[Setting]** → **[AXES Parameter]** → **[PTICH ERR(pitch error compensation)].**



PARA INDEX	NO.	NAME	VALUE	AUT	ACT
ALL	2000410	ENABLE PITCH ERROR COMPEN?	0	M	HOM
AXIS CFG	2000411	PITCH ERROR COMPEN TYPE	0	M	HOM
E-GEAR	2000412	ENABLE PITCH ERROR CYCLE COMPEN?	0	M	HOM
ABS POS	2000413	ORIGIN OF PITCH ERROR COMPEN	0.0	M	HOM
HOME	2000414	SPACE OF PITCH ERROR COMPEN	0.0	M	HOM
REF POS	2000415	DATA ADDR OF PITCH ERROR COMPEN(	0	M	HOM
SPEED	2000416	DATA ADDR OF PITCH ERROR COMPEN(	0	M	HOM
LIMIT	2000417	PITCH ERROR COMPEN COUNT	16	M	HOM
BACKLASH					
PITCH ERR					

Fig. 10- 2 Pitch compensation type setting screen



PARA INDEX	NO.	NAME	VALUE	AUT	ACT
ALL	4000000	AXIS0/DIR1/PITCH COMPEN[0](um)	0.0	M	HOM
AX0 PITCH	4000001	AXIS0/DIR1/PITCH COMPEN[1](um)	0.0	M	HOM
AX1 PITCH	4000002	AXIS0/DIR1/PITCH COMPEN[2](um)	0.0	M	HOM
AX2 PITCH	4000003	AXIS0/DIR1/PITCH COMPEN[3](um)	0.0	M	HOM
AX3 PITCH	4000004	AXIS0/DIR1/PITCH COMPEN[4](um)	0.0	M	HOM
AX4 PITCH	4000005	AXIS0/DIR1/PITCH COMPEN[5](um)	0.0	M	HOM
AX5 PITCH	4000006	AXIS0/DIR1/PITCH COMPEN[6](um)	0.0	M	HOM
AX6 PITCH	4000007	AXIS0/DIR1/PITCH COMPEN[7](um)	0.0	M	HOM
AX7 PITCH	4000008	AXIS0/DIR1/PITCH COMPEN[8](um)	0.0	M	HOM
AX8 PITCH	4000009	AXIS0/DIR1/PITCH COMPEN[9](um)	0.0	M	HOM
AX9 PITCH	4000010	AXIS0/DIR1/PITCH COMPEN[10](um)	0.0	M	HOM
AX10 PITCH	4000011	AXIS0/DIR1/PITCH COMPEN[11](um)	0.0	M	HOM
AX11 PITCH	4000012	AXIS0/DIR1/PITCH COMPEN[12](um)	0.0	M	HOM
	4000013	AXIS0/DIR1/PITCH COMPEN[13](um)	0.0	M	HOM

DATA-TABLE

HOME STOP EMG-STOP

SYS PARA | CHAN PARA | AXES PARA | SPDL PARA | DATA TABLE | FIND | HEAD TAIL

Fig. 10- 3 Pitch compensation value input screen

10.3 Straightness compensation

Straightness compensation compensates for straightness errors caused by deformation of the guideway.

■ Parameter settings

Straightness compensation is set by the axis parameters (where XX is the physical axis number).

- 2XX0420: Straightness compensation enable
- 2XX0421: Straightness compensation reference axis number
- 2XX0422: Starting point for straightness compensation
- 2XX0423: Straightness compensation interval
  - Compensation interval greater than 0: compensation in the positive direction from the starting point of compensation.
  - Compensation interval less than 0: compensation in the negative direction from the starting point of compensation.
  - Compensation interval equal to 0: no compensation.
- 2XX0424: Offset address of straightness compensation data in the data table
- 2XX0425: Straightness compensation data points

■ Note

After changing the straightness compensation parameters, it is necessary to go return to the reference point to take effect.

10.4 Reverse jump compensation

When the coordinate axes are reversed, the type of friction changes (kinetic friction → static friction → kinetic friction), resulting in the axes lagging behind the other axes when moving in reverse, and this lag reacts to the reverse jump error.

