Cyclmotion control system
Software Description

Software version No.: Android-A-2024-10-07-001

Cyclmotion (Tianjin) Intelligent Technology Co., LTD

Change Log

Date	Software release	Primary coverage	Remarks
2024.10.9	Android-A-2024-10-07-001	First release	
2024.10.11		Add Appendix 2 mobile phone remote control terminal description Add blow output settings to the knife length measurement Perfect the modbus instructions	

目录

一、 summary

This article mainly introduces the use of the Android version of the control software instructions.

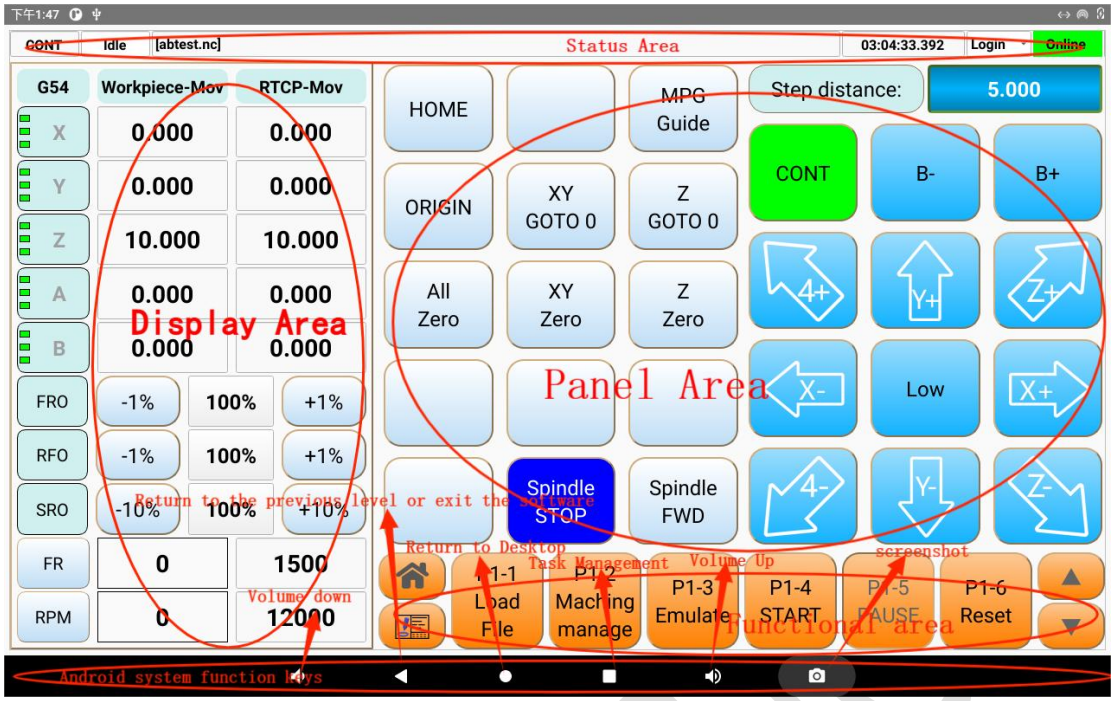
Secco off-line motion control system includes control card, control software and touch all-in-one machine. At present, the products are divided into three axis (basic model), three axis (with RTCP), four axis (basic model), four axis (with RTCP), six axis (with RTCP), suitable for android system.

二、 characteristic

Secco motion control system is a control system with RTCP function, which has the following main features.

- Adopt configuration design, support user deep configuration customization, rapid secondary development of interface, automatic deployment of functional components (all source codes are in / env directory, source code is written by extended macro program);
- Support three axis to six axis and other 18 kinds of machine tool structure forms, including double turn table, single turn single swing, double swing head, etc.;
- Support RTCP (G43.4) and fixed axis processing (G68.2) function;
- Support 3D dynamic processing simulation, knife-path and G code two-directional positioning;
- Support for MODBUS bus IO extension;
- Support the servo spindle, rigid tapping;
- Support custom function shortcut key, general input port can be configured to shortcut function key;
- Support array processing, single segment processing, cycle processing, breakpoint processing, nearby point processing and other processing strategies.
- Support axis mapping;
- Support the teaching function;
- Support for knife library customization.
- Support for absolute value encoder customization.

(一) Introduction to interface layout



Status bar: display the status and alarm information, etc.see details[status bar](#)introduce.

Display area: conduct coordinate display configuration, display coordinates, common parameters, etc.see details[display space](#)introduce.

Panel area: in the parameter settings-system settings-*[Panel configuration](#)*Custom in.

Function key: three pages, can turn pages, in the parameter setting-system setting-*[Function key configuration](#)*Custom in.

Android function keys: Android system function keys. See above.

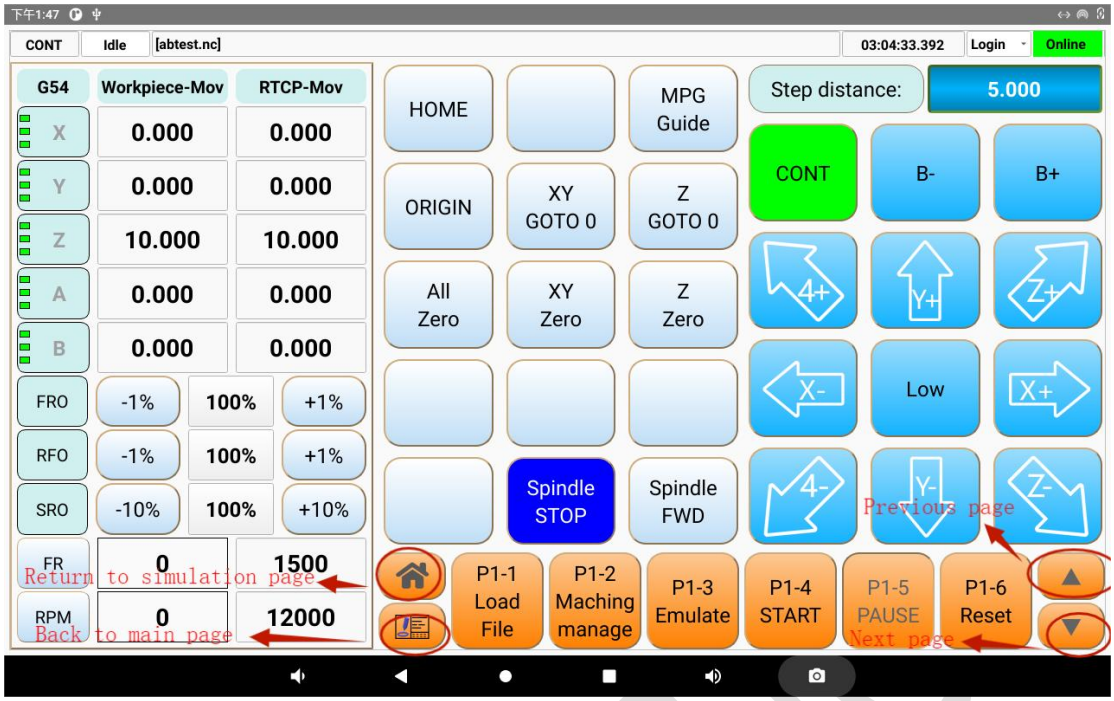
Note: When the first click of some functions enters into the subpage, there will be a little wait (loading time), the waiting time of some pages is slightly obvious (about 2 seconds), and it will be cut in seconds after the second time.

Heres the function of the home page and the simulation page icon.

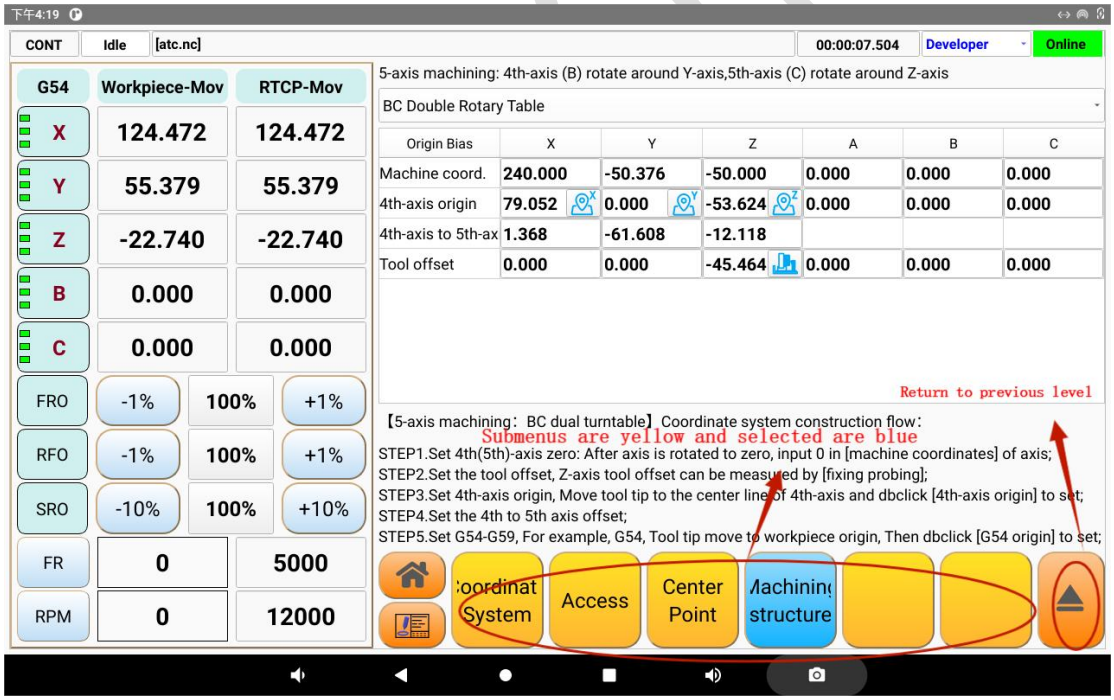
1. Home page icon

Under any page, click the icon and directly jump back to the main page. 

The following figure is for the main page, and the function key of the main page is orange-yellow.



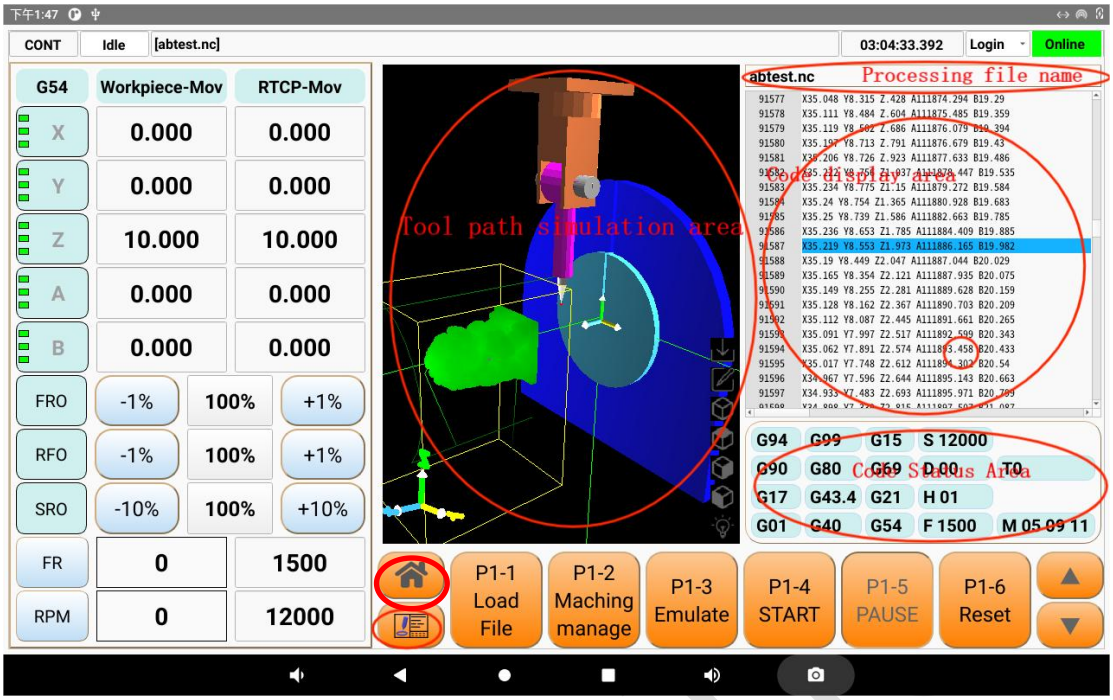
The following figure shows the example of the sub-menu interface. The sub-menu function key is yellow, and the right arrow shows the return to the previous menu.



2. Simulation page icon

Under any page, click the icon and jump directly to the simulation page. The following figure is the simulation page.

Due to limited resources, only a certain range of blades is displayed in large files. Range setting parameters are defined in parameter setting- -system setting- -knife window- -knife display capacity. After the general assembly led to a lag.

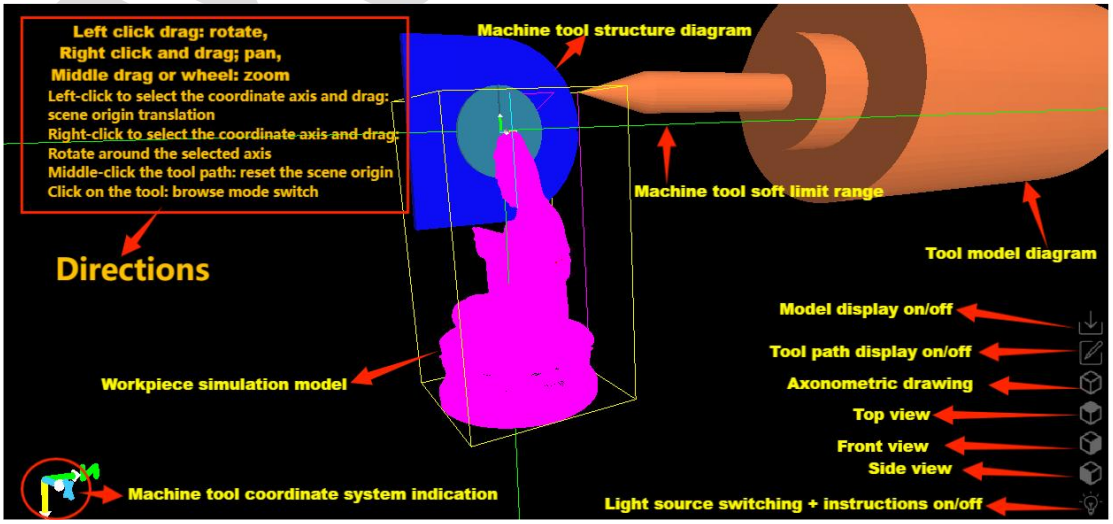


1) The knife road simulation area

Basic operation description of the simulation area:

- Rotation: Press the screen and drag around the center of rotation.
- Translation: Click the lower left coordinate axis to turn red and drag.
- Zoom: drag with two fingers.
- Click the tool: the tool turns red, the tip point is placed in the display center, which cannot be translated. Click again to cancel.
- Click the knife path: set the rotation center, but you can only choose in the workpiece or knife road. If there is no workpiece or knife road, you can simulate it first.

The meaning of the right icon of the simulation area is shown in the figure below.



2) Code display area

In file processing, the current processing line code shows blue and tracks in real time.

Click on the code line, and the corresponding knife path turns to red in the simulation area.

三、 status bar

The status bar indicates, from left to right:

1. **Control mode (automatic / continuous / inch / handwheel)**
   - When processed online, it will be automatically locked in the "automatic" mode;
   - Continuous / inch / hand wheel, select and control through the control area.
   - Continuous: press the movement direction key, will continue to move.
   - Move: move at the given pace.
   - Hand wheel: It is controlled by the hand wheel. The Host control is not valid.

Note: When running online,

- When the handwheel shaft is selected off, the handwheel control is invalid and can only be switched in continuous / inch and controlled by the main engine.
- When the handwheel shaft switches from off to another gear, or between the other gear, it automatically switches to the handwheel state, controlled by the handwheel.
- When the hand wheel shaft is not in off, the continuous / hand wheel / inch can be switched.

2. **Idle / reset / equipment is busy:**
   - Display is idle when not processing, and display equipment is busy when processing;
   - The reset will flicker, need to remove, press the function key "stop" or "reset" to eliminate the flicker;

3. **Processing documents / alarm information:**
   - Normal state shows the processing file name + current line number + current line code, alarm when display alarm information;
   - When the alarm message flashes, click to eliminate it.

4. **Processing time: refers to the processing time of a single file, without used.**

5. **Administrator permissions:**
   - Log in / administrator / developer, click to switch.
   - Administrators and developers can set up passwords.
   - Administrator authority: in the parameter setting-axis parameter setting, processing information zero, knife library configuration, machine tool structure selection, etc., need the administrator above authority;

- Developer permission: highest permission, can be all parameter setting, configuration of display area axis, panel area configuration, function key configuration, shortcut key definition, extension function definition, empty log, etc.

6. Simulation / Online

- After the online success, display "online" green, otherwise display "simulation" red;
- When multiple hosts are connected to the same control card, the first successful connection is the master controller, and the rest are monitoring machines, and the "monitoring" will be displayed here.

四、 display space

1. Axis configuration and display

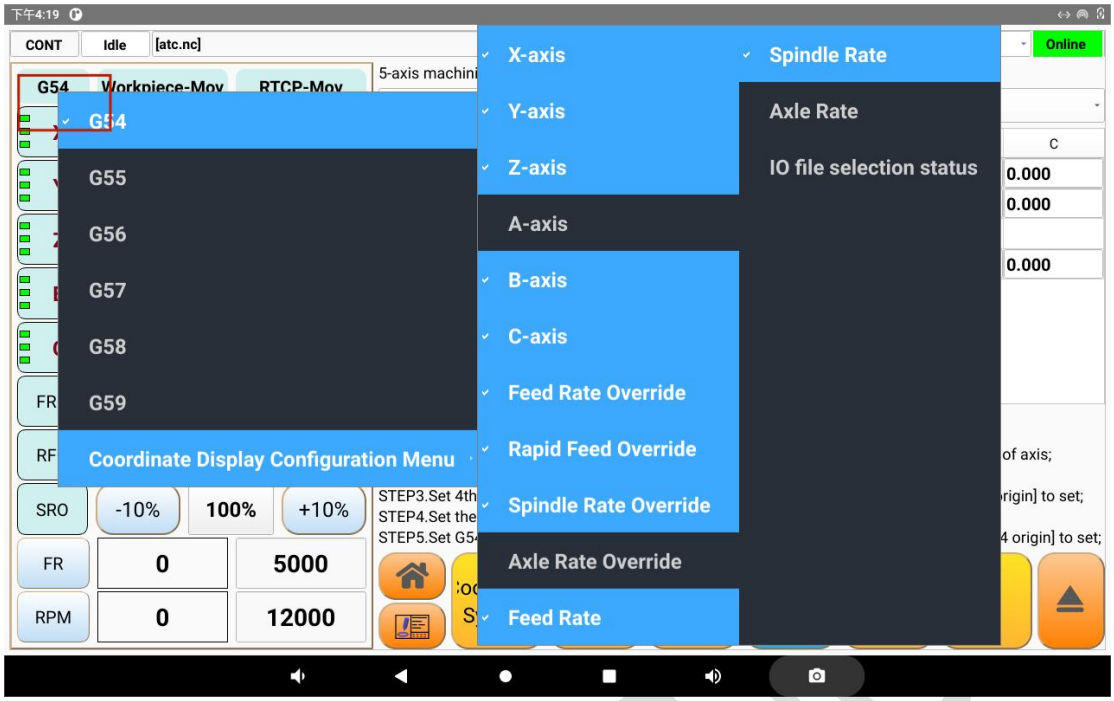
Click the G54 red box area of the title bar in the figure below. Different optional functions under different permissions. The blue band tick is displayed, and the black bottom is not displayed.

- Under login or administrator rights, you can only select the G54-G59 coordinate system.
- Under the developer permission, in addition to the G54-G59 coordinate system, coordinate display configuration, increase or decrease the number of display axes and axle related display.
- Four axis to six axis can check A / B / C axis as required;
- If equipped with an axle, it can increase the axle ratio, axle speed and other display;
- IO file selection status: Add the IO selection file bar to display the current IO selection file (withIO Select a file to startSupporting use).

X, Y, Z, A, B, and C Different colors indicate different states:
Gray: indicates undefined origin signal.
Dark red: after defining the origin signal.
Green: during the axis movement.
Blue: Zero-finding is complete.

The three red indicators on the left of each axis indicate:

- ✧ Drive alarm
- ✧ Hard limit alarm
- ✧ Soft limit alarm



For example, the display interface configured as the six-axis is as follows.

2. Work piece coordinate movement / location switch

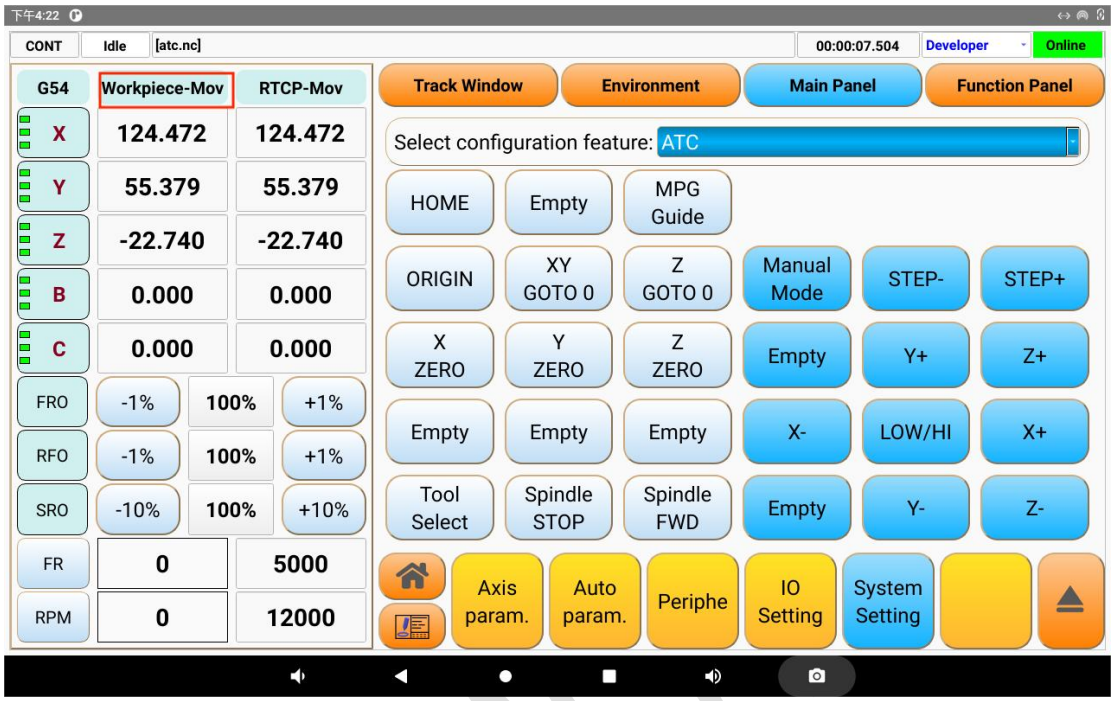
Click the red box area shown in the diagram to switch between the artifact coordinate movement / positioning.

The reference coordinate system is the currently selected artifact coordinate system (G54-G59).

- Work piece coordinates-Move: Move the tool to the specified coordinates. The movement speed is specified by the parameter setting- - -axis

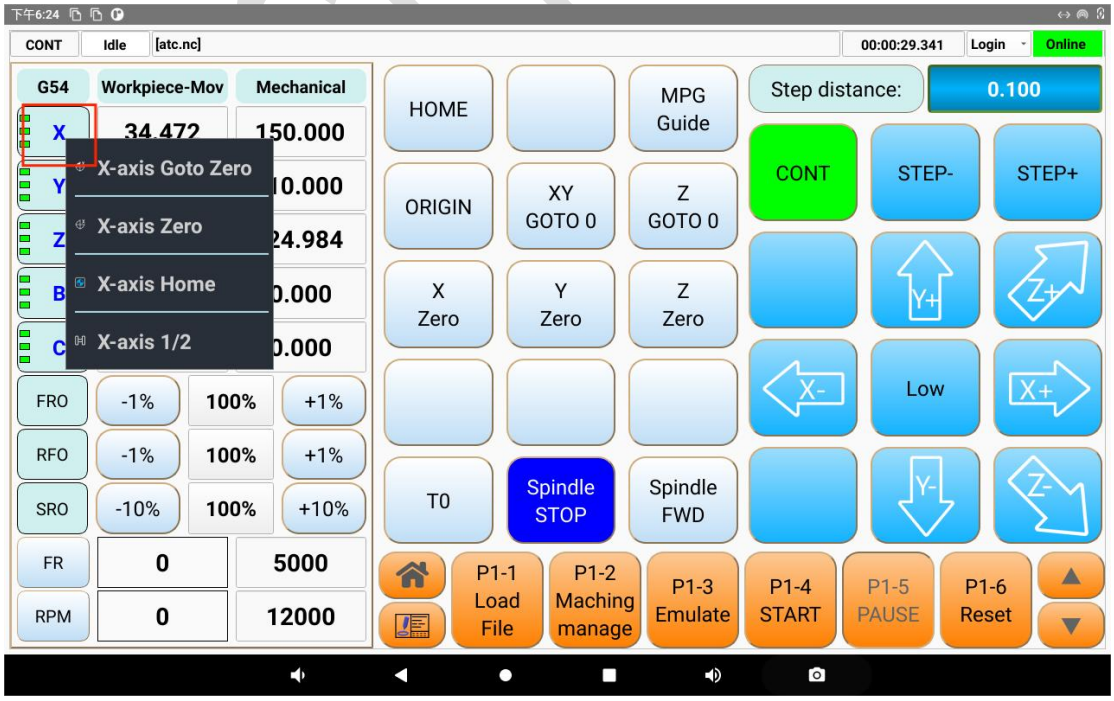
parameter-hand- -G0 speed.

- Workpiece coordinates-Positioning: Set the current artifact coordinates of the tool.



3. Each axis is returned to zero / zero / zero / point

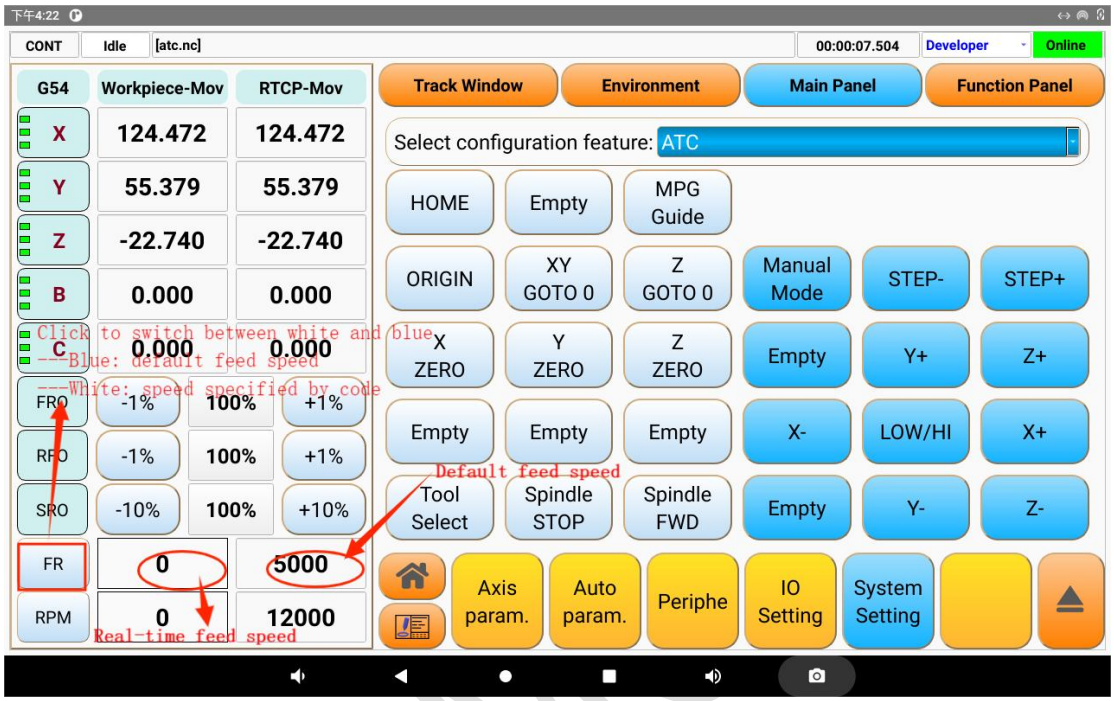
Click in the red box area of each axis diagram, and the diagram menu will pop up, which can be easily operated in zero / zero / zero / point of each axis.



4. Speed switch to default / specify

Similar to the feed speed, spindle speed, and axle speed (if configured), click the illustrated red box area to switch to light blue / dark blue.

- Dark blue: press the default speed / rotation speed;
- Light blue: Specify speed / speed by code.



五、 Panel area

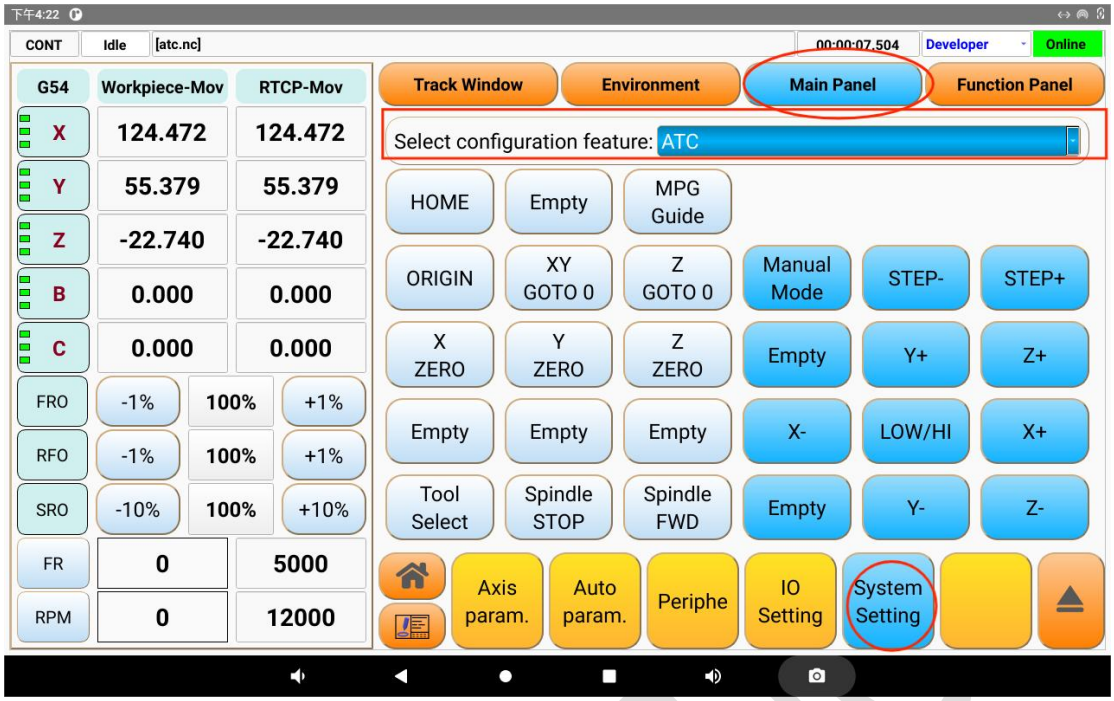
1. Panel configuration method

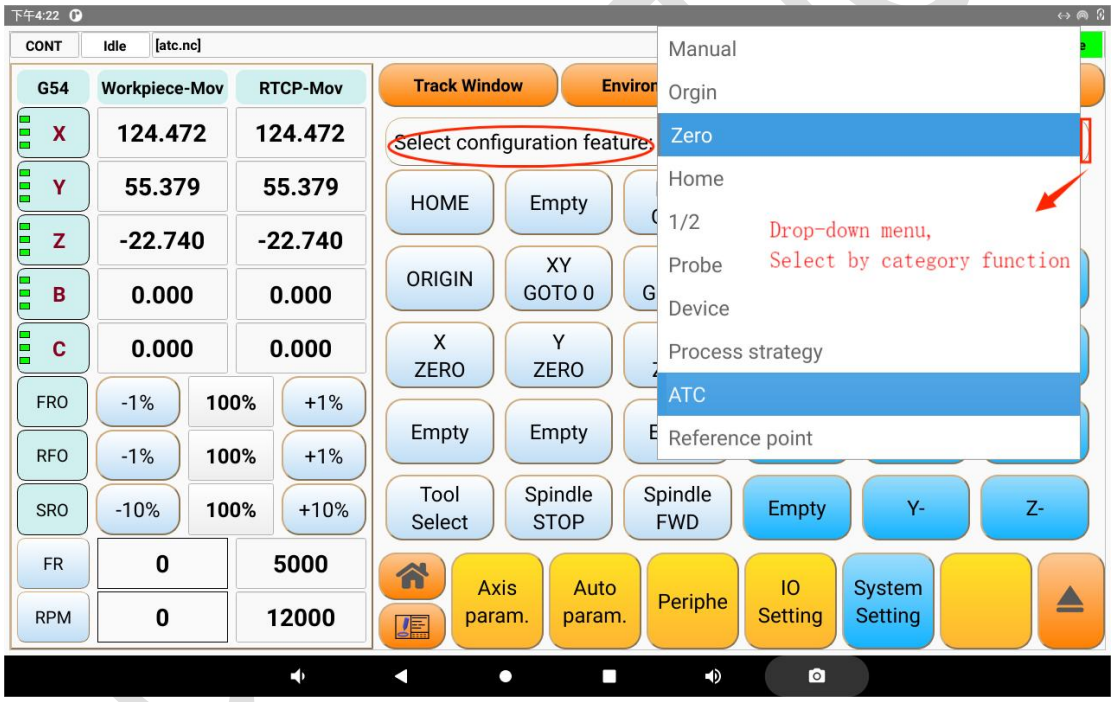
All buttons of the panel buttons can be configured in parameter setting-system setting-panel configuration. Personalized panels can be customized according to individual habits and preferences.

All options are related to the number of axis configurations. The own is a quad example.

Configuration steps:

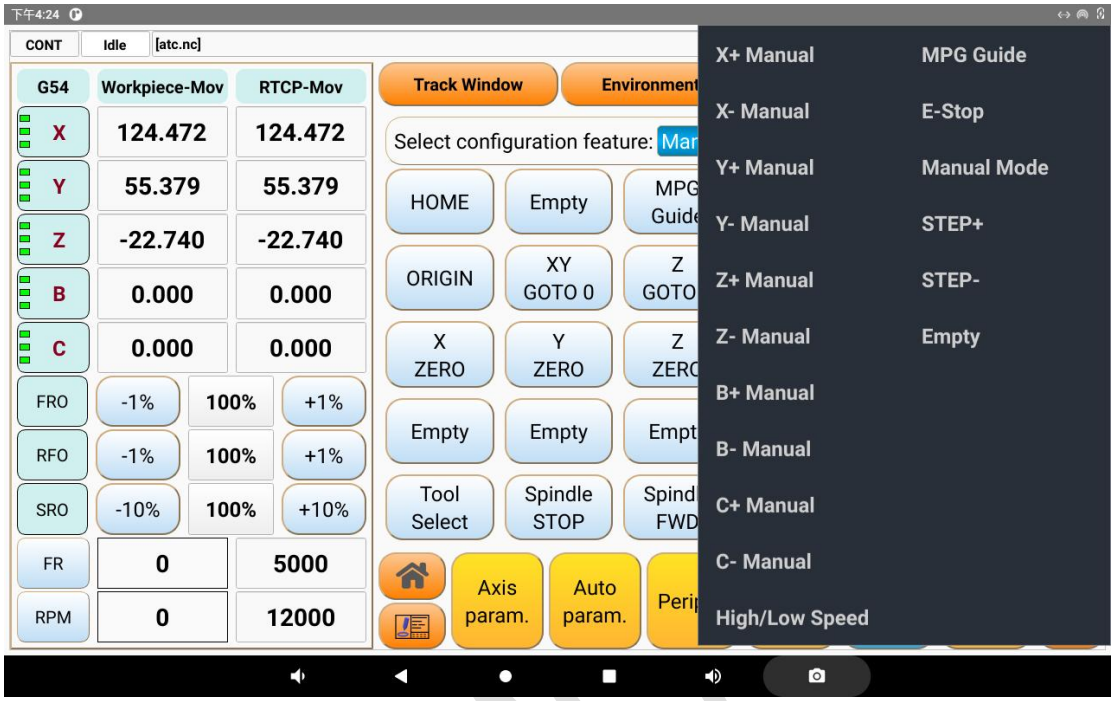
- Click the Select Configuration function drop-down menu to select the required function classification.
- Click on the button to be configured and select on the pop-up menu.


CONT
Idle
[atc.nc]
00:00:07.504
Developer
Online
G54
Workpiece-Mov
RTCP-Mov
X
124.472
124.472
Y
55.379
55.379
Z
-22.740
-22.740
B
0.000
0.000
C
0.000
0.000
FRO
-1%
100%
+1%
RFO
-1%
100%
+1%
SRO
-10%
100%
+10%
FR
0
5000
RPM
0
12000
Track Window
Environment
Main Panel
Function Panel
Select configuration feature:
ATC
HOME
Empty
MPG Guide
ORIGIN
XY GOTO 0
Z GOTO 0
Manual Mode
STEP-
STEP+
X ZERO
Y ZERO
Z ZERO
Empty
Y+
Z+
Empty
Empty
Empty
X-
LOW/HI
X+
Tool Select
Spindle STOP
Spindle FWD
Empty
Y-
Z-
Axis param.
Auto param.
Periphe
IO Setting
System Setting

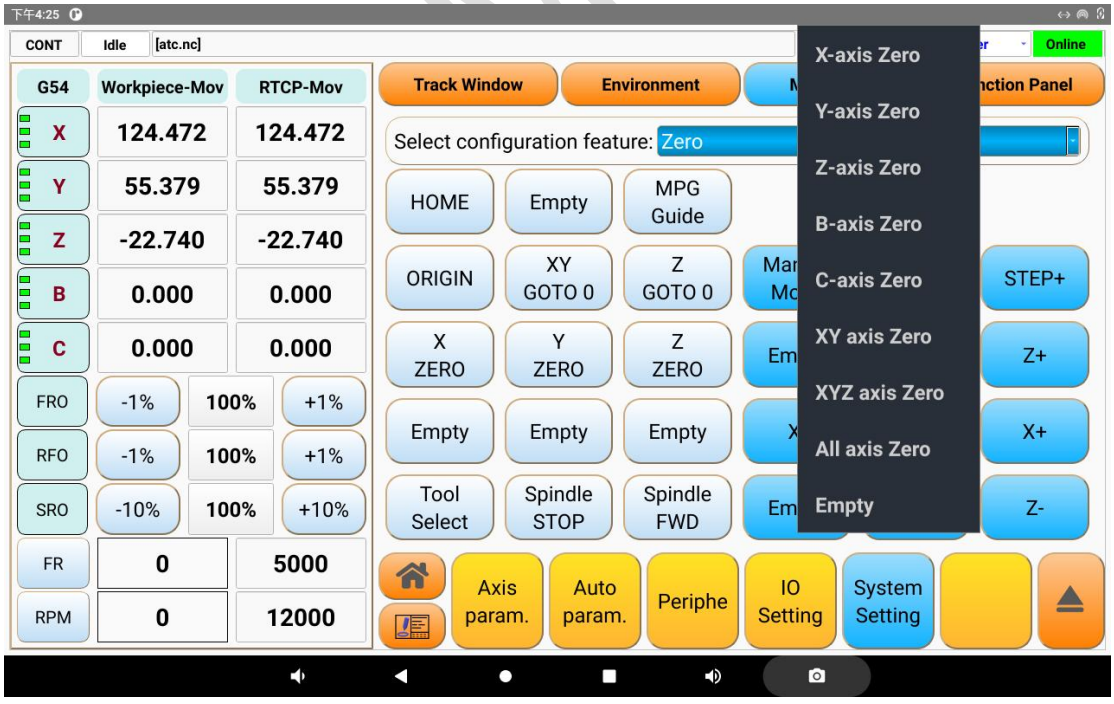

CONT
Idle
[atc.nc]
G54
Workpiece-Mov
RTCP-Mov
X
124.472
124.472
Y
55.379
55.379
Z
-22.740
-22.740
B
0.000
0.000
C
0.000
0.000
FRO
-1%
100%
+1%
RFO
-1%
100%
+1%
SRO
-10%
100%
+10%
FR
0
5000
RPM
0
12000
Track Window
Select configuration feature:
HOME
Empty
ORIGIN
XY GOTO 0
X ZERO
Y ZERO
Empty
Empty
Tool Select
Spindle STOP
Spindle FWD
Empty
Y-
Z-
Axis param.
Auto param.
Periphe
IO Setting
System Setting
Manual
Orgin
Zero
Home
1/2
Probe
Device
Process strategy
ATC
Reference point
Drop-down menu,
Select by category function

2. The panel is optional for the introduction

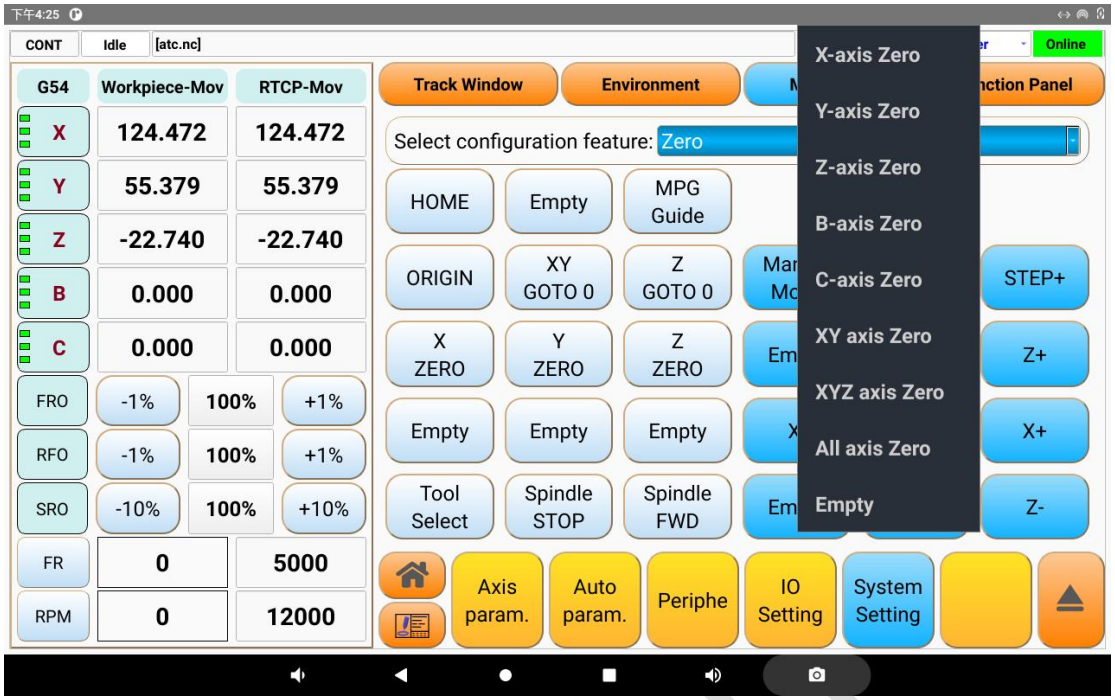
1) Manual control options:



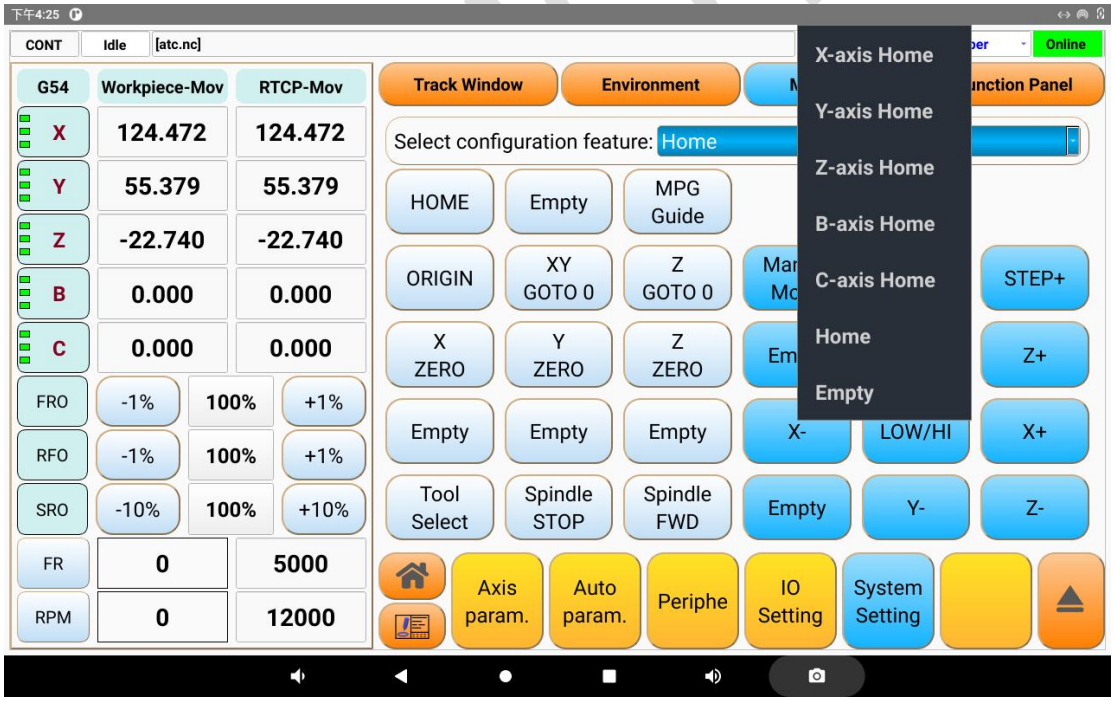
2) Return back to zero option:



3) Zero clearance option:



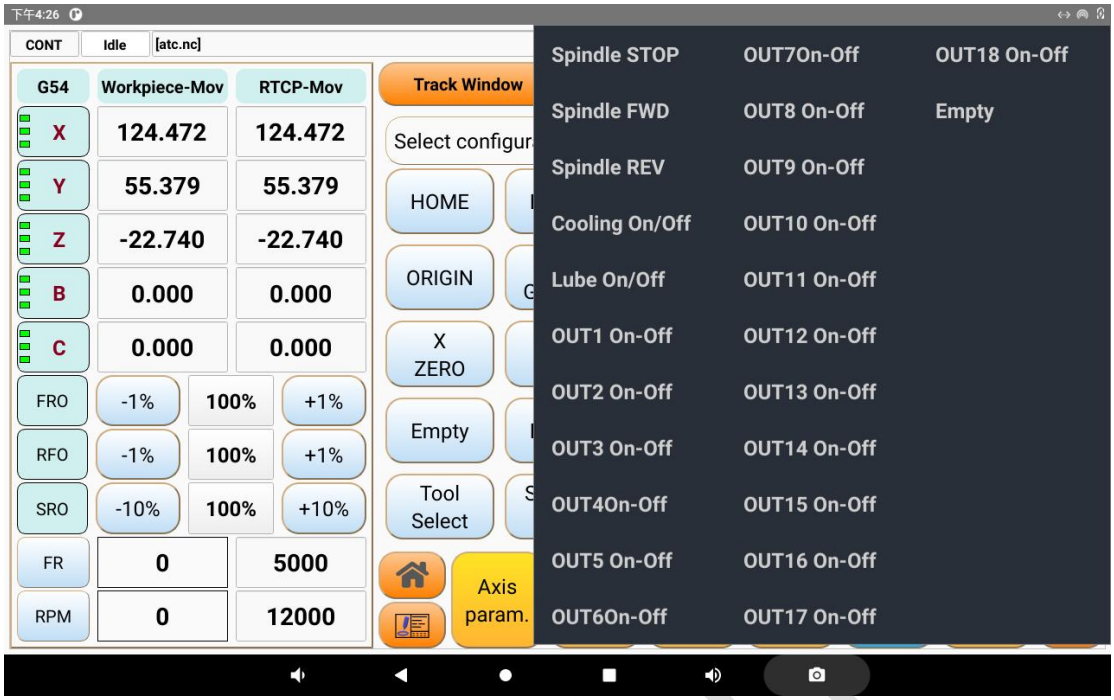
4) Mechanical return to zero is optional:



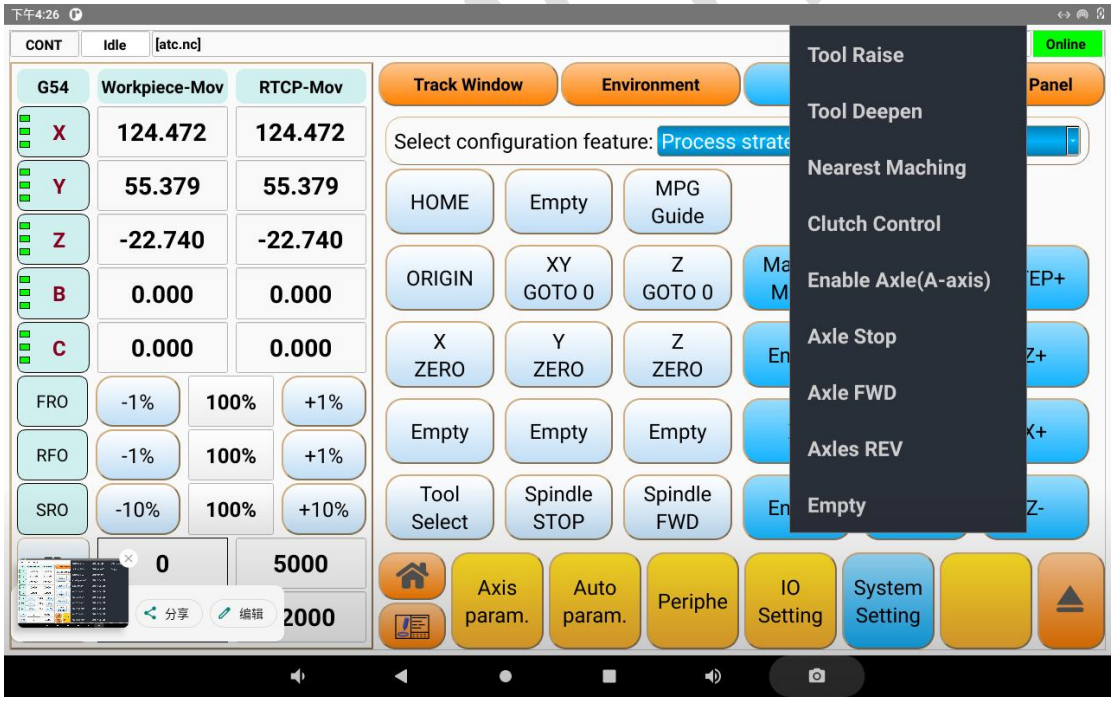
5) The option in the score:

6) For knife / probe options

7) Peripheral controls are optional

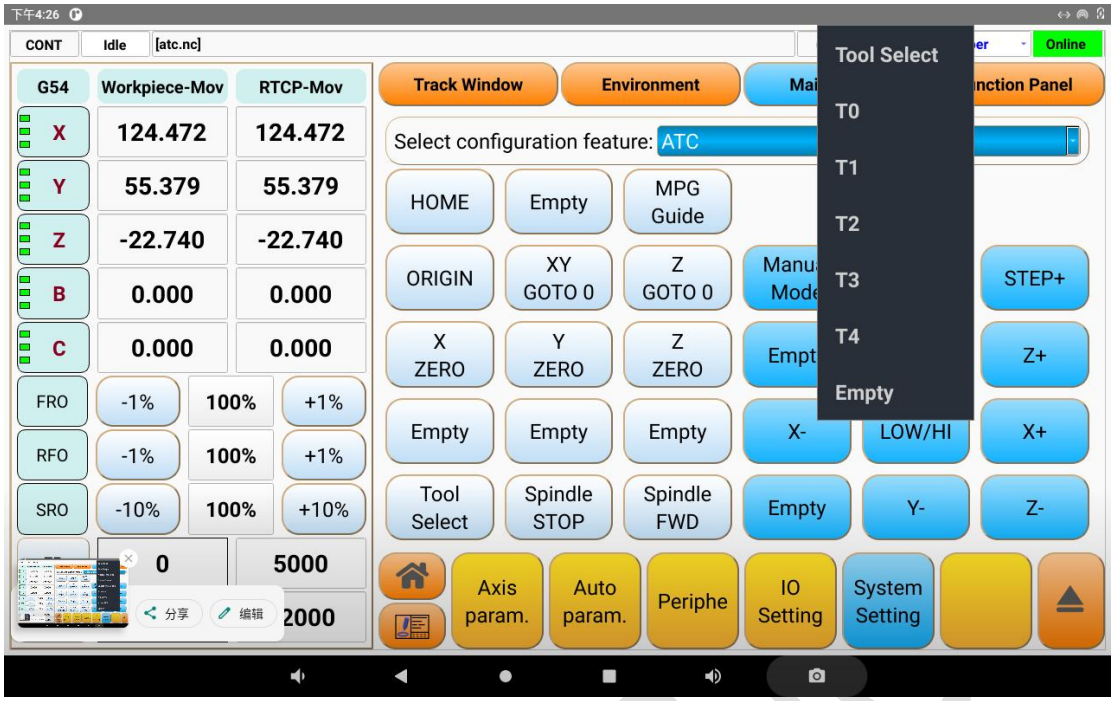


8) The process policy is optional



9) Knife library options

You need to configure the tool library (select the tool library type in the tool management-tool library configuration) before the following tool option.