The reverse jump compensation is set by the axis parameters (where XX is the physical axis number).

- 2XX0405: Reverse jump compensation enable
- 2XX0406: Reverse jump impulse compensation value (μm)
- 2XX0407: Backward jump compensation time (milliseconds)
- 2XX0408: Reverse jump-impulse compensation delay (milliseconds)

Taking the X-axis as an example.

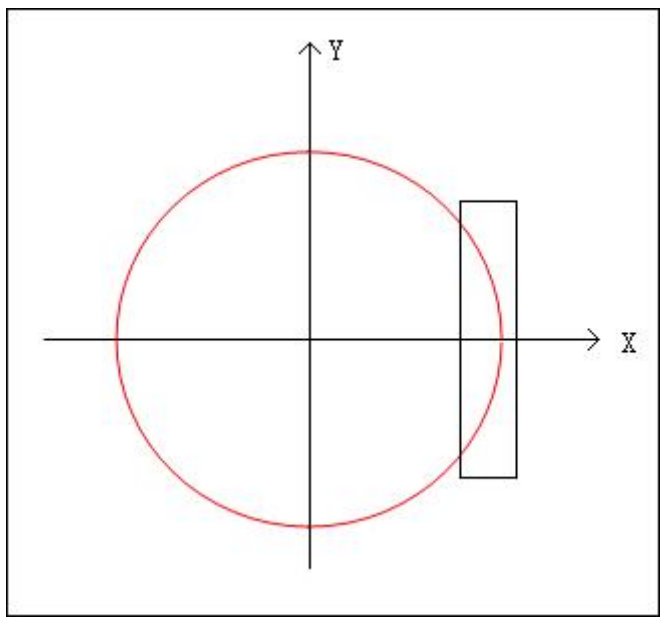


Fig. 10-4 X-axis over quadrant

After the correct setting of the parameters 2000405~2000408, it is generally possible to have a greater improvement in the machining accuracy of the arc over quadrant, as shown in the figure below.