10) Back to the reference point is optional

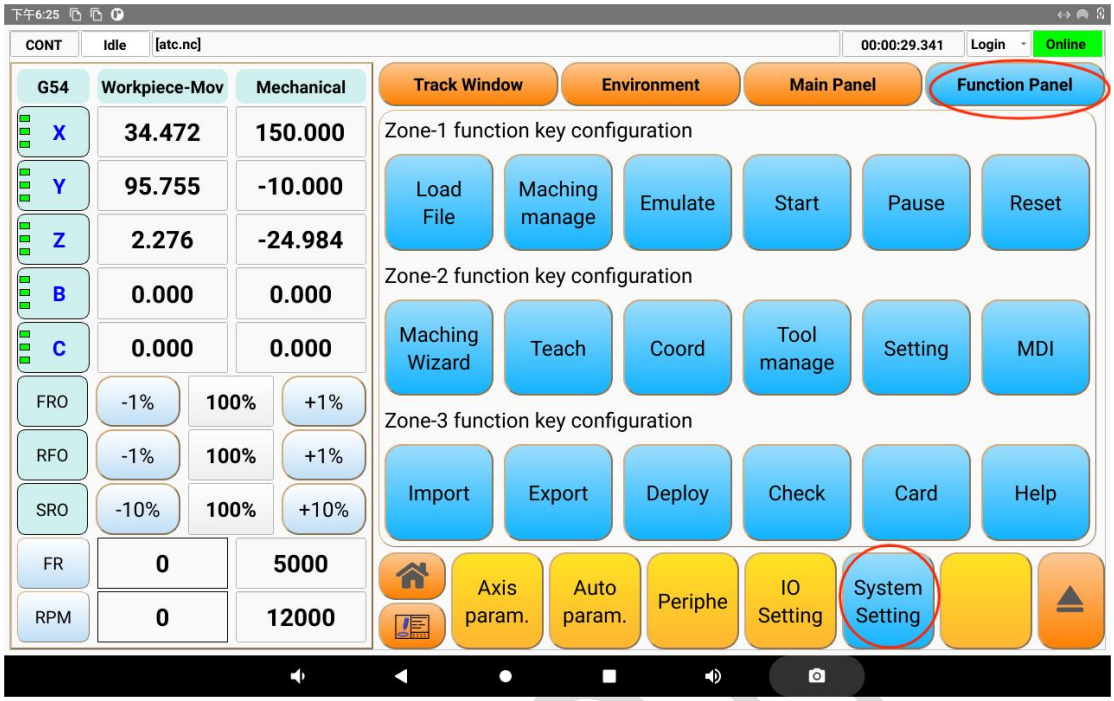
六、 function key

(一) Function key configuration method

Only the developer permissions can be configured.

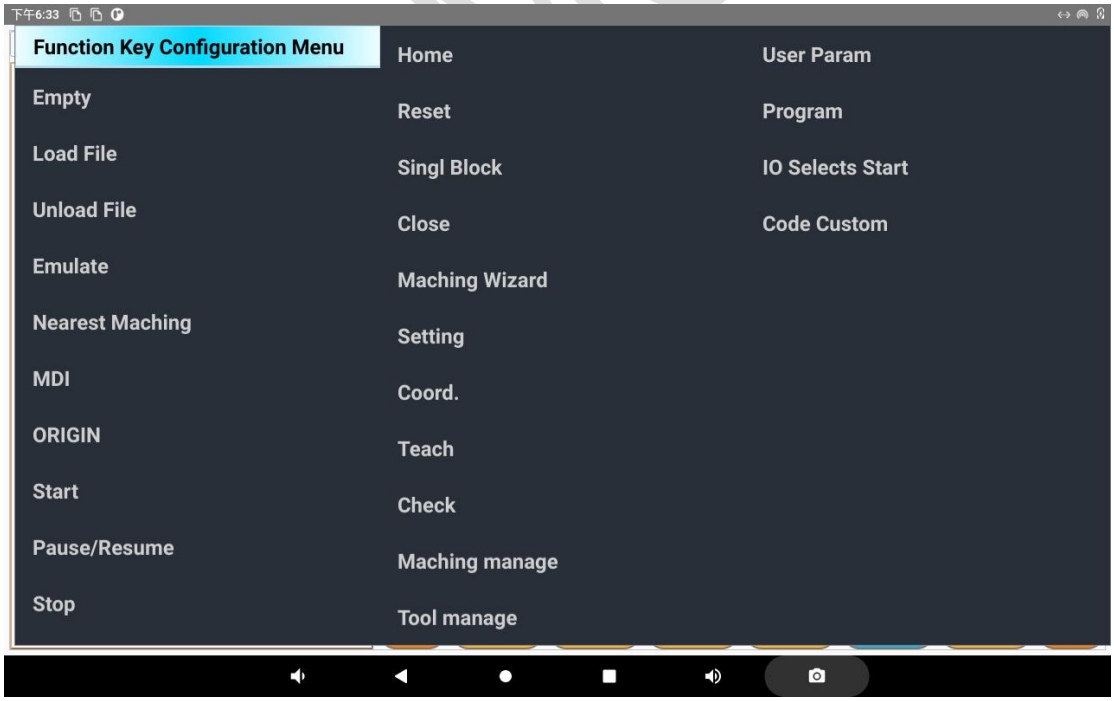
Function keys are 3 pages, which can be turned up and down, and can be customized in the parameter setting-System Settings-function key configuration.

Click parameter setting to enter the following interface, and select System Settings-function key configuration.



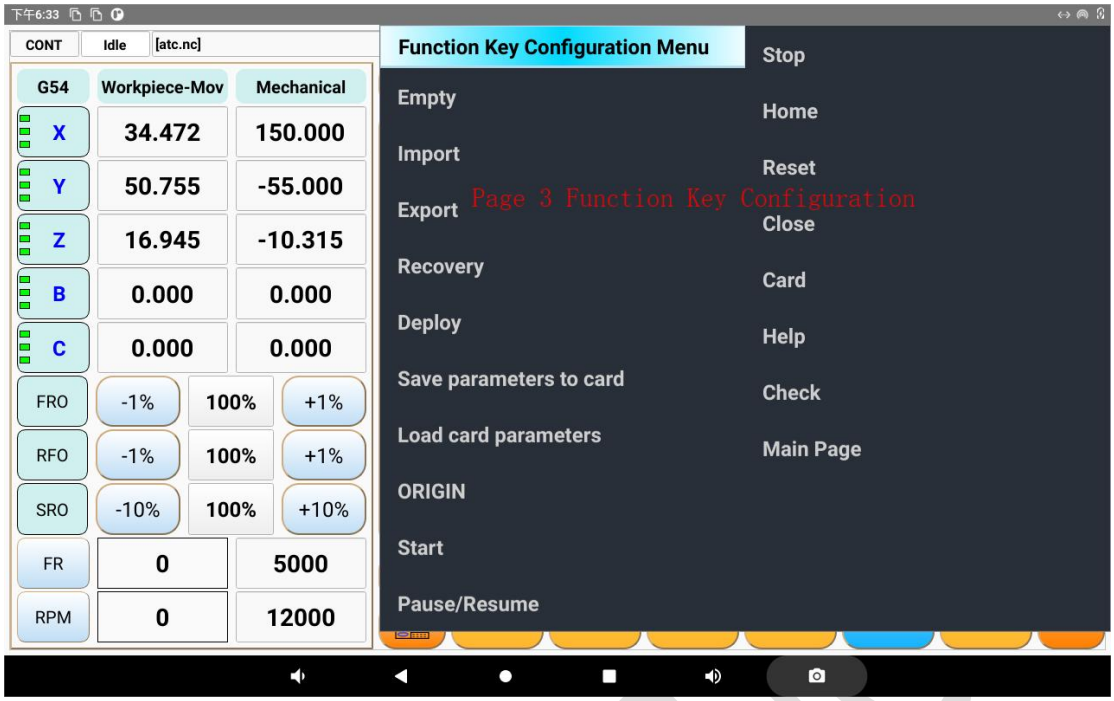
Click the button to be configured, pop up the following picture menu, and select the required function. The following figure shows the function key menu on pages 1 and 2.

See (2) to (15) on pages 1 and 2.



Page 3 The customization function is different from page 1 and 2, as shown in the figure below.

See the functions for details on page 3(sixteen)To (22).



The following functions are introduced.

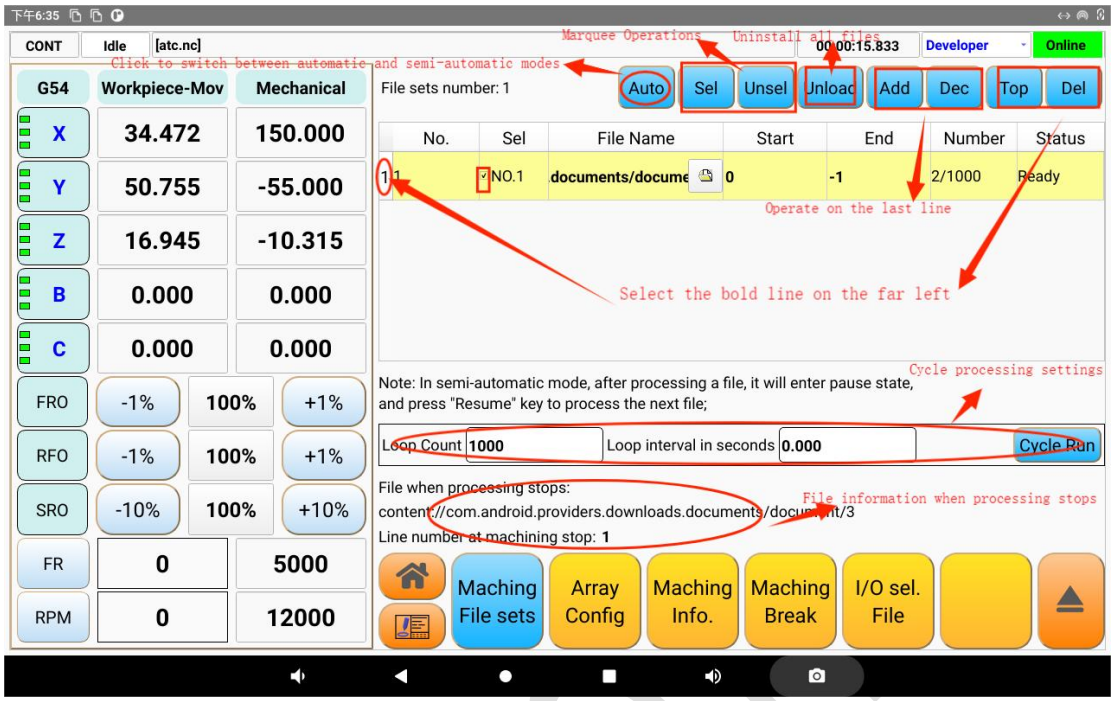
(二) Loading files

Click Load file, jump out of the file selection page, you can select a single file to load.

(三) Processing strategy

Click the processing strategy to enter the next level page. Including file set, array configuration, processing information, processing breakpoint, IO selection file function.

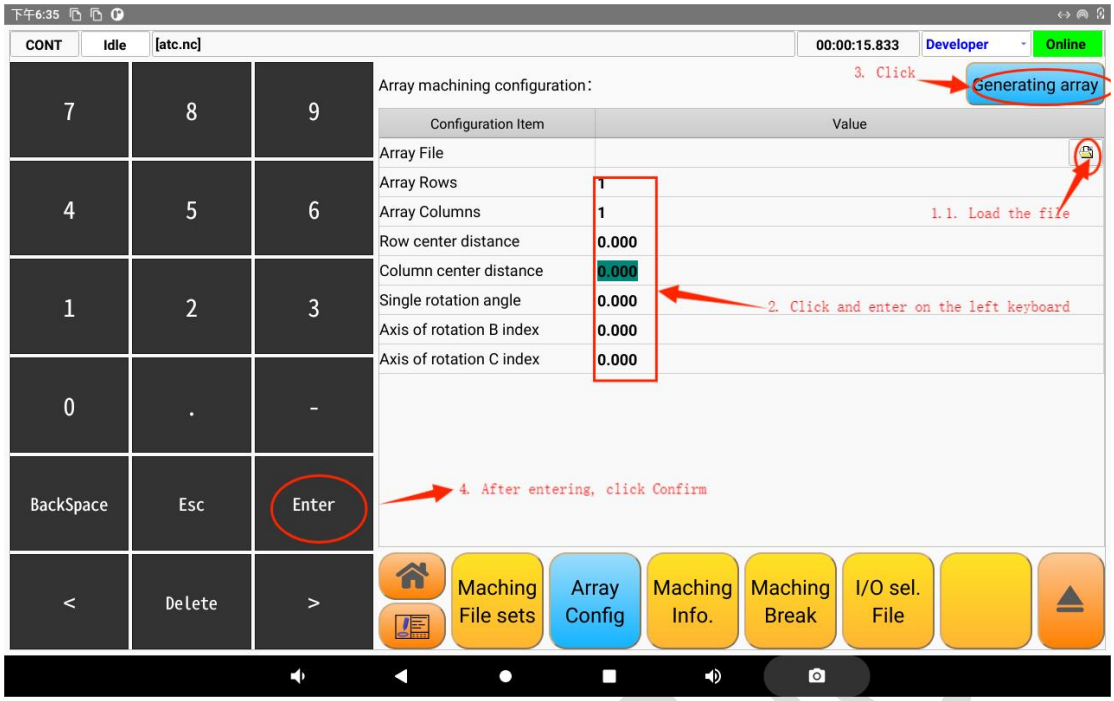
1. file set



- Automatic mode / semi-automatic mode: click to switch.
  - Automatic mode: according to the selected processing documents;
  - Semi-automatic mode: process the selected processing files. After each processing file, it will enter a pause state, wait for the "restore key" command, and then continue to process the next file.
- All / all: a column box for the diagram. The selection box is only valid for the processing selection.
- Uninstall files: Clear all files.
- Add: Add a blank row to select a new file.
- Reduce: Delete the last row of file rows.
- Top: Top the number in the leftmost column (note that instead of the selection box, bold the number in the leftmost column).
- Delete: Delete the bold row for the leftmost column (note that it is not the selection box, but the bold row for the leftmost column)
- Cycle processing: select the files that need to be processed, set the cycle number and interval time, and click the cycle start to start the processing.
- Start line number, stop line number column, click to input, can be regional processing.

2. Array configuration

In the figure order, load the file- - -input parameters- - -array generation.



After the array is generated, jump to the following interface to start the following two ways:

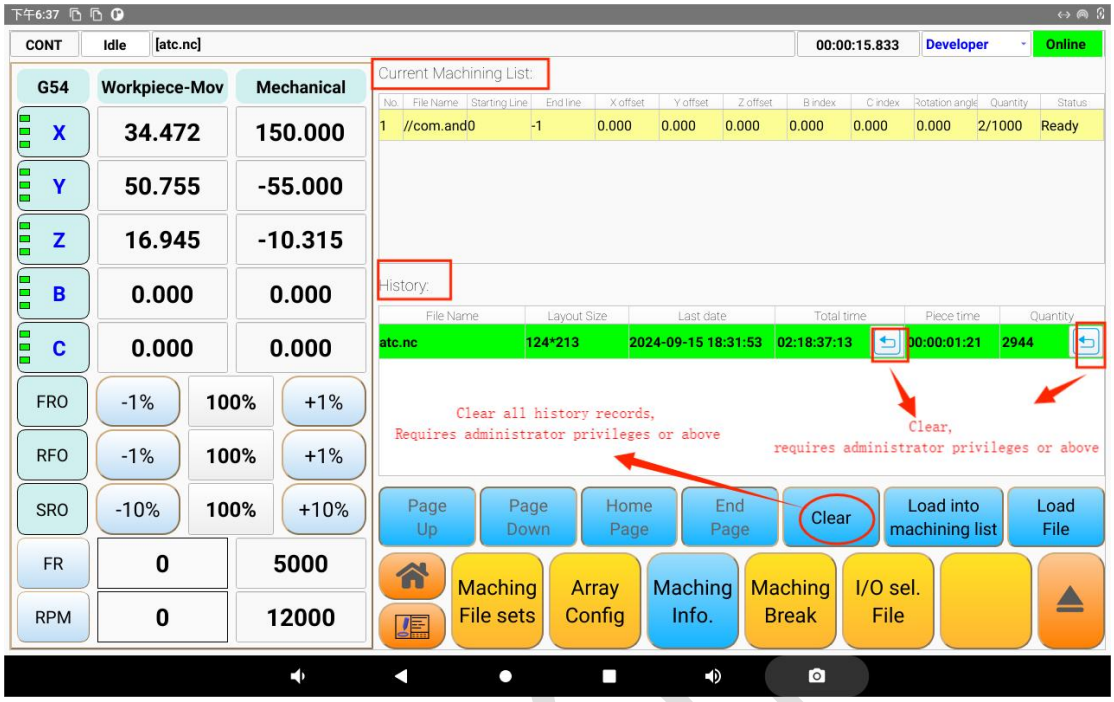
- Point or icon, point start processing.
- The number of cycles is set to 1, the interval to 0, and the point cycle is started.

3. Processing information

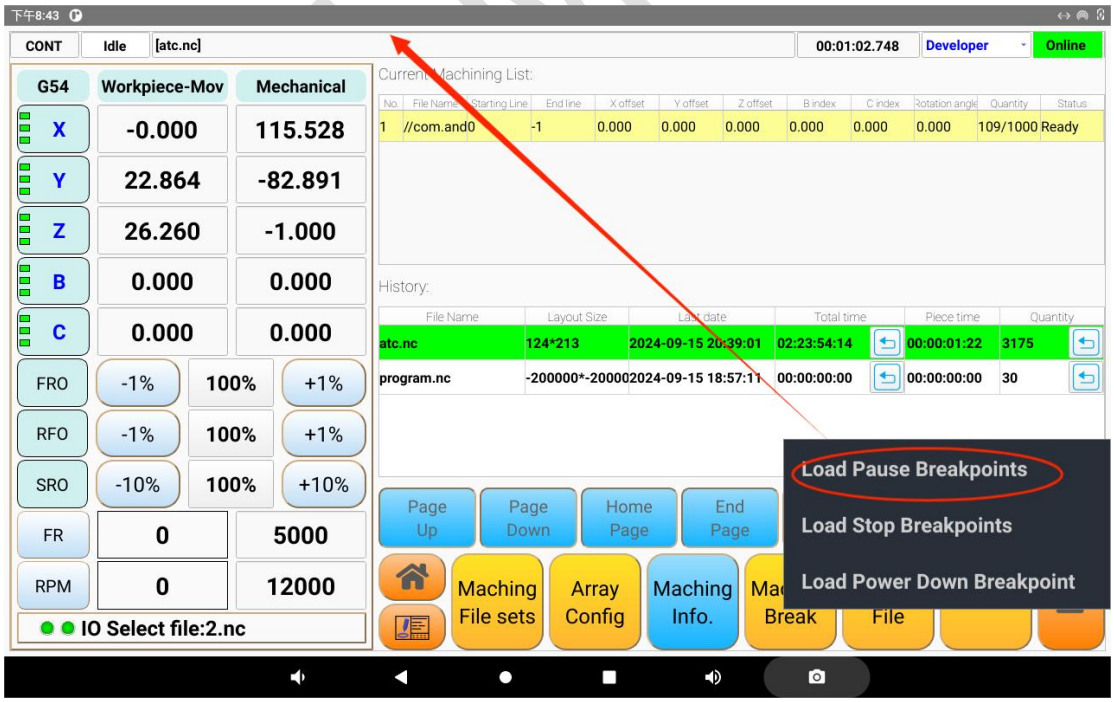
In two parts, the current processing list and the history record.

- **Empty: clear all history. Only the administrator above permissions can operate.**

- **Load the processing list: Add the selected history file to the processing list.**
- **Load file: Load the selected history file and jump to the simulation page window.**



4. Processing breakpoint



Click the load pause breakpoint to load the current processing file breakpoint position and load it in the status bar and the simulation page.

Load stop breakpoint and load power breakpoint are similar, the prompt box

pup when there is no corresponding breakpoint.

5. IO selection file

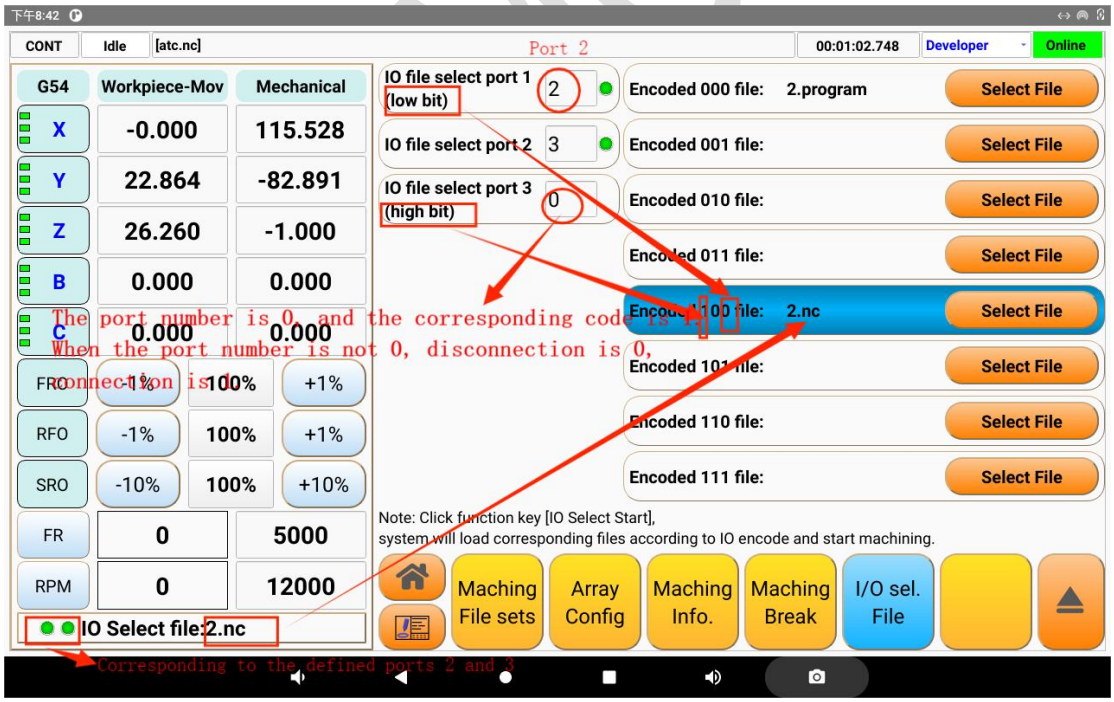
This function can combine different codes by defining the corresponding code 1 / 0 of multiple IO ports. After configthe port and corresponding file, configure an "IO select file start" key in the home page function key (see for detailsFunction key configuration), You can implement the IO selection file start function.

Open the IO file selection status in the display area configuration to display the corresponding file name and IO port status.see detailsAxis configuration and display。

When the port number is 0, the corresponding code is 1. When the port number is not 0, code 0 when disconnected and code 1 when connected.

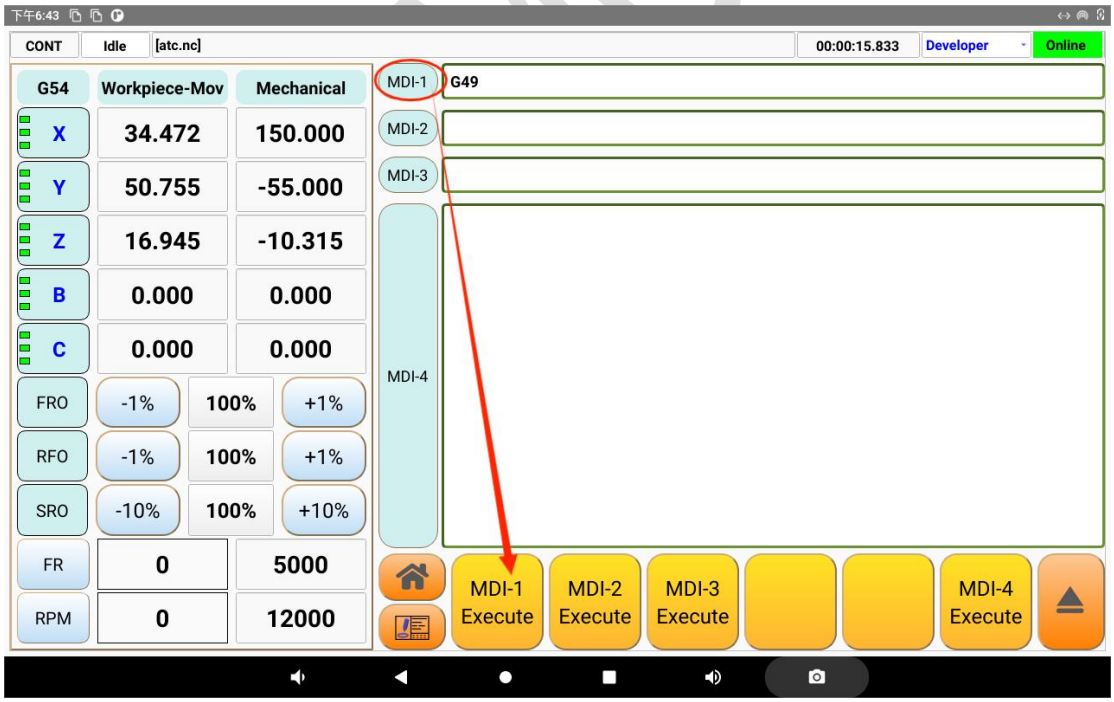
For example, in the following figure below, the high port number is 0 and the corresponding code is always 1. Both the middle and low levels have defined port numbers, and the illustrated port status is disconnected, and the corresponding code is 0, so the corresponding code is 100. The encoding 100 definition is named 2. nc, so the lower left IO selection file displays the file name 2. nc.

Therefore, 8 states can be combined from 000-111, and can correspond to 8 processing files. When the three IO ports are connected to three external buttons, the file processing is quickly switched by controlling the on / disconnection of the three buttons.



(四) MDI

The command can be entered in MDI, MDI-1, MDI-2 and MDI-3 are single lines, and MDI-4 can be entered in multiple lines.



(五) IO Select a file to start

take part in*IO selection file*Supporting use, you can quickly select different processing files through the IO port.

(六) Recent point processing

Used to quickly locate the breakpoint, find the cutting path closest to the current cutting point and start machining.

usage method:

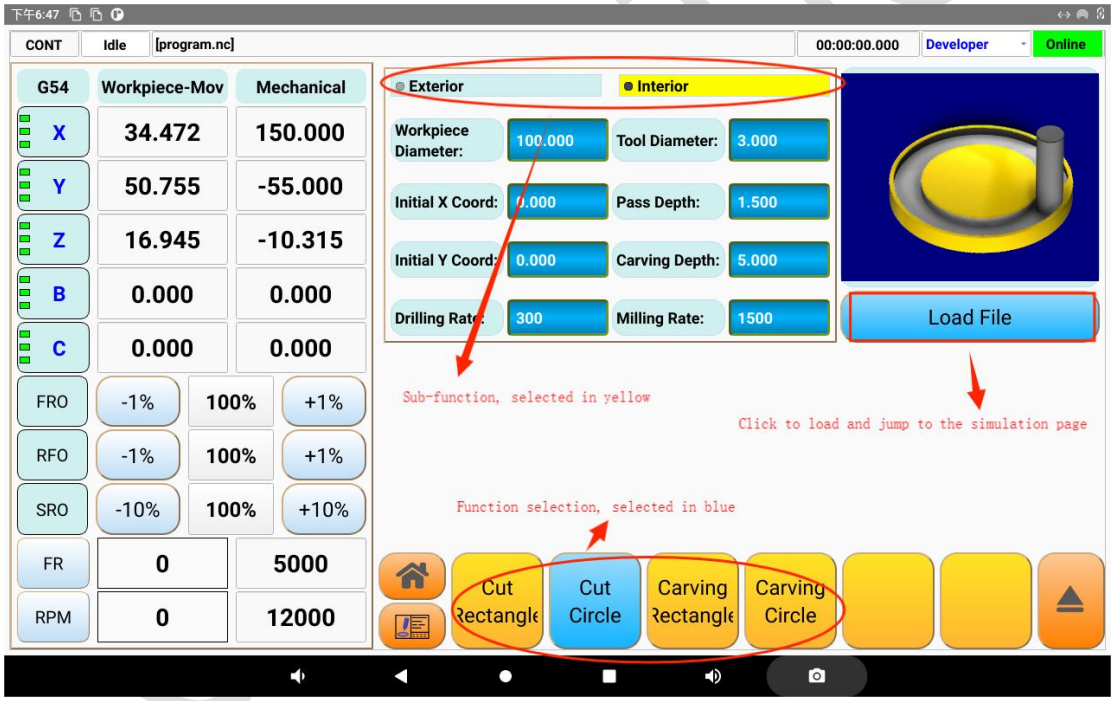
- Arrange the nearest point processing function on the function key first. SeeFunction key configuration method。
- Move the tool near the breakpoint (closer, more accurate).
- Click on the nearest point for processing.

Application premise:

- The soft limit should be in the enabling state.
- The Z axis can not be a layered blade, only a single layer blade.

(七) Processing guide

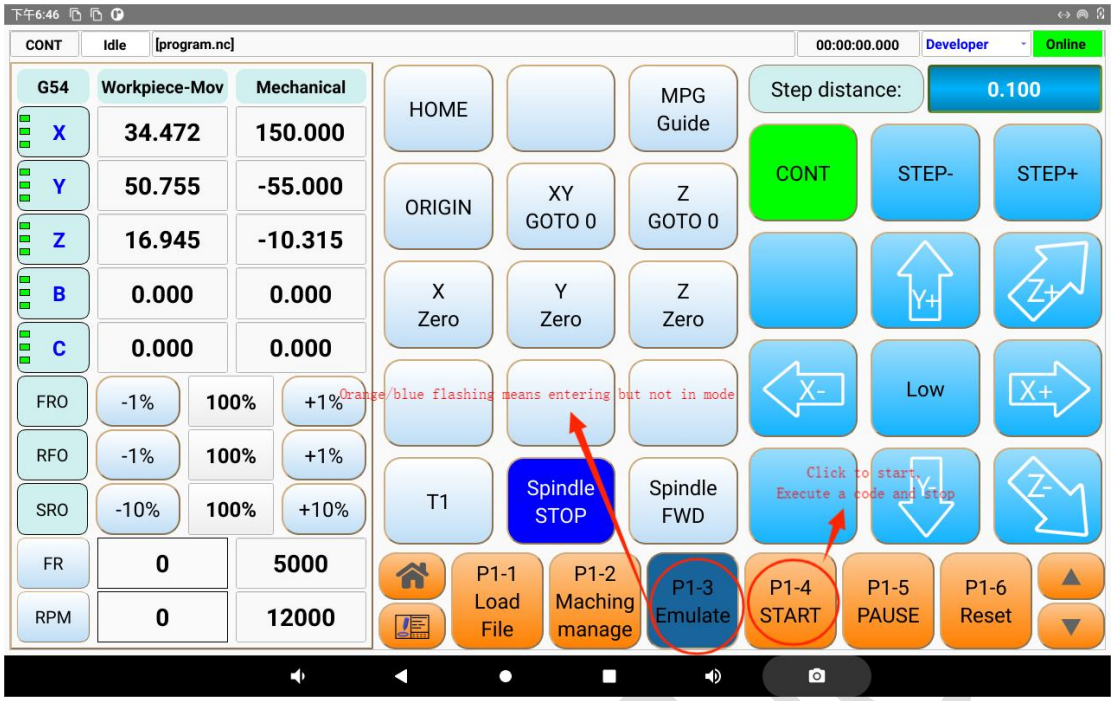
It provides four parametric programming functions of cutting square frame, cutting circular frame, milling rectangular bottom and milling circular bottom.



(八) One Step

After clicking the single step, the button flashes to enter the single step mode, then click the start button, press to execute a code.

After clicking the single step again, the flicker disappears, exit the single step mode, and then the program automatically performs until the end.

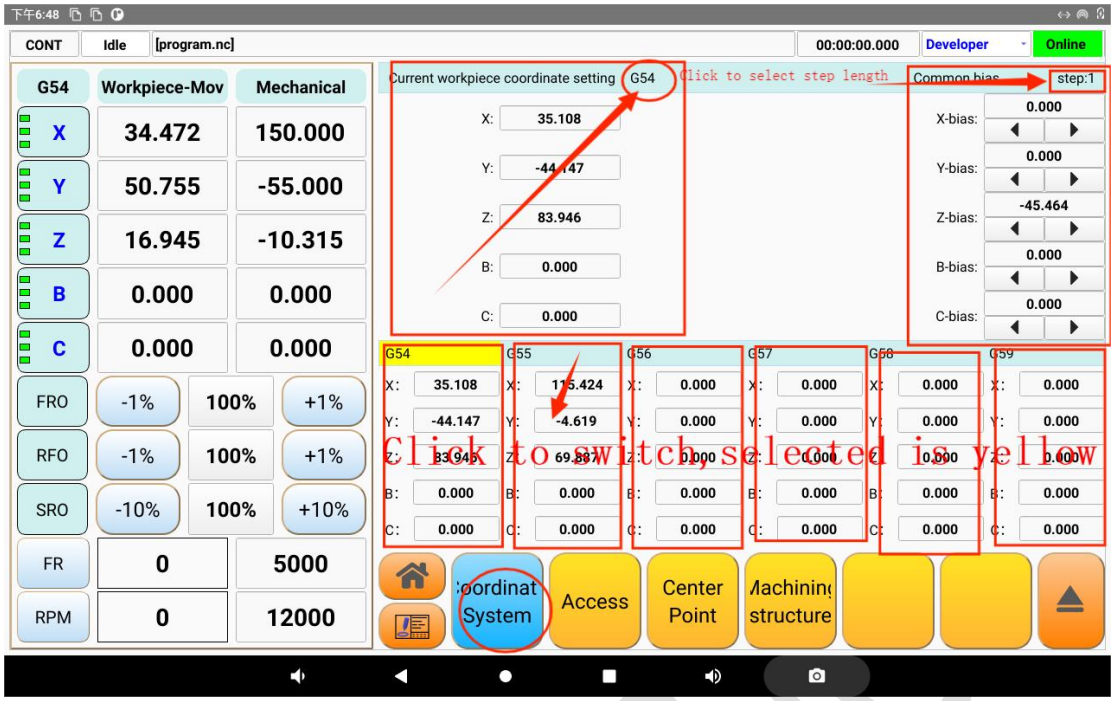


(九) Coordinate System

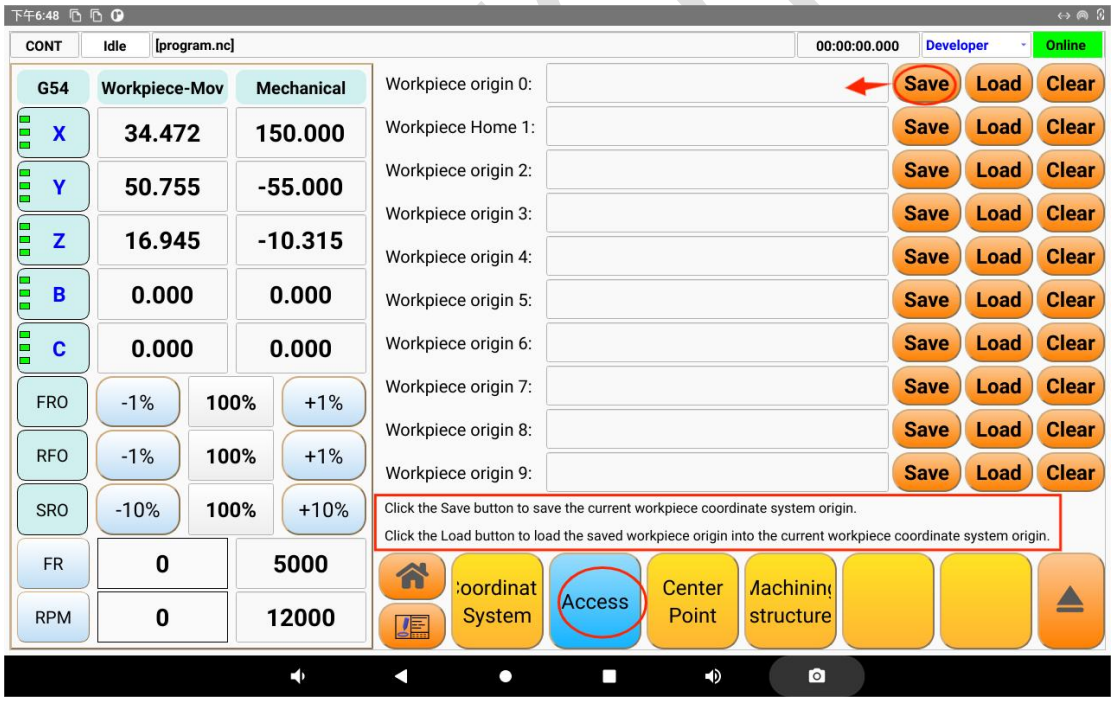
Including coordinate bias, access, center point measurement, machine tool structure.

1. Coordinate bias

- The G54-G59 coordinate offset can be set;
- Can set the public bias of each axis, click to edit, or adjust through the direction key below;
- Click to adjust the step distance to choose the steps of 0.01,0.1 and 1.



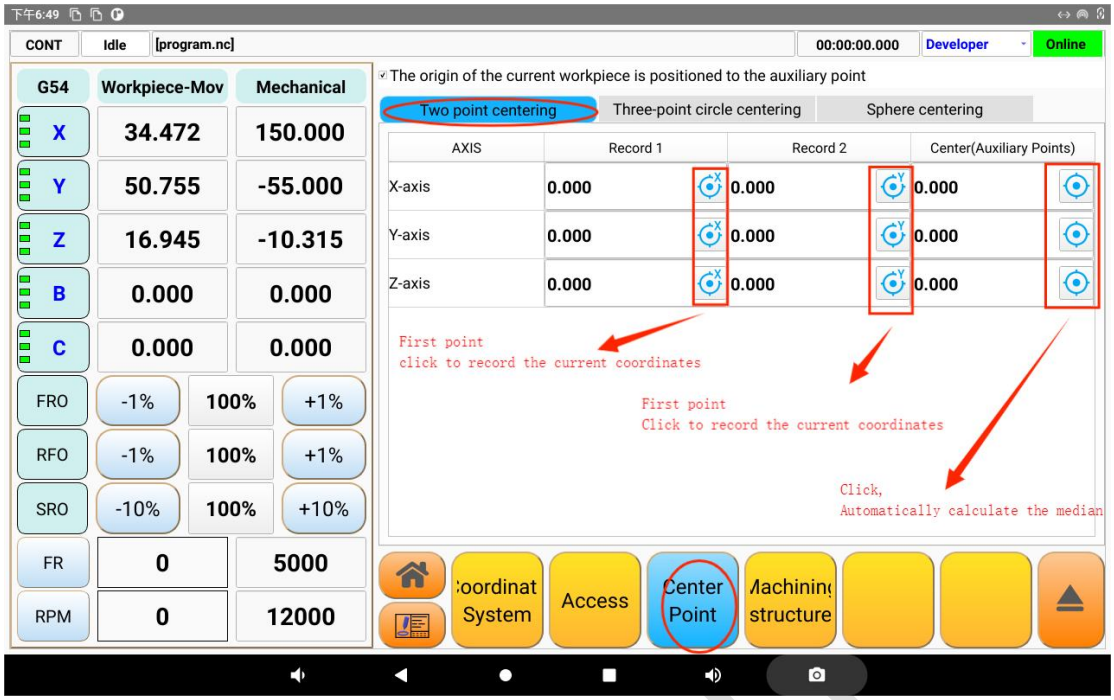
2. access



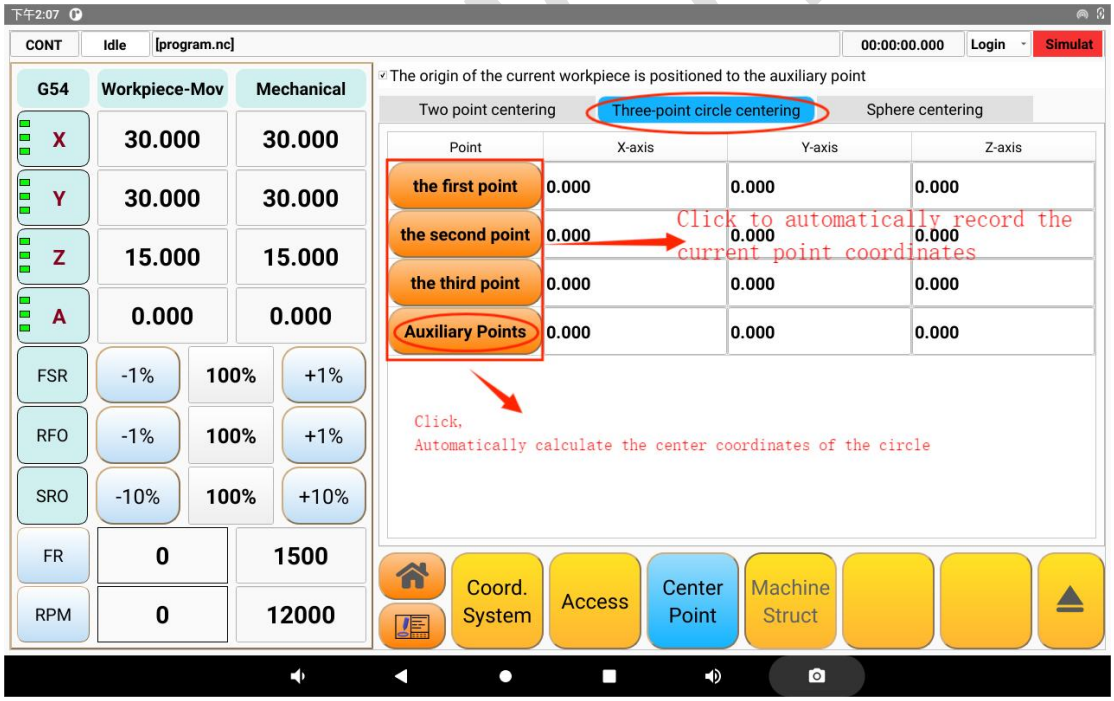
3. Central point measurement

Including two point center, three point center, three coordinate center.

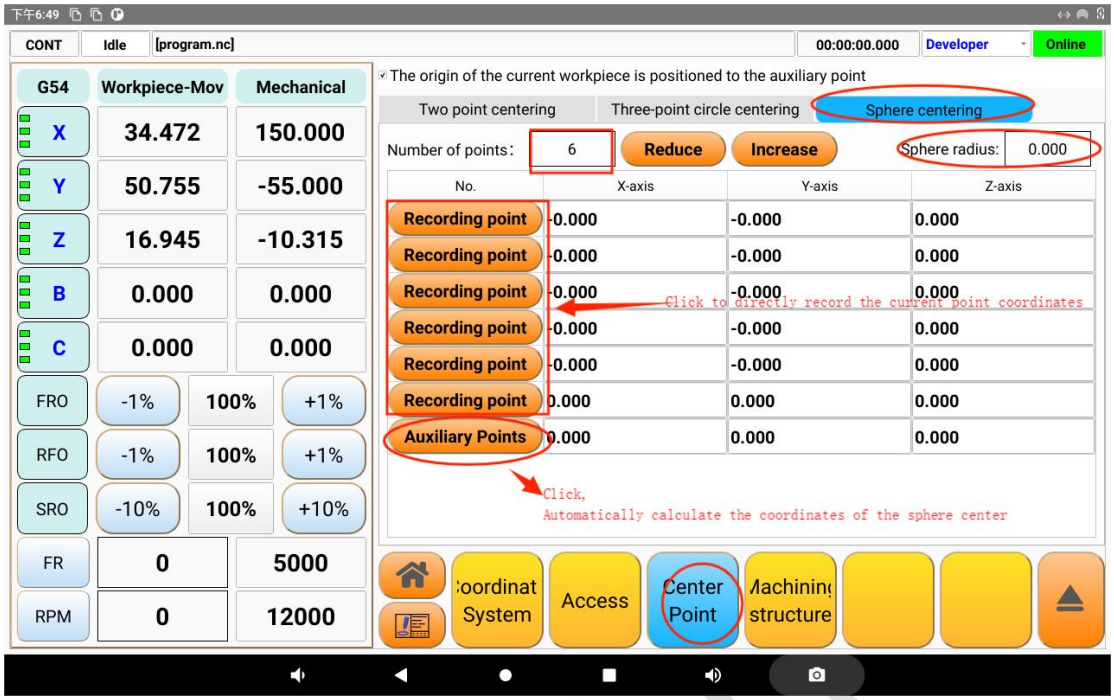
1) Two points:



2) Three points fixed circle heart:



3) Three-coordinate fixing center:

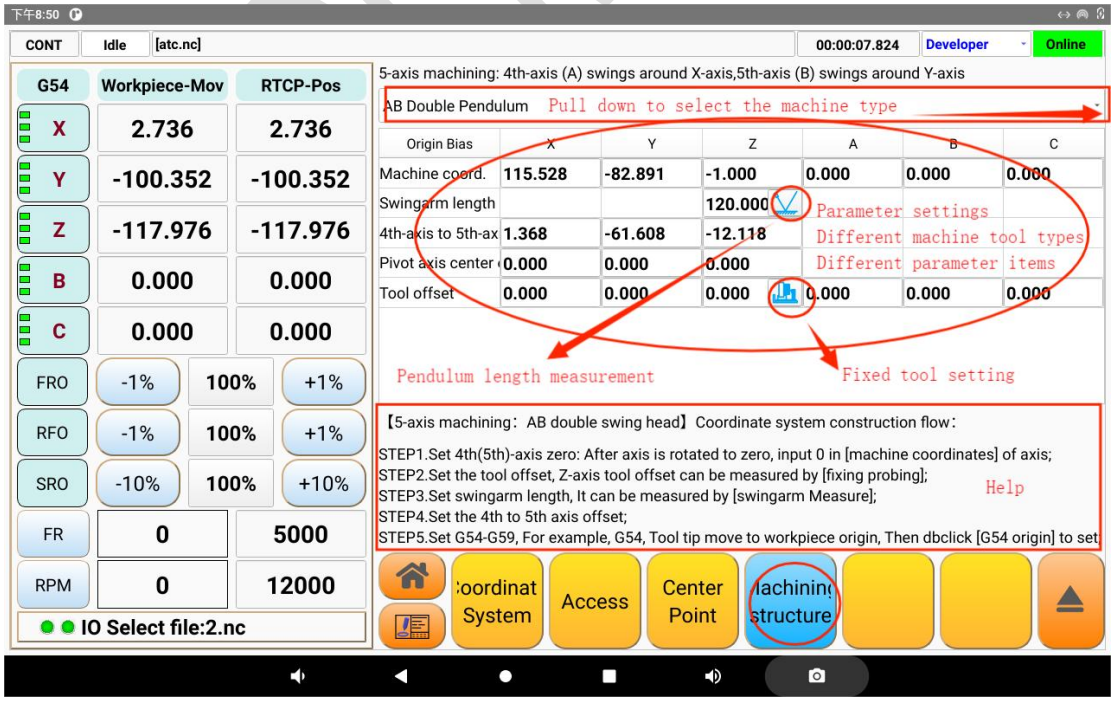


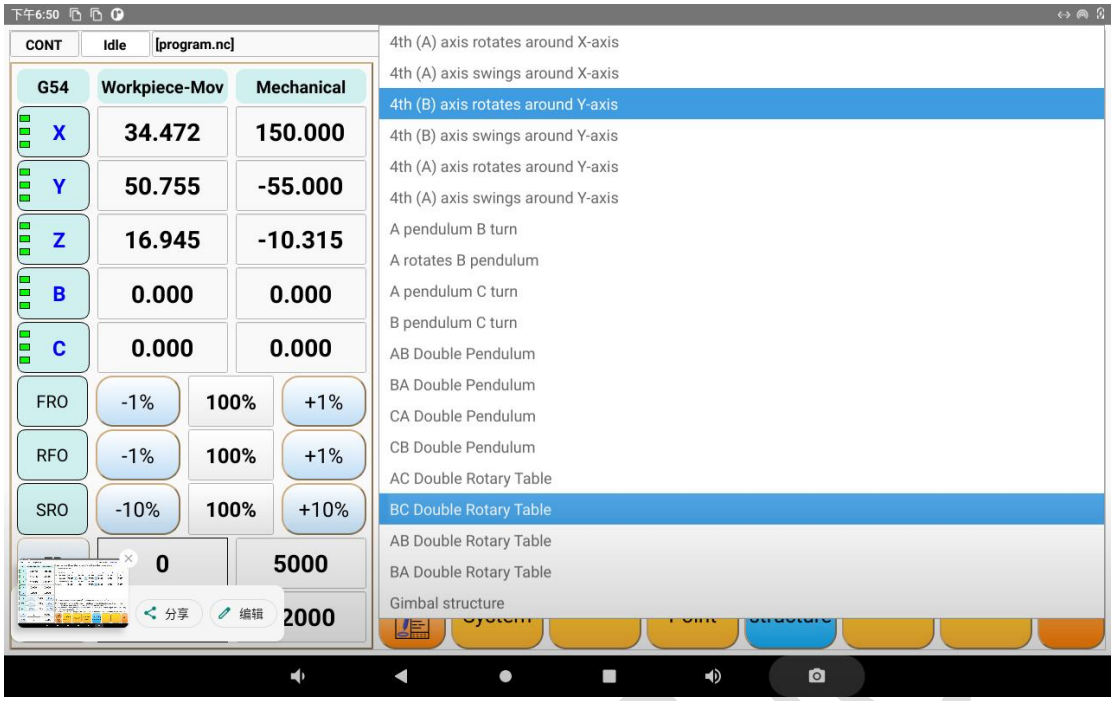
4. Machine tool structure

Ask the administrator above permission.