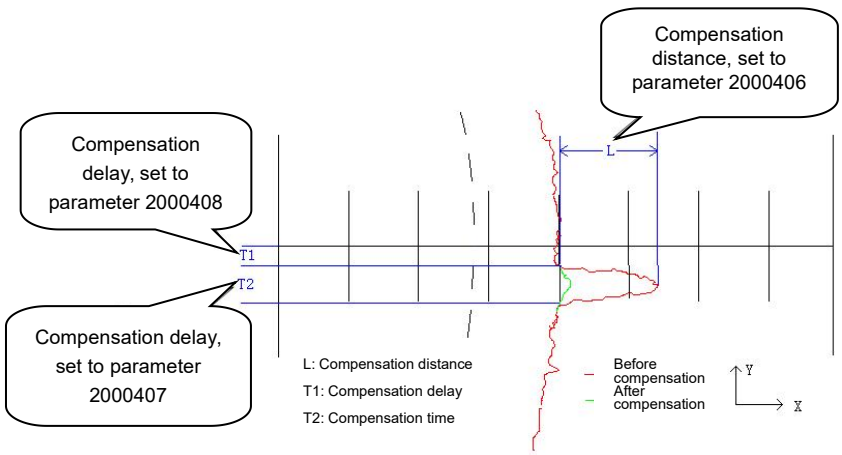


Fig. 10-5 Comparison of reverse jump before and after compensation

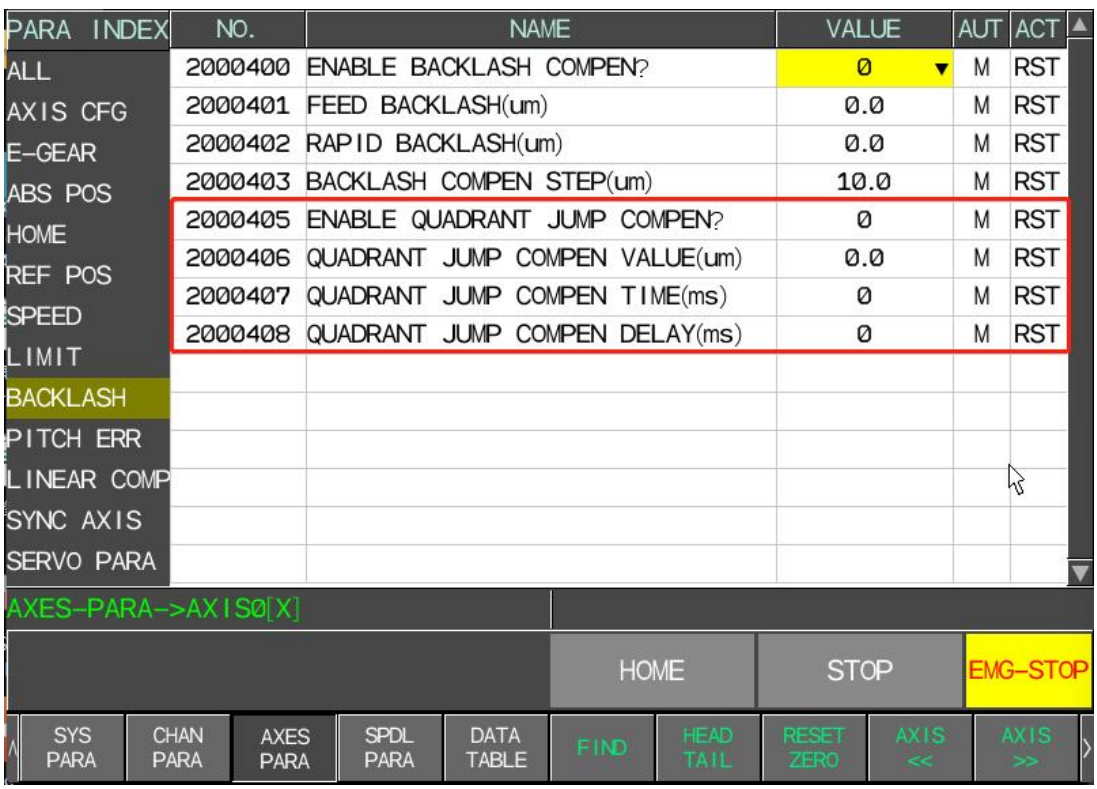


Fig. 10-6 Reverse jump compensation screen

11 Controller exceptions and handling

11.1 Controller alarm indication

The alarm status of the controller can be displayed in the alarm indication area at the top of the controller interface.

The alarm status is divided into the following categories.

- Controller reset

When the reset button is pressed, the emergency stop switch is released or the overtravel is released, the controller requires a certain amount of time to reset and during the reset period the controller cannot be operated.

Reset" is displayed in the alarm indication area during the reset.

- Overtravel (etc. external signal alarm)

When the table is pressed against the upper limit switch, an overtravel alarm appears and "Overtravel" is displayed in the alarm indication area.

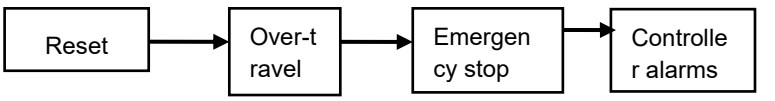
- Emergency stop

When the emergency stop switch is pressed, an emergency stop alarm appears and "Emergency stop" is displayed in the alarm indication area.

- Controller alarms

When a controller alarm other than the above occurs, "controller Alarm" is displayed in the alarm indication area.

When the above alarms occur simultaneously, they are displayed in the alarm indication area in the following order of priority.



11.2 Alarm information search

When an alarm occurs on the controller, the user can view the details of all current controller alarms in the alarm information window.

To access the alarm information window: **[Diagnostics]** → **[Alarm information]**

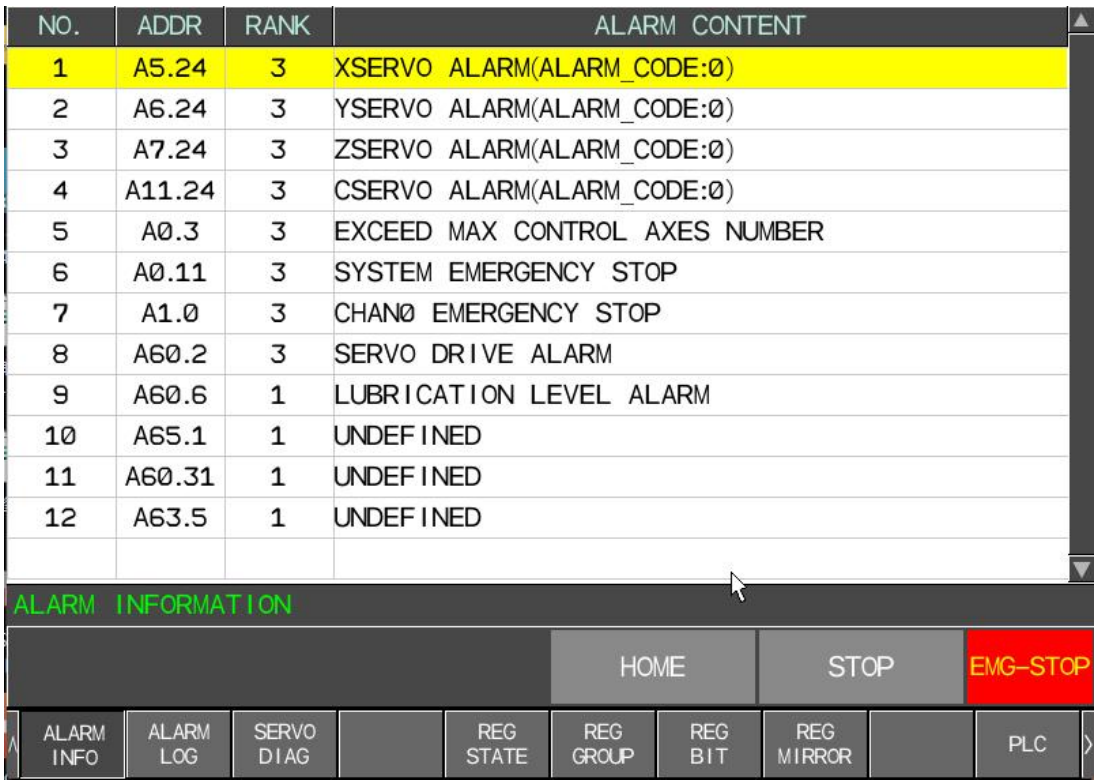


Figure 11- 1 Alarm information window

11.3 Alarm log

The controller can query historical alarms.

To access the alarm log window: [**Diagnostics**] → [**Alarm Log]**

The maximum number of records in the alarm log is set by the controller parameter 0000025.