Select the corresponding machine type according to the actual machine structure and set relevant parameters.

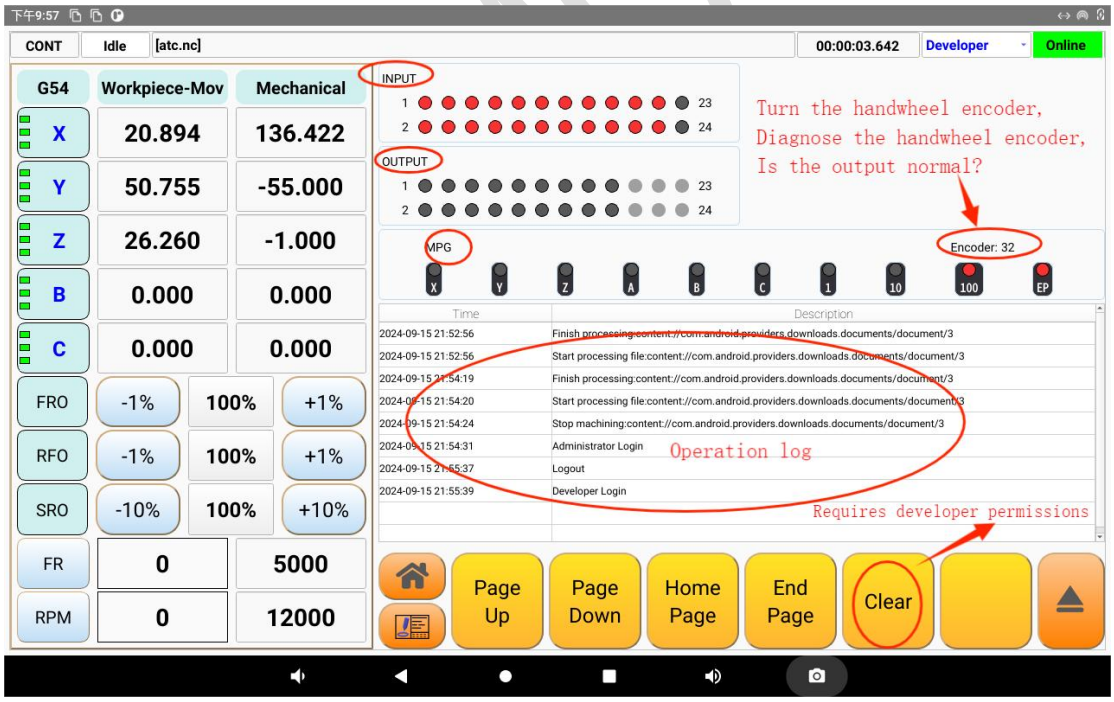
If it is a three-axis machine tool, you can choose arbitrarily.



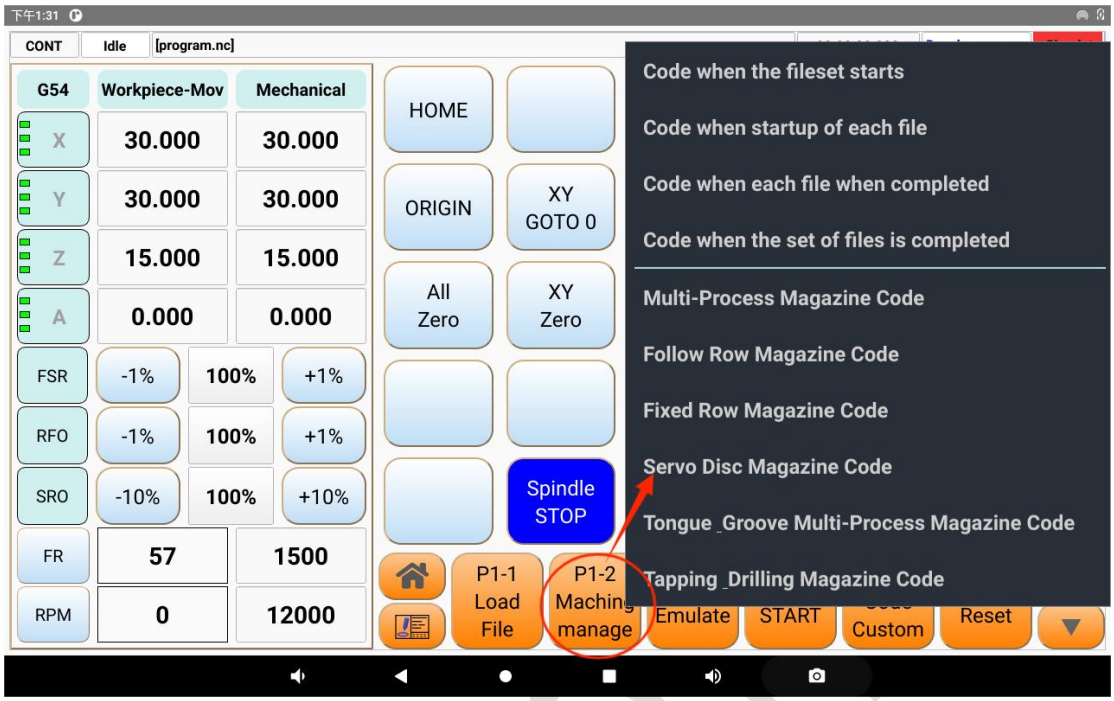


(十) Diagnose

All system logs are logged here and only cleared under developer permission.



(十一) Code customization



1. Usage Method

- Click on Code customization and select the type of code to be customized.
- Click Edit to edit the code (insert the exported code), click to cancel edit.
- Click to save.
- Click Close, then jump back to the original page (main page or simulation page). Complete the code customization.
- If you need to export them, you can point to export them after the editing is complete.

2. Code custom type

1) Code at the processing file set startup

This code is executed before each processing file set starts.

2) Code when each file starts

This code is executed before each single processing file starts.

3) Code when each file completes

This code is executed at the end of each individual processing file code.

4) Process file set completion code

This code is executed at the end of each processing fileset code.

5) Multi-process knife library code

Provides a common swap code for multi-process library and execuexecuted when M6 swap code. It can be modified as needed.

6) Gantry straight line knife library code

A universal swap code for the gantry cutter library is provided and executed with the M6 swap code. It can be modified as needed.

7) Fixed direct library code

A universal swap code for the fixed straight knife library is provided and is executed when encountering the M6 swap code. It can be modified as needed.

8) Disk Knife Library Code

A universal swap code for the disc knife library is provided and is executed when an M6 swap code is encountered. It can be modified as needed.

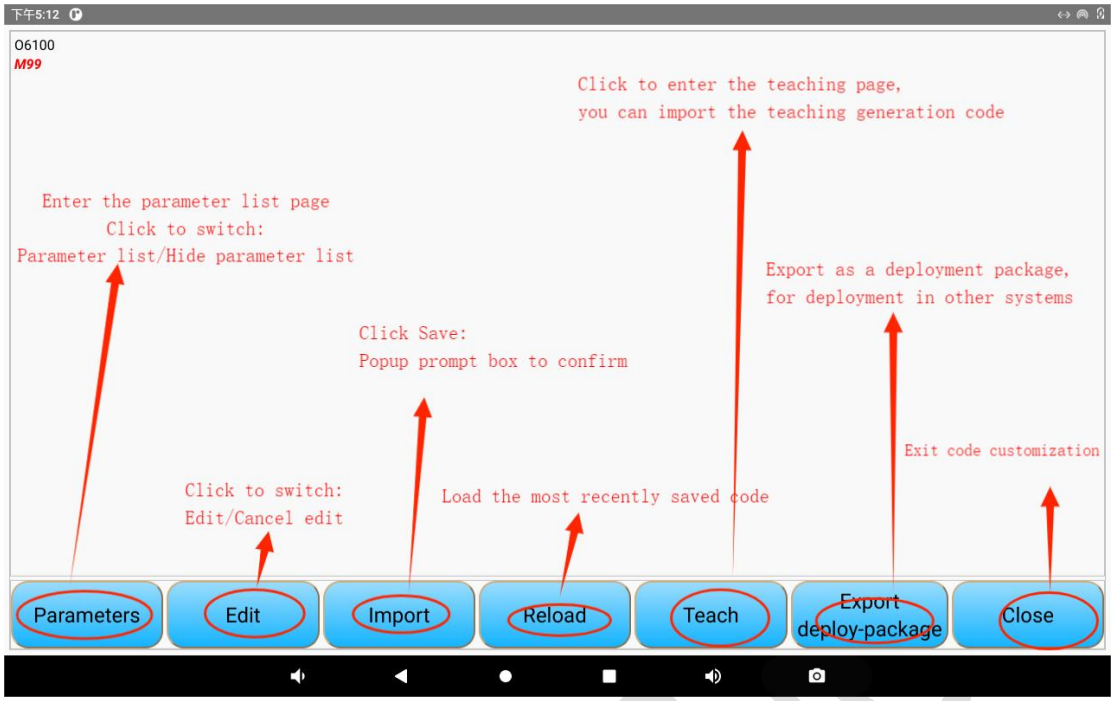
9) Tenon and groove multi-process knife library code

The general swap code for the multi-process knife library is provided and is executed when M6 swap code. It can be modified as needed.

3. Function Declaration

Consider the code of the processing file set.

After clicking code customization to select customization type, the code editing interface is up:

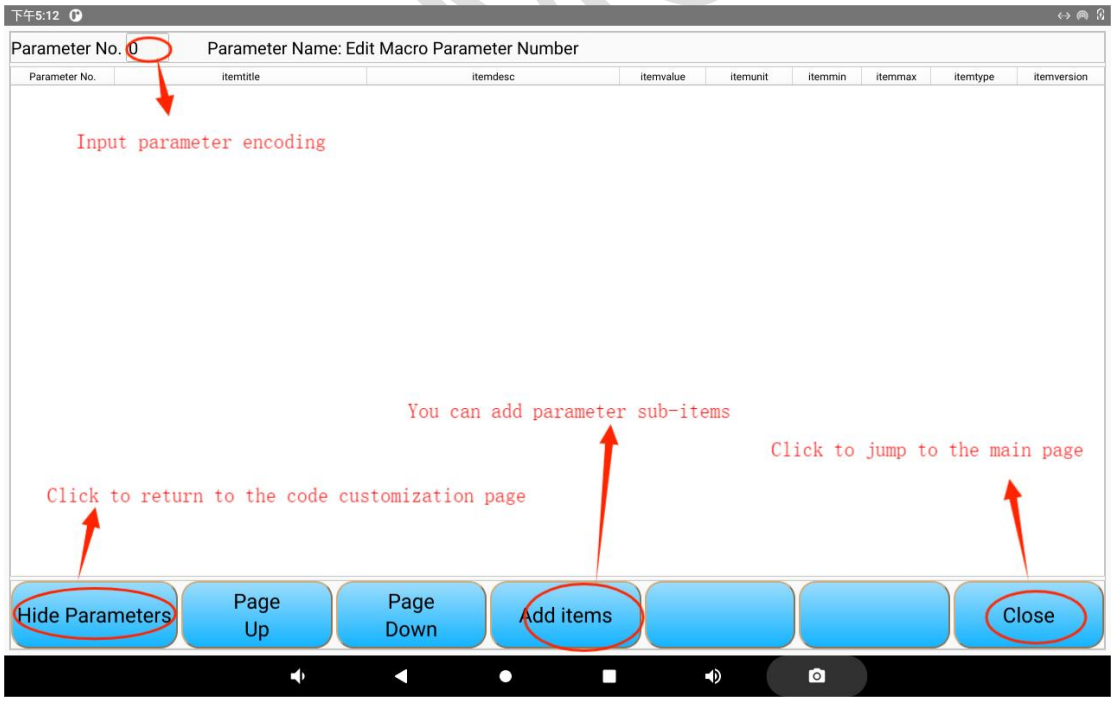


1) List of parameters

Click on the parameter list and the following page pop up to view and modify the parameters, but need to modify carefully.

To cut back to the code customization page, press the hidden parameter list.

Press Close and jump directly to the main page.



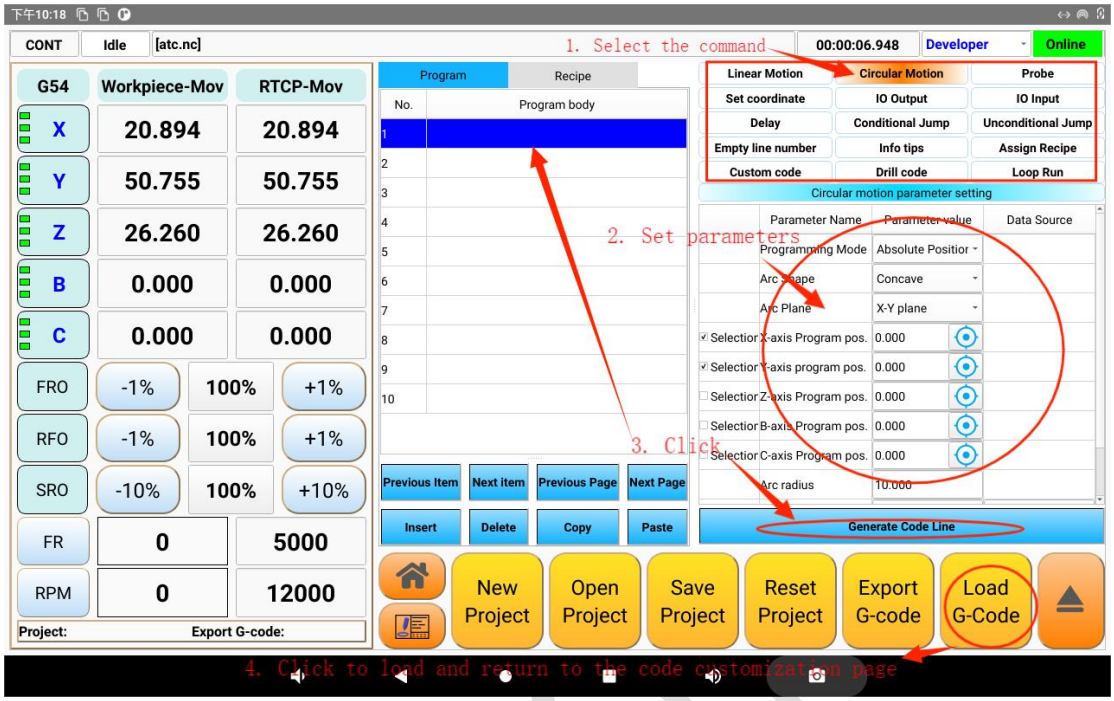
2) Instruct

The teaching function is introduced in the code customization, which is convenient to generate the code and import G code to simplify the programming

difficulty.

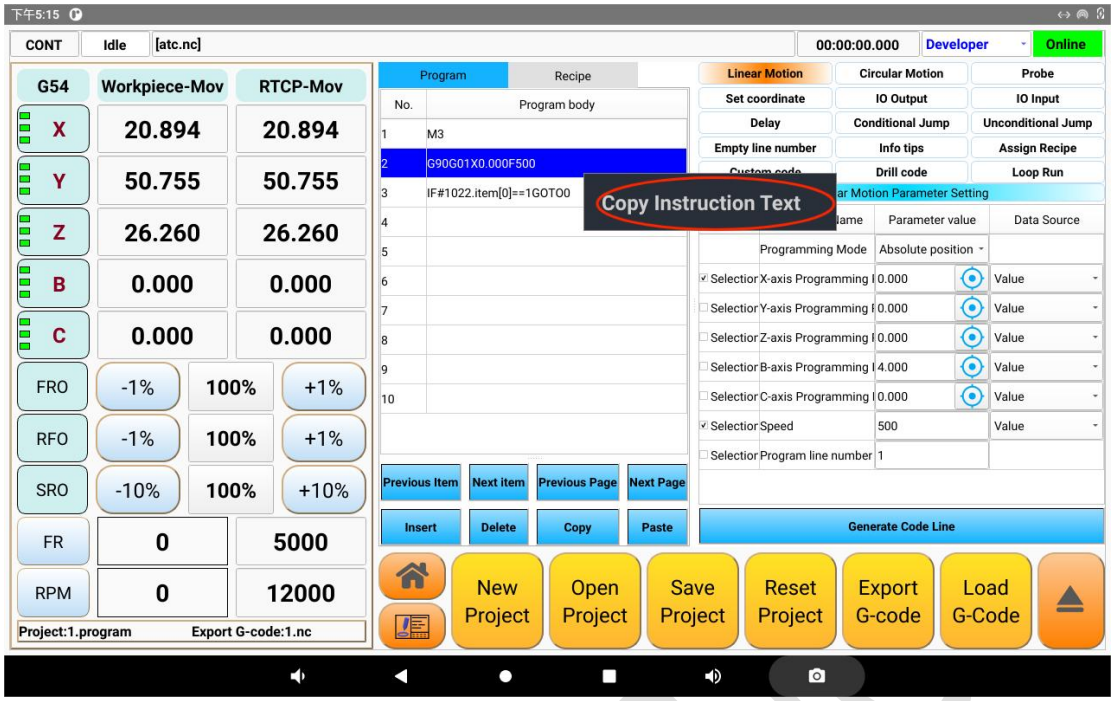
Click Teach and the page popup below.

Except for the load G code function, see the main function key"instruct”。



pay attention to:

- The load G code in the main function key is to load the G code into the processing file code area, and the load G code function here is to load and return to the code customization page.
- If you return to the main page during the teaching process (not click closed), then click the instruction in the main function key, you are still in the teaching state of code customization, and the G code will return to the code customization page.
- The loading G code here is to load all the generated code lines into the code customization. If you only need to insert a line of it to generate code, you can click the line of code, copy text, click back, and paste in the code customization. See below.

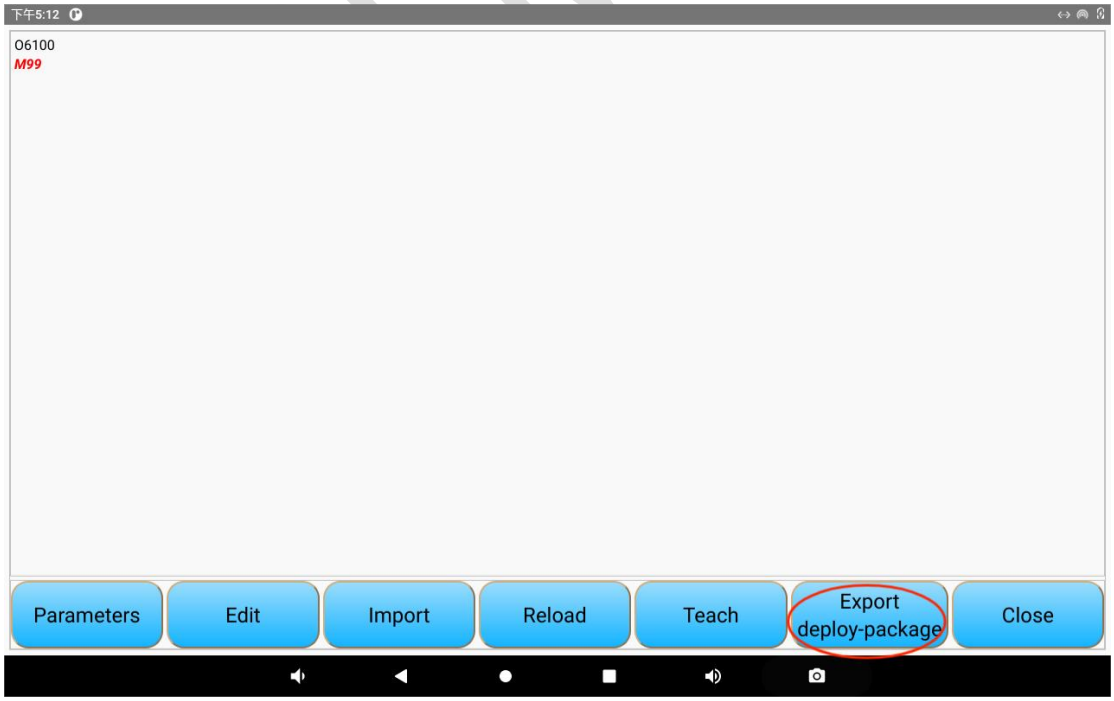


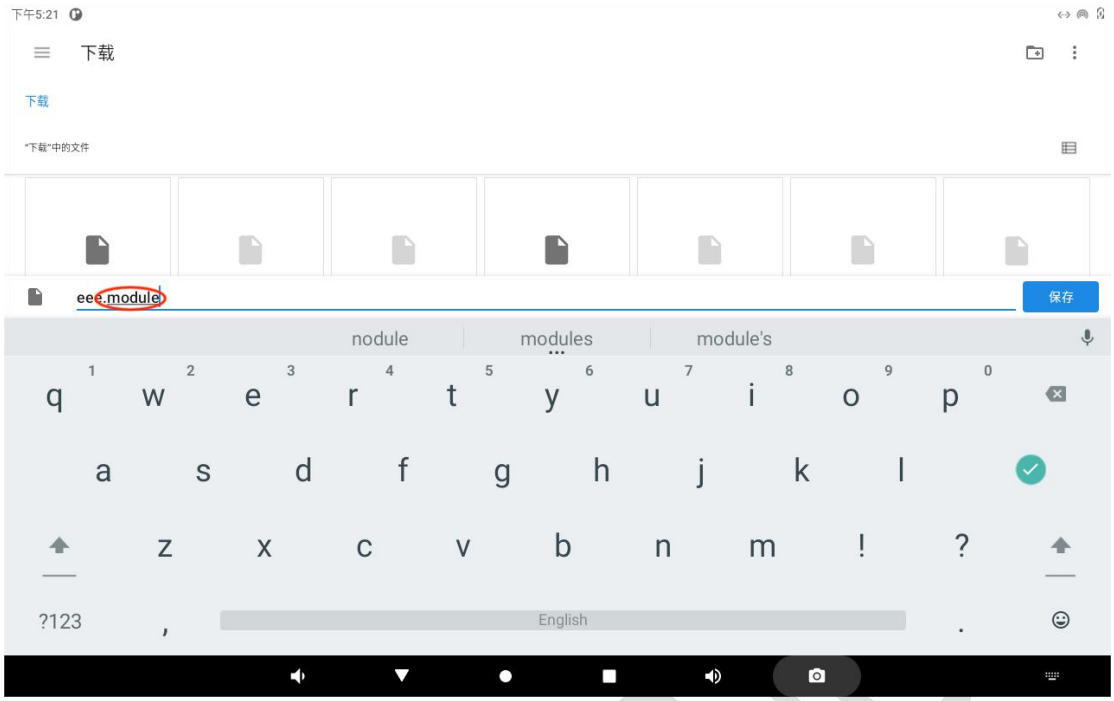
3) Export the deployment package

Click Export Deployment Package, pop up the deployment package save page (as shown below), save path can be selected, but be sure to enter the suffix name. module.

The Deploy Export prompt box appears after saving.

Here is the deployment package that generates the custom code and can be deployed on other systems.





(十二) Instruct

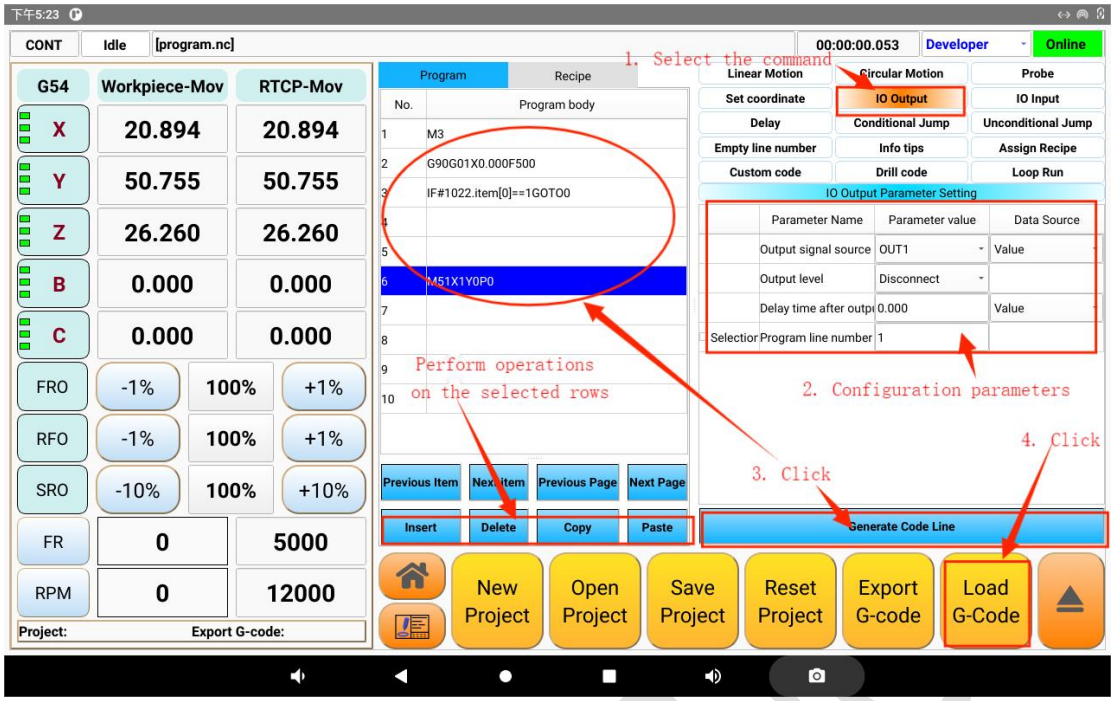
The teaching page is as follows. Various instructions can be customized by setting parameters to realize parametric programming.

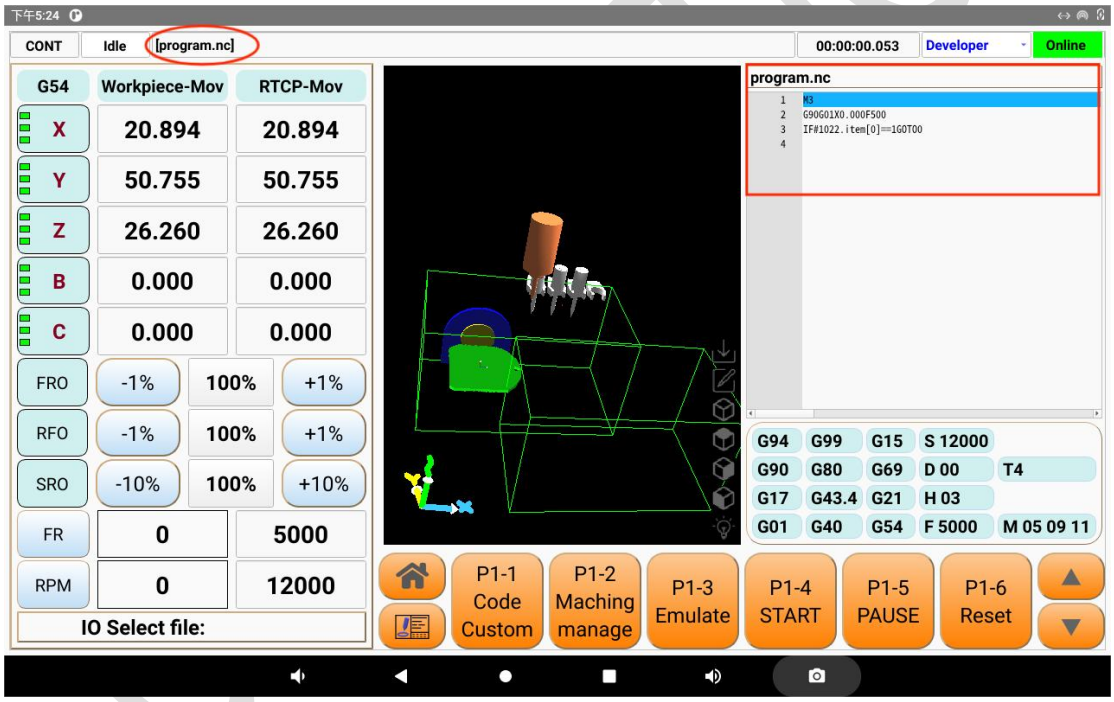
1. usage method

- In Step 1, choose different instructions according to the requirements
- The second step, set the parameters;
- Step 3, click on the generation line of code;
- Step 4, repeat steps 1-3 to continue generating the lines of code;
- Step 5, after all the instruction lines are completed, click load G code, will jump to the simulation page, click start, then start the processing.as illustrated in following figure.

pay attention to:

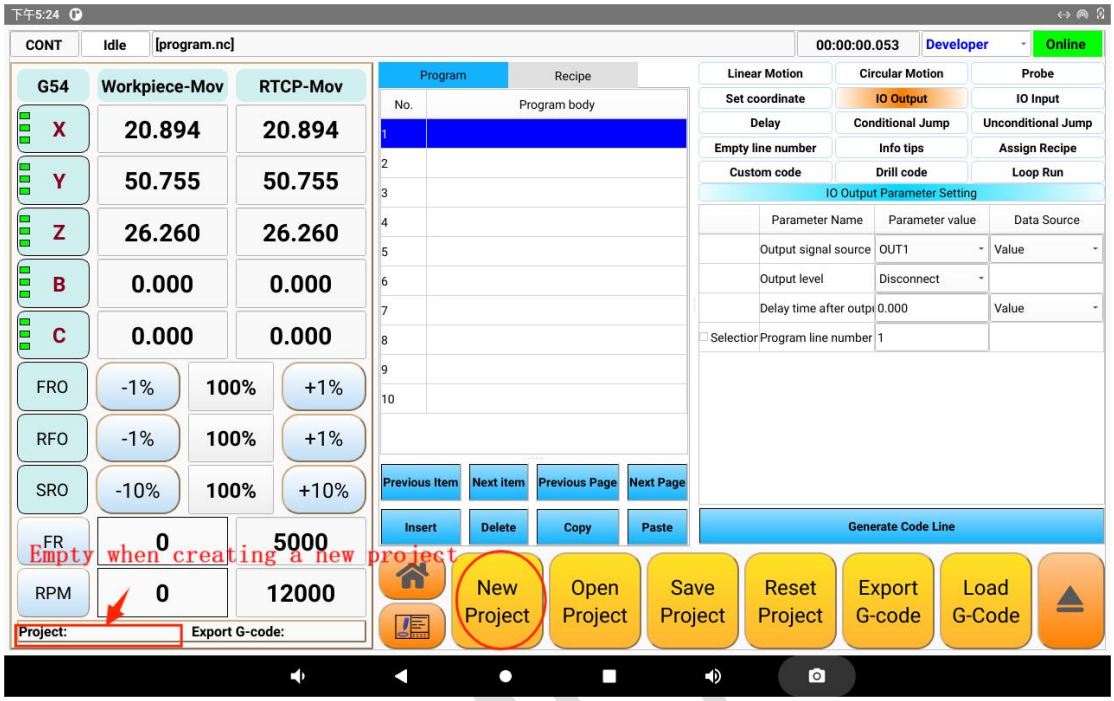
- When generating multiple lines of code, if the speed is set in the front, the following programs move at the nearest specified speed above.
- If the whole speed is the same, there is no need to set here, will press the default move input speed (parameter setting) movement;


1. Select the command
2. Configuration parameters
3. Click
4. Click
Perform operations on the selected rows
Program
Recipe
Program body
Linear Motion
Circular Motion
Probe
Set coordinate
IO Output
IO Input
Delay
Conditional Jump
Unconditional Jump
Empty line number
Info tips
Assign Recipe
Custom code
Drill code
Loop Run
IO Output Parameter Setting
Parameter Name
Parameter value
Data Source
Output signal source
Output level
Delay time after outp
Program line number
Previous Item
Next Page
Previous Page
Insert
Delete
Copy
Paste
Generate Code Line
New Project
Open Project
Save Project
Reset Project
Export G-code
Load G-Code
CONT
Idle
[program.nc]
00:00:00.053
Developer
Online
G54
Workpiece-Mov
RTCP-Mov
X
Y
Z
B
C
FRO
RFO
SRO
FR
RPM
Project:
Export G-code:

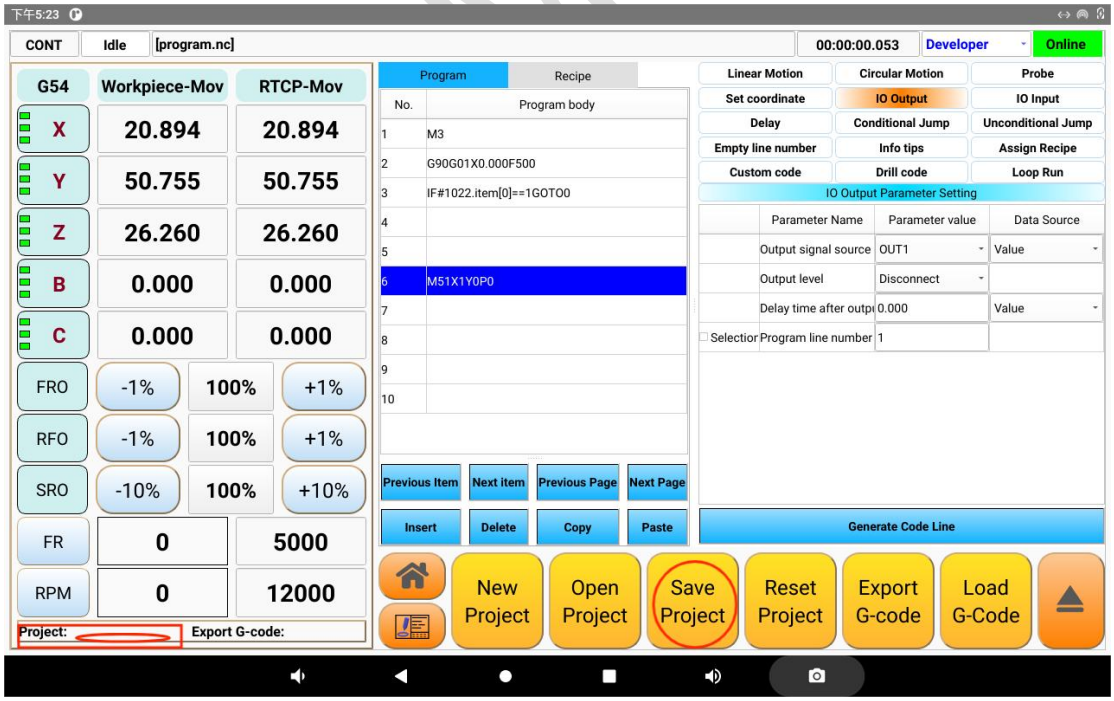

CONT
Idle
[program.nc]
00:00:00.053
Developer
Online
G54
Workpiece-Mov
RTCP-Mov
program.nc
G94 G99 G15 S 12000
G90 G80 G69 D 00 T4
G17 G43.4 G21 H 03
G01 G40 G54 F 5000 M 05 09 11
P1-1 Code Custom
P1-2 Maching manage
P1-3 Emulate
P1-4 START
P1-5 PAUSE
P1-6 Reset
IO Select file:

2. Function key instructions

1) new construction:



2) Save the project:



The save path can be arbitrarily defined, note the input file suffix. program The prompt box Ffile is saved. And the saved file name appears in the lower left corner project.

If the lower left project has a file name, the point save project will no longer pop

the save path, but directly in the existing file. Similar to the save function in word.

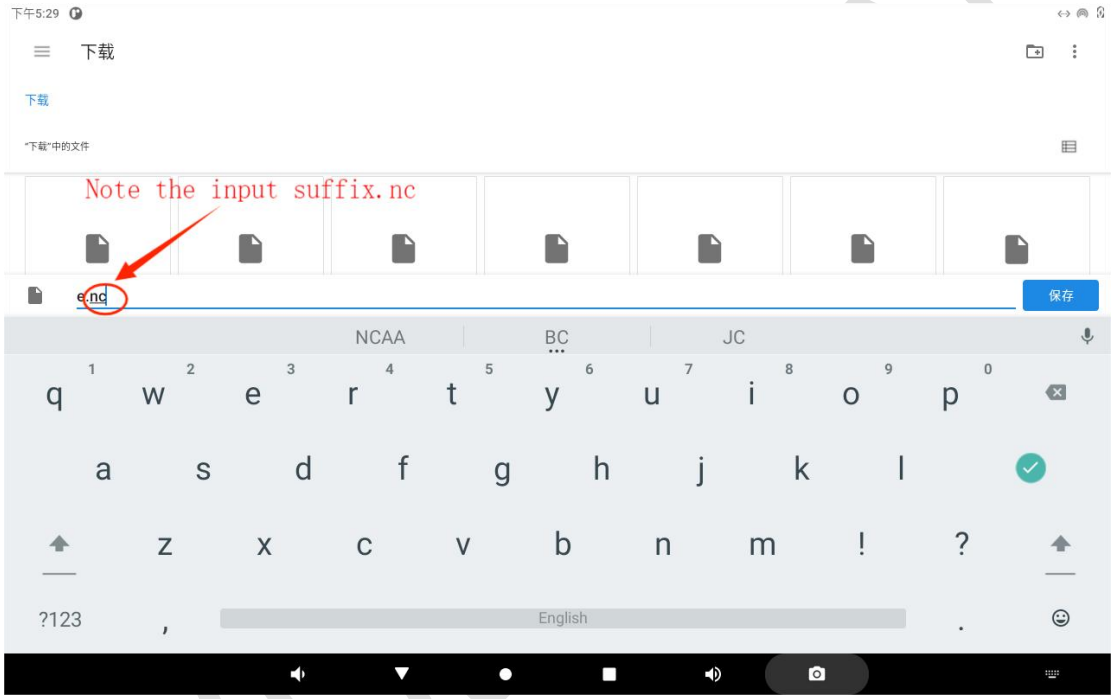
3) Save the project

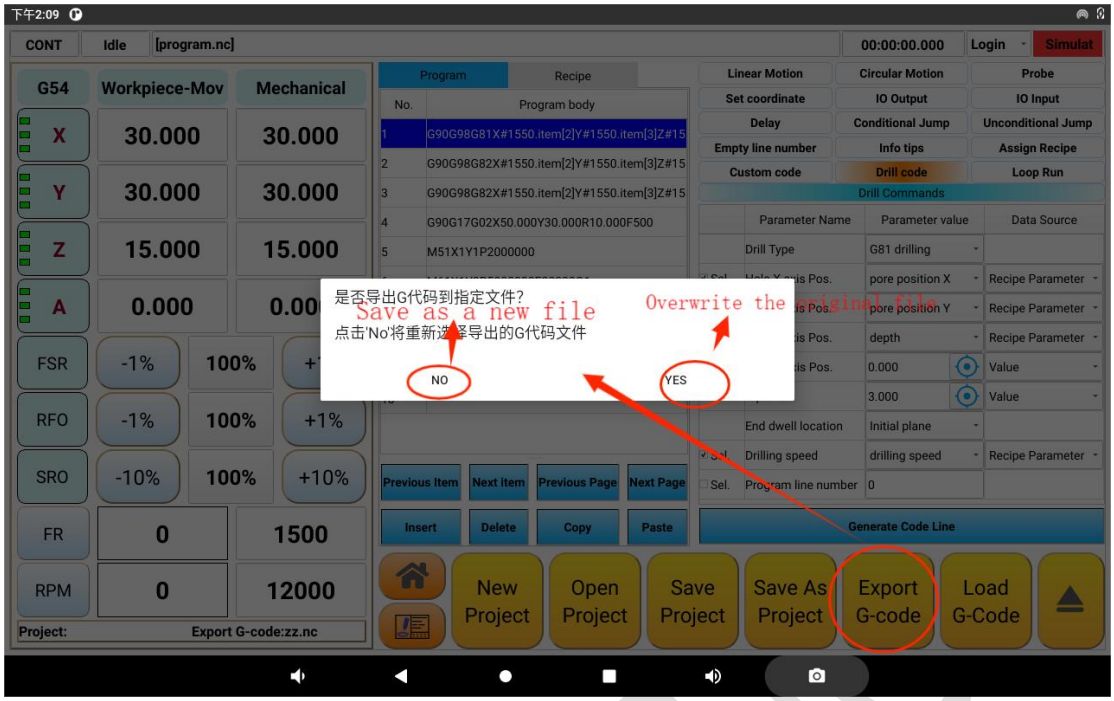
Can be saved as an other project, note the input file suffix. program.

4) Export G code

If "Export G Code:" is empty in the lower left corner, the save path page will pop up. note the file suffix. nc.

If the file name is "Export G code:" in the lower left corner, then the dialog box appears, select yes overwrite the original file. Select no to pop the save path page and save it as a new file.





5) Load G code

Click load G code, will jump to the simulation page, click start, that is, press the generated command code to start processing.

3. Formula parameter table for application

There are two types of data sources, directly specified values or indexed by formula parameters, formula parameters can be set on the formula parameters table page.

The formula parameter function is equivalent to the introduction of variables in the instruction. The flexible use of the formula function can increase the versatility of the program and reduce the program changes.

1) usage method

In the first step, click the instruction-formula parameter table to define the parameter names and numerical values.

In the second step, select the instruction type and select the formula parameter index in the corresponding parameter data source column.

In the third step, select the parameters in the corresponding parameter parameter value bar. Repeat steps 1-3 until all parameter settings are complete.

Step 4, click the line of code.

Step 5, click on the program page to see the generated lines of code.

Step 6, click load G code, switch to the simulation page, click start, then start processing.

Step 7, click the instruction-formula parameter table, modify the parameter values, and save the formula.

Step 8, click, directly switch to the simulation page, click start, that is, start the processing according to the new parameters.

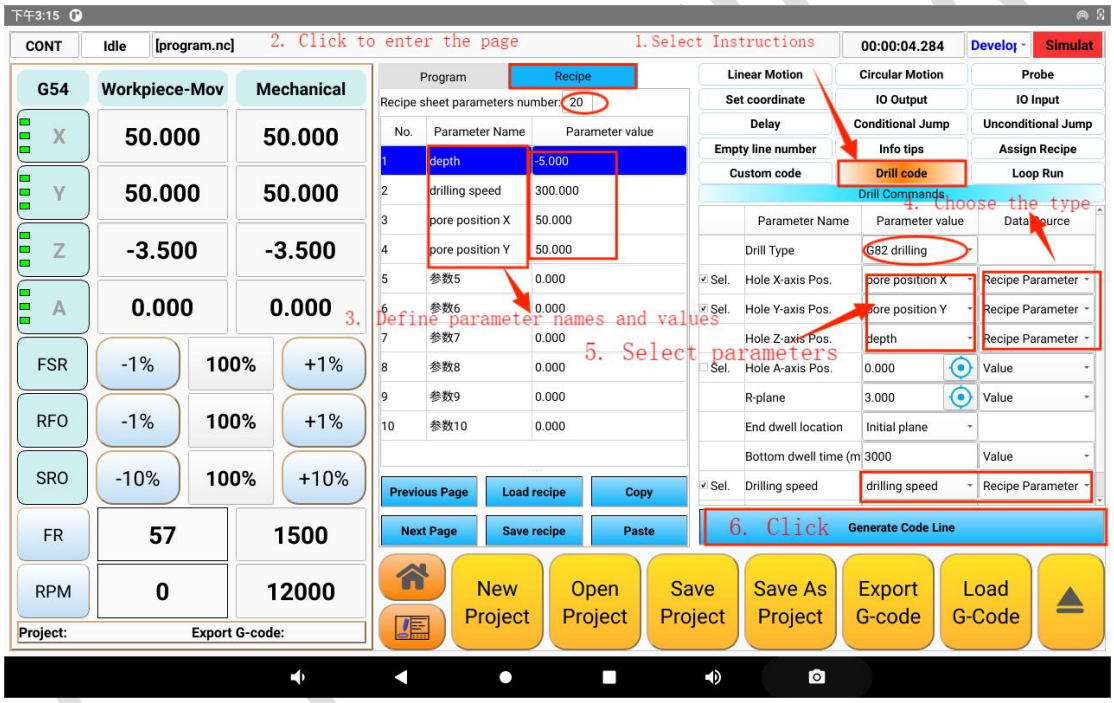
2) explain

- The number of parameters can be edited, the number of changed, the number of the list below follows the change, click "last page", "next page" to turn the page view.

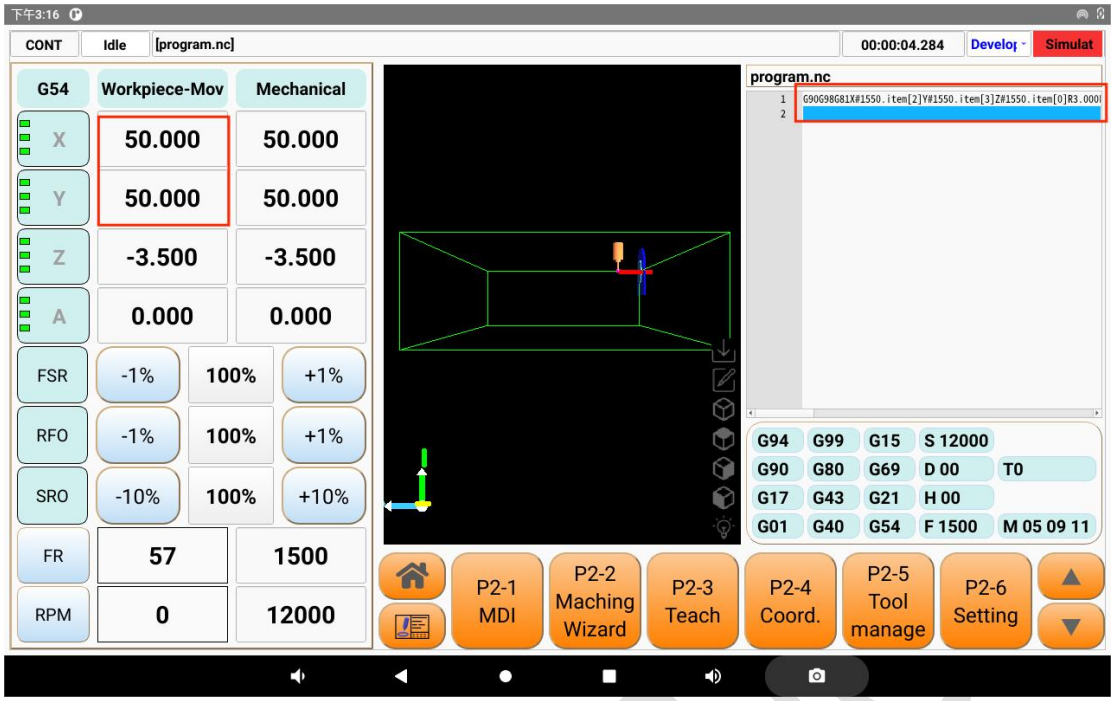
- Parameter names can be customized for easy identification.
- Click on the parameter value to edit.
- Each set of parameter values can be saved as a formula, and different schemes can be loaded into different formulas. The save path can be customized.

3) Application examples

Take the drilling instruction as an example, see the figure below.

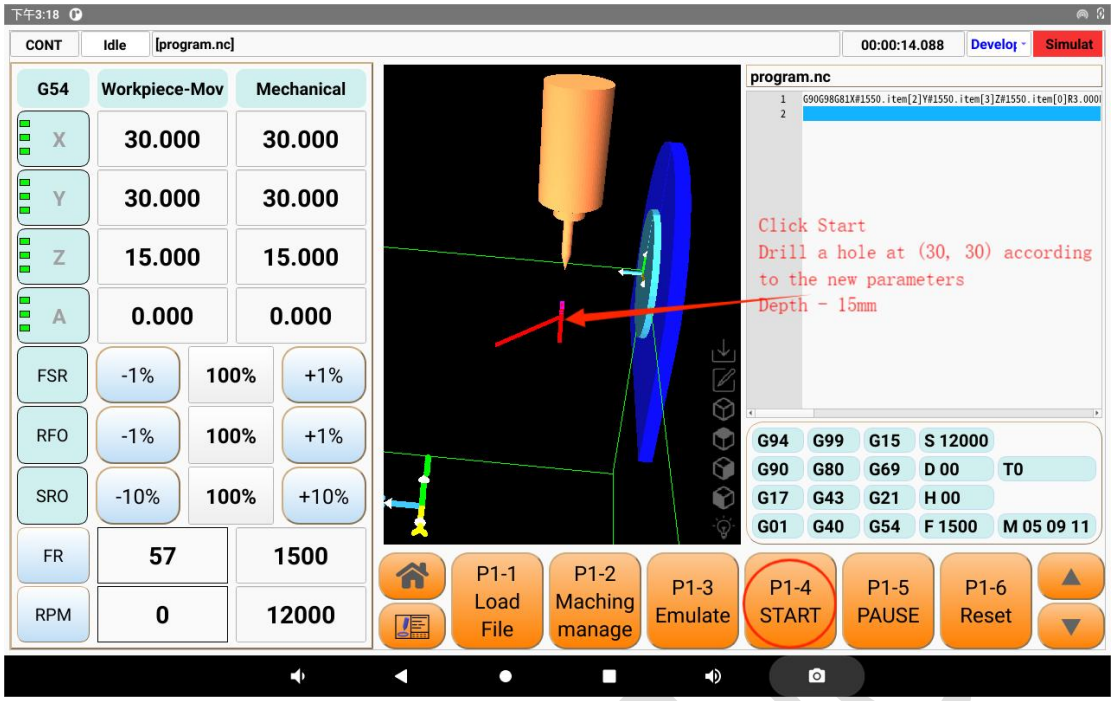


Click the program page to see the code just generated. Click Start, drill G82 according to the parameters in the figure above and below (50,50):

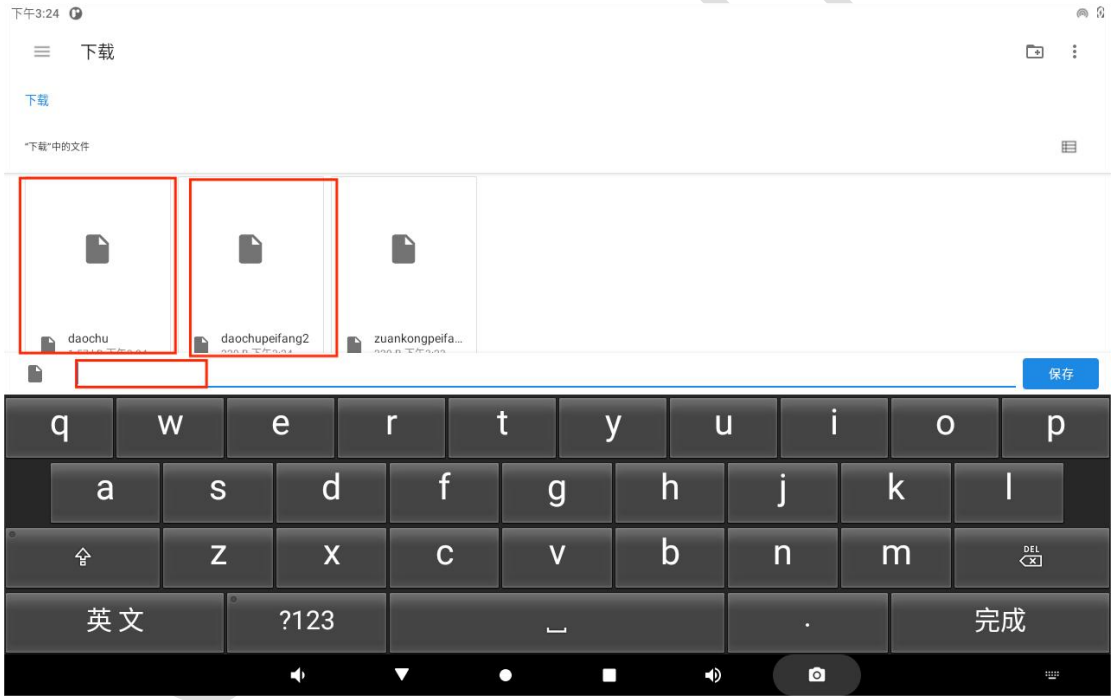


At this point, change the parameters, drill in (30,30), depth-15mm, then return to the teaching-formula parameter table page, as shown in the figure below.

Point, cut back to the simulation page, start at the new location (30,30).



Two sets of parameters can be saved separately to facilitate the next time.



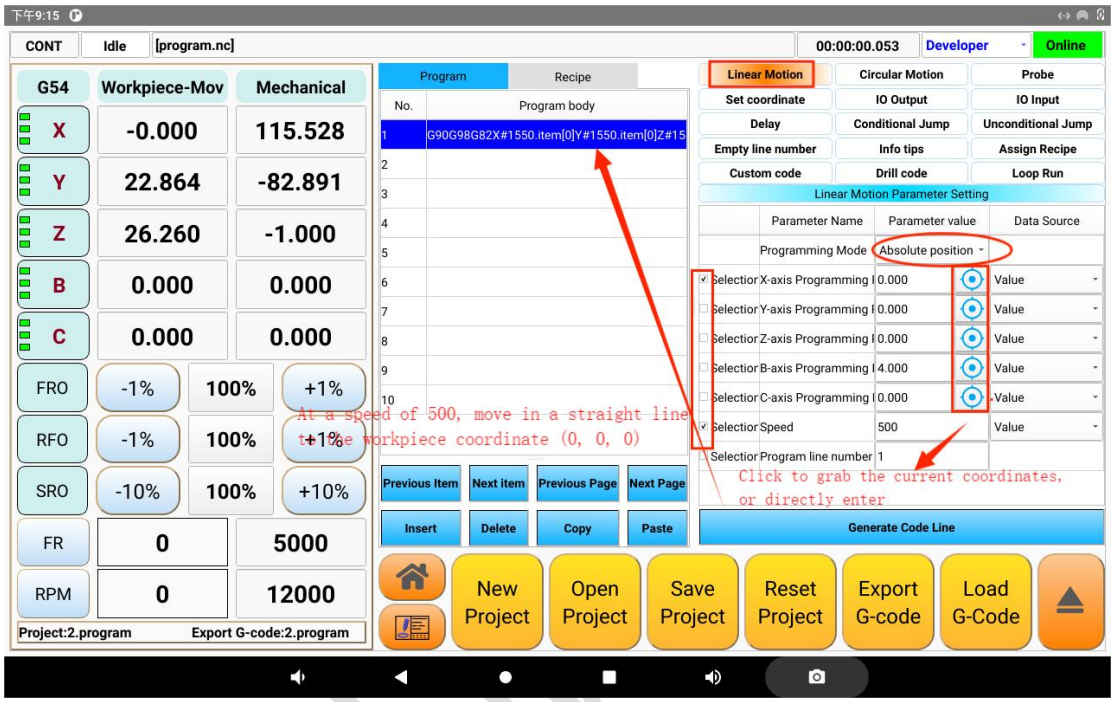
4. Directive introduction

1) Straight line motion instructions

- Move in a straight line to the specified position at the specified speed;
- When the data source selects a value, you can directly input or click the right icon to grab the current position.similarly hereinafter;
- It can move 4 axes at the same time (if there are 6 axes, you can choose 6

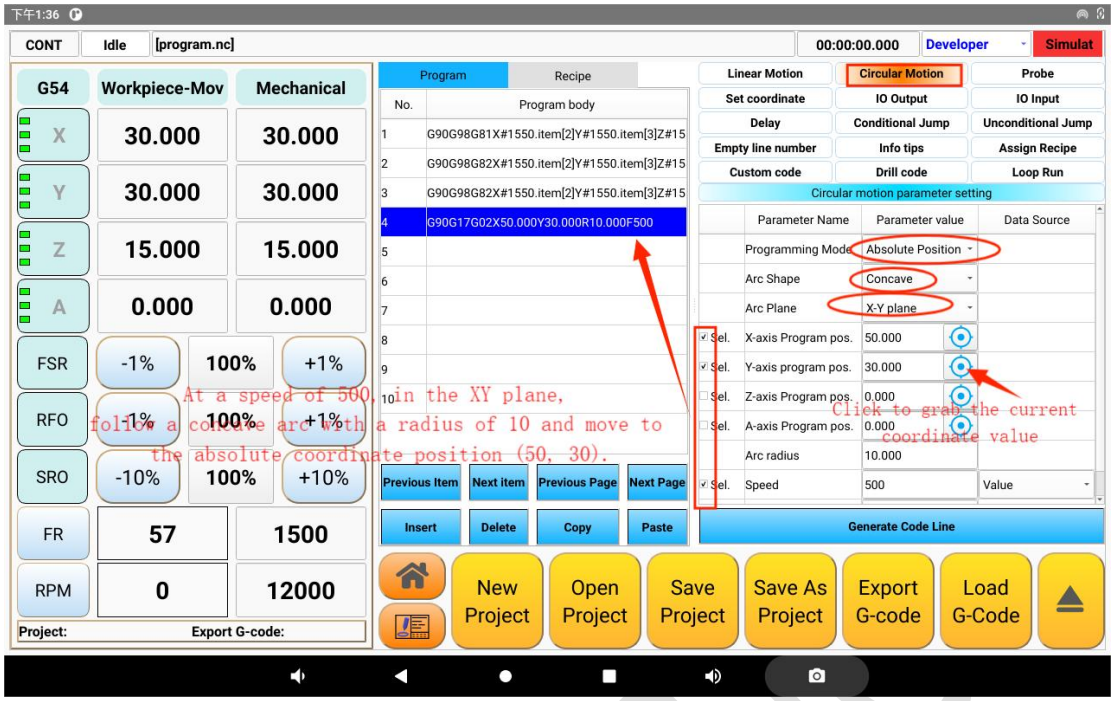
axes).similarly hereinafter;

- The absolute position, relative position, absolute position of mechanical parts;
- Here, axes 1 – 4 correspond to the XYZA axes, respectively.similarly hereinafter;
- The program line number is only set if the condition is required. After selection, Nxx (xx is line number) will be added before this instruction.similarly hereinafter;



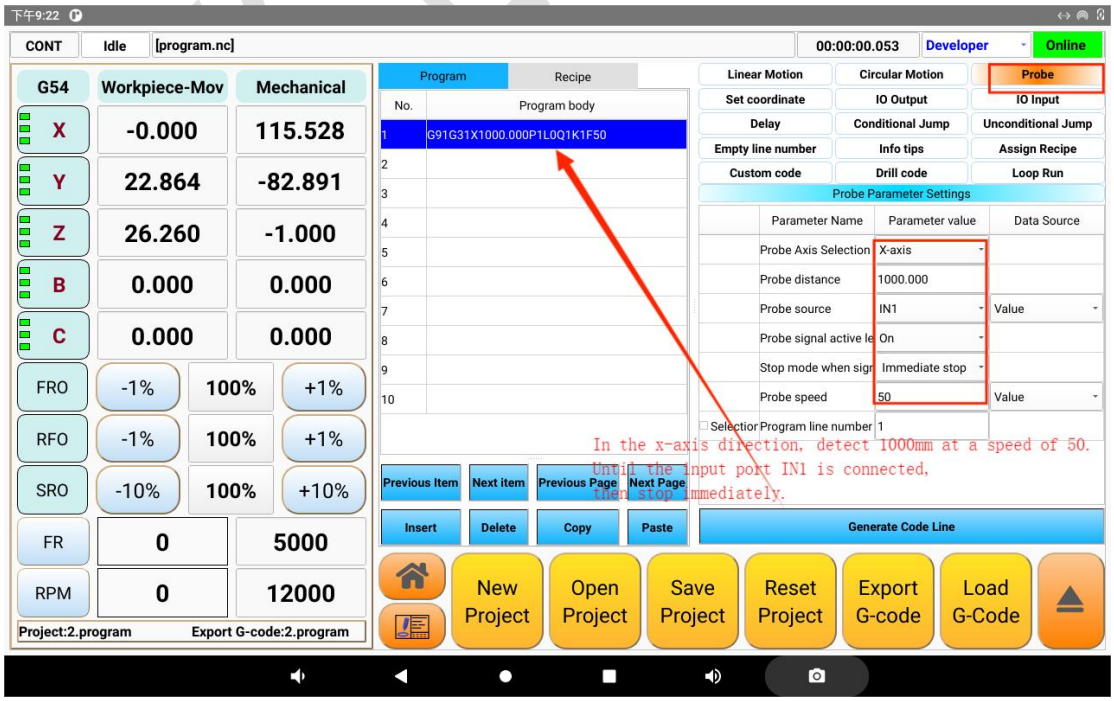
2) Circular arc motion instructions

- Draw the arc to move to the specified position at the specified speed;
- Programming mode: optional absolute position, relative position.
- Arc shape: optional convex arc, concave arc.
- Arc plane: optional xy, yz, xz plane.



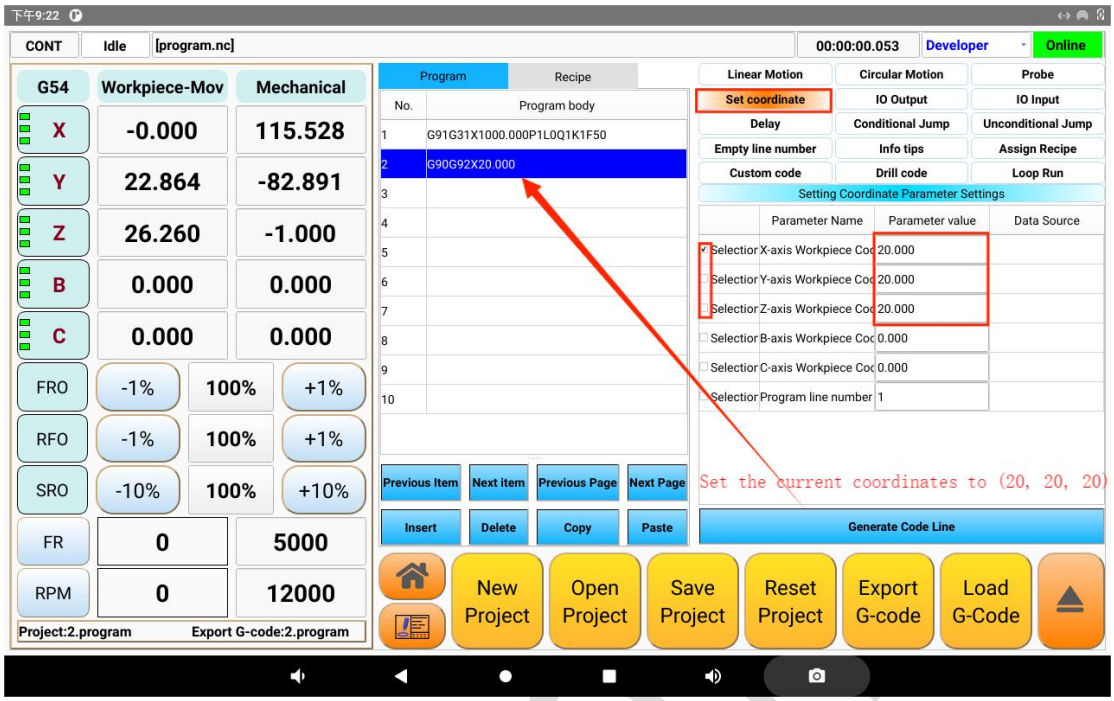
robe instructions

- Detection the signal in the selected axis direction at the specified speed;
- Probe distance can be clicked to edit, if unlimited detection is needed, can input a large enough distance;
- When the detection signal source is a numerical value, the input signal IN 1-24 can be selected by pull-down, corresponding to the input port number. The name can be changed in the parameter setting- -IO setting- -input IO customization;
- Stop mode when the signal is detected, optional immediately stop, deceleration stop;



3) Set the coordinate system instructions

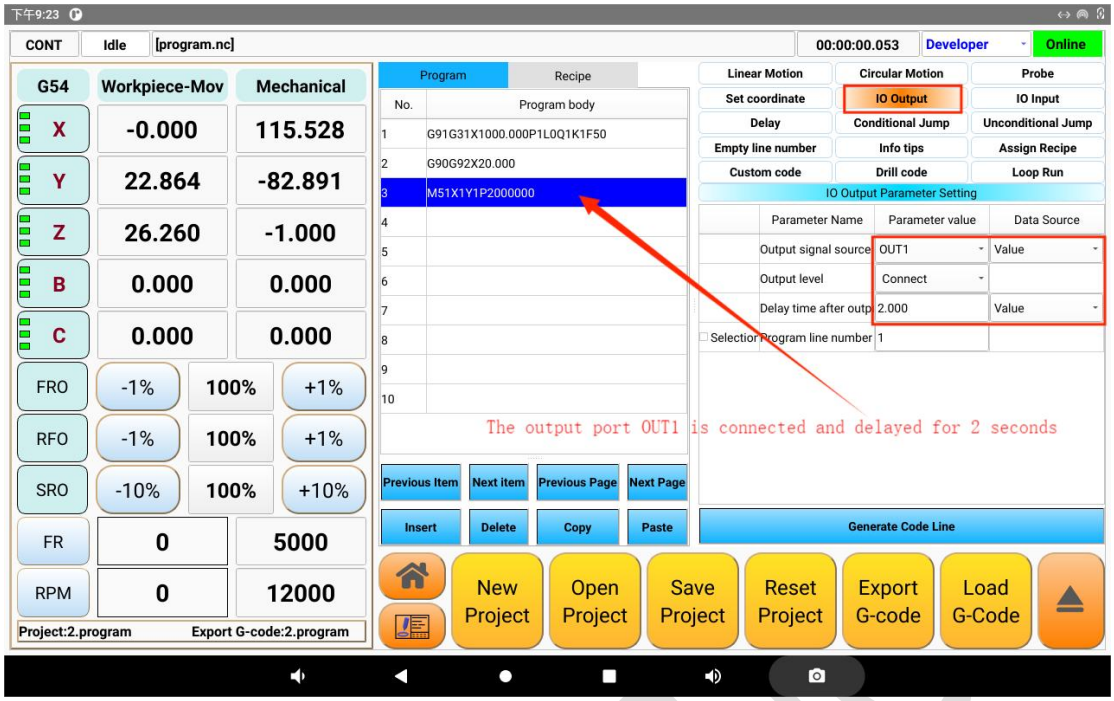
- Change the current work piece coordinates;



4) The IO outputs the integration instruction

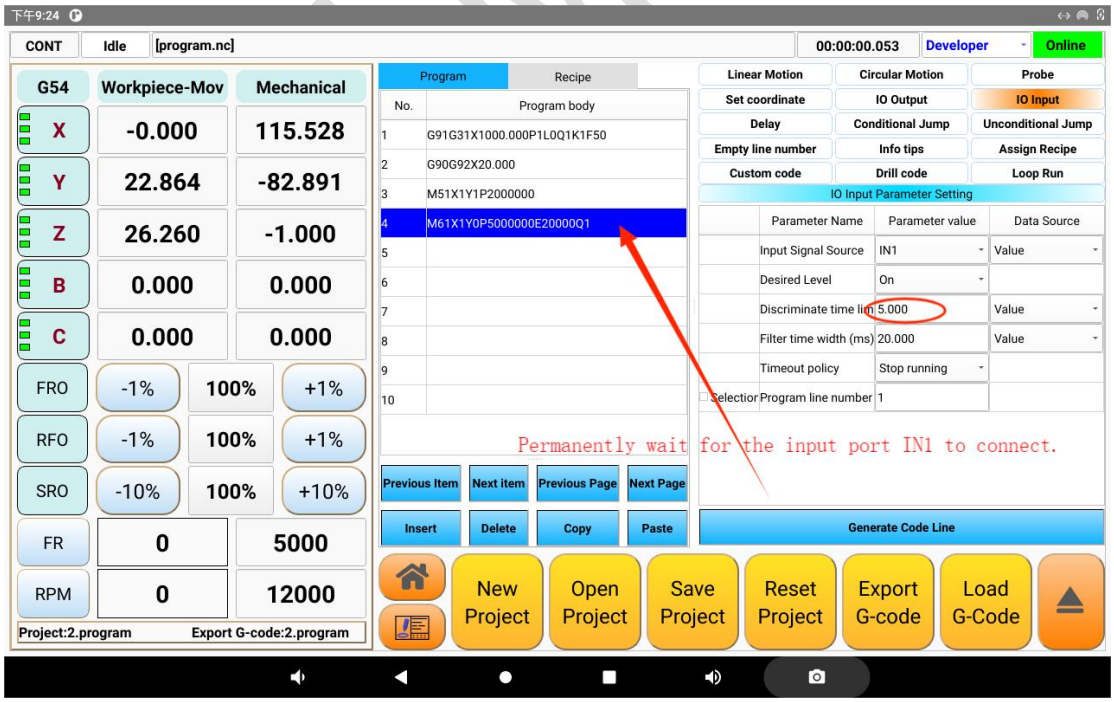
- The output signal source drop-down optional output port number, the name can be changed in the parameter setting- -IO setting- -output IO customization;

- Tip: The data source of the output signal source can choose the formula, and the signal name can be defined in the formula parameters for convenient identification. This way, when the port is replaced, only need to change the parameter value, do not need to modify the instruction.



5) IO input the integration instruction

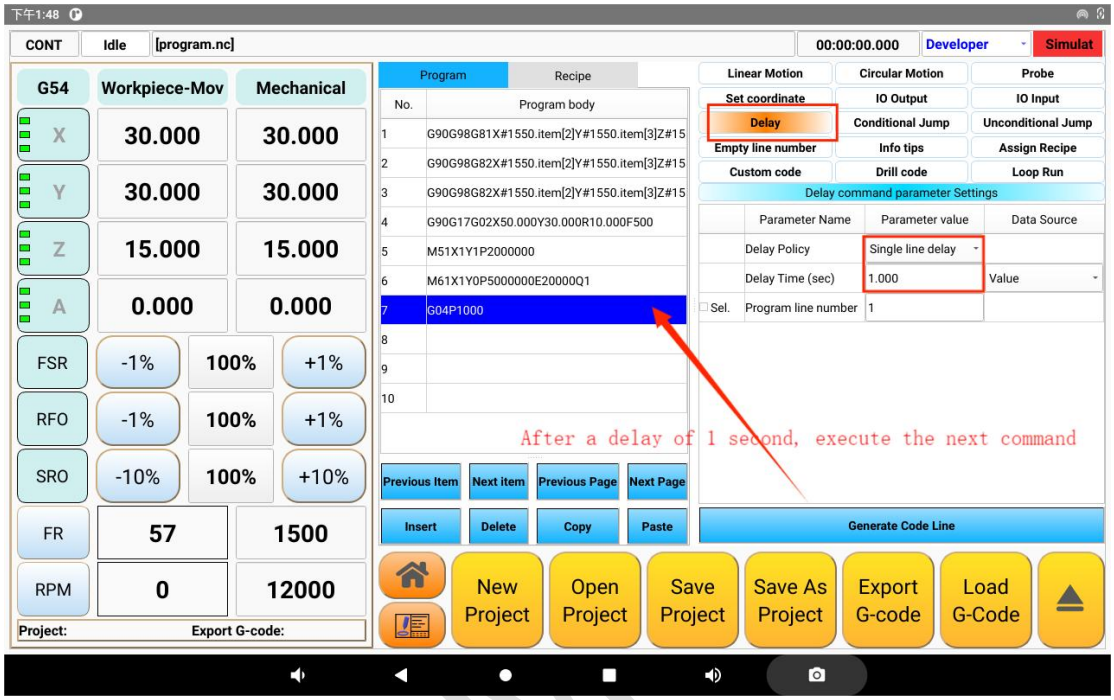
- Discrimination time: if 0 means permanent waiting, if 5, it is 5 seconds according to the timeout processing policy;
- Filter time width: generally according to the default 20 can be;
- Timeout processing policy: optional to continue and stop running;

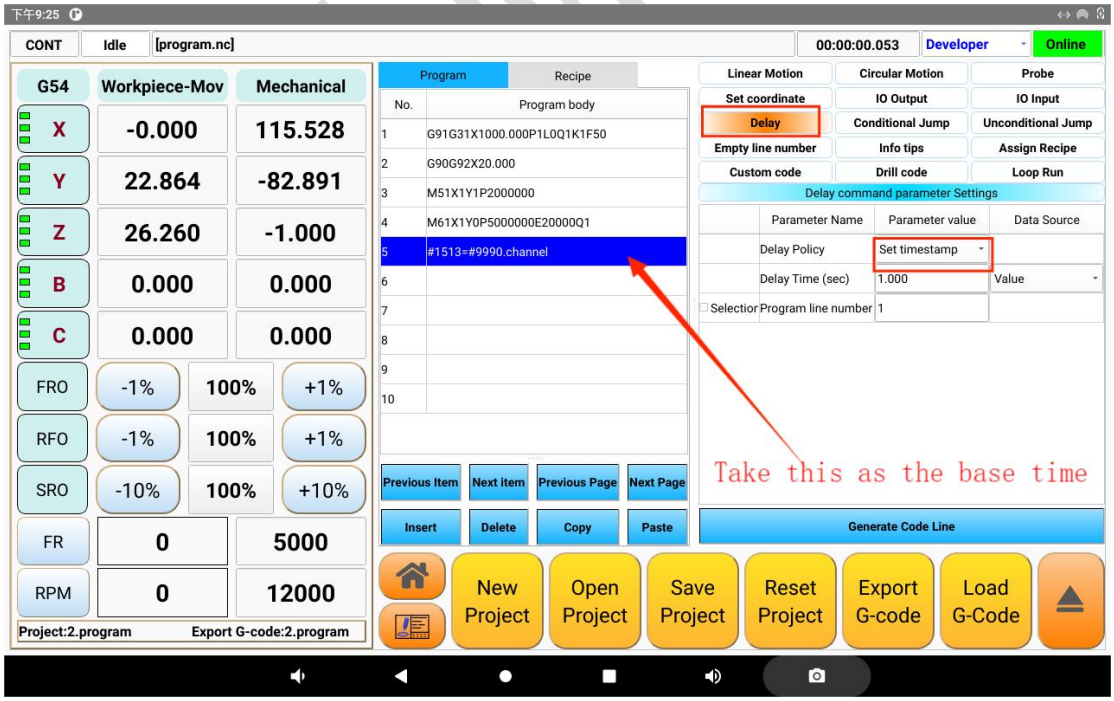


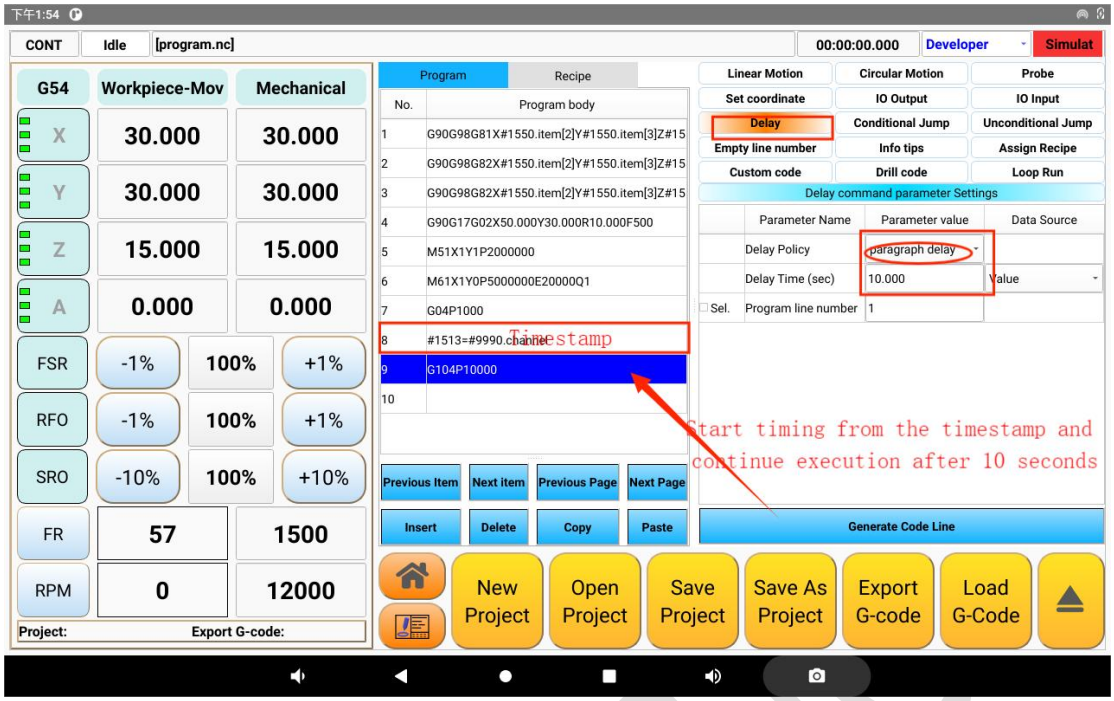
6) time-delay command

Generate a time-delay command. The time-delay policy is optional:

- Single-line delay: start the command line, and after the set time, continue the next instruction. Examples are shown below.
- Set time stamp: this instruction line time as the reference point, with paragraph delay. The delay time setting is invalid for this policy. Examples are shown below.
- Paragraph delay: The delay time is the timing time starting from the Set timestamp. Examples are shown below.

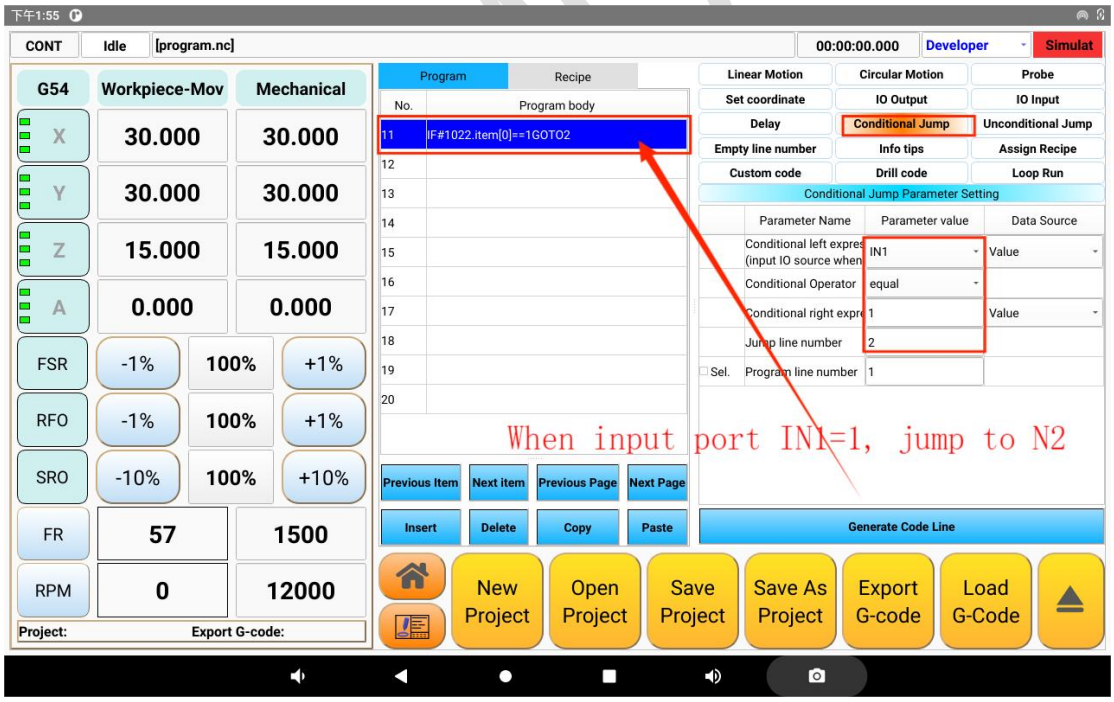






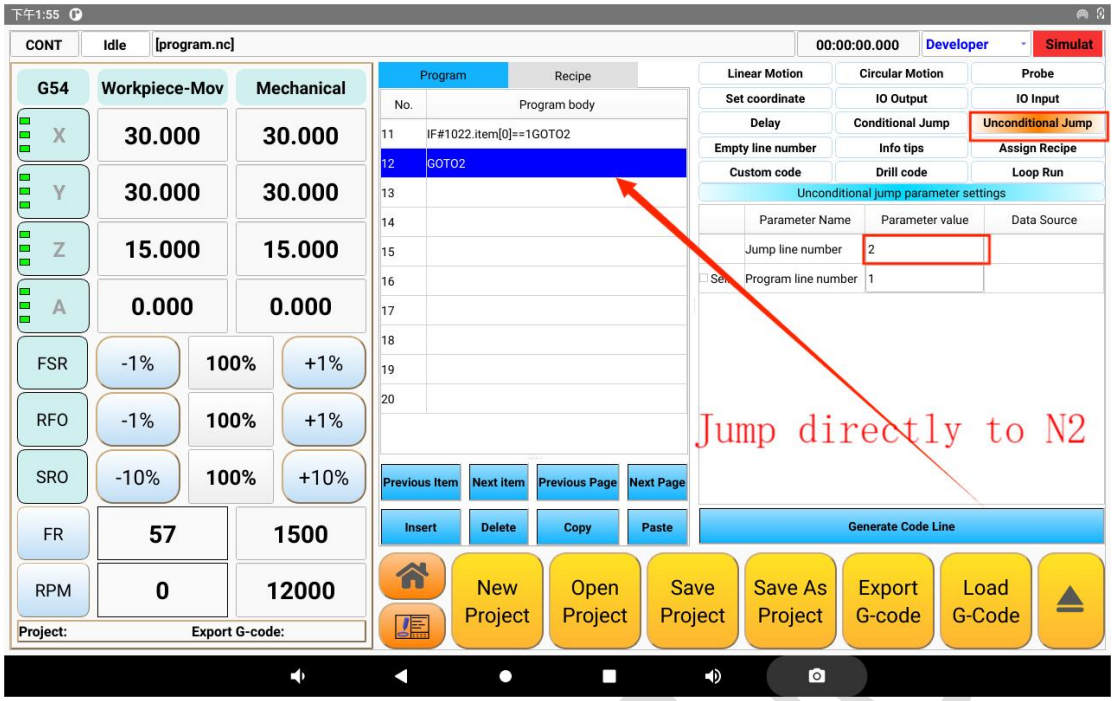
7) Conditional jump command

- When the condition is met, jump to the specified line; when the condition is not met, execute the following instructions successively.



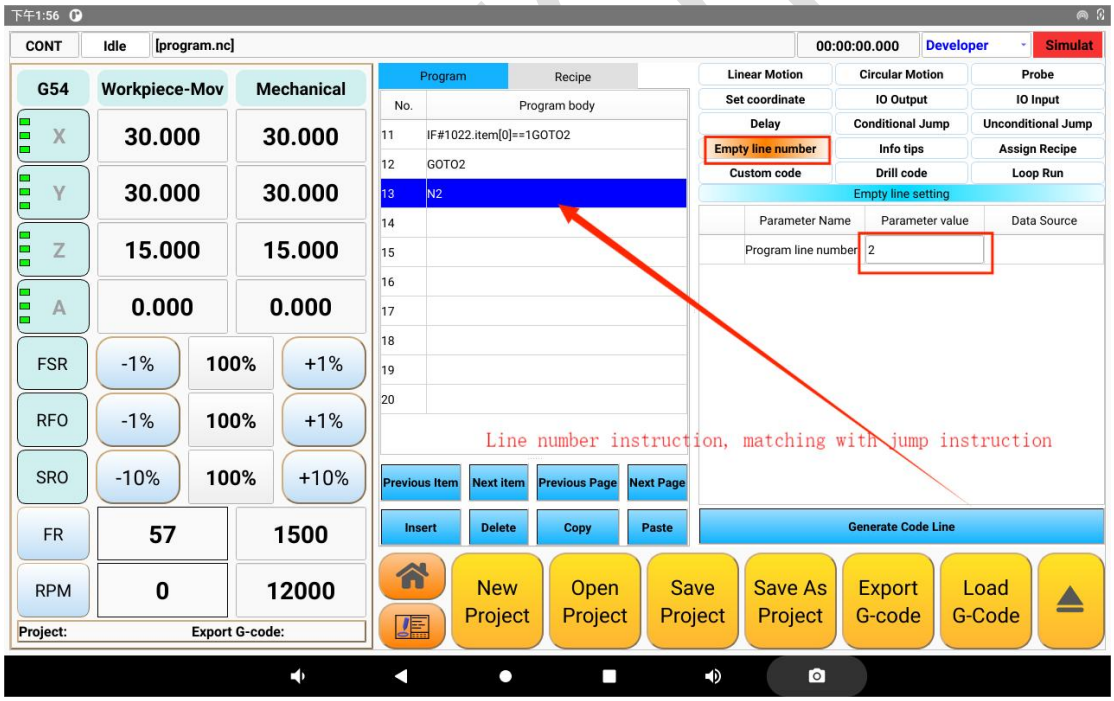
8) Unconditional jump instruction

Generate the GOTO instructions.



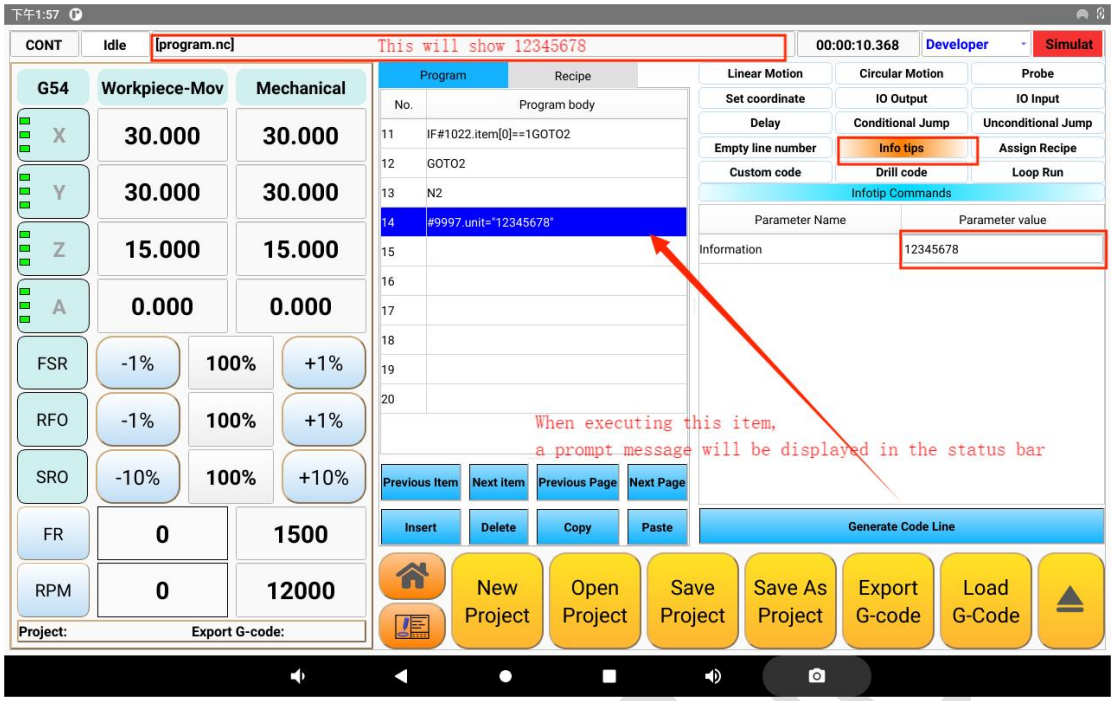
9) Empty line number

Generate jump change number command Nxx (xx is jump change number).



10) Information prompt instruction

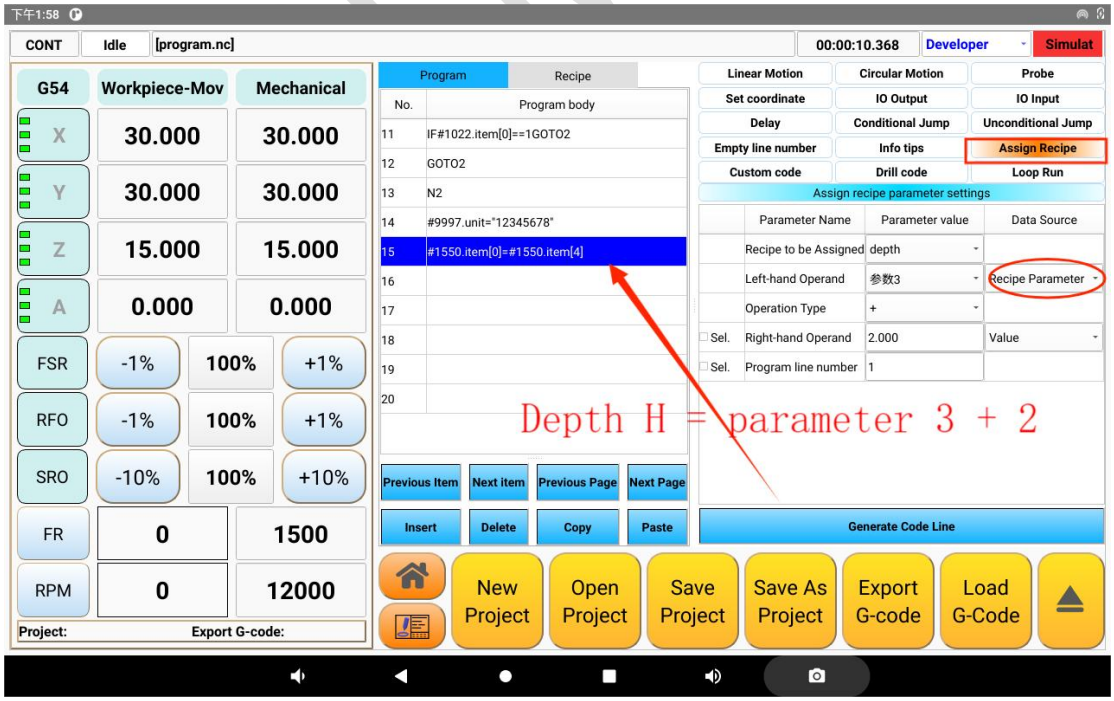
When you run this instruction, the message is prompted at the status bar.



11) Formula assignment instruction

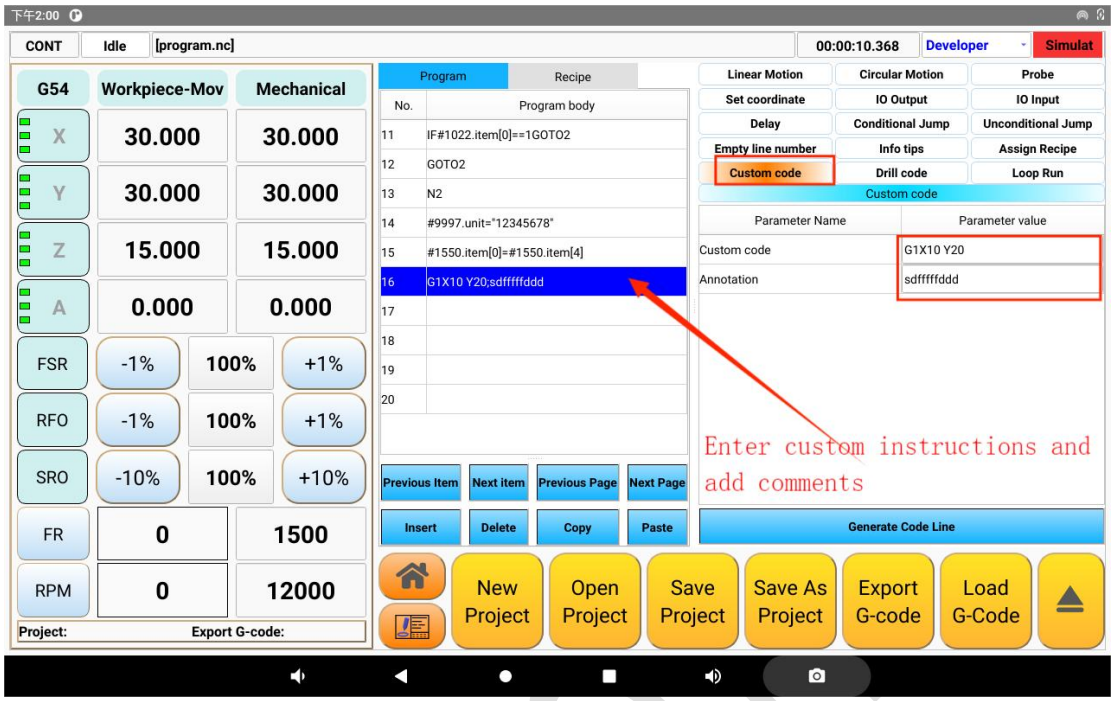
Parametric operations are introduced.

- Operation type: optional to add, subtract, multiply and divide.
- Data source: optional formula parameter index, numerical value, axis workpiece coordinates, axis mechanical coordinates.
- When selecting axis workpiece coordinates and axis mechanical coordinates, each axis can be selected.



12) Custom instructions

You can insert custom instructions and add comments.

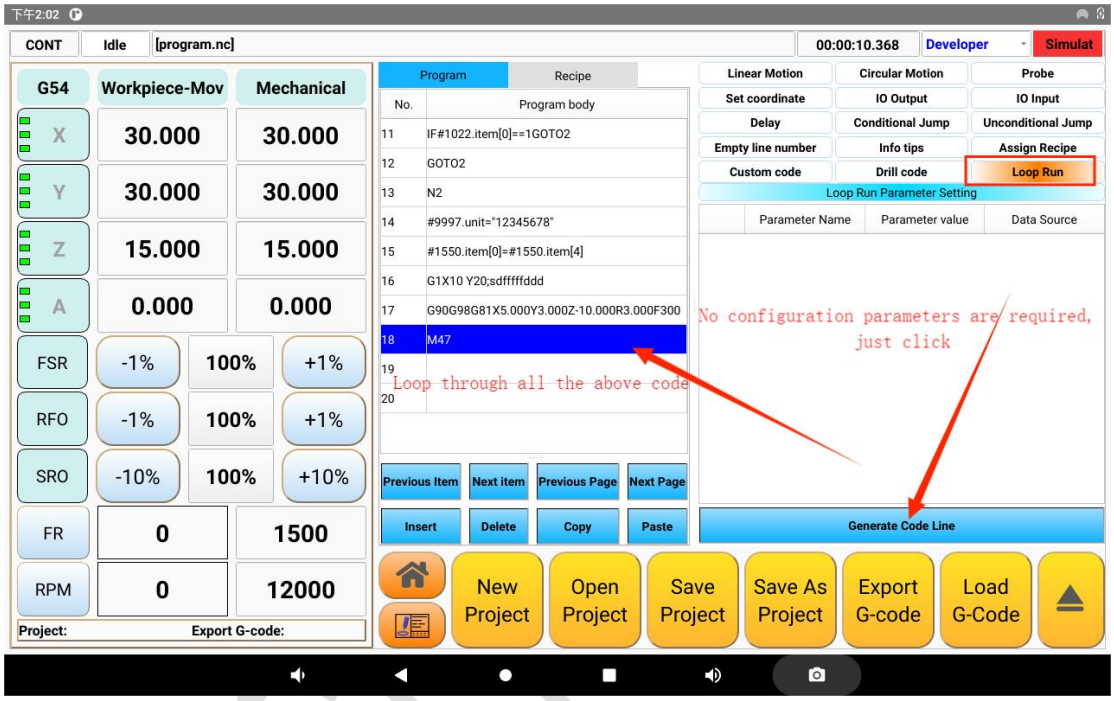


13) Drilling instructions

- Type: optional G81 / G82 / G83 / G73.
- Stay position when drilling is completed: optional initial plane, R plane.
- If the hole position XY coordinate is not checked, it is drilled in the current position.