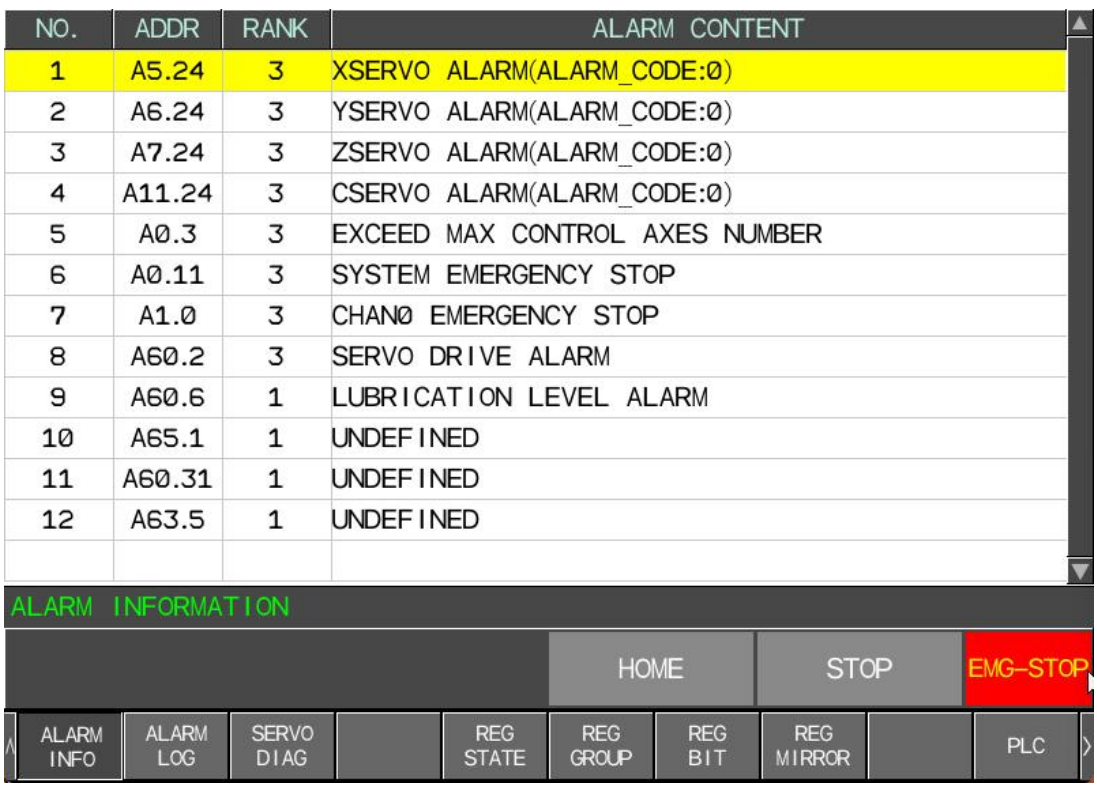


Figure 11-2 Alarm log window

11.4 Controller soft limit alarm handling

The soft limit is the limit of motion for each axis set by the controller parameters. When the movement of an axis is outside this range, a soft limit alarm occurs and all axes slow down and stop.

In the event of a soft limit alarm, programs and MDI commands cannot be run in the automatic mode and can only be operated manually.

To resolve the soft limit alarm, simply switch to manual or incremental mode, move the axis in the safe direction and the controller automatically disengages the alarm after moving out of the soft limit.

The positive and negative soft limits are set by the axis parameters (where XX is the physical axis number).

- 2XX0150: Positive soft limit enable
- 2XX0151: Negative soft limit enable
- 2XX0152: Positive soft limit (mm)
- 2XX0153: Negative soft limit (mm)
- 2XX0154: Soft limit safety distance (mm)
- 2XX0155: 2nd positive soft limit enable
- 2XX0156: 2nd negative soft limit enable
- 2XX0157: 2nd positive soft limit (mm)
- 2XX0158: 2nd negative soft limit (mm)
- 2XX0159: 2nd soft limit safety distance (mm)

11.5 Internal controller alarms and descriptions

The table below shows the internal alarms of the controller, other extended alarms can be found in the machine tool

Alarm points	Alarm messages	Detailed description
A0.0	Serial number error	Please contact the controller manufacturer.
A0.1	Controller not registered	Please contact the controller manufacturer.
A0.2	Controller usage expiry	Please contact the controller manufacturer.
A0.3	Exceeding the maximum number of control axes	
A0.5	Inconsistent controller versions	Please contact the controller manufacturer.
A0.10	Controller reset	
A0.11	Controller emergency stop	
A0.15	PLC has stopped running	
A1.0	Passage emergency stop	
A1.2	Channel reset	
A1.3	Program error	There is a syntax error in the program, see the program alarm message for details.
A1.5	Machine tool not in breakpoint position	
A1.10	Probe not triggered	The G31 program segment reaches the end point without detecting a probe input signal. This alarm can be blocked by setting the channel parameters.
A5.0	Not zeroing	
A5.1	Failure to zeroing	
A5.2	Absolute position initialization failed	
A5.3	Servo parameter upload failed	
A5.4	Failed to download servo parameters	
A5.5	Forward overtravel	
A5.6	Negative overtravel	
A5.7	Positive soft limit	
A5.8	Soft limit in the negative direction	
A5.10	Too large following error	
A5.11	Overspeed	Please contact the controller manufacturer.