14) Cycle running instructions

- Without setting content, directly click the generate code line to complete adding circular running instructions;
- Note that this instruction can only be used in the end, only recycling all the previous instructions;
- This instruction is an infinite loop instruction, if a limited number of cycle processing, do not need to cycle operation instruction, but point load G code, click program- -processing strategy-processing file set, can be matched with the number of cycles;

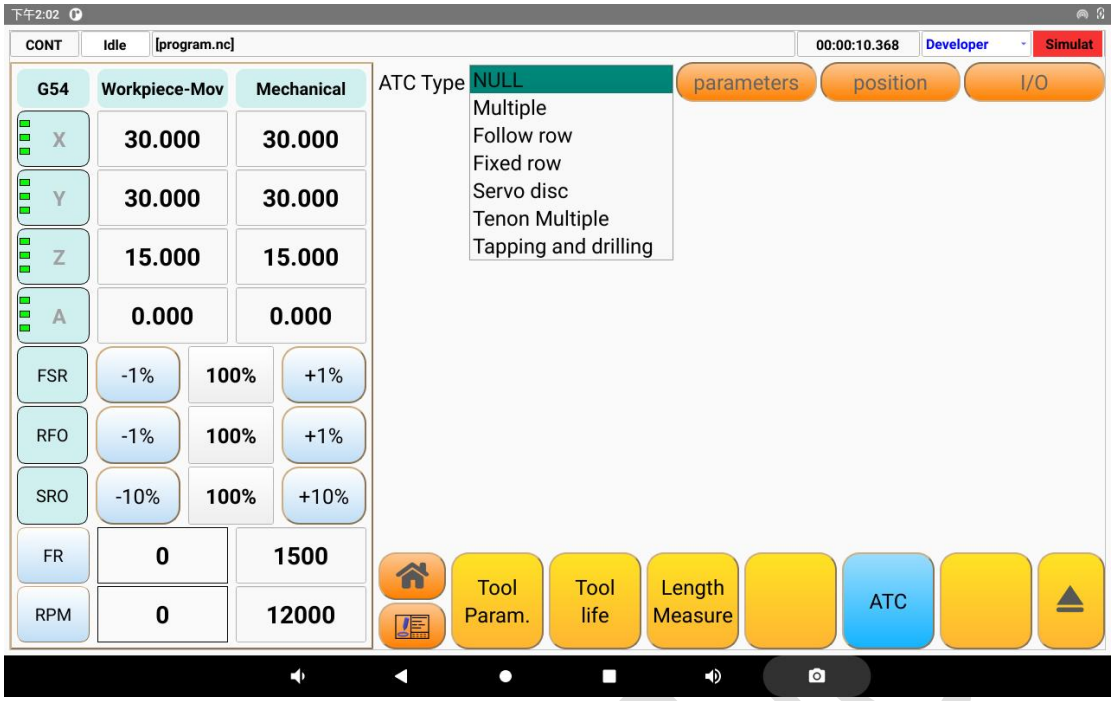


(十三) Tool management

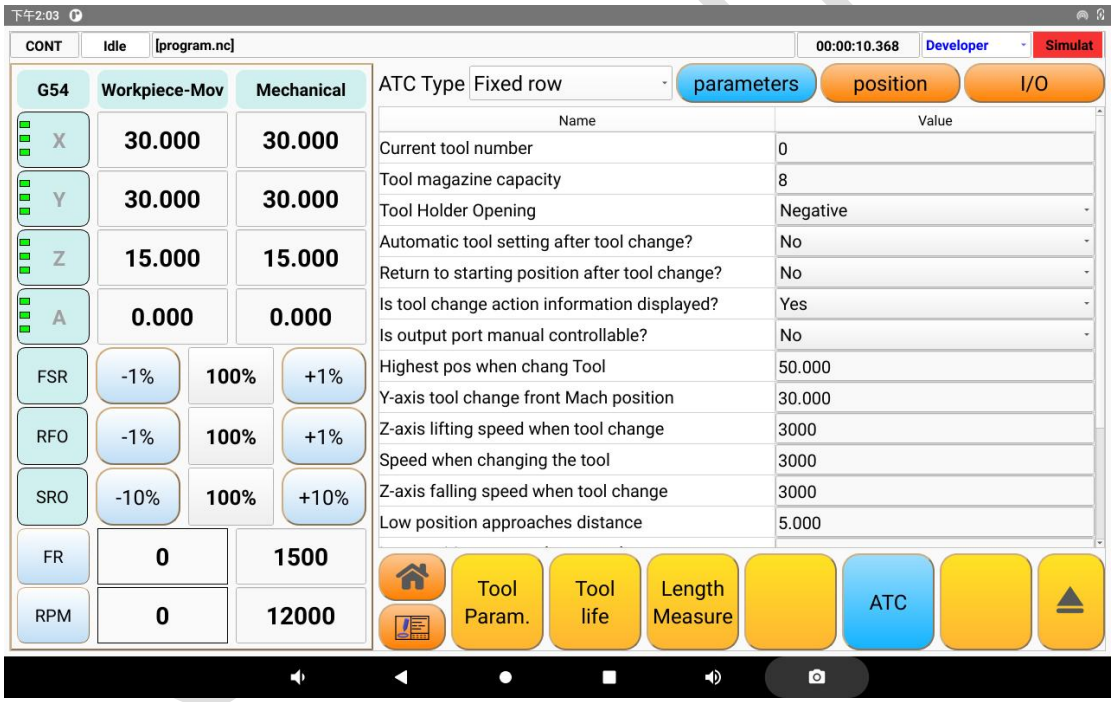
Provide multi-process, gantry straight row, fixed straight line, disc knife library, mortise and groove multi-process tool management.

1. Knife library configuration

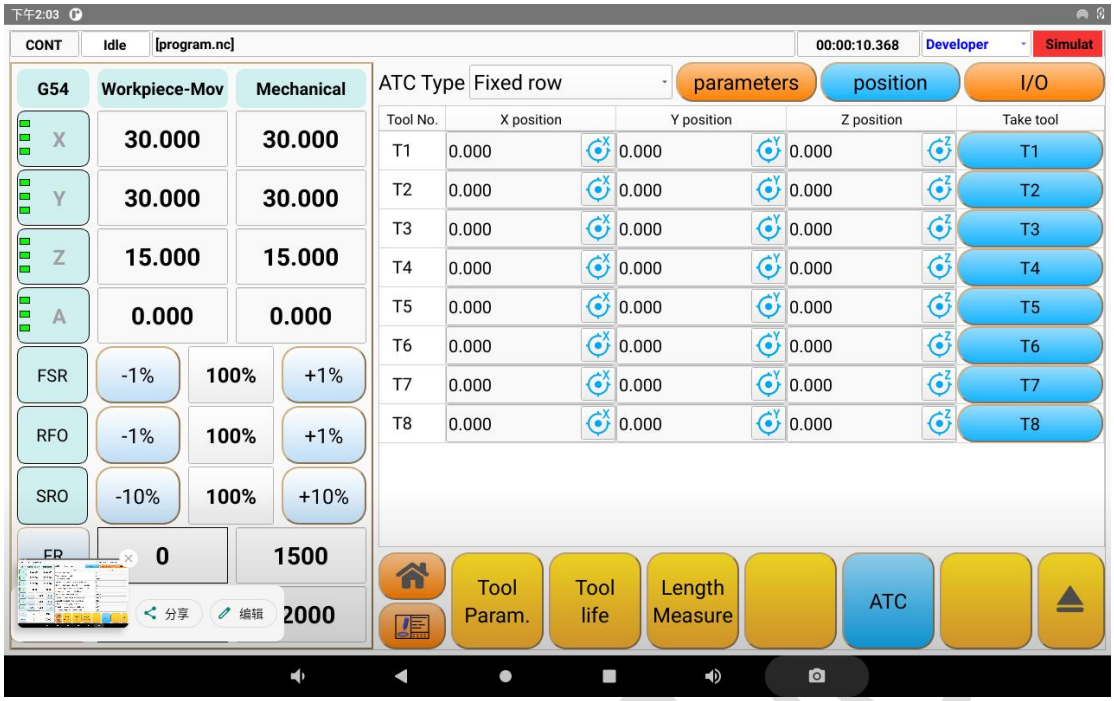
Administrator-above authority is required.



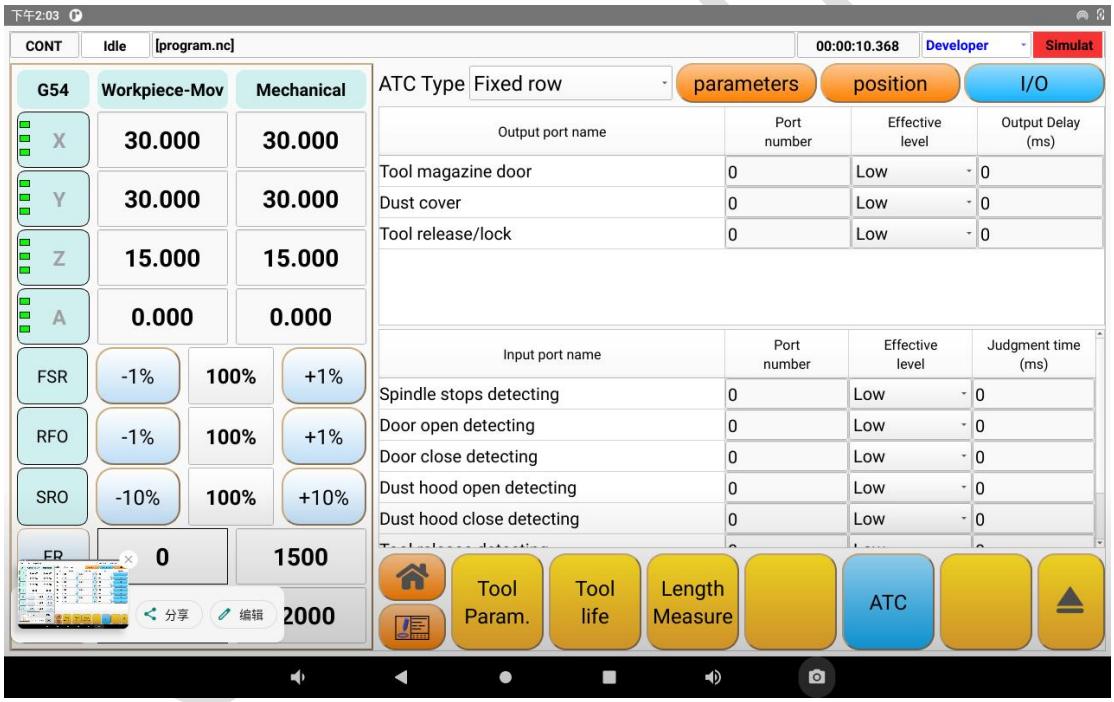
Knife library parameters



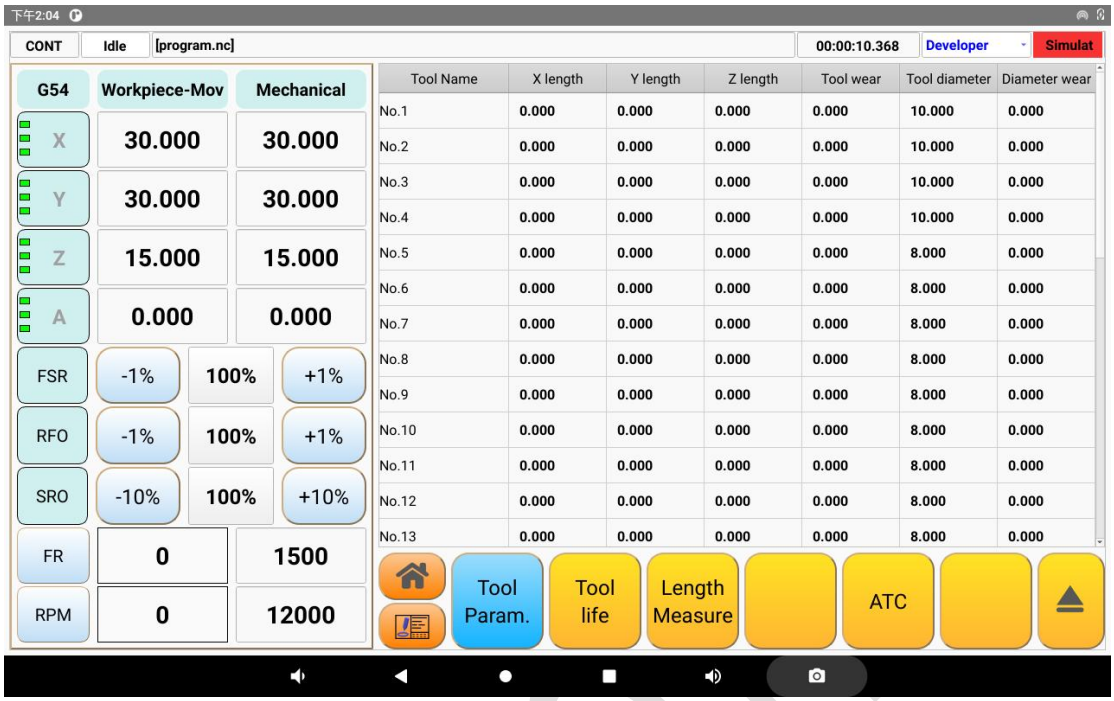
tool position



tool magazine I/O



2. Tool parameters

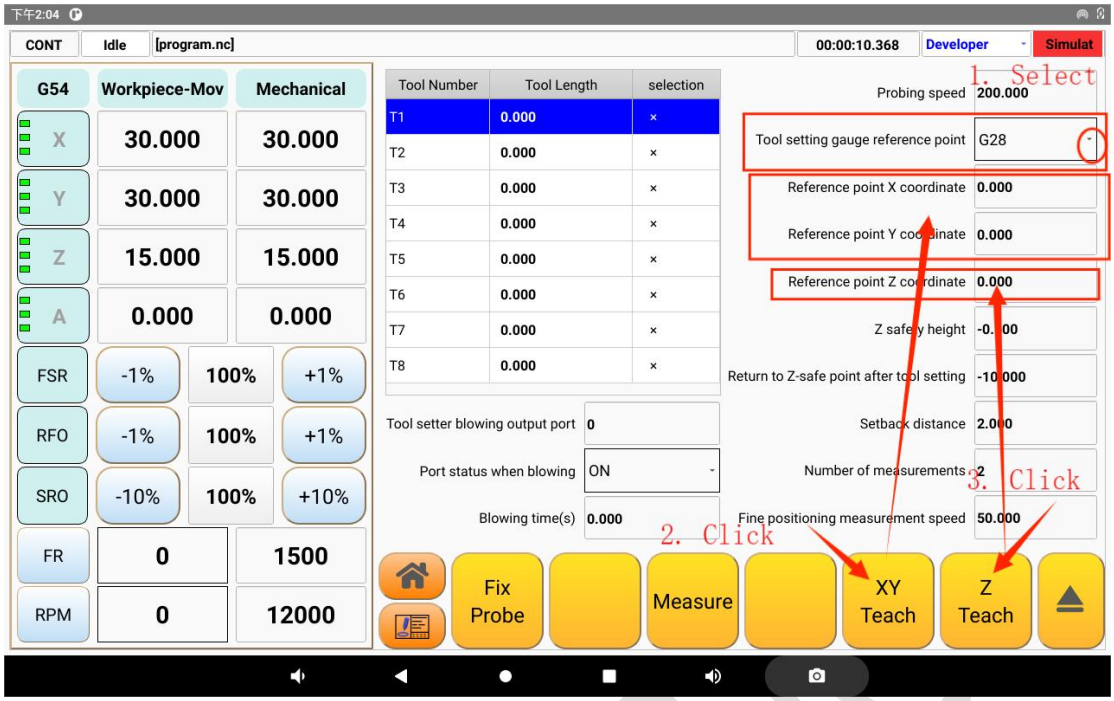


3. life of cutting tool

Zero clearance requires the administrator above permission.

4. The length of the knife measurement

The coordinates here are all the machine tool coordinates.



1) XY teaching, Z teaching

Grasp the reference point and tool:

- First, select the reference point of the knife instrument;
- Move the tool to the knife point above the knife instrument;
- Click XY instruction, Z instruction, will automatically grasp the coordinate value of the reference point of the knife instrument;
  Note: The coordinates of the knife instrument reference point can also be set in the axis parameter —— reference point.

2) *Instructions for fixing knife:*

- Because the fixed knife instrument is installed in the fixed position of the machine tool, it is necessary to complete the X / Y / Z axis mechanical zero operation before fixing the knife;
- Set the knife signal source correctly (—— IO setting —— input port allocation —— knife signal) and check whether the detection signal is correct;
- Set the parameters and check whether the relevant parameters are correct;
- Click the fixed knife and the machine moves as follows:
  - The Z-axis is raised to the Z-safety height;
  - Move the X / Y axis to the reference point XY position (above the knife block);
  - Move the Z axis to the reference point Z (fixed knife point);
  - The Z-axis goes down at the measured speed, and when receiving the detection signal, it falls back according to the fallback distance;
  - Then according to the speed of precision measurement for detection, the number of times according to the number of precision measurement value;
  - Write the machine coordinates of the detection point to the Z-axis tool

offset;

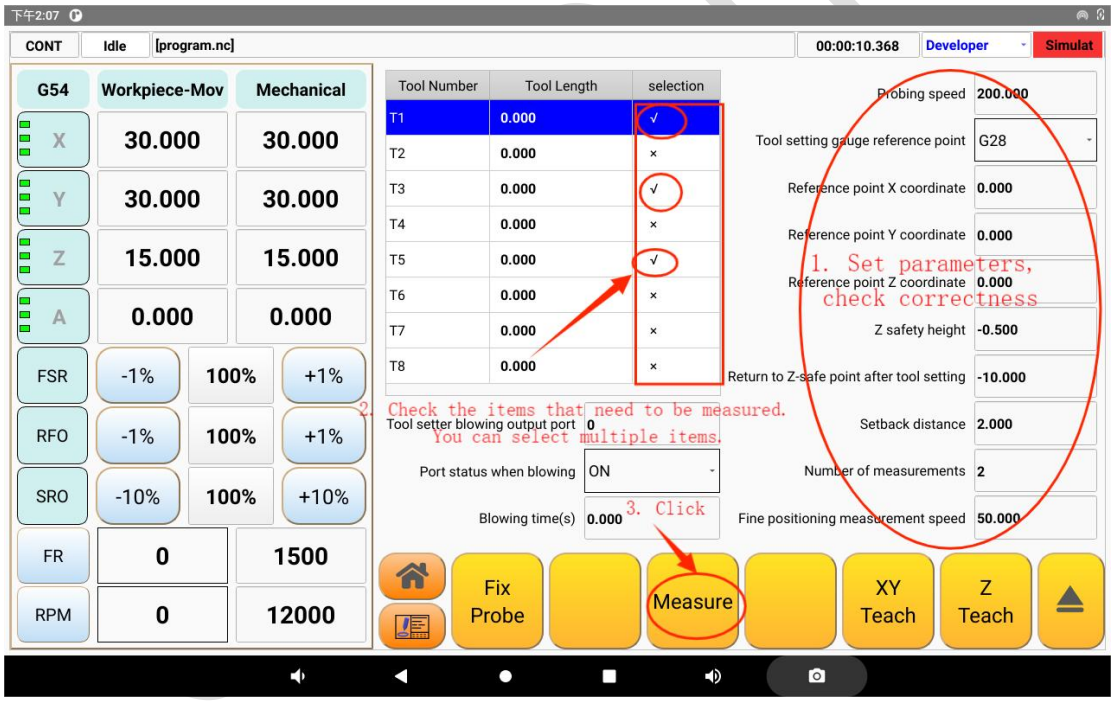
- After the Z axis steps back to the knife, return to the Z safety point, and fix the knife.

pay attention to:

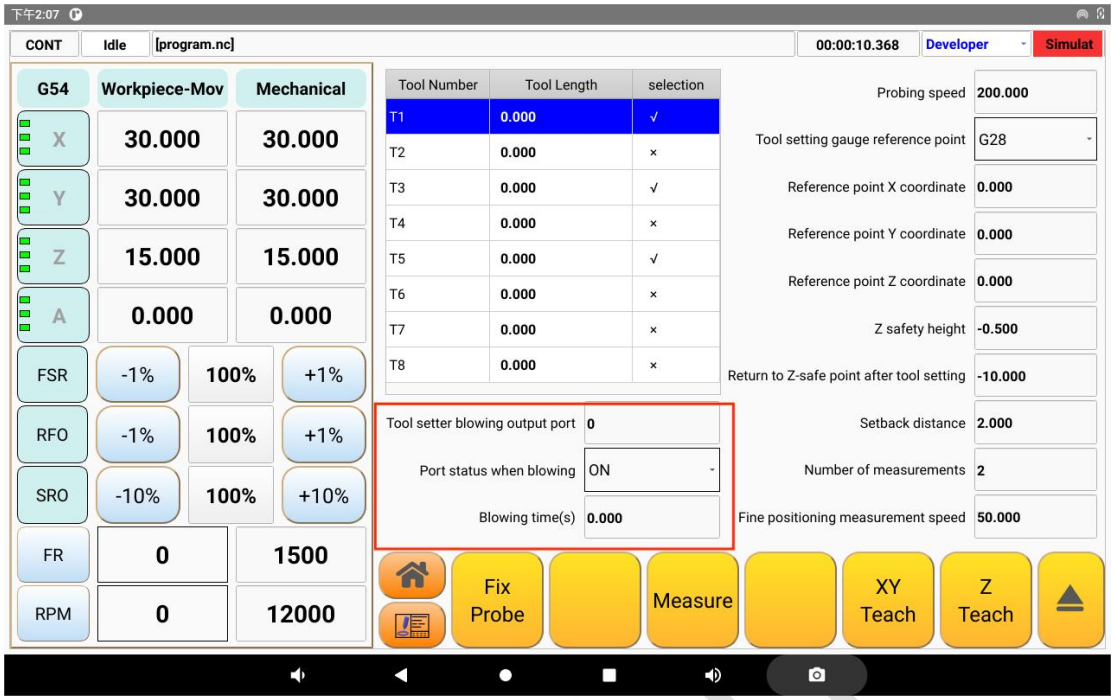
- When the first use of the product or the position of the tool changes, it is necessary to fix the knife (the first knife);
- In the case of no knife library, after replacing the tool, it is necessary to fix the knife again;
- In the case of a knife library, after the knife is fixed, the length of the knife needs to be measured for each knife (one key to the knife). When one of the knives is replaced, we only need to measure the length of the replaced knife again, and there is no need to be fixed again.

3) *One-click to knife operation instructions:*

- It is for the knife library, which completes the one-click measurement.
- Also check whether the relevant parameters are correct before use.
- One-button knife is the calculation of tool length compensation by using the fixed knife detection result [Z-axis tool bias];



4) Blow port on the knife



(十四) parameter setting

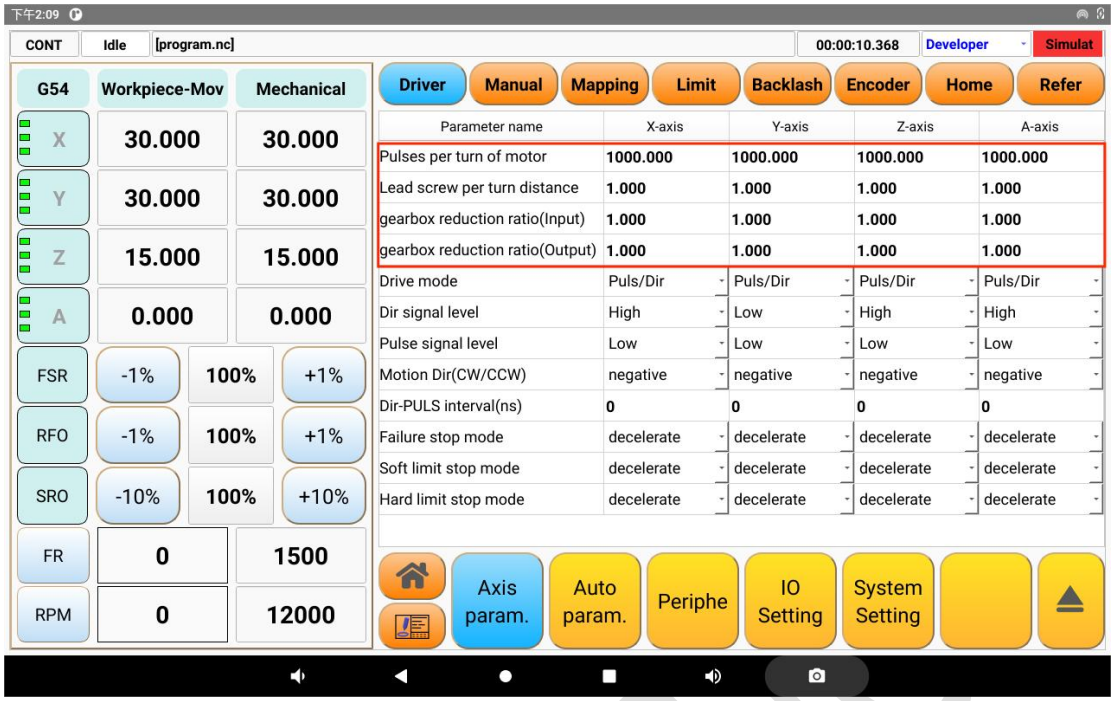
1. Axis parameters

Display only under Administrator and developer rights.

1) drive

a) **Pulse-equivalent-related parameters:**

The first 4 settings are all parameters related to the pulse equivalent.



b) **Axis output mode**

Optional pulse direction / double pulse 2 ways.

i. Pulse direction

One is a direction output signal indicating the motion direction; the other is a pulse output signal, the pulse waveform related to the motion parameters.

When selecting the "pulse direction" mode, the correlation parameters are "direction signal level", "pulse signal level" and "pulse direction delay". The relationship is shown in the figure below.

- **The Orientation signal level:**

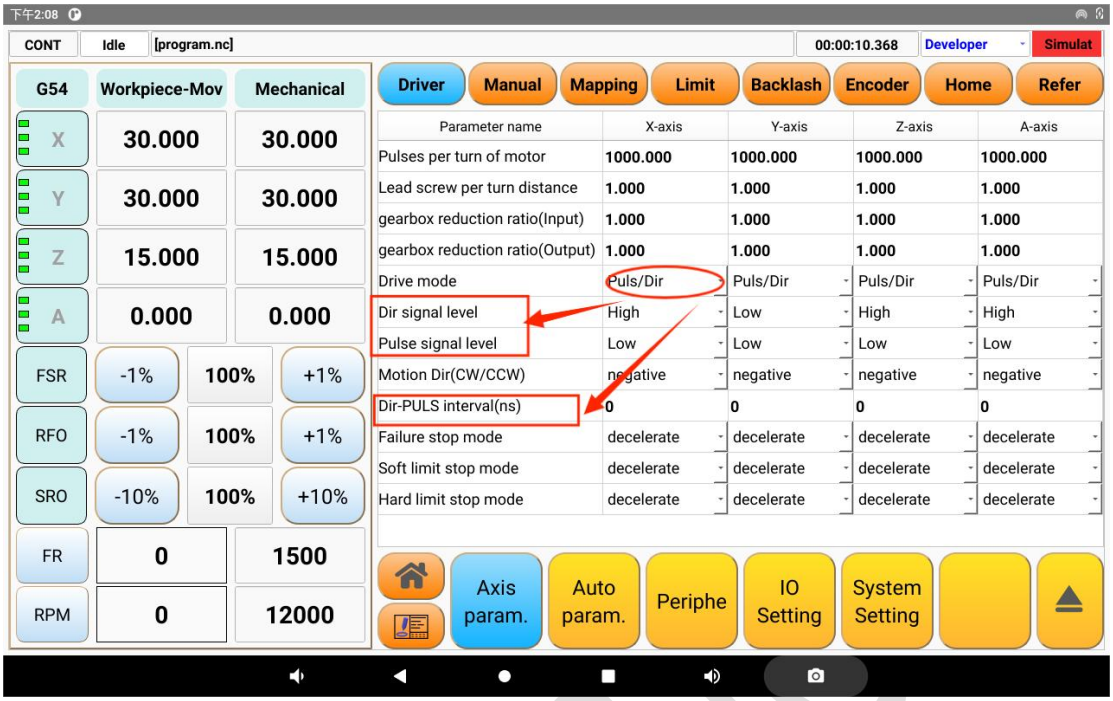
  In the process of machine tool debugging, it is necessary to determine the positive direction of each axis according to the coordinate system of the right-hand rule. After determining the positive direction of each axis according to the right hand rule, operate the machine movement manually to determine whether the axis movement is correct. If the axis movement is found in the opposite direction, the definition of the "direction signal level" level can be changed to change the direction.

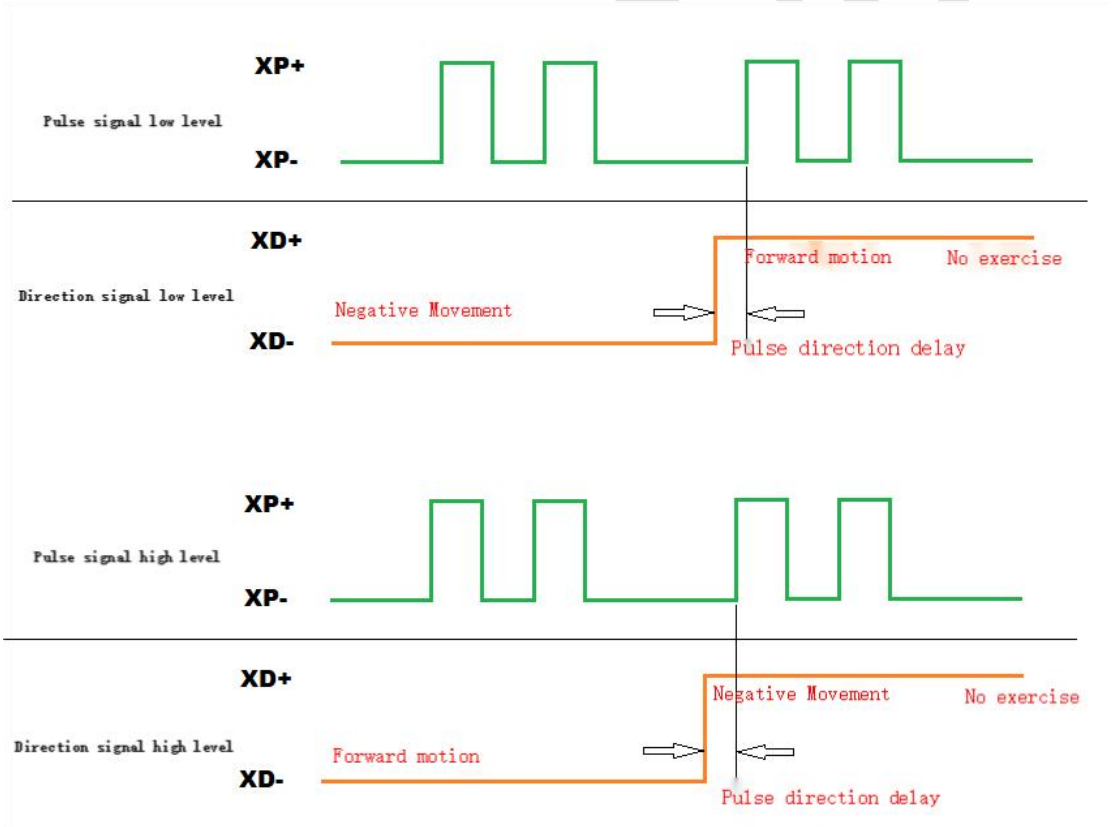
- **"Pulse signal level"**

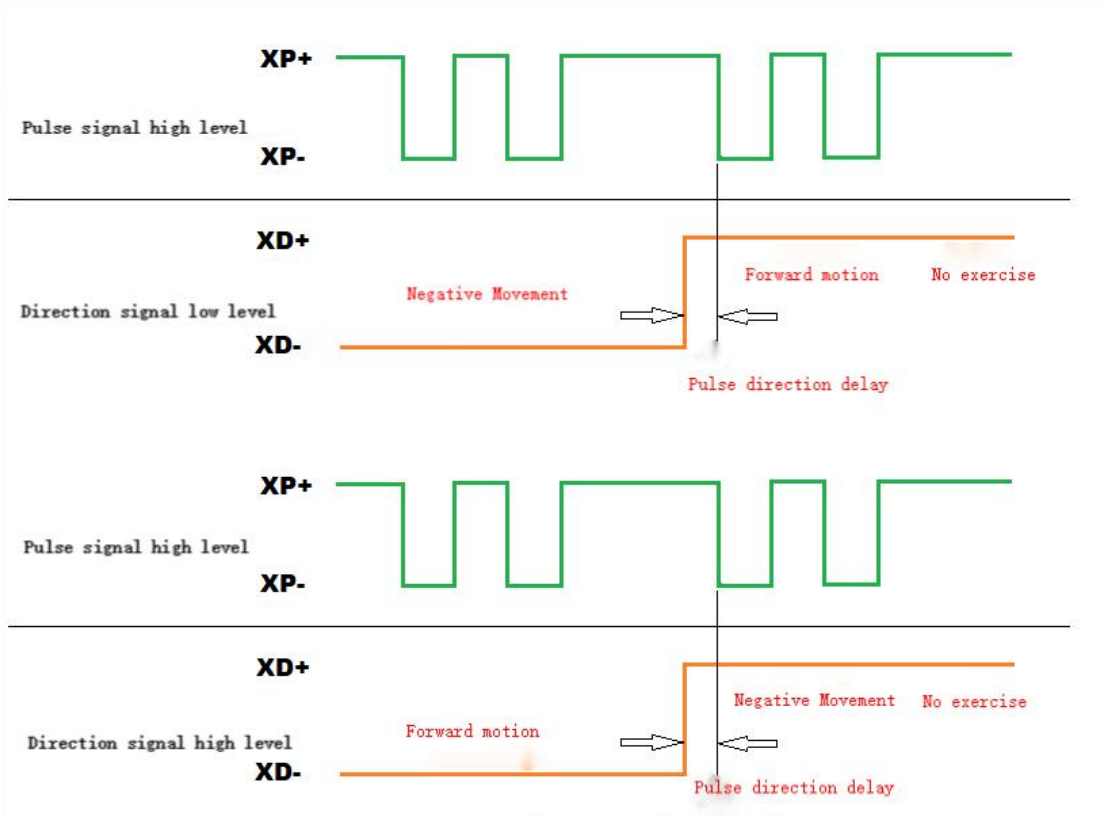
  High level refers to high level when there is no pulse input, and low level refers to low level when there is no pulse input. Default low level.

- **"Pulse direction delay"**

  In the changing direction, the working principle of the pulse direction mode is to receive the change of the direction signal level first, and then receive the pulse signal, which may have a delay. If the two signals are synchronized, the loss of part of the pulse signal may be caused. By setting this delay, this situation can be
  avoided


CONT
Idle
[program.nc]
00:00:10.368
Developer
Simulat
G54
Workpiece-Mov
Mechanical
X
30.000
30.000
Y
30.000
30.000
Z
15.000
15.000
A
0.000
0.000
FSR
-1%
100%
+1%
RFO
-1%
100%
+1%
SRO
-10%
100%
+10%
FR
0
1500
RPM
0
12000
Driver
Manual
Mapping
Limit
Backlash
Encoder
Home
Refer
Parameter name
X-axis
Y-axis
Z-axis
A-axis
Pulses per turn of motor
1000.000
Lead screw per turn distance
1.000
gearbox reduction ratio(Input)
gearbox reduction ratio(Output)
Drive mode
Puls/Dir
Dir signal level
High
Low
Pulse signal level
Motion Dir(CW/CCW)
negative
Dir-PULS interval(ns)
0
Failure stop mode
decelerate
Soft limit stop mode
Hard limit stop mode
Axis param.
Auto param.
Periphe
IO Setting
System Setting


XP+
Pulse signal low level
XP-
XD+
Direction signal low level
XD-
Forward motion
No exercise
Negative Movement
Pulse direction delay
XP+
Pulse signal high level
XP-
XD+
Direction signal high level
XD-
Negative Movement
No exercise
Forward motion
Pulse direction delay



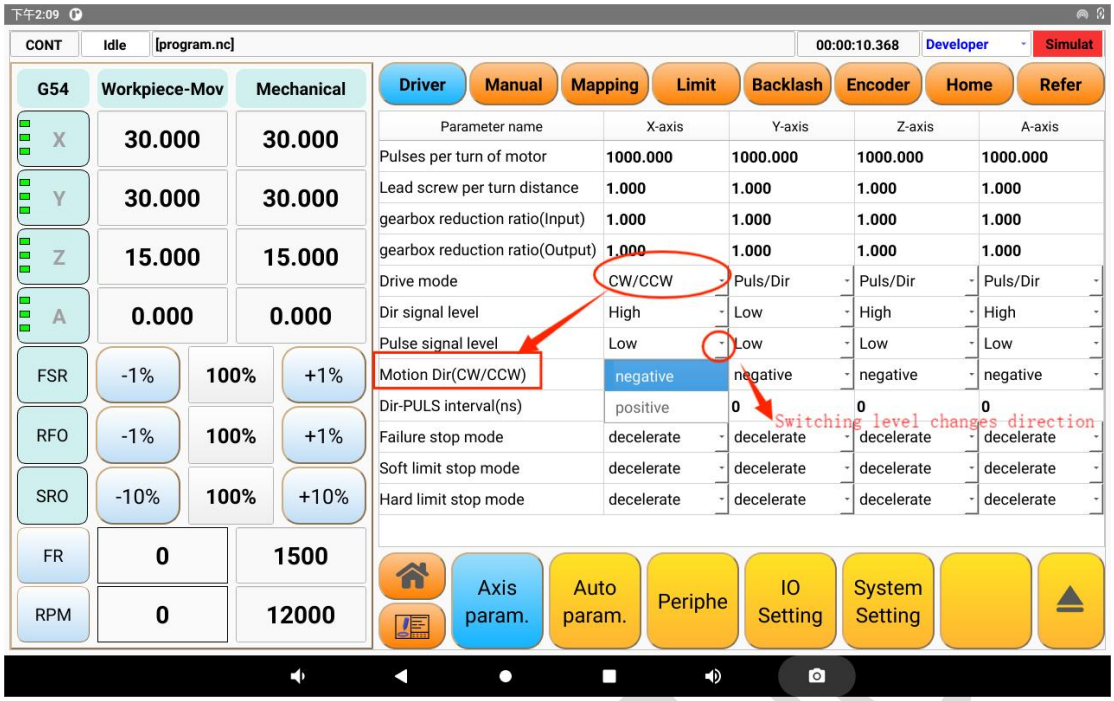
ii. dipulse

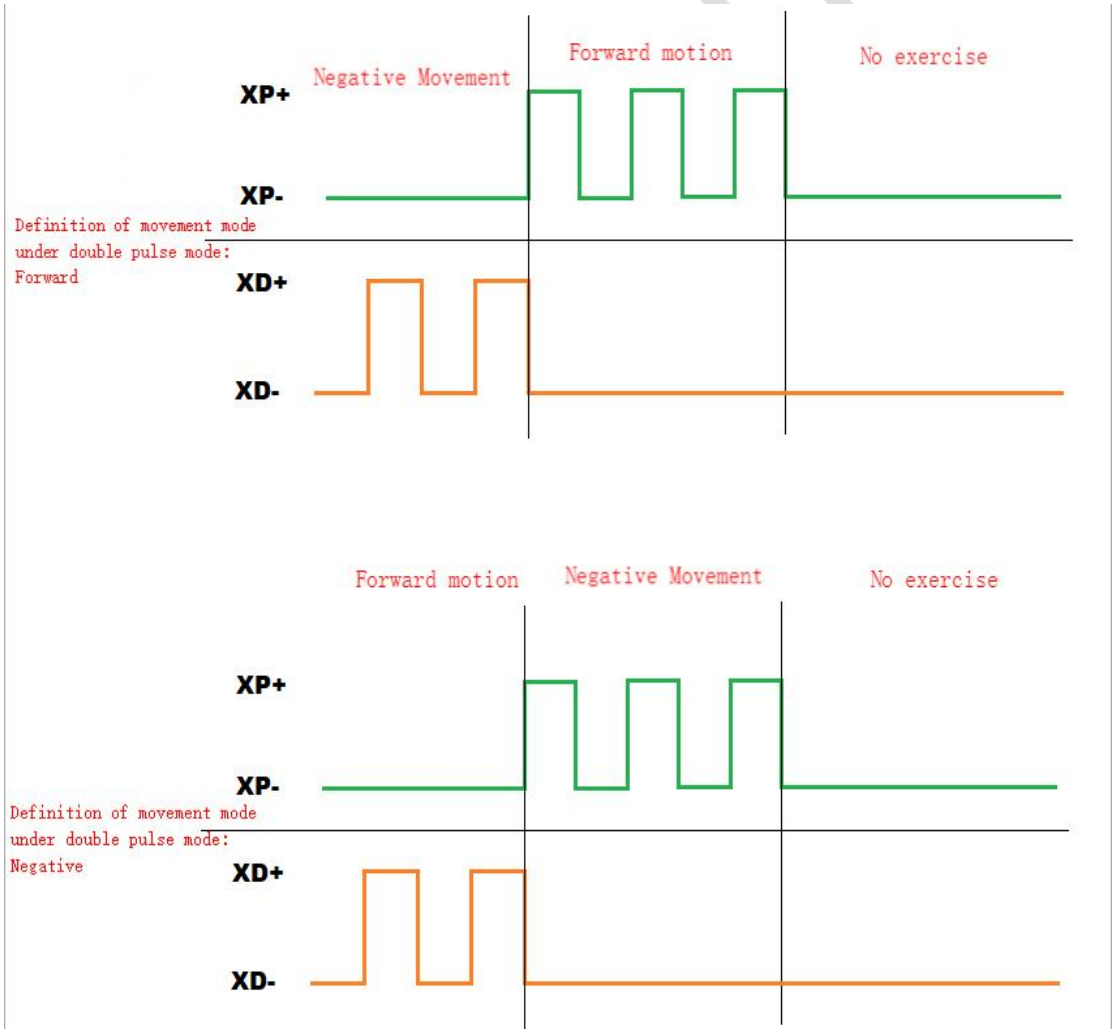
Double pulse refers to both pulse output and direction output, but the one is positive and the other is negative.

When the "double pulse" mode is selected, the correlation parameter "Definition of motion direction under the double pulse mode". The principle is shown in the figure below.

- **Definition of motion direction in dual-pulse mode:**

  When the machine moves manually, if the shaft movement direction is opposite to the desired direction, it can be changed.


CONT
Idle
[program.nc]
00:00:10.368
Developer
Simulat
G54
Workpiece-Mov
Mechanical
X
30.000
30.000
Y
30.000
30.000
Z
15.000
15.000
A
0.000
0.000
FSR
-1%
100%
+1%
RFO
-1%
100%
+1%
SRO
-10%
100%
+10%
FR
0
1500
RPM
0
12000
Driver
Manual
Mapping
Limit
Backlash
Encoder
Home
Refer
Parameter name
X-axis
Y-axis
Z-axis
A-axis
Pulses per turn of motor
1000.000
1000.000
1000.000
1000.000
Lead screw per turn distance
1.000
1.000
1.000
1.000
gearbox reduction ratio(Input)
1.000
1.000
1.000
1.000
gearbox reduction ratio(Output)
1.000
1.000
1.000
1.000
Drive mode
CW/CCW
Puls/Dir
Puls/Dir
Puls/Dir
Dir signal level
High
Low
High
High
Pulse signal level
Low
Low
Low
Low
Motion Dir(CW/CCW)
negative
negative
negative
negative
Dir-PULS interval(ns)
positive
0
0
0
Failure stop mode
decelerate
decelerate
decelerate
decelerate
Soft limit stop mode
decelerate
decelerate
decelerate
decelerate
Hard limit stop mode
decelerate
decelerate
decelerate
decelerate
Switching level changes direction
Axis param.
Auto param.
Periphe
IO Setting
System Setting


Negative Movement
Forward motion
No exercise
XP+
XP-
Definition of movement mode
under double pulse mode:
Forward
XD+
XD-
Forward motion
Negative Movement
No exercise
XP+
XP-
Definition of movement mode
under double pulse mode:
Negative
XD+
XD-

Stop mode only in manual mode. In automatic mode, stop by "fault drop rate" in automatic parameter.

c) **Failure stop mode**

It refers to the stop mode used when the fault situation (external emergency stop, hand wheel emergency stop, reset, servo alarm), you can choose the deceleration stop / immediately stop.

d) **Soft limit, hard limit to stop mode**

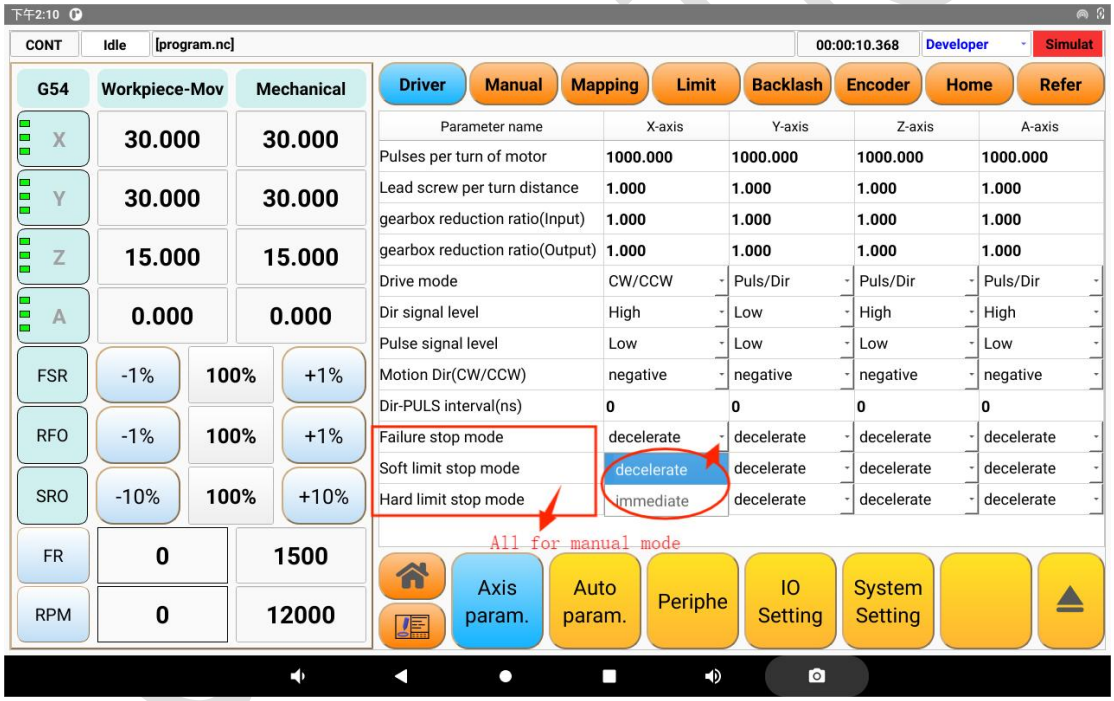
It refers to the stop mode used when triggering the soft limit / hard limit, with optional deceleration stop / immediate stop.

- Slow down stop:

In hand control mode, it refers to the "emergency stop reduction acceleration" set in "hand control mode motion parameters (continuous, inch, hand pulse, G0 quantitative motion mode)".

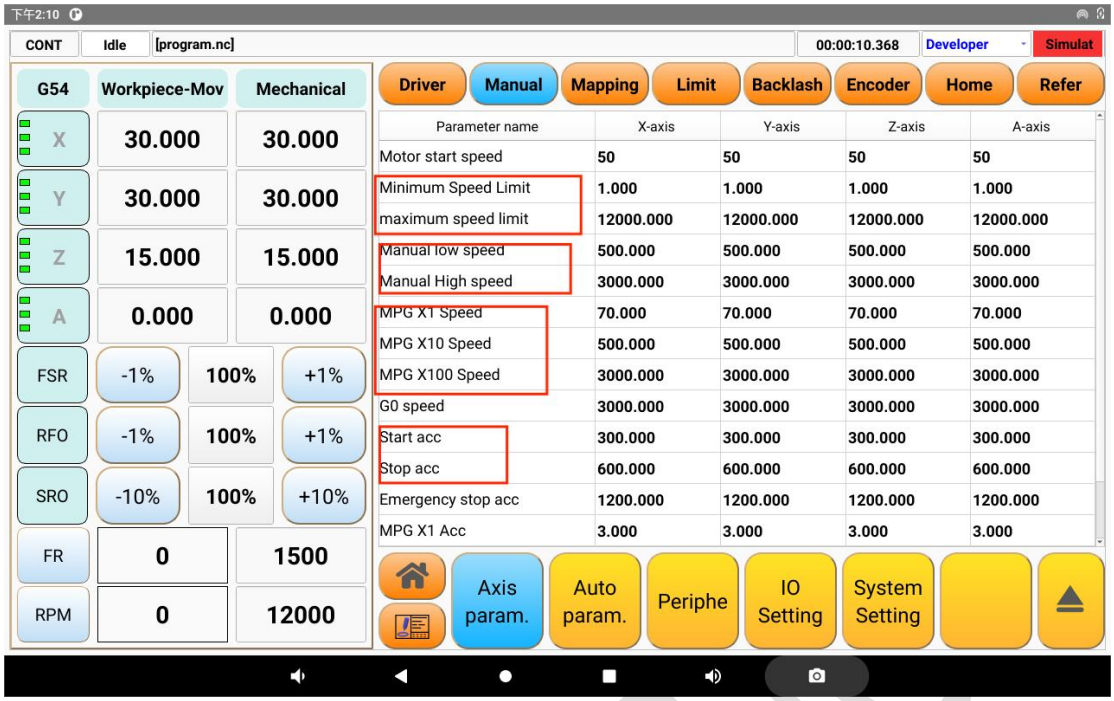
- Stop immediately:

Direct stop, and no deceleration process. Note that this mode has a high impact at high speeds.



2) manual control

Suitable for manual control mode. Each axis can be set up separately.



- **initial velocity**

You can set the starting speed for each axis.

- **Minimum / maximum speed limit**

It refers to the minimum and maximum speed limit after the combination of manual multiplier.

- **Running low speed / running high speed**

Running low speed: Running speed in low speed mode.
Running high speed: running speed in high-speed mode.

- **Hand pulse gear running speed**

It refers to the highest speed of the hand pulse X1, X10 and X100 gears respectively. Is the basal rate, excluding the multiplier.

Each gear can be set parameters, can ensure that the hand pulse in different gear, the movement speed of the machine can be stable.

- **G0 speed**

Speed of motion in the Work Coordinate-Move state.

- **Acceleration of the acceleration section / deceleration section**

Acceleration segment acceleration: the acceleration value during the acceleration process from a low speed.

Acceleration of deceleration section: the acceleration value during deceleration from high speed in normal operation.

pay attention to:

- **Usually, the acceleration value of the deceleration section should accelerate the**

segment acceleration value, otherwise, speed diving may occur during the deceleration process.

- **When the "inch" mode, the system accelerates according to the acceleration section and slows down according to this acceleration, that is, symmetrical trapezoidal acceleration and deceleration, without the deceleration value of the deceleration section.**
- **When "continuous", accelerate the acceleration acceleration and the deceleration acceleration.**
- **When the "hand wheel" mode and the hand wheel is "stop mode", accelerate at the acceleration section and the deceleration section. Same as the "continuous" mode.**
- **When the "hand wheel" mode and the hand wheel is "precision mode", accelerate the acceleration section and slow this acceleration, that is, symmetrical trapezoidal acceleration and deceleration, without the acceleration value of the deceleration section. Same as the "inch move" mode.**

- **Rapid stop and decrease acceleration**

In hand control mode, when "Drive parameter- -Stop mode" selects "deceleration stop", the deceleration process presses this acceleration value.

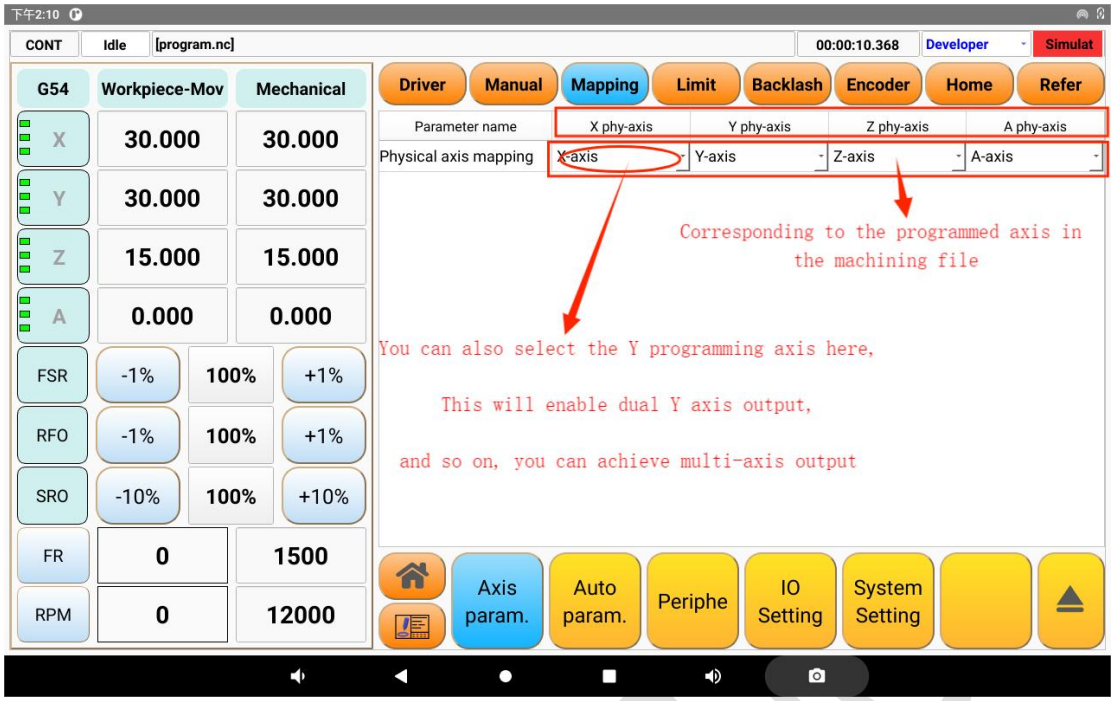
- **Hand pulse gear position acceleration**

The acceleration value of the hand pulse X1, X10 and X100, respectively. Is the basal rate, excluding the multiplier.

3) Axis mapping

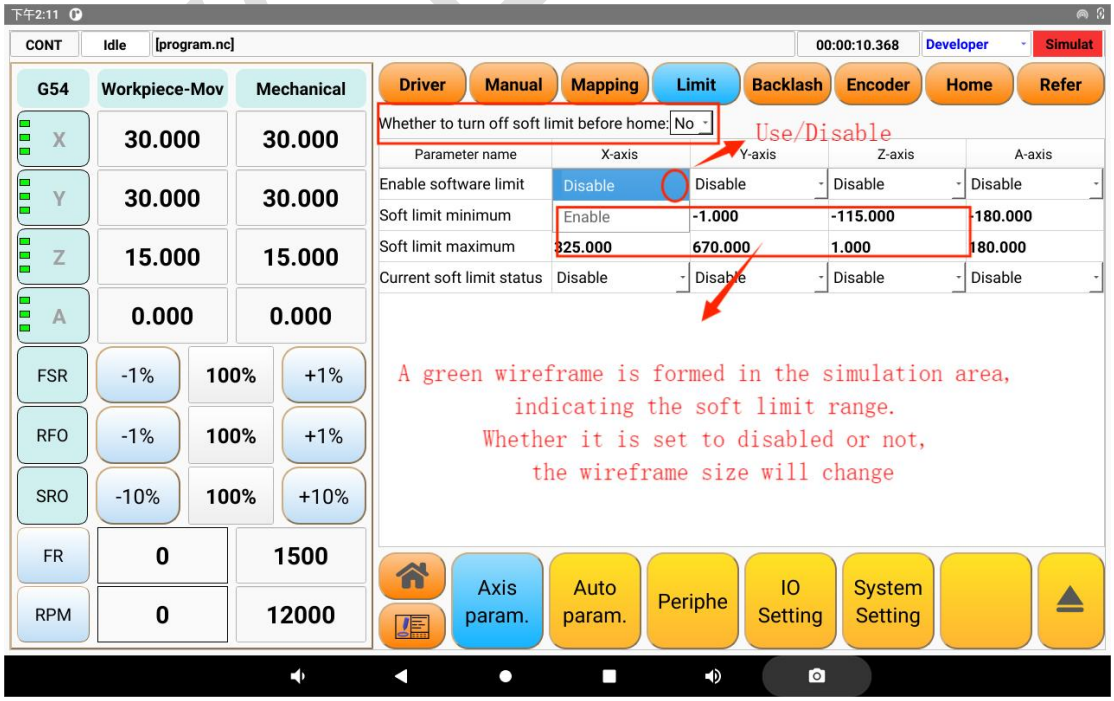
Set the corresponding relationship between the programming axis and the card movement port in the machining code. The number of axes corresponds to the number of card axes.

Biaxial simultaneous output or multiaxial simultaneous output can be achieved through setting.



4) Soft limit

- Close the soft limit switch before zero search is set.
- The soft limit range is displayed in the simulation area as a green wire frame.
- Soft limit enable / disabled can be set. When enabled, the alarm message will appear during the online processing, and the corresponding axis letters in the coordinate display of the control area will turn red.
- Current soft limit state: Since the soft limit setting is closed before zero, there is an inconsistency between the soft limit state and the soft limit enabling setting, which is used to indicate the current actual soft limit state.



For AC structure, for G68.2/G53.1 instruction, if axis A soft limit is set:

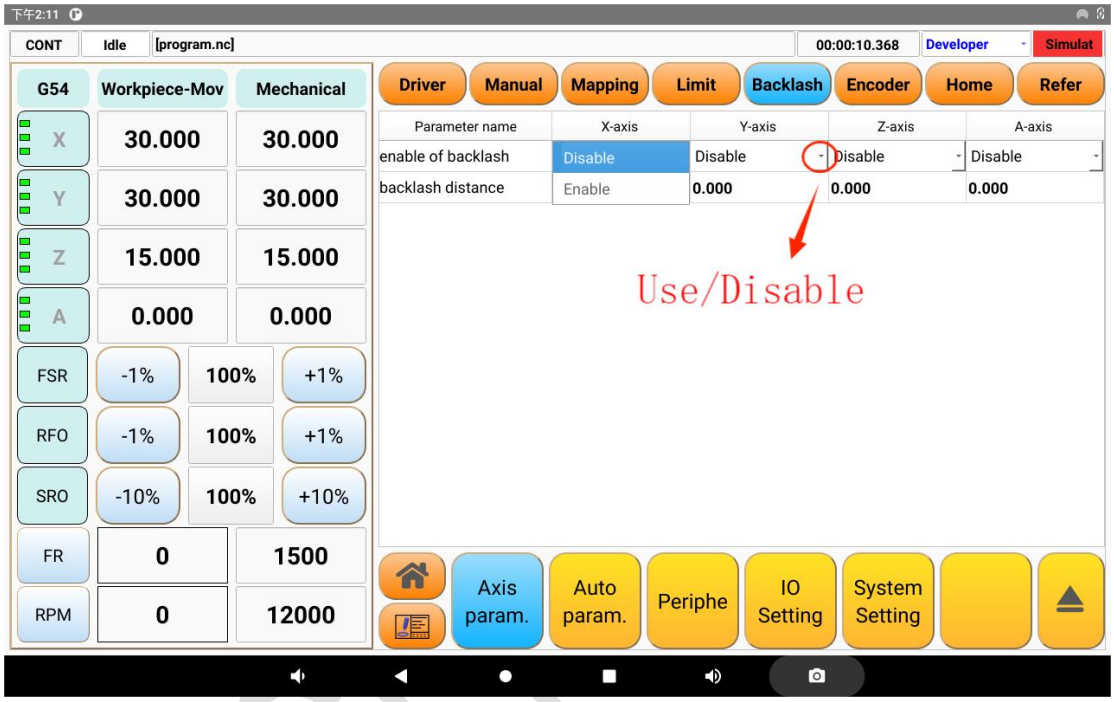
- (Max + minimum) / 20, the kinematic forward restriction rule is used.
- (Max + minimum) / 2 <0, and the kinematic negative limit rule is used.

For BC structure, for the G68.2/G53.1 instruction, if axis B soft limit is set:

- (Max + minimum) / 20, the kinematic forward restriction rule is used.
- (Max + minimum) / 2 <0, and the kinematic negative limit rule is used.

5) return difference

Set the difference of each axis, select enable / disabled.

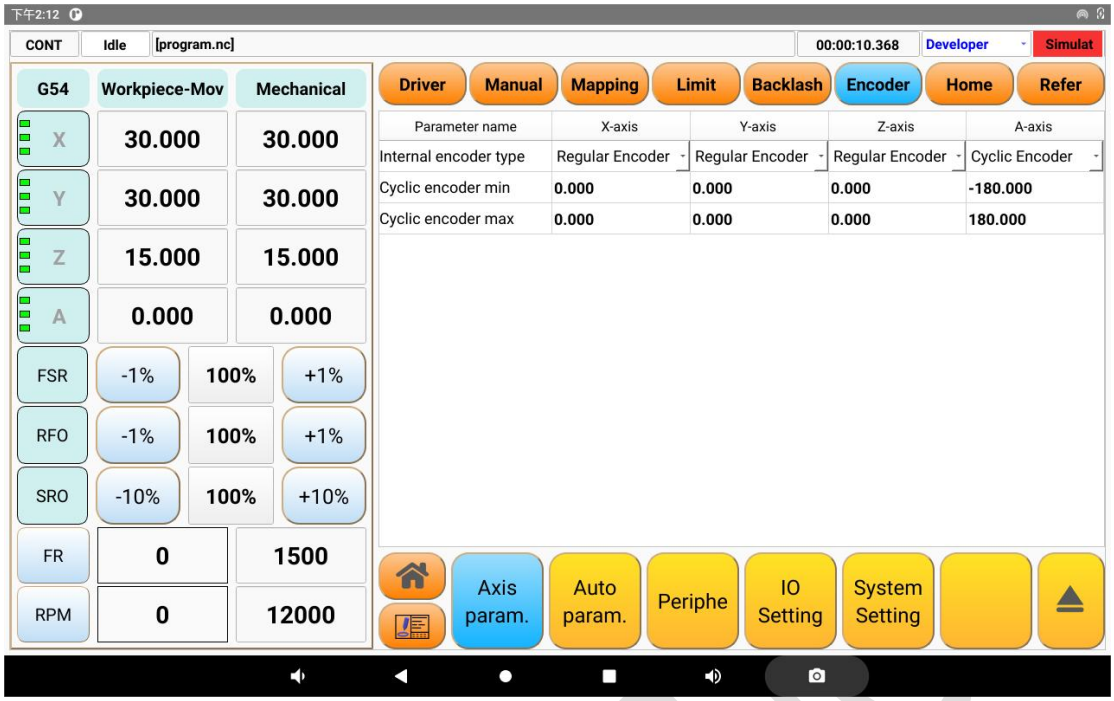


6) encoder

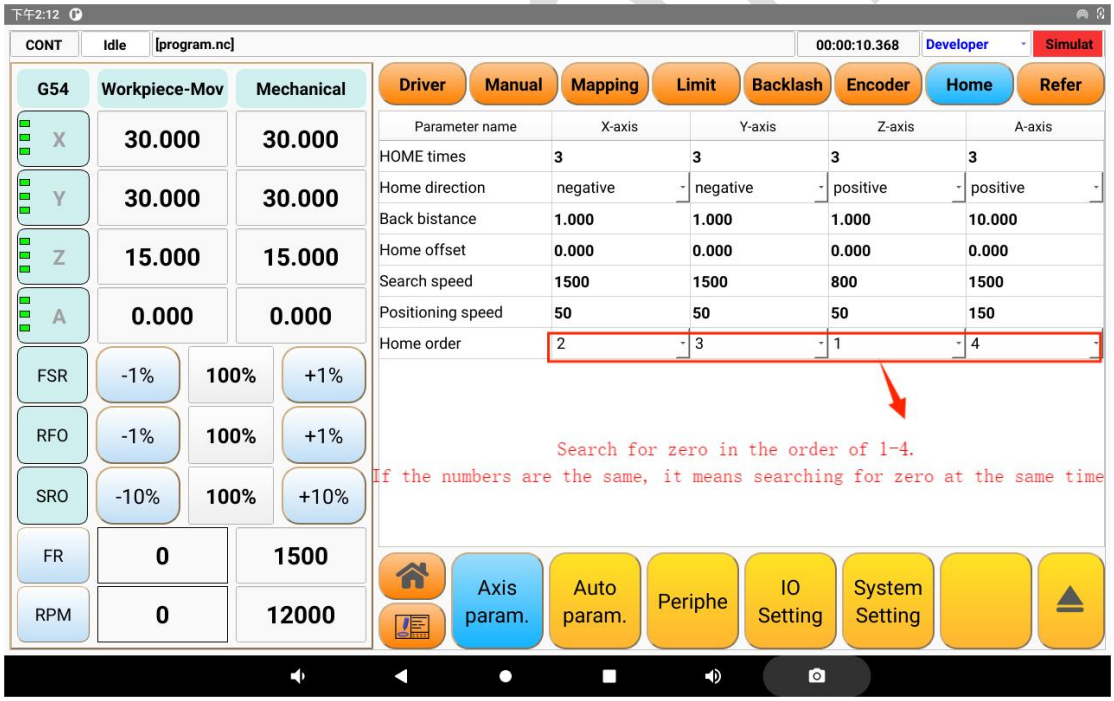
Usually the rotation axis is set to the cyclic encoder.

In particular, the corresponding rotary processing file, if the conventional encoder is used, the display Angle may always increase, when back to zero operation, it takes a long time.

When it is a cyclic encoder, if the minimum value is set to-180- - -180, the display Angle changes between-180- -180. If the minimum value is set to 0- -360, the display angle will only change between 0- -360.



7) Find zero



- **Find zero times:**

Number of times the zero-point switch is triggered. For example, it is set to 3 times, indicating that the first zero switch is withdrawn from the switch area, the zero switch is triggered and then turned back, and the end of the zero switch is triggered, a total of 3 times. The main purpose of many times is to improve the zero-finding accuracy.

- **Looking for zero direction:**

According to the installation position of the zero point switch in the movement range of the machine tool. If the zero switch is installed in the middle of the trip, then it can be zero whether positive or negative, it will move according to the set direction first, and when the limit switch is touched, it will move back to find the zero switch.

Note: If the zero switch is used and the limit switch is used, the setting of the zero direction will be ignored and the default direction of the system will be changed.

- **Return distance:**

It refers to the distance of the tool after zero. The general Z-axis backback distance is set to 0.

- **Zero point bias:**

Refers to the machine tool coordinates of the zero-point switch position.

- **Home Search Speed:**

Running speed of finding zero.

- **Home positioning speed:**

When the zero switch is touched, press "Accelerated section Acceleration" in "Manual mode motion parameter" to slow down to 0, and exit the switch area with the positioning speed, and then stop immediately.

- **Find zero order:**

The six axes can be found at the same time, or in order. If you choose the same serial number, it means finding zero at the same time. For example, if the Z axis is 1 and the X and Y axes are 2 at the same time, it means that the Z axis is zero first, and then the X and Y axes are zero at the same time.

8) reference point

- The positions of the reference points of G28, G30P1, G30P2, and G30P3 can be set.
- The G28 reference point is usually used as a return knife reference point.
- The G28 reference point is based on the mechanical coordinate system, so when using the G28 reference reference point, the zero-finding operation must be completed to establish the machine coordinate system. When the zero search operation is not performed, it will first move to the specified point, without going back to the G28 reference point.
- To ensure that the position is still located to the G28 reference point without the mechanical return zero operation, execute the instruction again in G91 mode.

Example 1:

G90G28X0Y0Z0A0; Move to the work coordinates (0,0,0,0) and then return to reference point 1

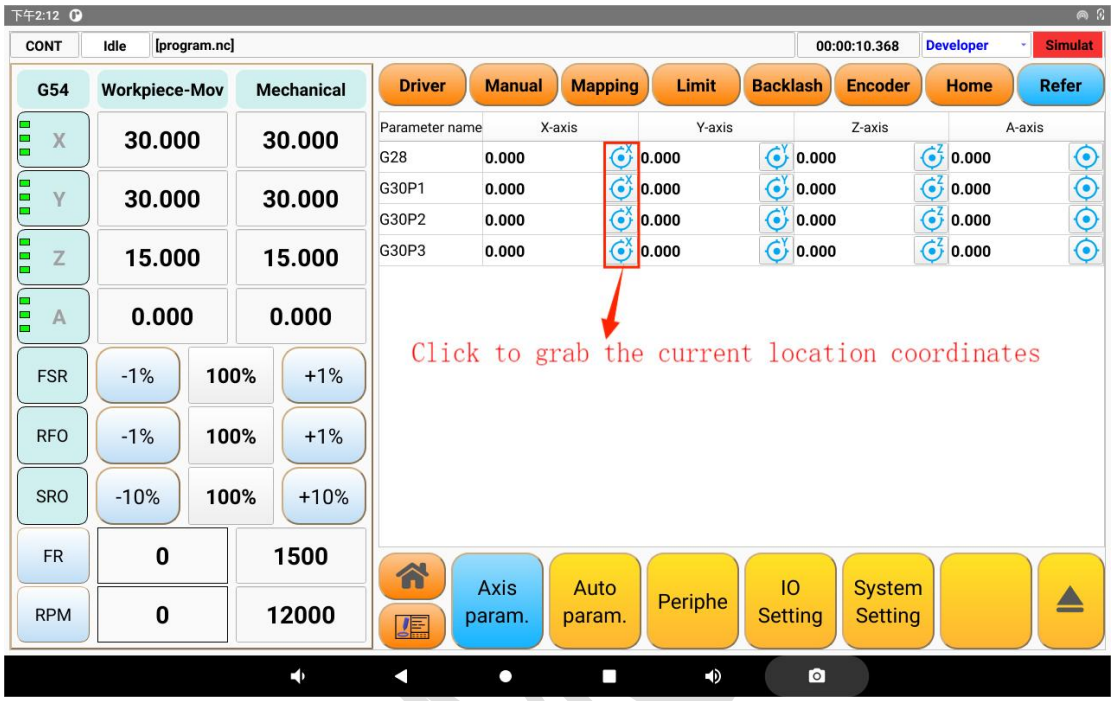
G91G28X0Y0Z0A0; Ensure the final positioning to the G28 reference point without

mechanical zero

Example 2:

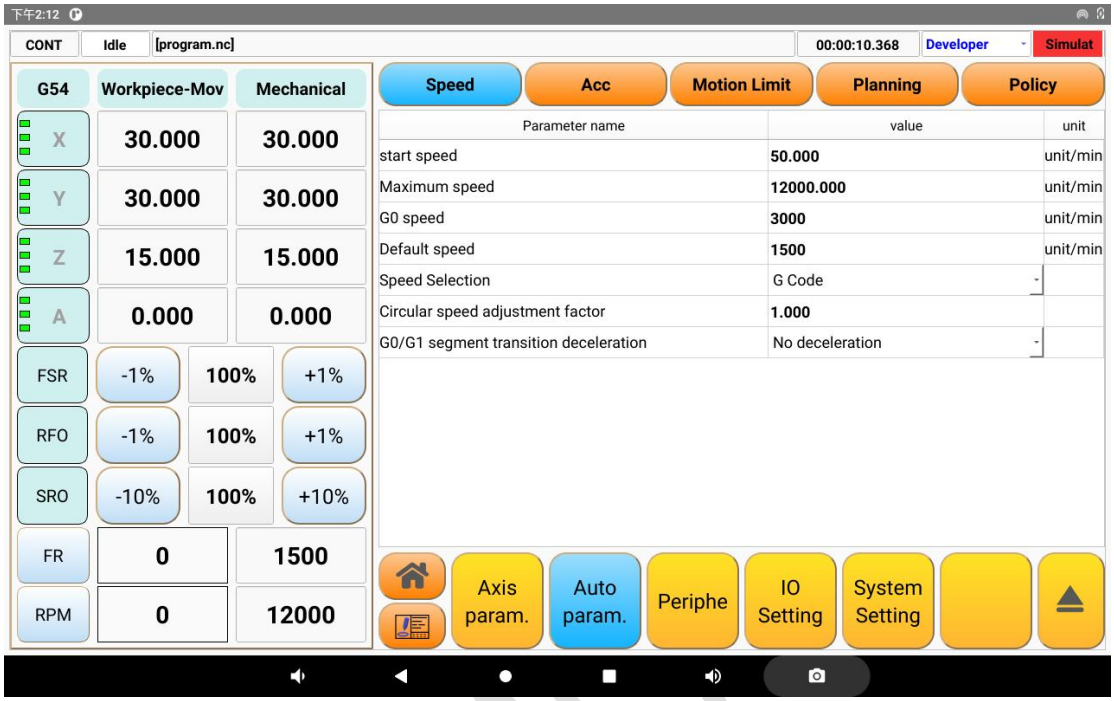
The G91G28Z0; Z axis returns from the current position to the G28 reference point

G91G28Z0; Call again to ensure that the Z-axis is finally located to the G28 reference point without performing the mechanical return to zero operation



2. Automatic parameters

1) velocity correlation



- **Start-off speed: it is the start value of the synthesis speed.**
- **Maximum speed: it is the maximum value of the synthesis speed.**
- **G0 segment speed: G0 segment command speed.**
- **Default feed rate: It is displayed in the display area.**
- **Feed rate selection: optional G code, default feed rate.**
- **G2 / G3 arc segment velocity ratio: take the proportion coefficient of the velocity * when forward as the given velocity of the arc segment.**
- **G0 / G1 section transition deceleration: optional deceleration, no deceleration.**

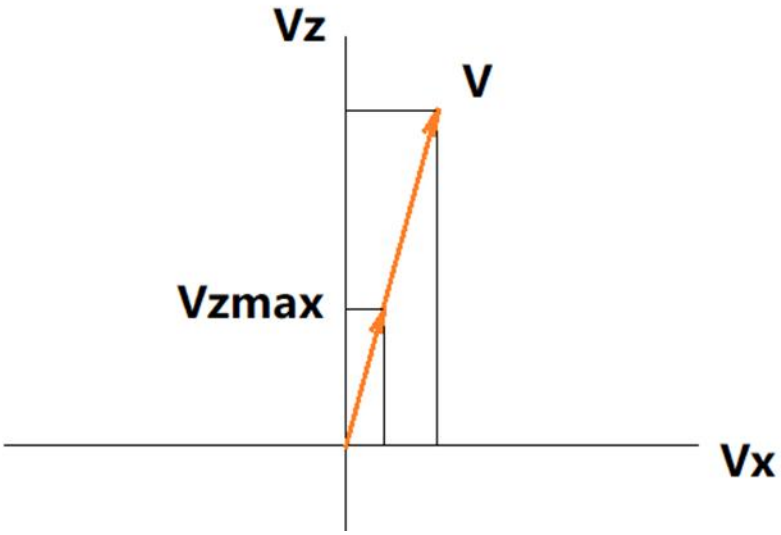
2) Acceleration related

- **Interpolation cycle: Interpolation cycle can be set for 0.001-0.01 seconds. It is generally recommended to use 0.004 or 0.005.**
- **S-type calculation coefficient of acceleration and deceleration: when it is not 0, the J acceleration and turning acceleration will be determined automatically, according to this coefficient and the line acceleration. If =0, J acceleration and turning acceleration must be set respectively.**
- **G1 segment line acceleration: refers to the acceleration of the synthetic motion vector.**
- **G1 segment line J acceleration: refers to the acceleration of the synthetic motion vector acceleration.**
- **G1 transition acceleration: two G1 transition acceleration.**
- **G0 segment line acceleration: refers to the acceleration of the synthetic motion vector.**
- **G0 segment line J acceleration: refers to the synthetic motion vector J acceleration.**
- **G0 transition acceleration: two G0 transition acceleration.**
- **G2 / G3 circular arc section centrifugal acceleration: when this parameter is 0, the G1 section linear acceleration is used as the centrifugal acceleration.**
- **Pause drop rate: In automatic mode, press the drop rate.**
- **Failure drop rate: in automatic mode, the fault rate (external emergency stop, hand wheel emergency stop, reset, servo alarm).**

3) Axis speed limit

This setting is mainly for some machine tools between the axis of the movement characteristics are very different cases, by setting different parameters for each axis to adapt to this kind of machine tools.

For example, if the Z-axis speed of the machine can only reach 100, and the X-axis speed can reach 1000, if there is no independent speed limit of each axis, in the situation below, the Z-axis speed will be too high, resulting in motion failure and other problems. If the Z-axis speed limit is set, the movement speed of the other axes is planned according to the Z-axis limit speed.



- **Automatic mode axis acceleration**

  Set in automatic mode, the acceleration value in each axis.

  If 0, it is not limited by uniaxial acceleration and only controlled by linear

acceleration in automatic parameter setting.

- **Automatic mode axis J acceleration**

Set in automatic mode, the J acceleration value in each axis. That is the rate of change in acceleration.

If 0, it is not limited by uniaxial J acceleration and only controlled by "line J acceleration" in the automatic parameter setting.

- **Positive / negative motion protection speed**

Set the maximum protection speed of uniaxial forward and negative motion in automatic mode.

Positive and negative protection speeds can be set separately to meet the different needs of each axis. For example, the Z-axis is generally cut downward, so the movement speed of the upward lifting knife is higher than that of the downward processing.

If 0, no uniaxial speed protection. Controlling only by the maximum speed used in the automatic parameter settings.

4) Path planning

- **Default programming unit of linear axis: optional metric mm / inch inch. The programming unit change only affects some of the features of the G20 / G21 instruction. In metric programming unit mode, the unit selection command group command default is G21 (mm); in English programming unit mode, the default is G20.**
- **Maximum shaping error: the maximum deviation distance between the actual contour and the theoretical contour.**
- **Processing path smoothing distance: the distance of two straight line arc transition.**
- **Shortest motion path of rotation axis (amplitude determines direction of axis**

motion): Optional on / off.

- **G00 link G00 path allows mixing: optional on / off.**
- **G00 link G01 path allows mixing: optional on / off.**
- **G01 link G00 path allows mixing: optional on / off.**
- **Pway mixing transition distance: When the path allows mixing, transition at this distance.**
- **G2 / G3 arc split string height error: used to control the arc machining error.**
- **Number of path planning small line segments: the maximum number of pre-planned rows.**
- **Parsed link table capacity: the maximum number of pre-parsed rows.**

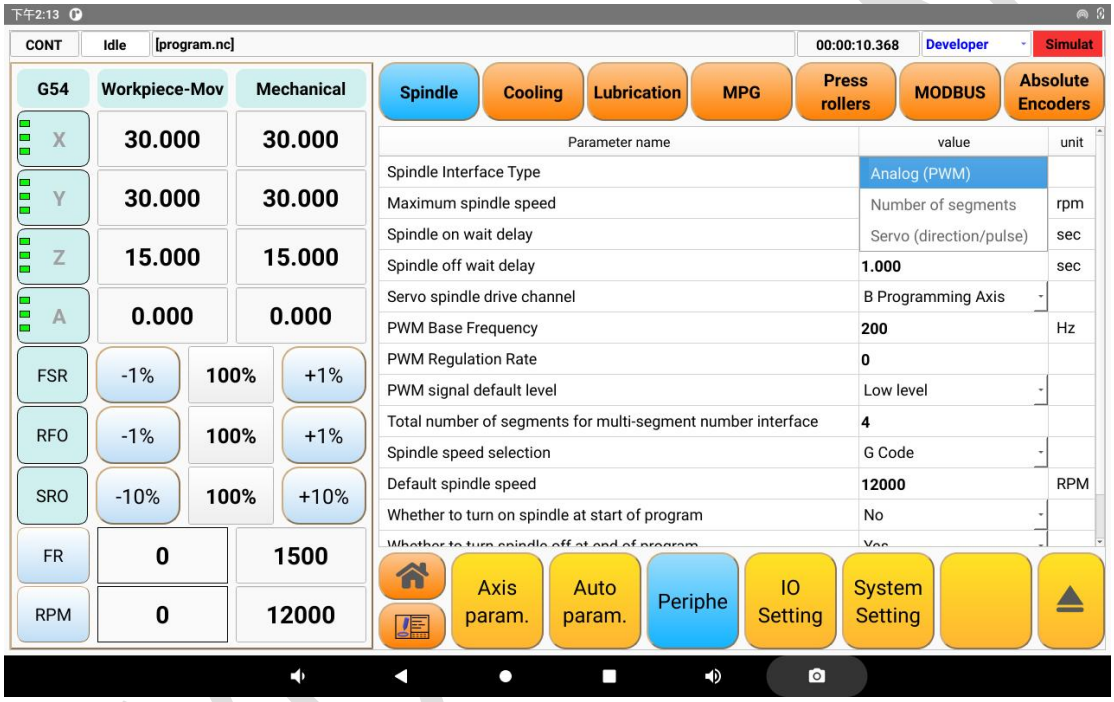
5) Policy related

- **RTCP workpiece reference transformation: if the machine is five-axis structure, select open. Display only under Developer Rights.**
- **Pause the knife lifting distance: press the automatic knife lifting distance after the pause.**
- **Back to zero strategy: optional back to zero / return to safety height;**
- **safety height**
- **Approach distance when the blade shaft returns to zero: use with approach speed.**
- **Approach speed when the blade shaft returns to zero: move at this speed when approaching the distance.**
- **Check back to the mechanical origin before processing: optional yes / no.**
- **Whether it automatically returns to the mechanical origin after going online: optional yes / no.**
- **Back to the mechanical origin program selection: optional host program / onboard program.**

- **Shaaxis strategy for RTCP pause recovery: optional back to G28 / G30P1 / G30P2 / G30P3 reference point.**
- **Stay position at the end of processing: optional current position / G28 reference point / G30P1 reference point / G30P2 reference point / G30P3 reference point.**
- **Spindle M code: optional built-in code / NC custom code.**

3. External Equipment

1) Principal Axis



- Spindle interface type: optional analog / multiple segments / servo (direction / pulse).
- Servo spindle drive channel: optional X / Y / Z / A / B / C programming axis.
- Spindle speed selection: Optional G code / default spindle speed.
- Default spindle speed: It has a display in the control area.
- You can select whether to close the spindle at the program start / program end / reset.

2) Cooling

3) Lubricating

4) Hand Wheel

a) **Handwheel accuracy**

Define the distance corresponding to 1 rotation of the hand wheel at X1 gear. The range can be set to be 0.001- - -0.01.

b) **Handwheel Control Mode**

Optional on-stop mode / precision mode.

- Stop mode: as long as the hand wheel is stopped, slow down and stop regardless of whether the number of pulses input by the hand wheel has been executed. The deceleration mode is to press the "programming axis- -hand control mode motion parameter- -deceleration section acceleration" deceleration stop.
- Precision mode: completely corresponding to the input of the hand wheel, all the pulses input by the hand wheel will be stopped after the execution. The phenomenon seen is that after the hand wheel is stopped, it will not stop immediately. The deceleration mode is to slow down according to the "programming axis- - -hand control mode motion parameter- -acceleration section acceleration", that is, symmetrical trapezoidal acceleration and deceleration, without the deceleration value of the deceleration section.

c) **The hand wheel shakes the movement direction counterclockwise**

Define the direction of movement of the machine when turning the hand wheel counterclockwise. Can choose positive / negative, positive reference coordinate positive change, negative reference coordinate to negative change.

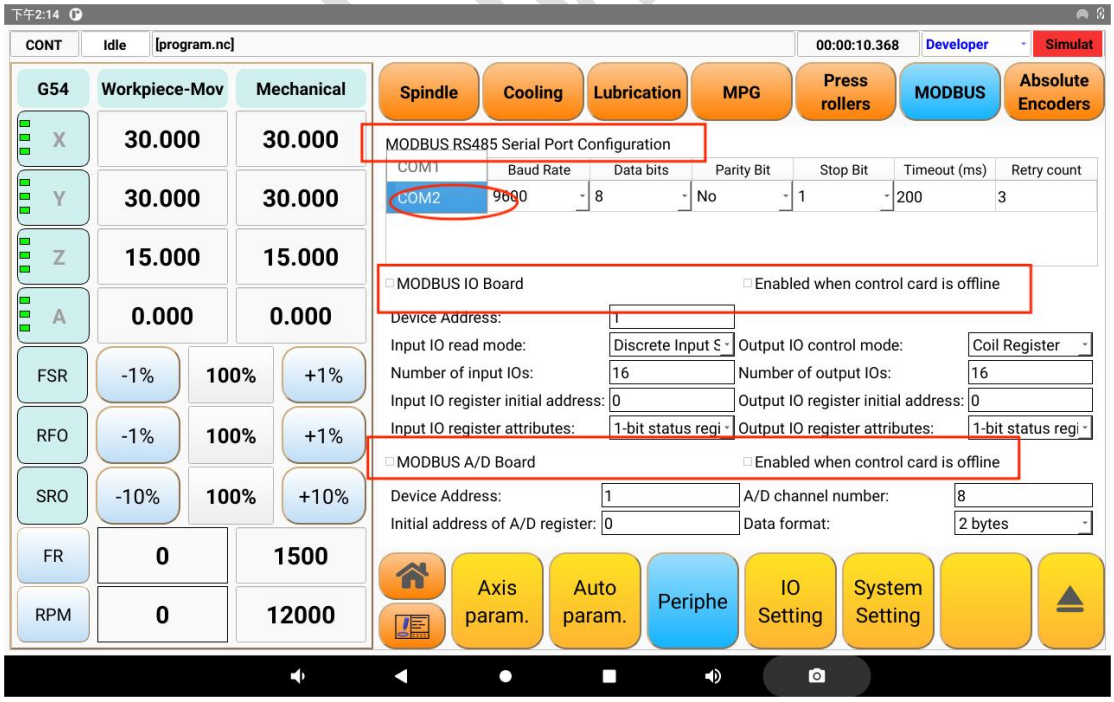
5) Compression Roller

Note: When the end coordinate of the axis, the axial direction.

6) MODBUS

MODBUS RS485 Serial port configuration:

- The ttys 9 corresponds to the port number RS485_1.
- The ttys 3 corresponds to the port number RS485_2.



7) Absolute value encoder

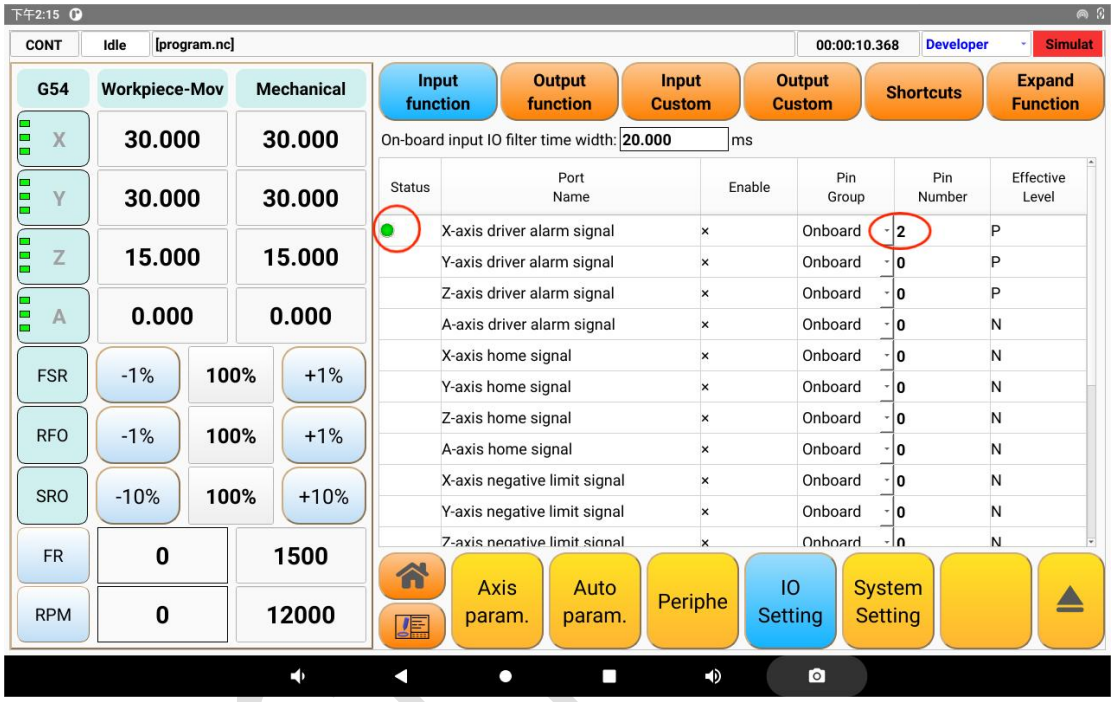
Customizable, see for details[Appendix 1](#)。

4. IO set up

pay attention to:

- The input / output port allocation is a port number based on a fixed input / output signal.
- Input / output IO customization is based on the port number as the traction, define the input / output signal.

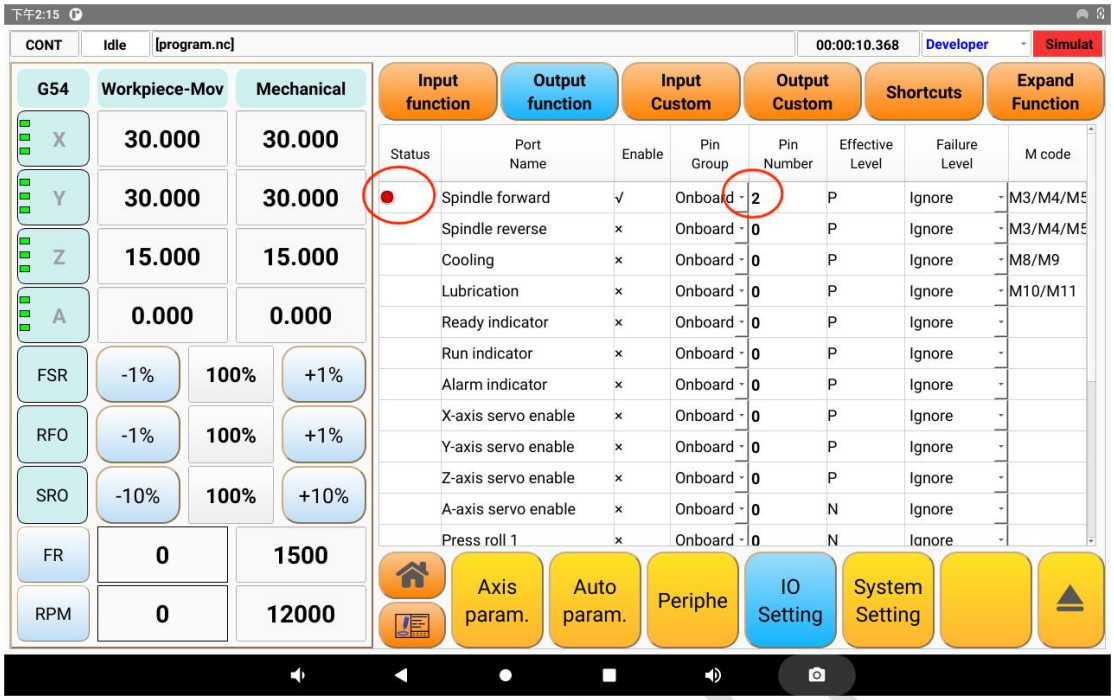
1) Enter port assignment



After defining the pin number:

- There is a green flag in the left status bar;
- Click the enable bar to switch to enable () or disable (x);
- Click the effective level to switch N or P.
- The pin group can be switched after the MODBUS is configured.

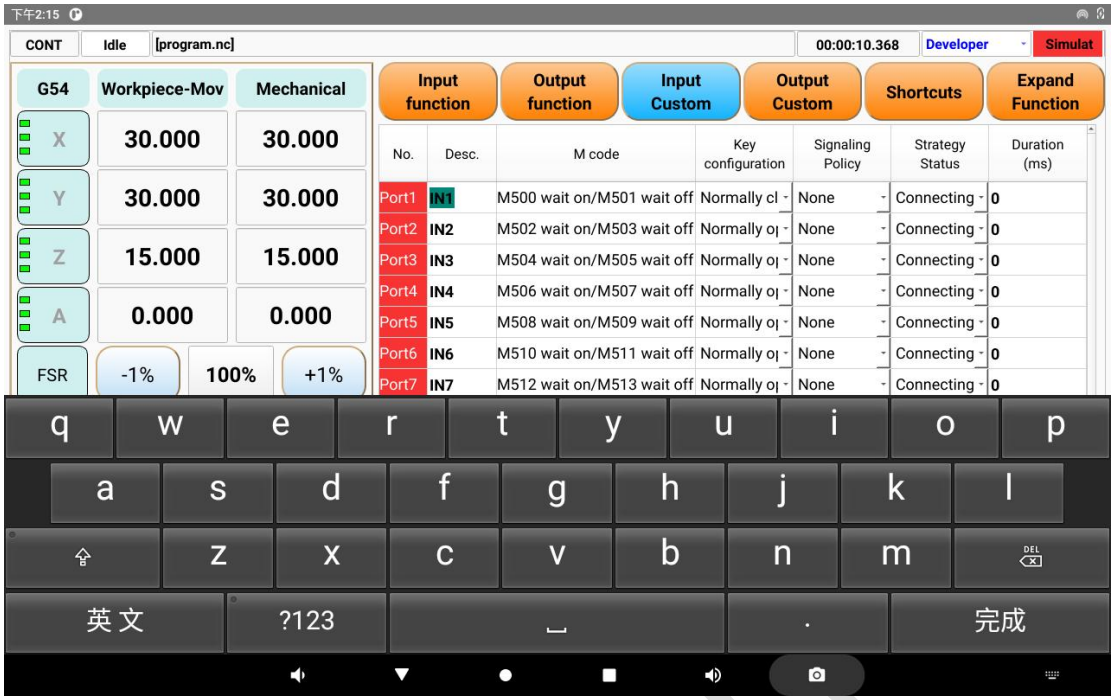
2) Output port assignment



After defining the pin number:

- There is a green light or red light mark in the left status bar, which can switch the status by clicking;
- The status of the enabling bar changes to enable the enabling bar ();
- Click the effective level to switch either N or P;
- System fault level optional ignore, N, P;
- The relevant M code is the relevant M instruction related to the corresponding output signal, as follows, refer to M Instruction Instructions for details.
  - M3 / M4 / M5: spindle forward / spindle reversal / spindle stop
  - M8 / M9: Coolant on / coolant off
  - M10 / M11: lubrication on / off
- The pin group can be switched after the MODBUS is configured.

3) Enter IO Custom



- Port description is editable name.
- Signal policy is optional: no / only alarm / alarm reset / alarm pause / execution code.

4) Output IO Custom

- Port description is editable name.
- Control policy optional: unlimited / run only M code control allowed / not control when spindle is on.
- Fault state trigger time is optional: when fault occurs / when fault is

removed.

- Status optional when fault occurs: hold / off / on.

5) IO keyboard shortcuts

Only the developer permissions are available.

usage method:

- Click the shortcut bar, will flash prompt "Enter shortcut key (ESC cancel recording)", then press the shortcut key to be defined.
- If you press ESC to cancel, the originally defined shortcut keys will also disappear.
- If you mistakenly click the existing shortcut key, you can click the blank line of the shortcut key, and then click "Cancel the shortcut key", so that the original shortcut key will not change.

All shortcuts are customizable, with the default definition.

The external device can also be defined as a shortcut key. For example, the start processing shortcut key is defined as the external button to realize the "one-button start".

6) Extended functional sets

Only the developer permissions are available.

Customizable function, linked to the corresponding one. The nc file, mounted into the shortcut key, accordingly will be added to the IO shortcut key list, when you can define the shortcut key. This executes the linked nc file content when the shortcut key is pressed.

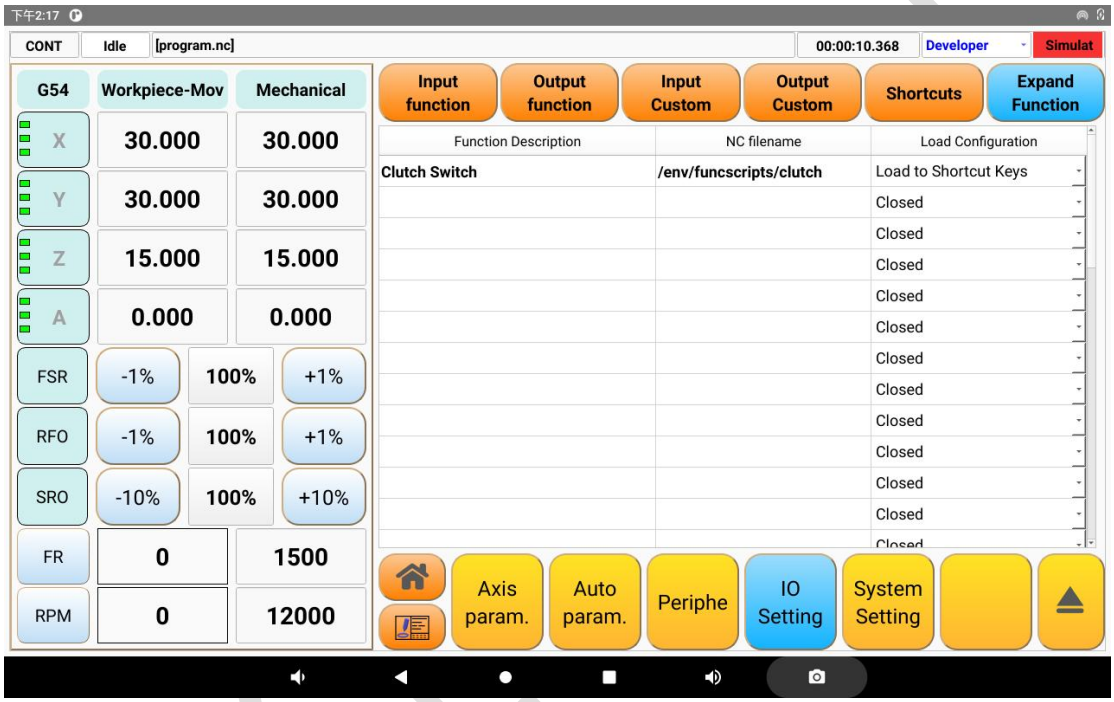
usage method:

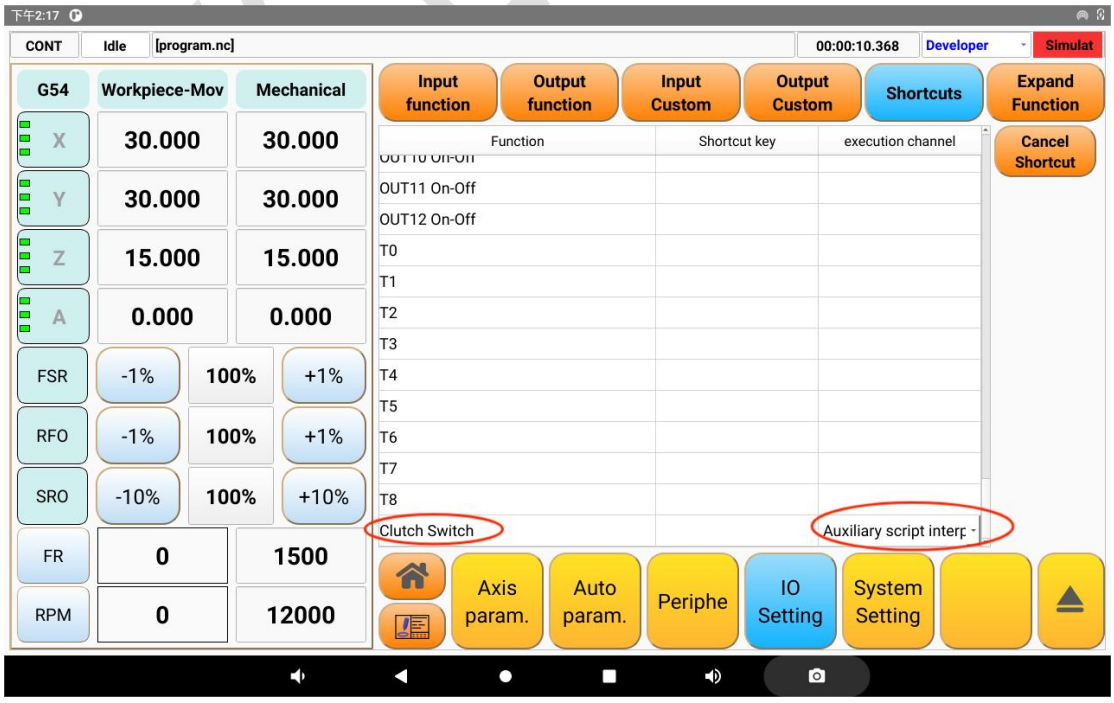
- Click on the function description bar to edit the function name.
- Click on the function NC file name bar and enter the link. nc document.

Note: If using script interpreter, the directory must be / env /...; if using NC interpreter, no add / env.(Perute bar selection in the IO shortcut list script)

- *Load Configuration Bar Select the Mount to shortcut key.*
- *The IO shortcut list for script execution channel bar Select NC Interpreter / Master Script Interpreter / Auxiliary Script interpreter.*
- Define the shortcut keys, and press the[IO shortcut key use method](#)。

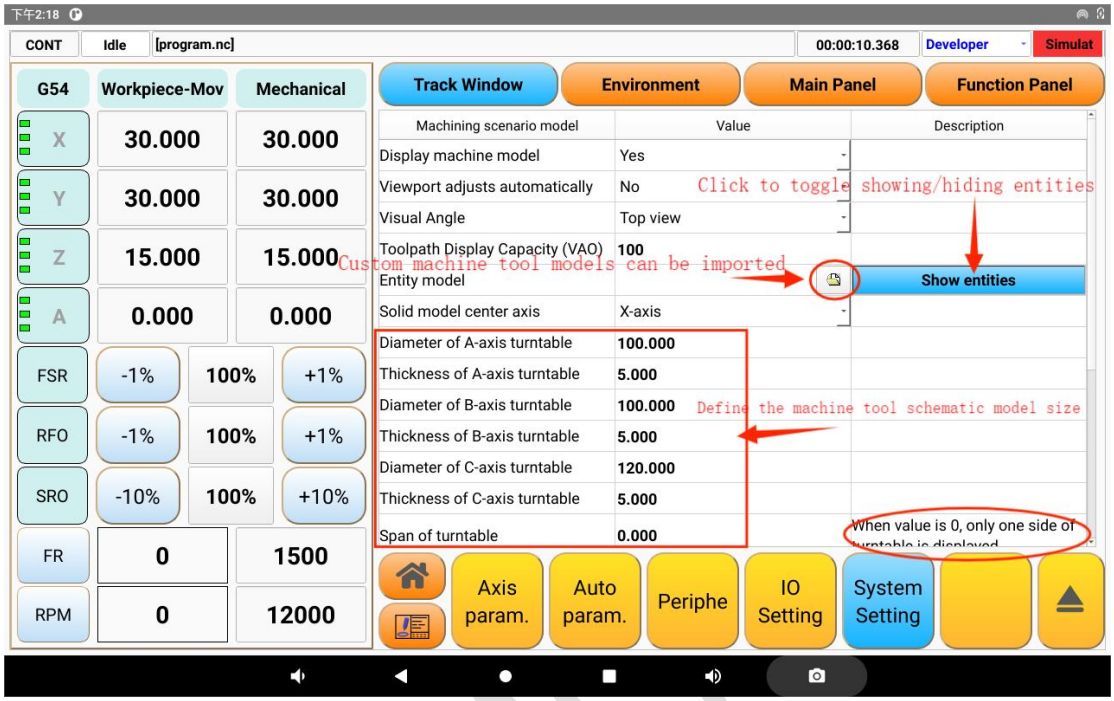
For example, the clutch switch in the figure below is also listed in the IO shortcut key.



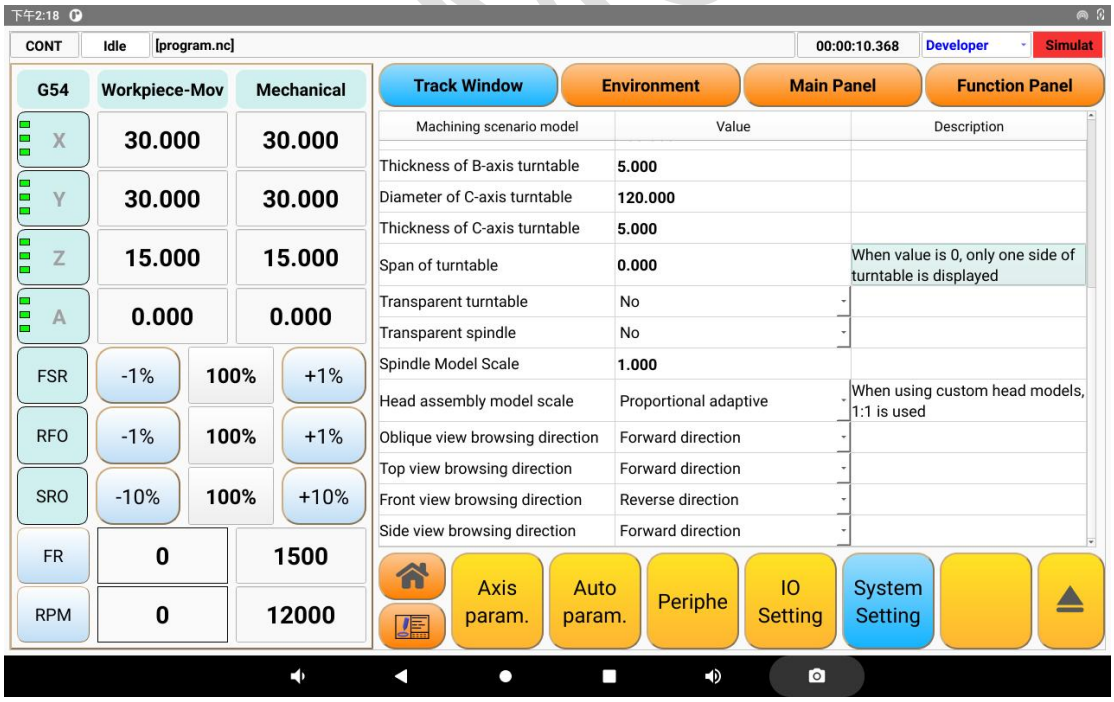


5. System Settings

1) Dao road window

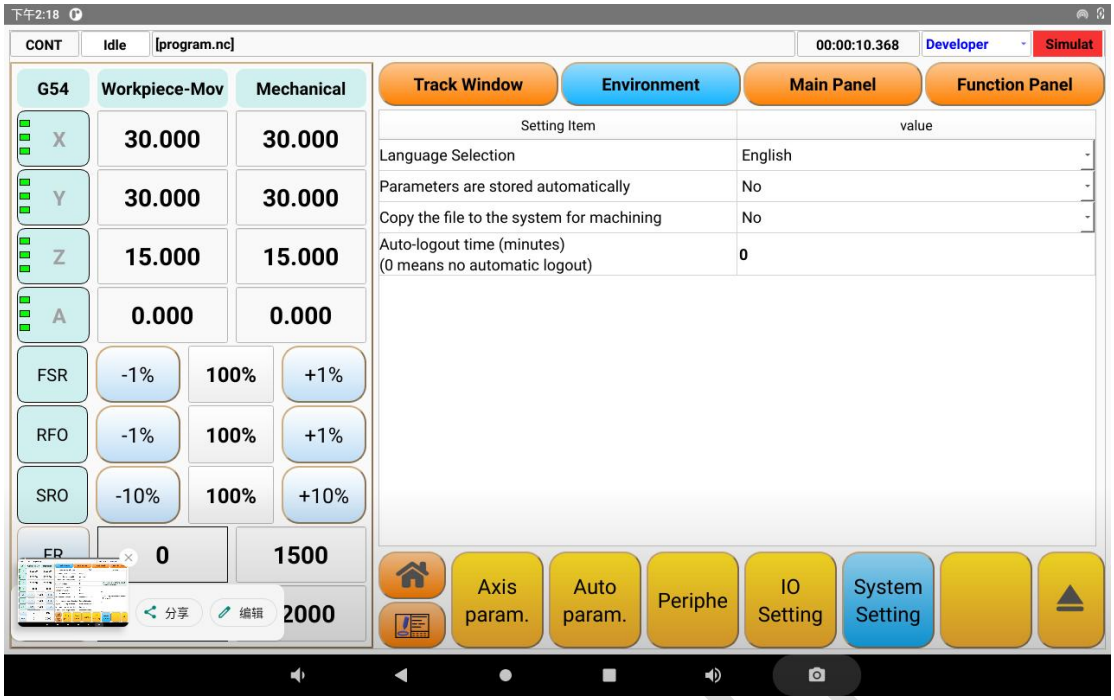


We define the machine model related parameters in the simulation window. If the machine structure is AB double turntable, the C-axis parameters are ignored.



- Spindle model scaling: the spindle model scale can be adjusted. For example, when using British units, you can adjust the spindle proportion here, so that it matches the machine model.

2) Environmental configuration



a) **Language selection: Chinese / English can be switched.**

b) **Copy the files to the system for processing**

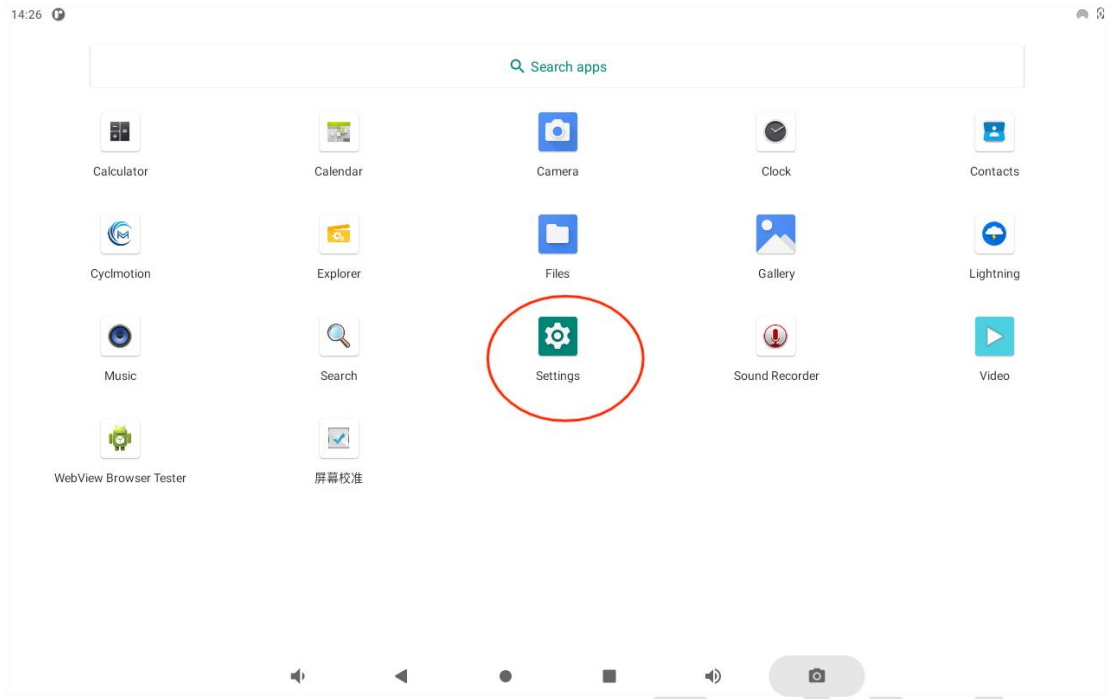
Optional yes / no. When the selection is yes, after selecting the file on the U disk to start the processing, the file will be automatically copied to the system cache, when the U disk will not affect the processing.

merit:

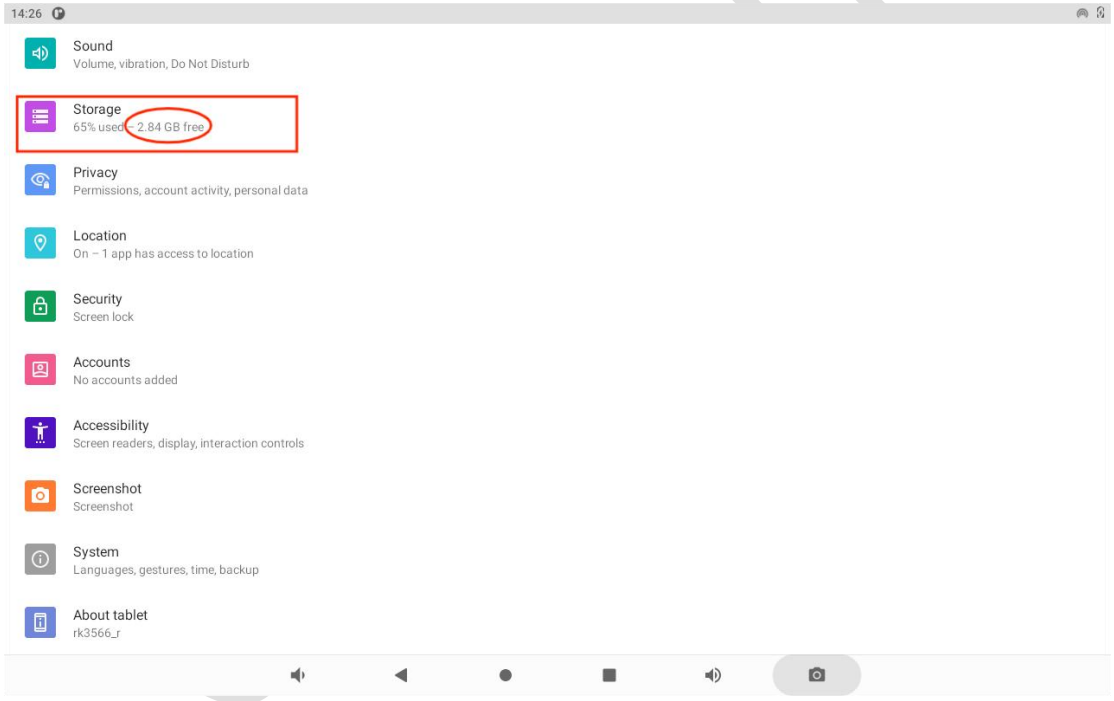
- Processing is not disturbed by the U disk interface and plug.
- When you need to call the cached file again, there is no need to insert the U disk, you can load the file directly in the processing policy-processing information.

pay attention to:

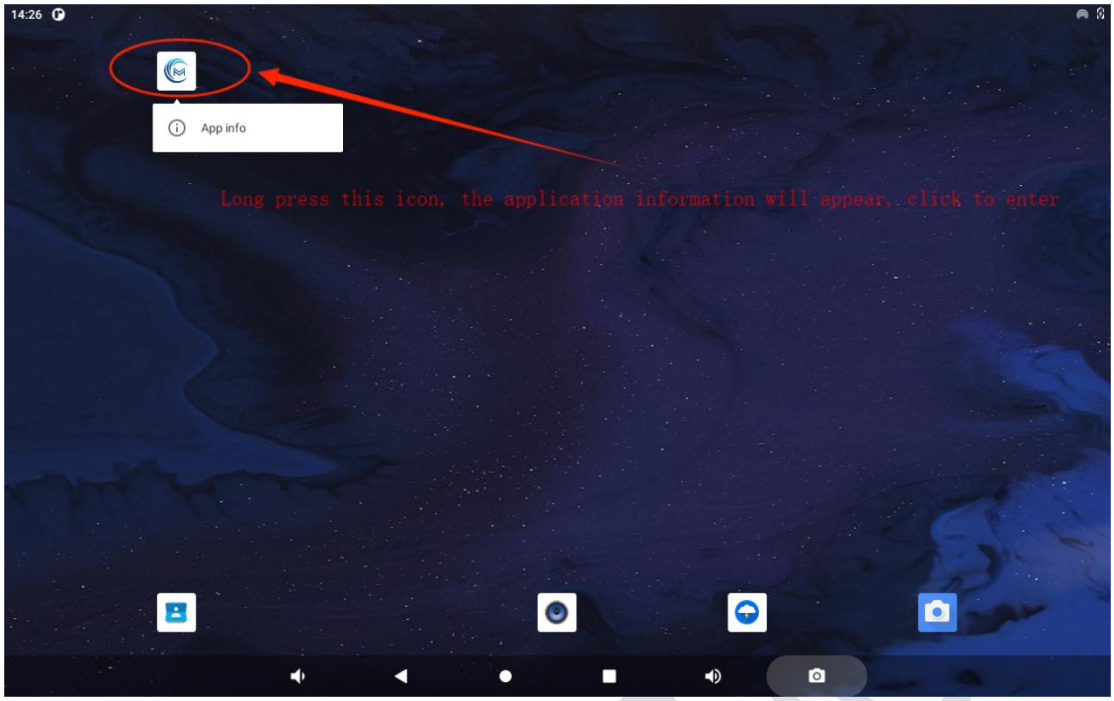
- The cache requires regular cleaning, and you can view the remaining space in the all-in-one Settings-Storage.
  On the all-in-one home page, the following interface appears:



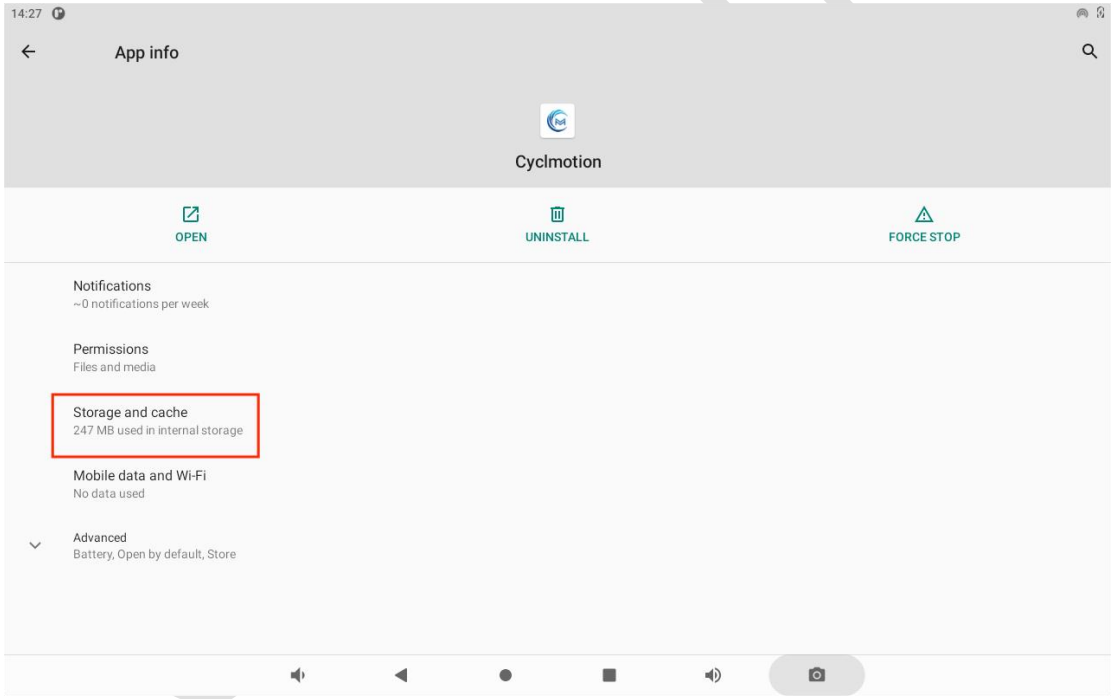
Click on Settings, see the figure below.



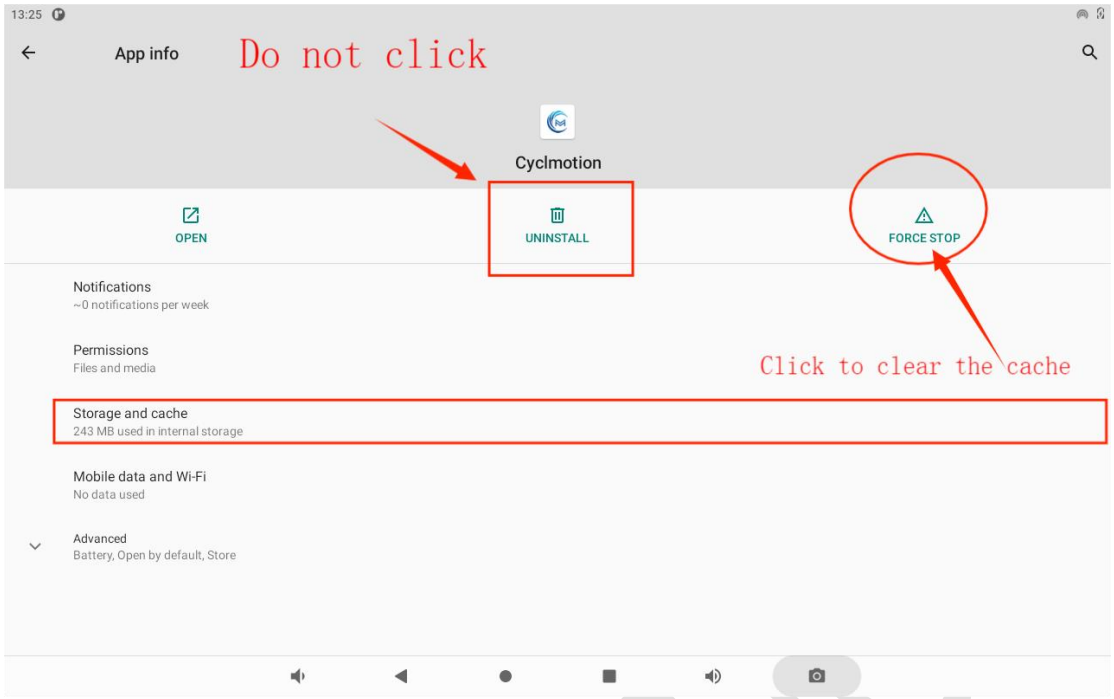
- Cleaning method: long according to the software icon, the application information appears, click to enter, click the storage and cache, click to clear the cache, pay attention to must not click to clear the storage space!!!



Click the Application Information, and the following interface appears:



Click on Storage and Cache:



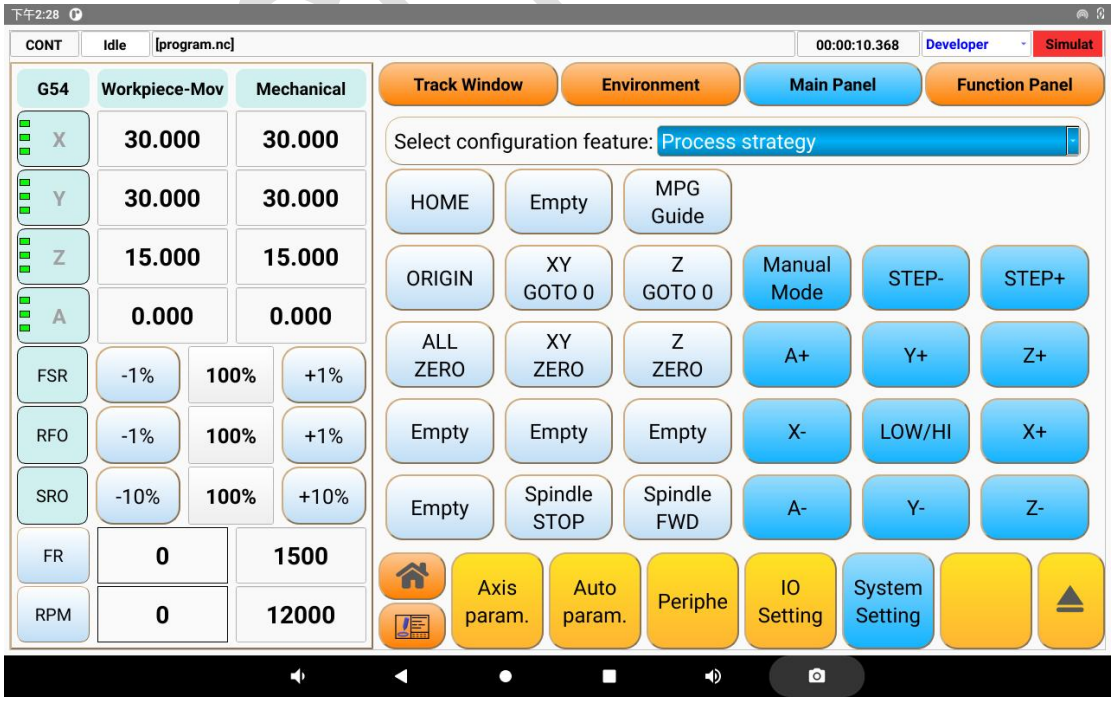
c) **Automatic exit login time (minutes)**

Set to 0, which means that it does not exit automatically.

The setting time is started from the login administrator or developer permission, and automatically returns to the login permission after the time exceeds.

3) Panel configuration

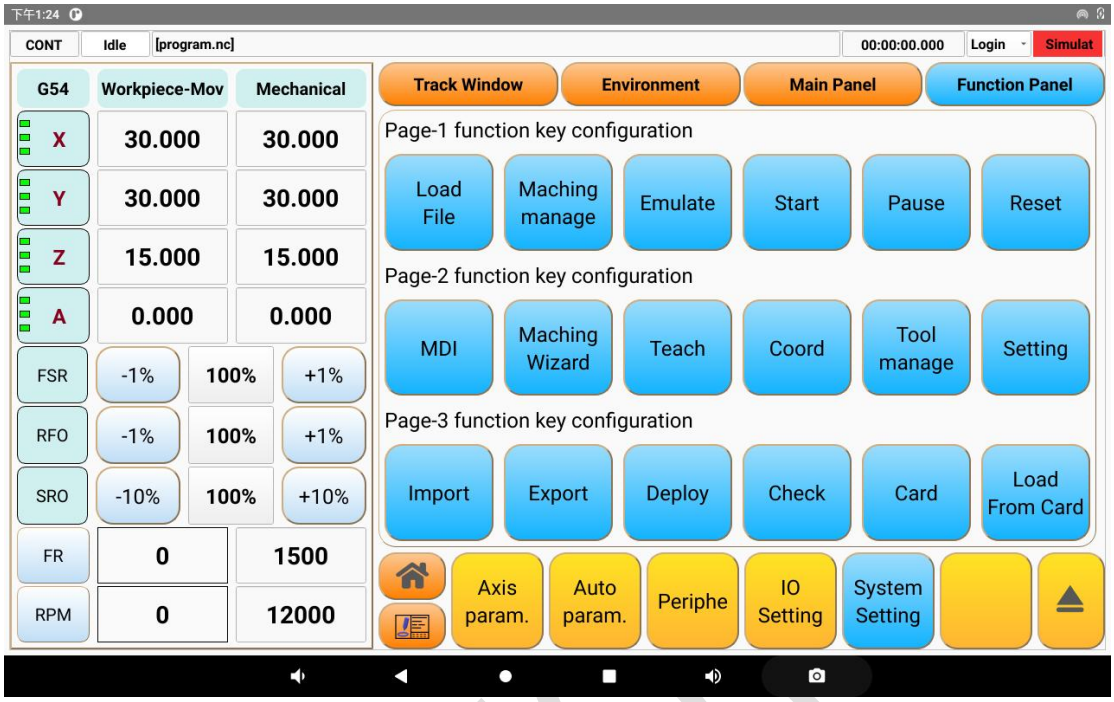
Only the developer permissions can be configured.see details[Panel configuration method](#)。



4) Function key configuration

Only the developer permissions can be configured.

In[Function key configuration method](#)Has been introduced.



(十五) Parameter import

Use to import the saved project files and quickly restore the system environment.

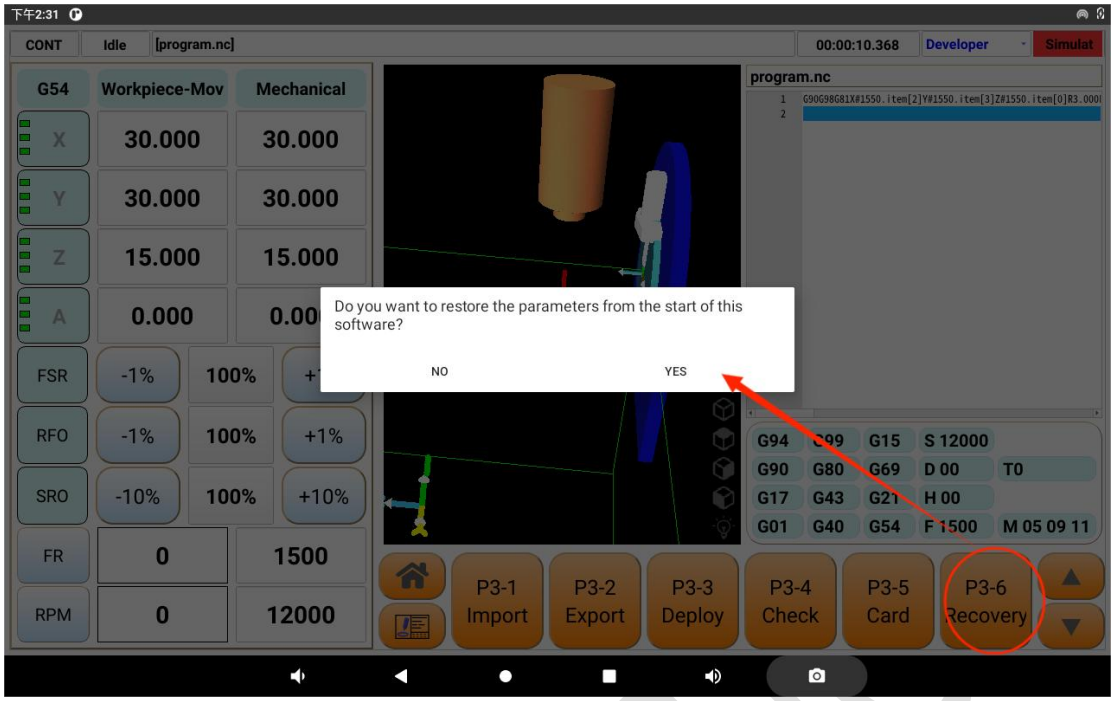
(十六) Parameter export

Used to save the current system environment as an engineering file for later import.attention in. program Suffix name.

(十七) Recovery parameters

Return to the state when the software is on.

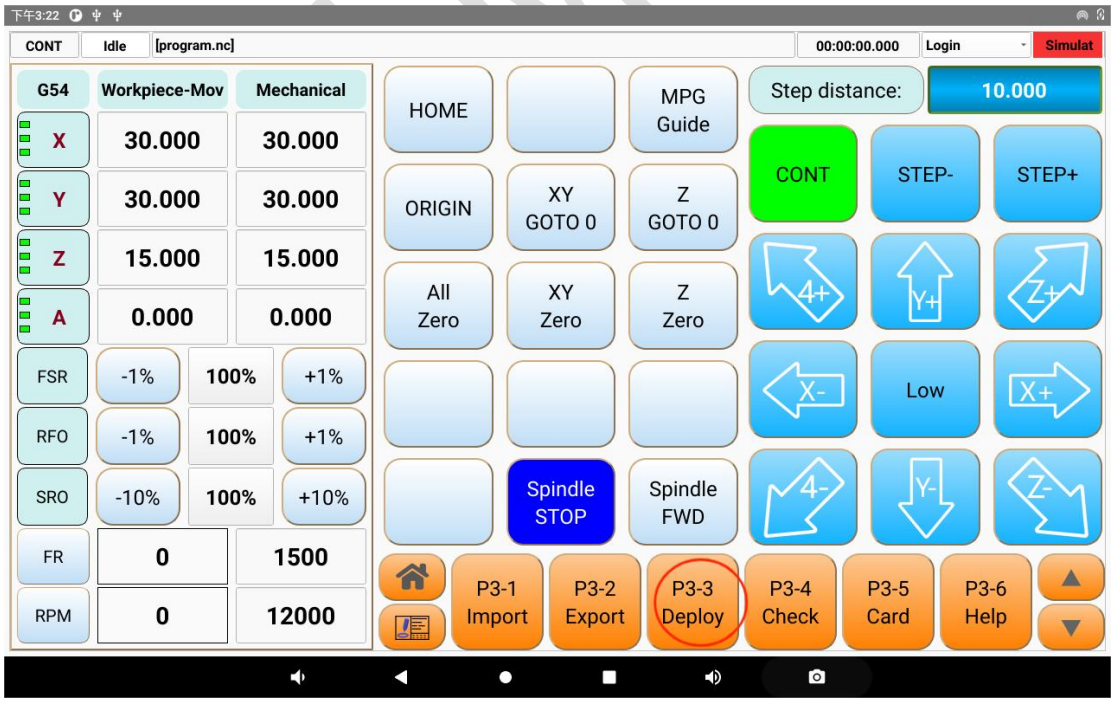
Apply when you the parameter or do not want to save the current state. Because the software will automatically save the parameters during use, if the software is reopened directly, it will return to the state of the last close.



(十八) Component deployment

To import the component deployment packages, upgrade, or update the system.

For example, a developer modifies and updates a file or folder, makes a deployment package and sends it to the customer, and the customer component is deployed.



1. Generate the deployment package method

Deployment packages are generally generated in the windows environment.

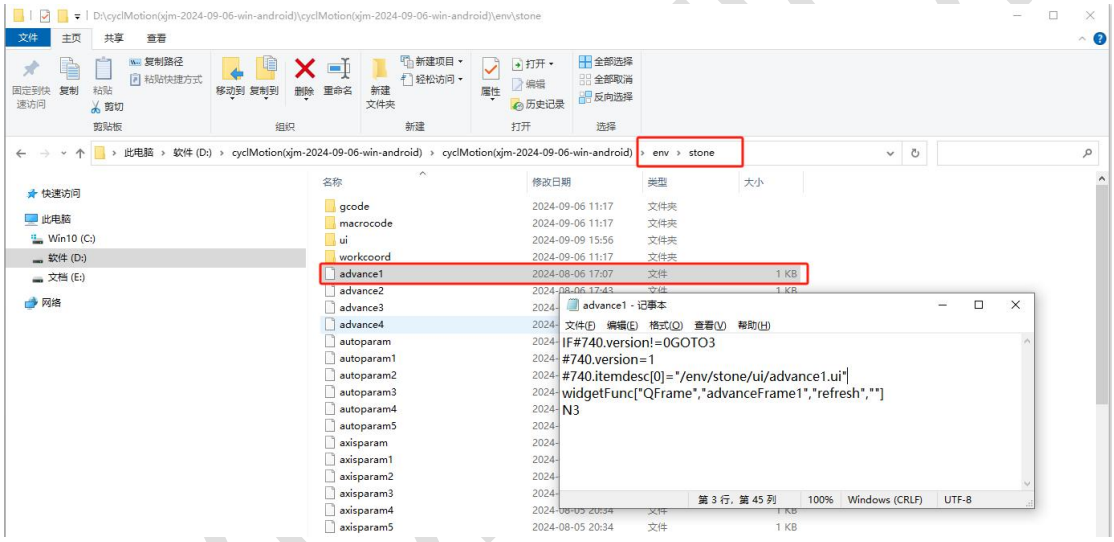
Since all the scripts of the software are in the \ env directory in the software installation directory, the updated files need to follow the corresponding folder under the \ env \ structure hierarchy.

Just put the file to be updated into the corresponding folder, and then go back to the \ env directory, and compress the corresponding file or folder into. zip form.

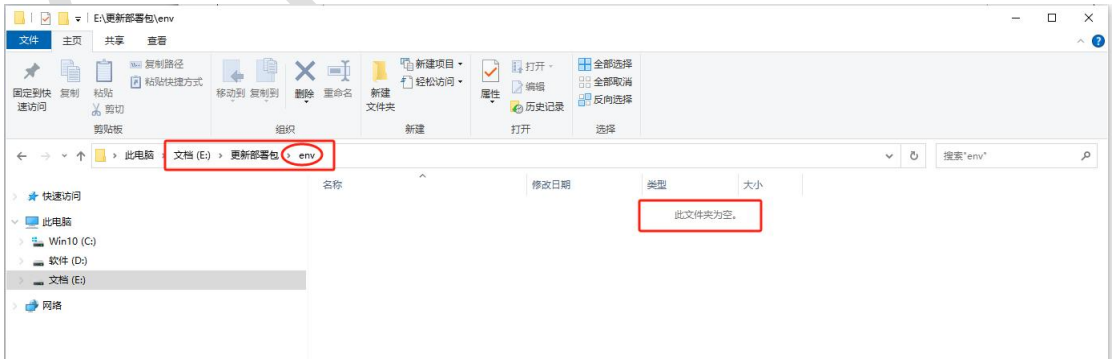
Note It must be compressed in the \ env directory so that the file is updated to the path location when the components are deployed.

2. Application examples

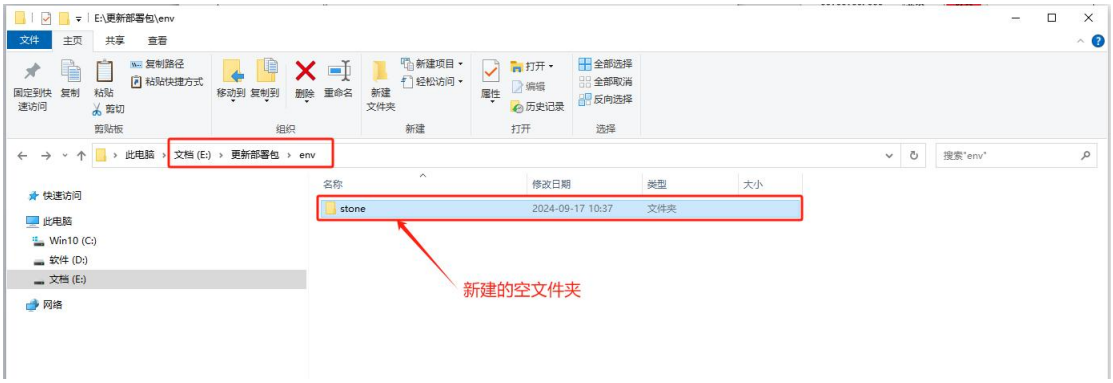
For example, prepare to make and deploy the current software in accordance with the \ env \ stone \ advance1 file (as shown in the figure below).



The first step is to build a folder where you want to save the package. An E: E. empty folder is established under disk E.



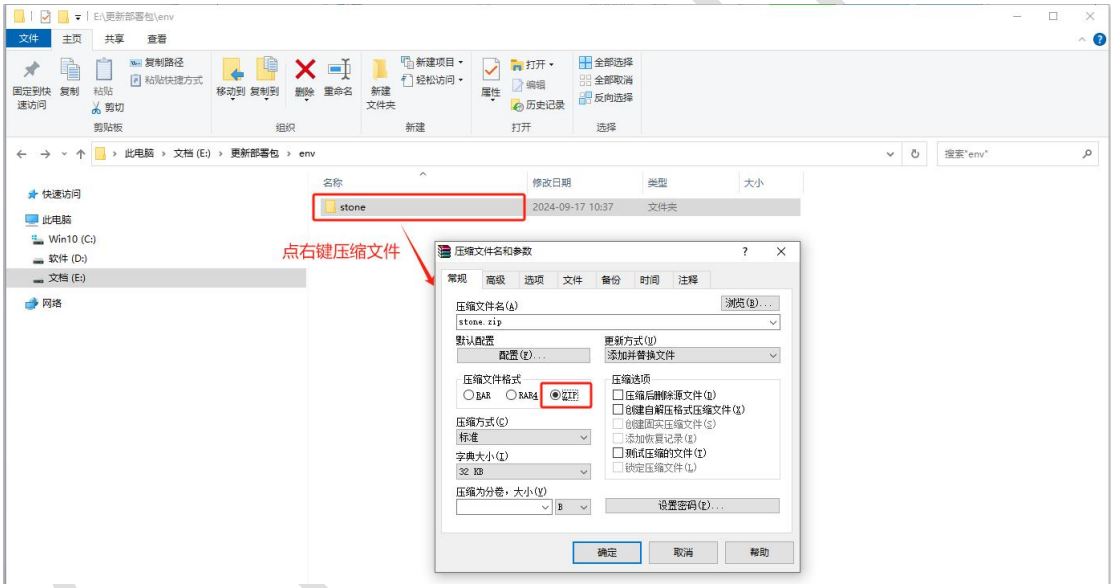
Step 2, continue to build the \ stone empty folder here.

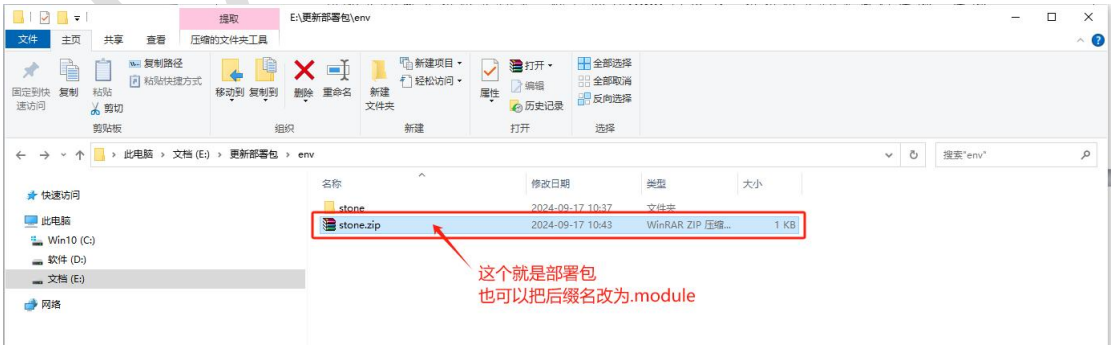


The third step, the D:\cyclMotionxjm-2024-09-06-win-android)\cyclMotion(xjm-2024-09-06-win-android)\env\stone\advance1file copy to the new empty folder E: \ update deployment package \ env \ stone.

Step 4, return to the E: \ Update Deployment Package \ Env directory.

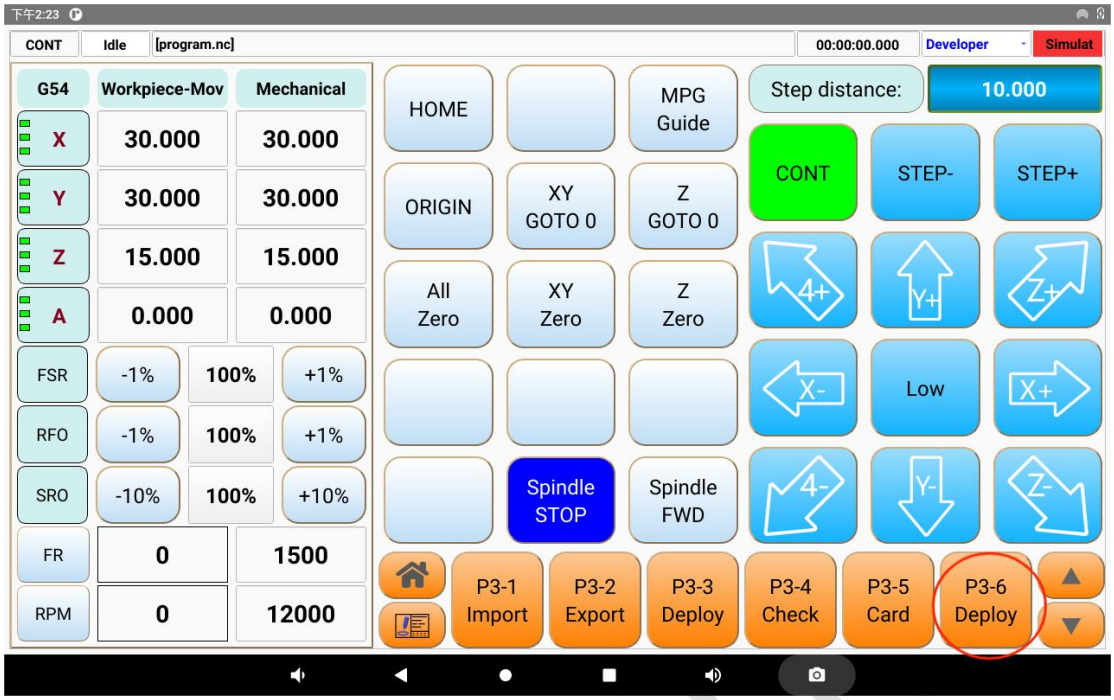
Step 5, compress the stone directory into. zip zip file. This compression package is a deployment package, or you can change the suffix name to. module.





Step 6: To deploy the components. Click on the function key component deployment, select the deployment package just generated, open, pop up the

following prompt box, yes continue, or complete the update.



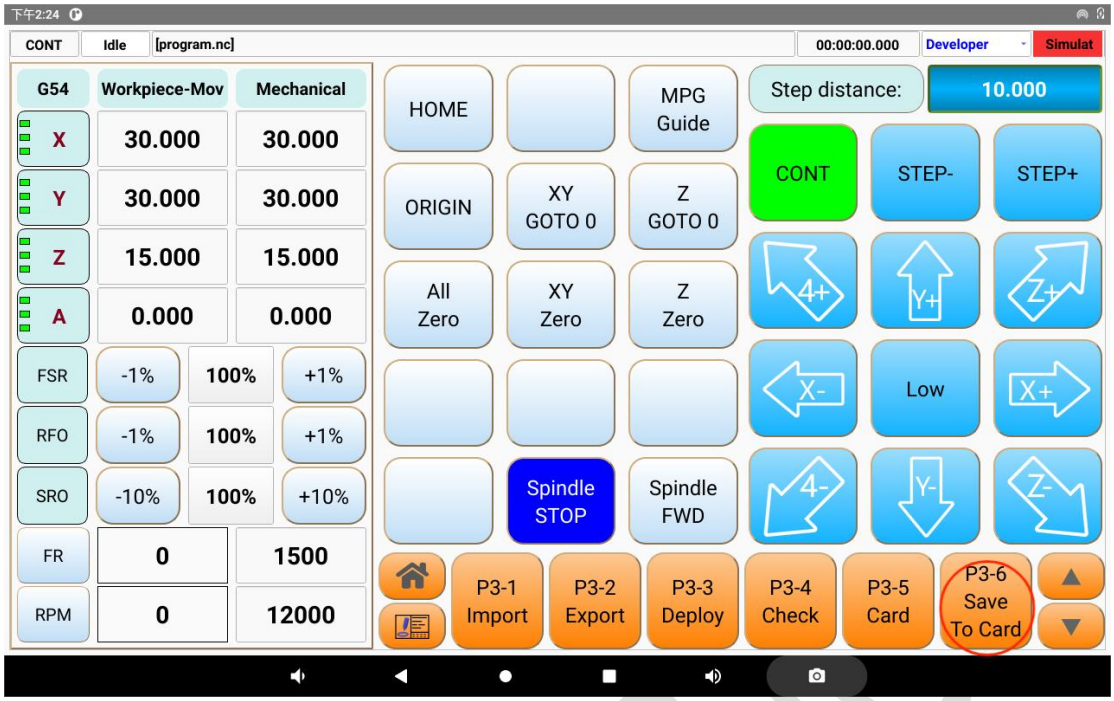
In addition, you can be exported directly in the code customization function. See the code for detailed customization-Export the deployment package。

(十九) Save the parameters to the control card

Movement control parameters (only motion control parameters, excluding interface, etc.) can be saved into the control card, only for V1.9.00 and above. Available only when available online.

Can be used to replace the all-in-one machine, first save to the control card, and then replace from the control card.

After clicking, the prompt box appears:

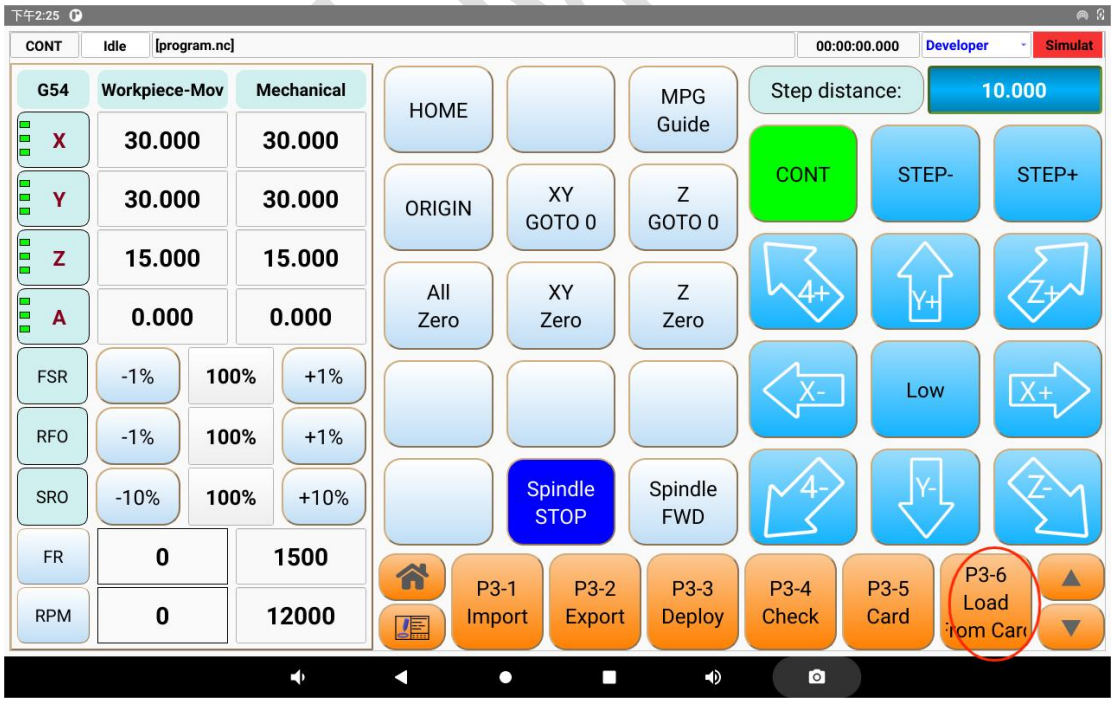


(二十) Load the control card parameter

Available only when available online.

The motion control parameters (the motion control parameters only, excluding the interface, etc.) can be loaded only for V1.9.00 and above.

After clicking, the prompt box appears:



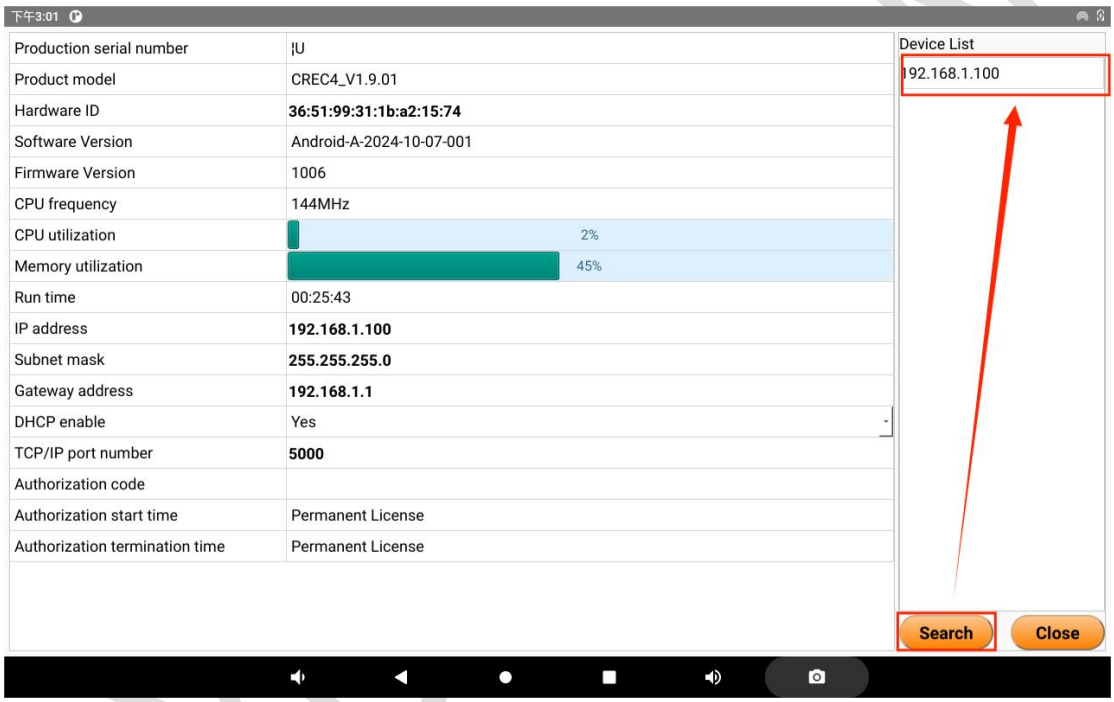
(二十一) Control Card

When online, click Search device, the existing devices will be listed in the device list, and click the device to be connected to connect.

When the control card is successfully connected, the prompt box will pop up and the green "Online" will be displayed in the status bar.

If there are more than two hosts connected to the same set of control cards, the first one connected to the main controller, and the others are monitoring machines, only monitoring authority, no control authority. The Monitoring opportunity (red) on the right of Online (green).

Running time: refers to the power-on time of the control card.



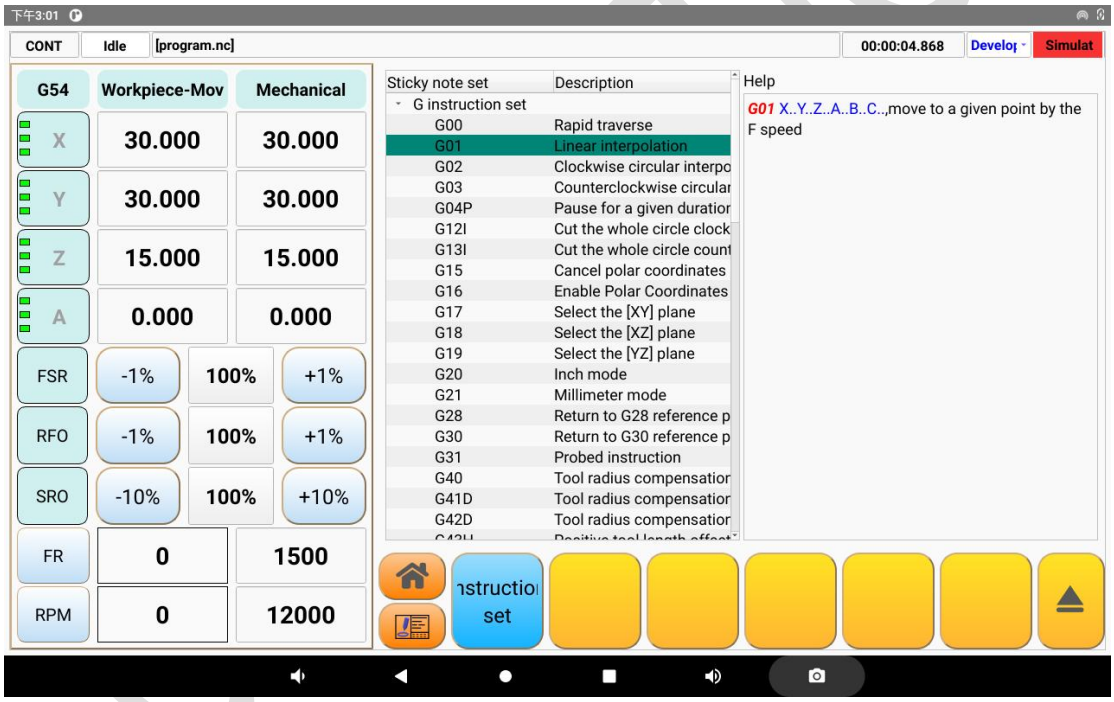
(二十二) Help

1. Order Set

There are detailed instructions and application examples for each G instruction, M instruction, and operational macro function set.

You can long press the selection and copy the instruction format to paste the application where needed.

See instructions for detailsAppendix 2 G instruction set，Appendix III. M instruction set， Appendix IV operations macro function set。



2. Contact Us

Company homepage web address:www.cyclmotion.com

Appendix I Absolute value encoder

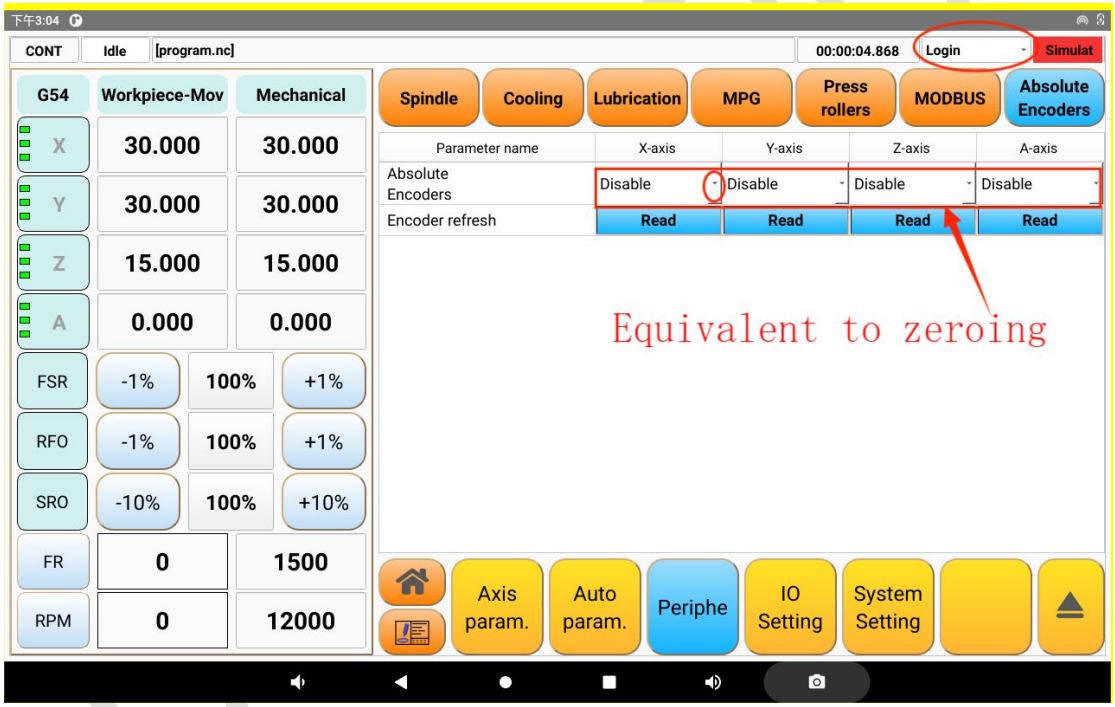
Since there is no unified standard for each absolute encoder on the market, the calibration and working principle are different. In order to have good versatility, the software provides the parameters and customized instructions of the common absolute encoders to help users to customize.

Note: Because the parameters of many types of encoders are covered, the parameters are selectively filled according to the selected type used, and the remaining parameters can be ignored.

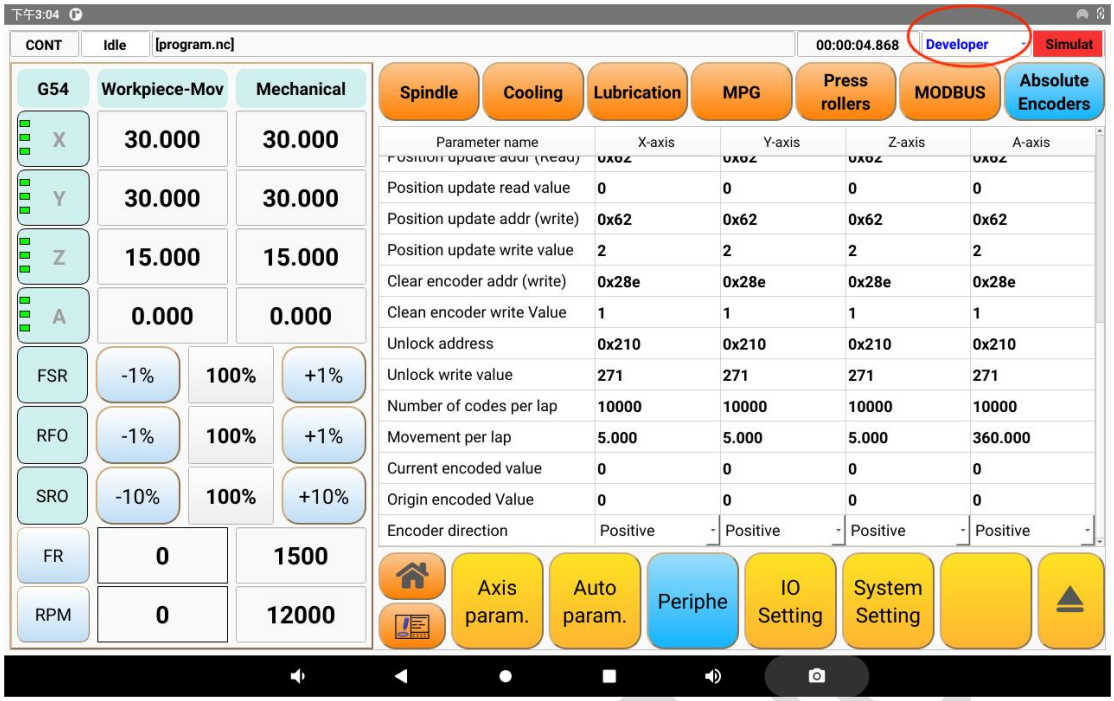
Different displays under different permissions.

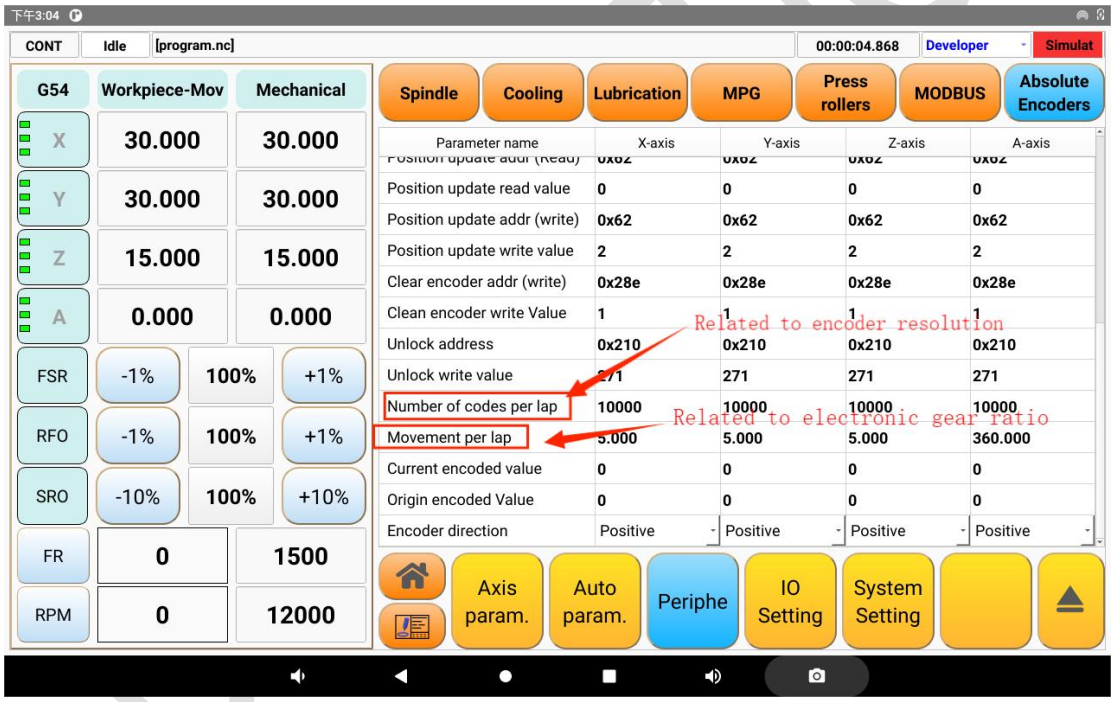
- Login permission: only use / disabled absolute value encoder and read the encoder.as shown in the figure.
- Administrator rights: you can set the address of related devices and registers, etc.
- Developer permission: can calibrate, customize the absolute value encoder.

The following interface of login permission is as follows:

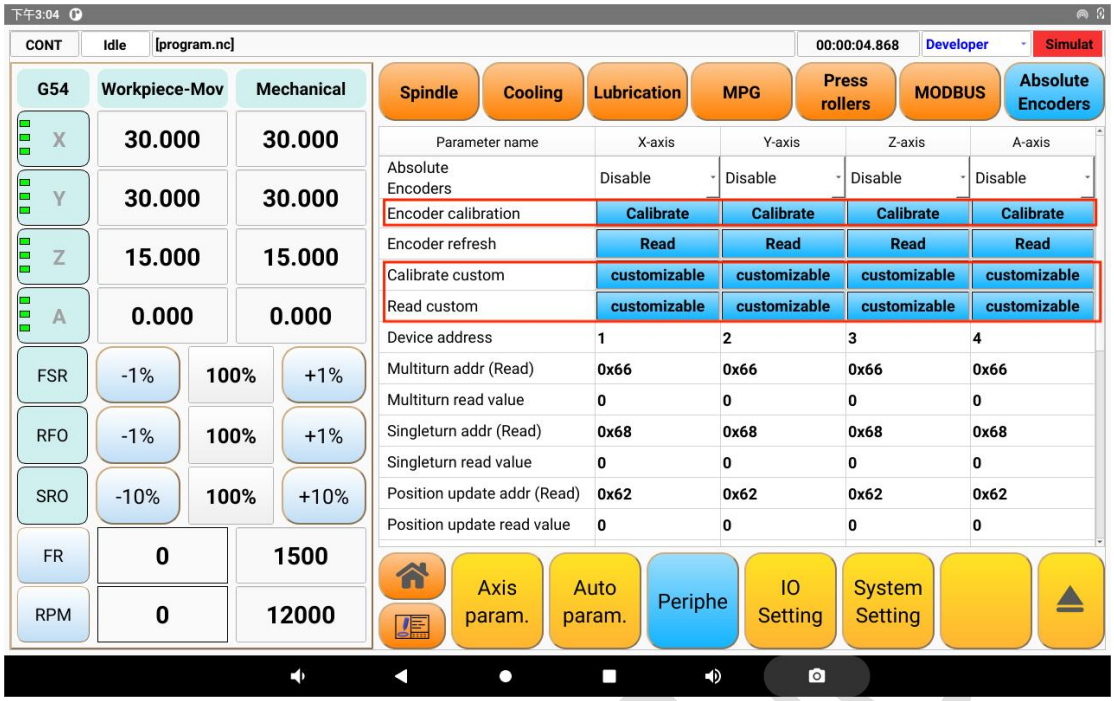


The following interface is as follows:





The interface under the developer permission is as follows:



1. Usage Method

In the first step, change the axis of applying the absolute value encoder to use instead.

The second step, the calibration code customization.see detailscustom made。

The third step, read the code customization.see detailscustom made。

Step 4, click the encoder calibration to complete the calibration. Calibration only needs to be completed once in the factory.

2. Custom Made

The right note set provides programming help:

- **Register read instruction, register write instruction: it is the instruction code that provides a single function.**
- **Related macro variable: it is the code that provides the relevant variable.**
- **Related formulas and instructions: it is the instruction code that provides common integration functions. Usually, the customization can be done by directly calling the functional combination here. When a special type of encoder, the relevant macro variables and single function instruction code can be combined and customized.**

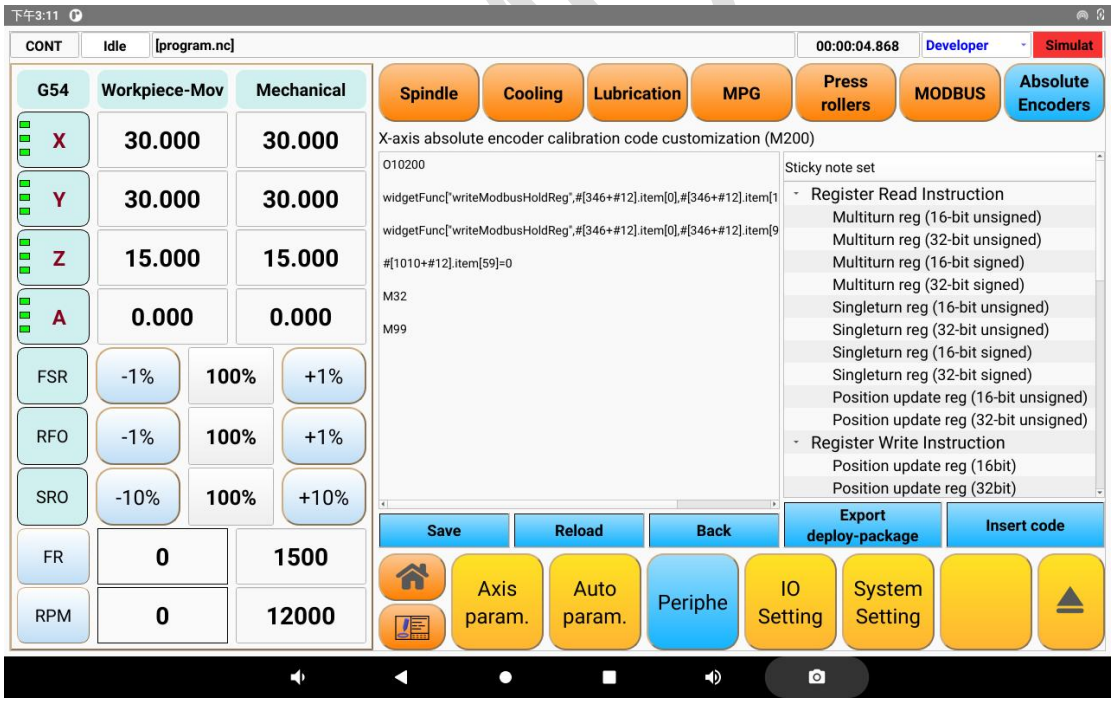
Custom method:

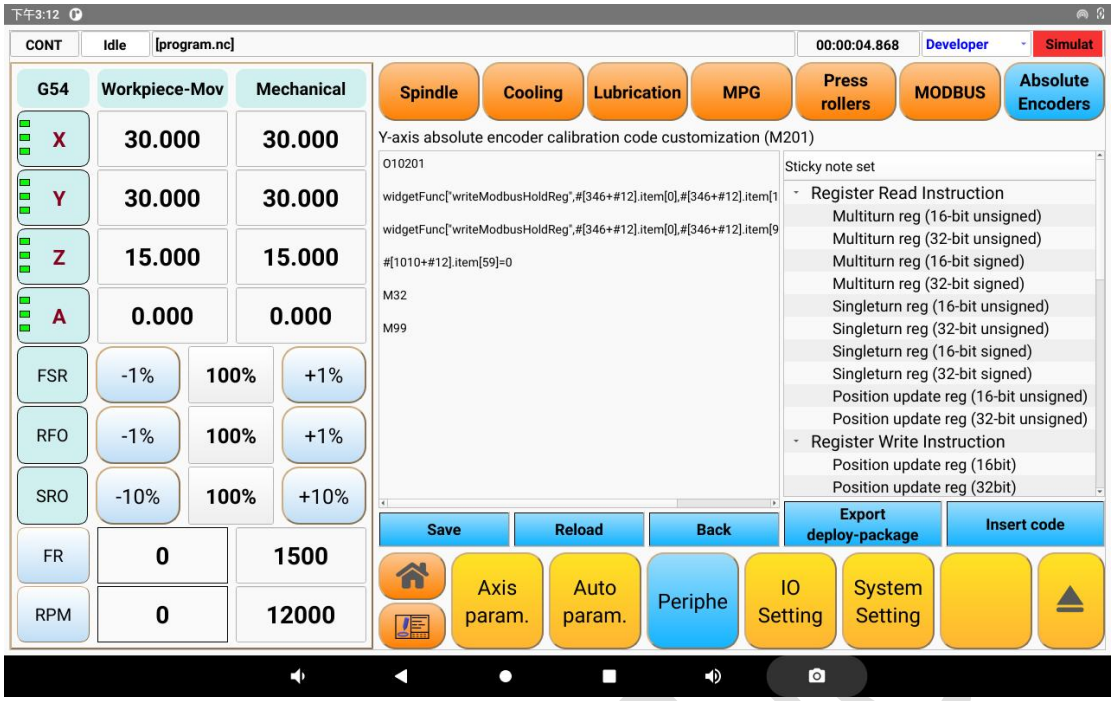
- After selecting the note set function on the right, click Insert code.
- After the customization is done, click on Save.
- If you need to export a package, click Export package.
- Click to return.

3. Application examples

Example 1, Customized calibration code for multiple loop and single loop absolute encoder:

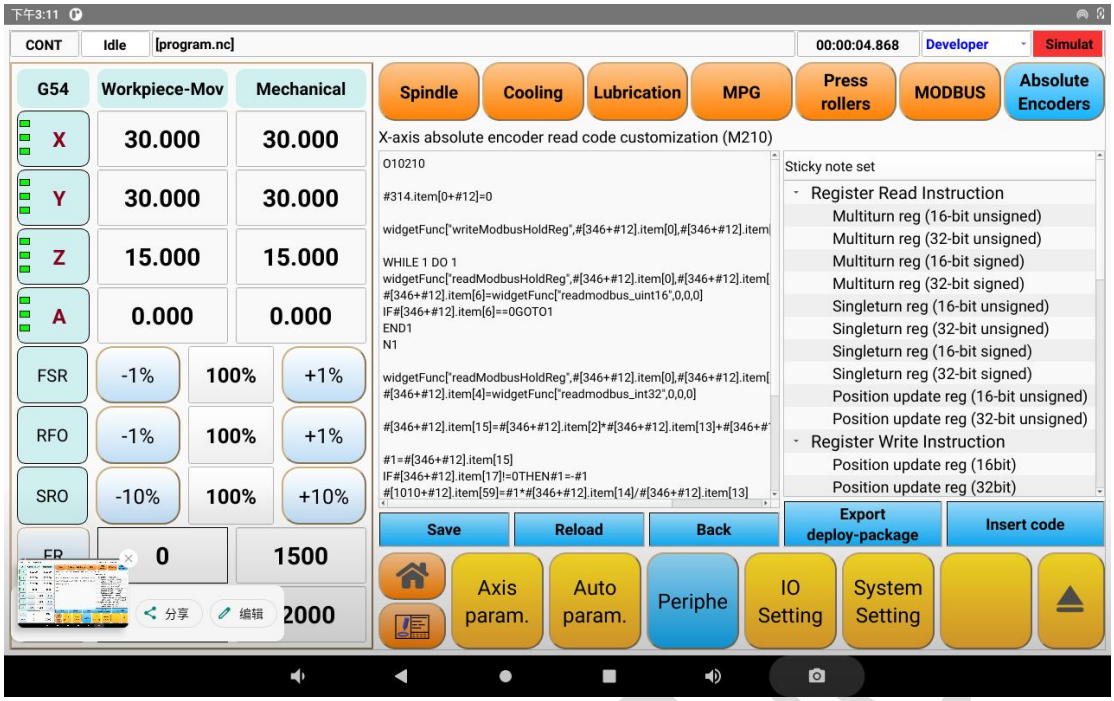
- Click on the customization to enter the customization page.
- Click the register to read the instruction —— multi-circle register (32-bit unsigned) and click the insert code.
- Click the register to read the instruction —— single circle register (32-bit unsigned), click the insert code.
- Click the relevant formula and instruction —— to calculate the current coding value according to the number of multiple circles, single circle and each circle, click to insert the code.
- Click the relevant formula and instruction —— to write the current coding value to the origin coding value, and click the insert code.
- Click the relevant formula and instructions —— Mechanical coordinate coordinates (calibration) and click insert code.
- Click the relevant formula and instructions —— to set the mechanical origin establishment completion mark and click the insert code.
- Click to save.
- Click to return.
- After moving the actual machine tool axis to the zero position, click calibration.





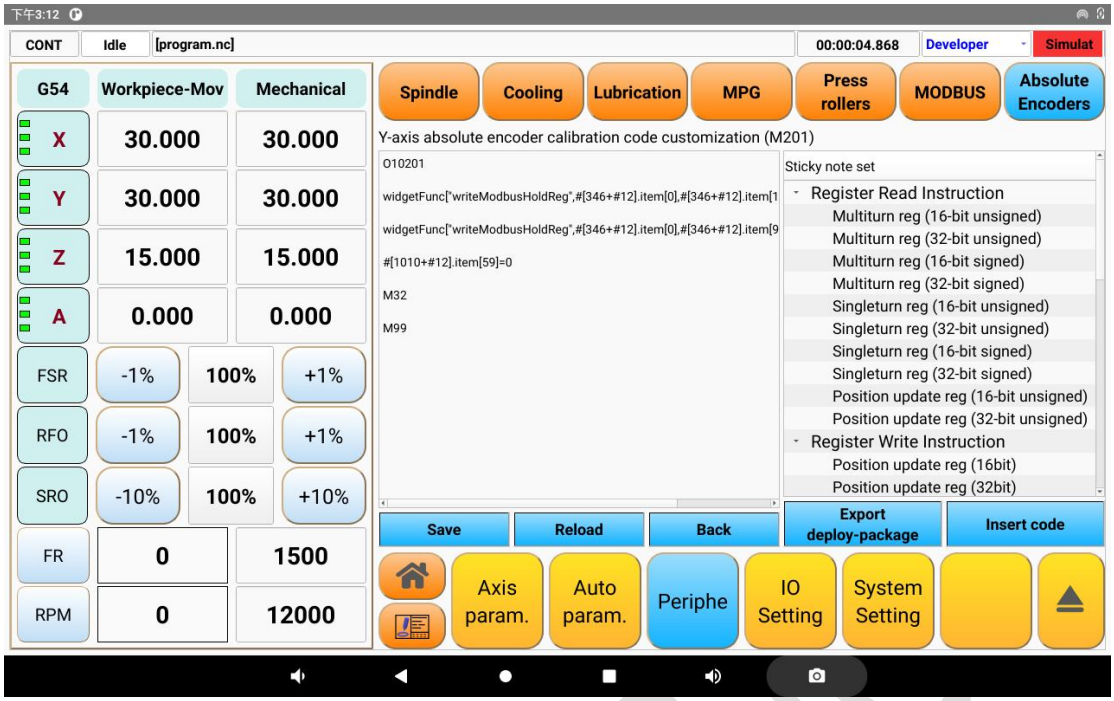
Customized read code for unsigned 32-bit multiple-loop and single-loop absolute encoders:

- Click Read customization to enter the customization page.
- Click the register to read the instruction —— multi-circle register (32-bit unsigned) and click the insert code.
- Click the register to read the instruction —— single circle register (32-bit unsigned), click the insert code.
- Click the relevant formula and instruction —— to calculate the current coding value according to the number of multiple circles, single circle and coding per circle, click to insert the code.
- Click the relevant formula and instructions —— according to the current code value and the origin code value, computer bed position, click the insert code.
- Click the relevant formula and instructions —— to set the mechanical origin establishment completion mark and click the insert code.(Zero completion mark, indicating zero completion)
- Click to save.
- Click to return.



Example 2, custom calibration code for zero class encoder:

- Click Read customization to enter the customization page.
- Click the register to write the instruction —— to unlock the register and click the insert code.
- Click register to write instruction —— clear register (32-bit unsigned), click insert code.
- Click the relevant macro variable —— machine coordinates, click the insert code, and manually enter "=0".
- Click the relevant formula and instructions —— to set the mechanical origin establishment completion mark and click the insert code.(Zero completion mark, indicating zero completion)
- Click to save.
- Click to return.



Appendix 2 Description of mobile phone remote control terminal

For Android phones or tablets only.

The mobile phone is connected to an all-in-one machine through hot spots, without the need to connect to the Internet.

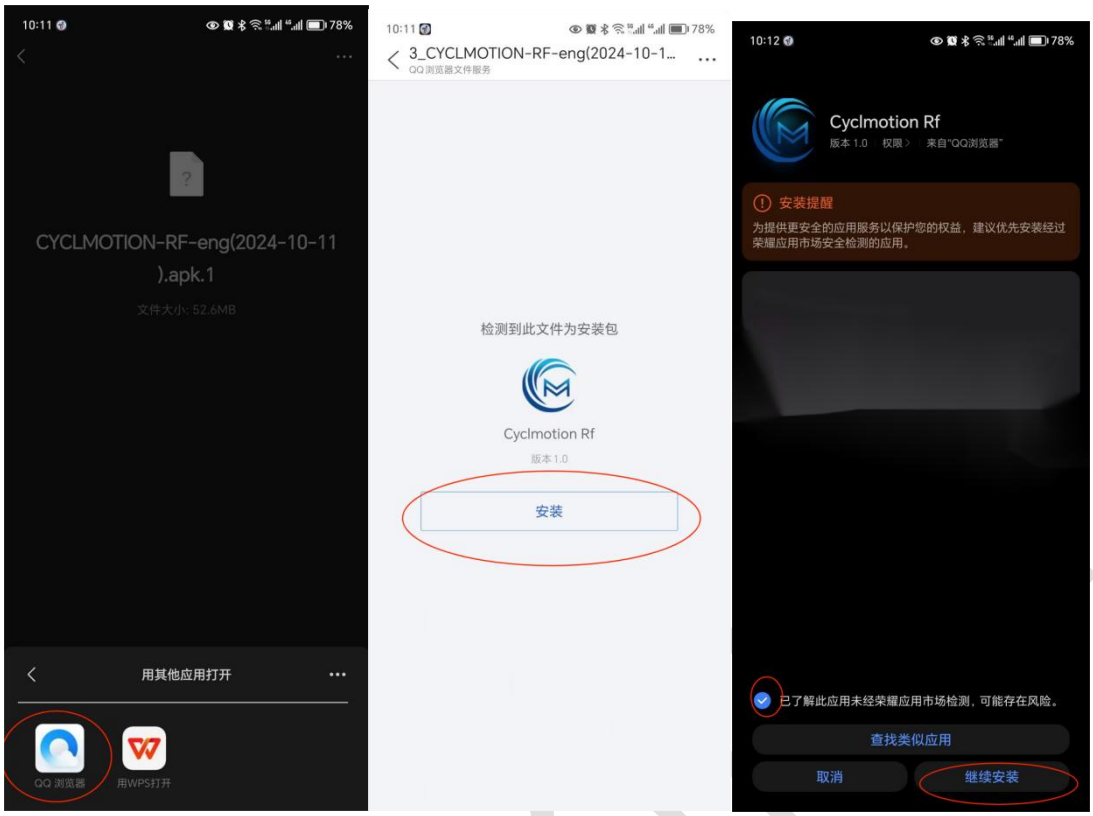
After connection, all displays and controls are synchronized with the all-in-one machine, and the virtual hand wheel can replace the hand wheel for wireless control. **Note: The remote control of the mobile phone is only a plug-in device, the shutdown or phone call will not affect the work of the host.**

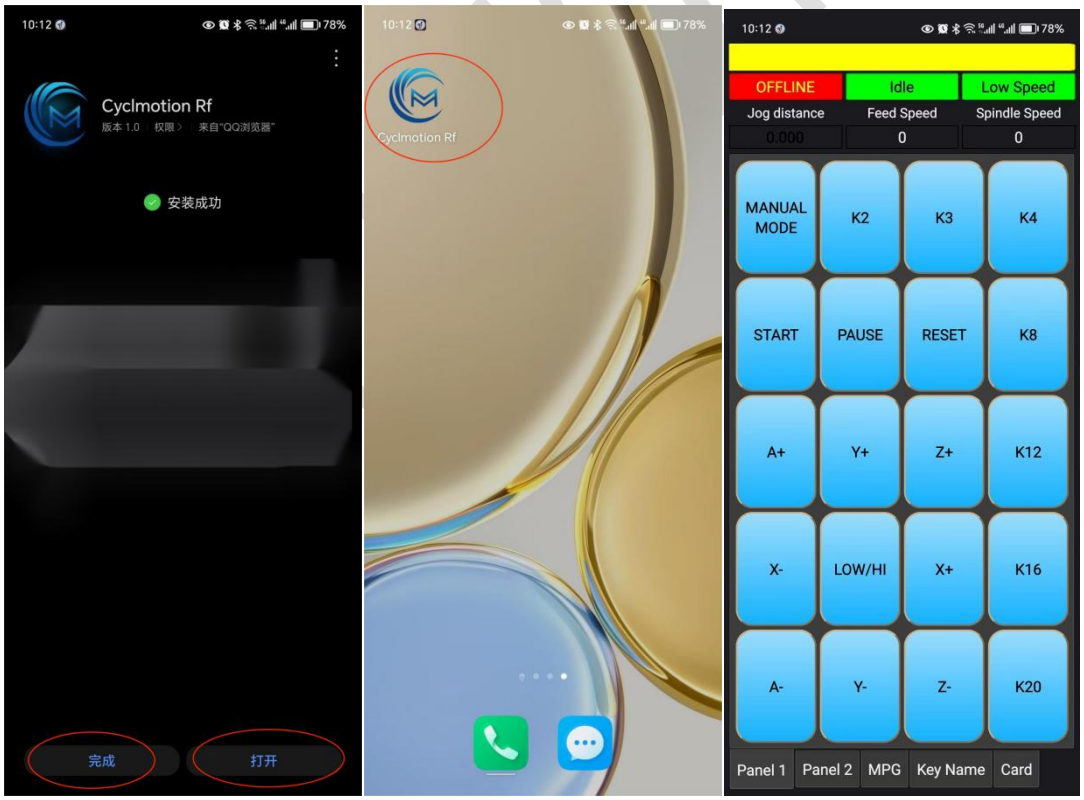
(一) Install

Install the installation package CYCLMOTION-RF-Chinese. apk.

Note: WeChat or qq will automatically be behind when receiving the installation package. 1, when the installation of the handle. 1 The suffix can be deleted.

The installation process of each brand may be slightly different, the following is the reference process.

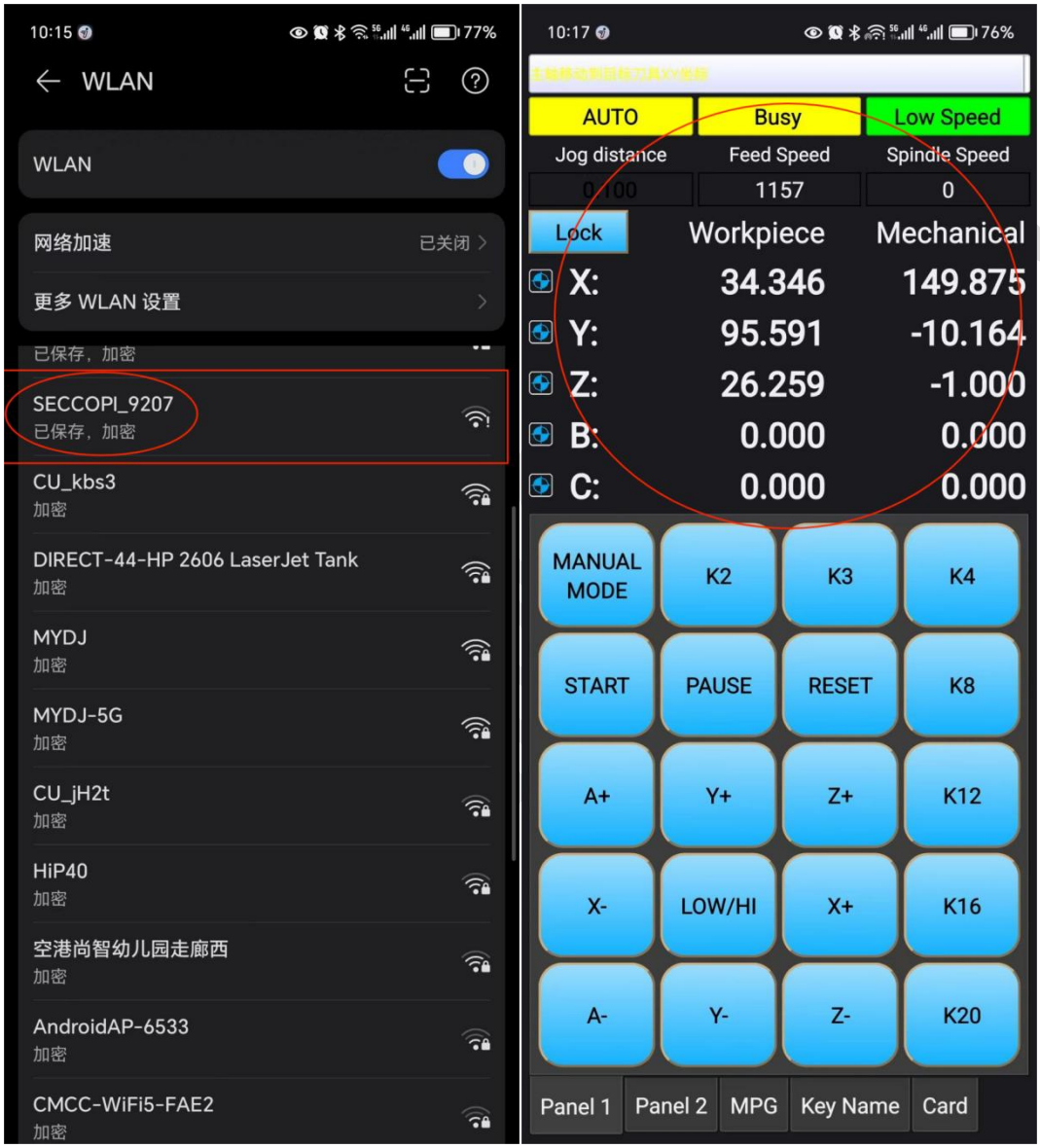




Since it is not connected to the all-in-one machine at this time, open the app after the first installation, the display interface is shown in the above, the unaxis coordinate display, etc.

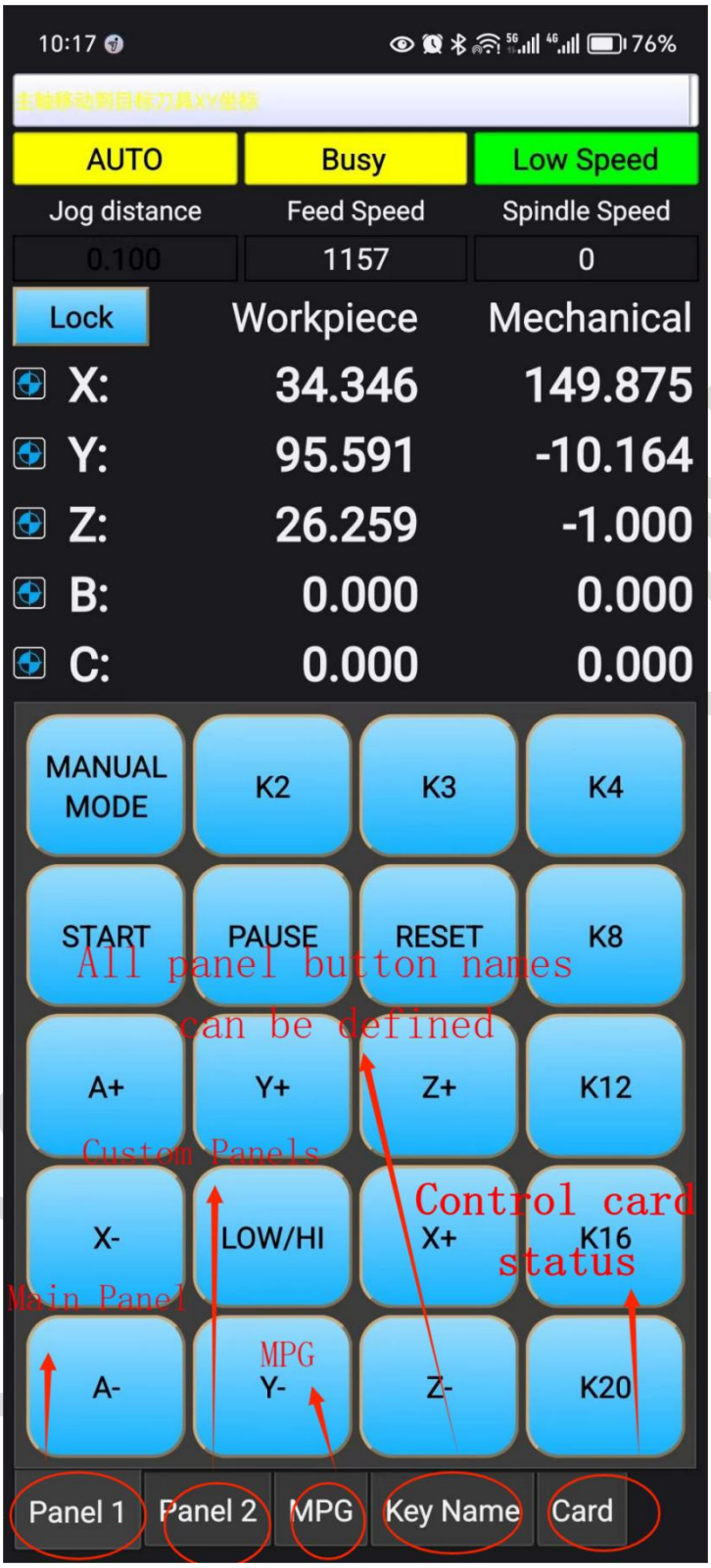
(二) Linkage

- Click Mobile Phone Settings- -WLAN to select the network starting with SECCOPI_xxxx (all-in-one hotspot, factory set), with the default password "12345678".
- Open the app again, and then automatically synchronize with the all-in-one machine, as shown in the figure below.



(三) Direction For Use

In 5 pages, as shown in the figure below.



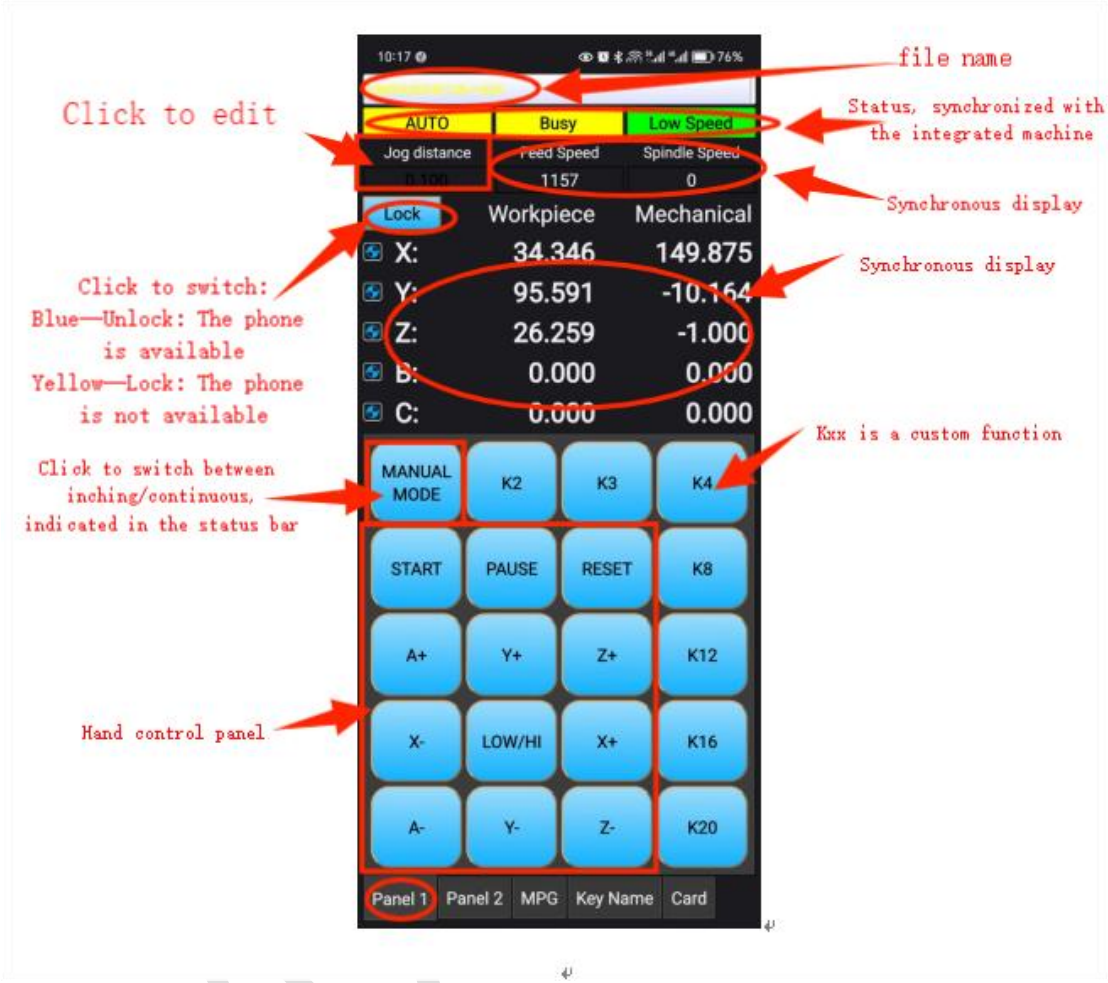
1. Main panel

All the display contents are synchronized with the all-in-one machine.

The control of the mobile terminal panel is only affected by the online status

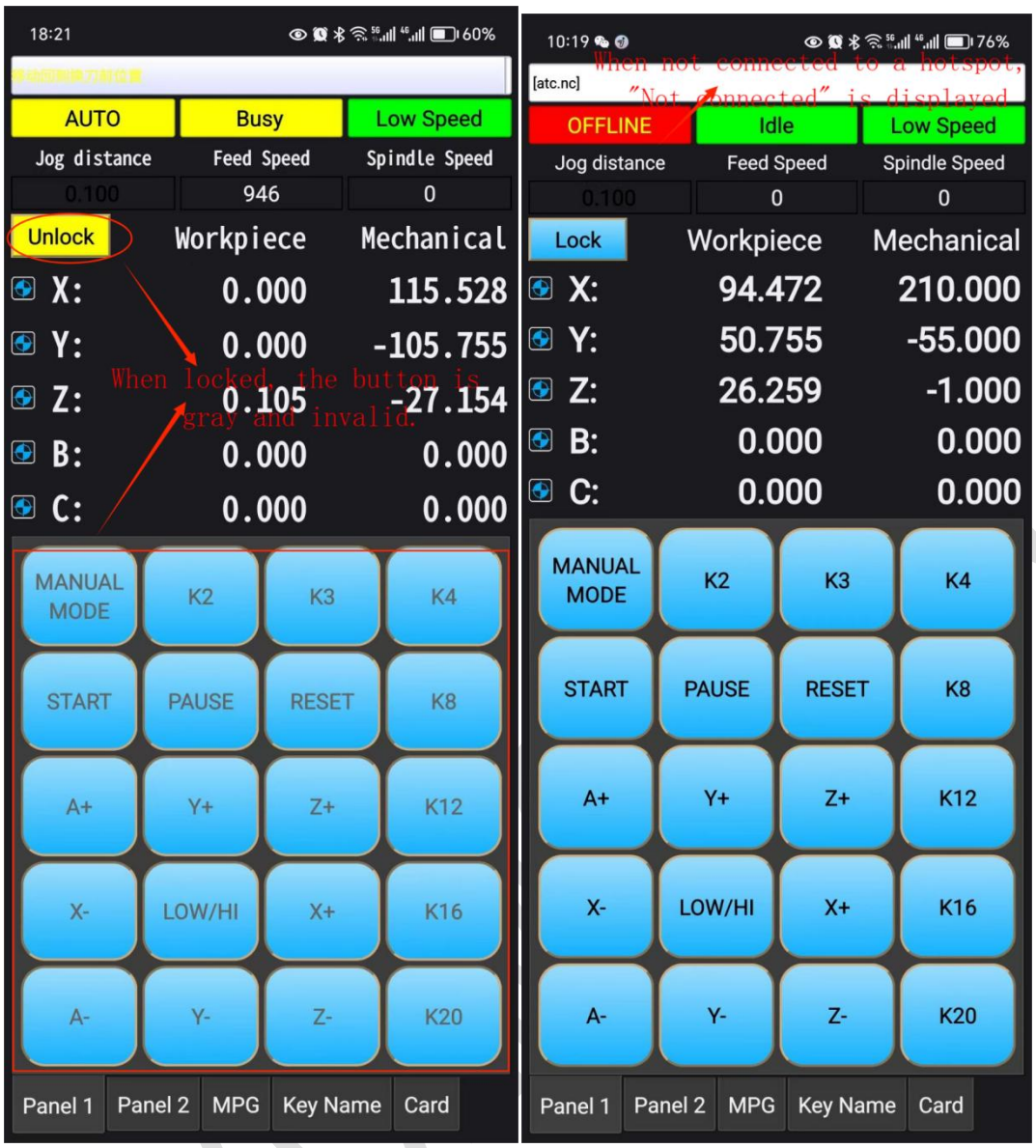
and locking status, and has nothing to do with the control of the virtual hand wheel on the mobile terminal.

All function keys can be customized, and the factory default configuration is shown in the figure below. Where Kxx is an undefined key, refer to[Custom method]Change the definition.



When the phone is locked, the buttons of the phone panel become gray, and the operation is invalid, but it can still be displayed synchronously.as illustrated in following figure.

When the phone is not online, the status bar displays red red.as illustrated in following figure.



2. Custom Panel

All keys can be customized functions and names.see[Custom method](#)。

1) Custom method

- The all-in-one machine requires permission in the developer.
- Select the all-in-one parameter setting —— IO settings —— IO shortcut key page.
- In the selected function line, click the shortcut bar to prompt "Enter shortcut key (ESC cancel recording)". At this time, click Kxx on the phone.(For more information, see alsoIO keyboard shortcutsintroduce)
- On the phone button name page, change Kxx to the corresponding function name.
- Cut back to the panel page, and click the button to customize it.

3. Virtual handwheel

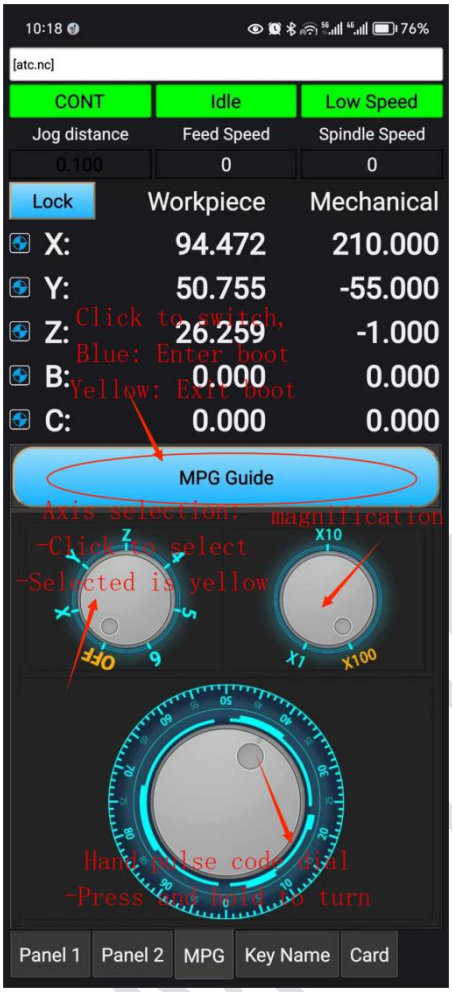
The control logic of physical wheel and virtual wheel is as follows:

- When the all-in-one machine and the board card are online, the physical hand wheel will be placed effectively.
- When the mobile phone terminal is online, the virtual hand wheel will be placed effectively.
- When the mobile phone terminal is locked, the physical hand wheel will be placed effectively. When the mobile phone is unlocked, the virtual hand wheel will be placed effectively.
- When the mobile phone is shut down normally, the virtual hand wheel exits the control right, and the physical hand wheel is valid.

  Note: Normal shutdown refers to the left from the right side of the screen and closes the software dialog box. All others are abnormal shutdown.

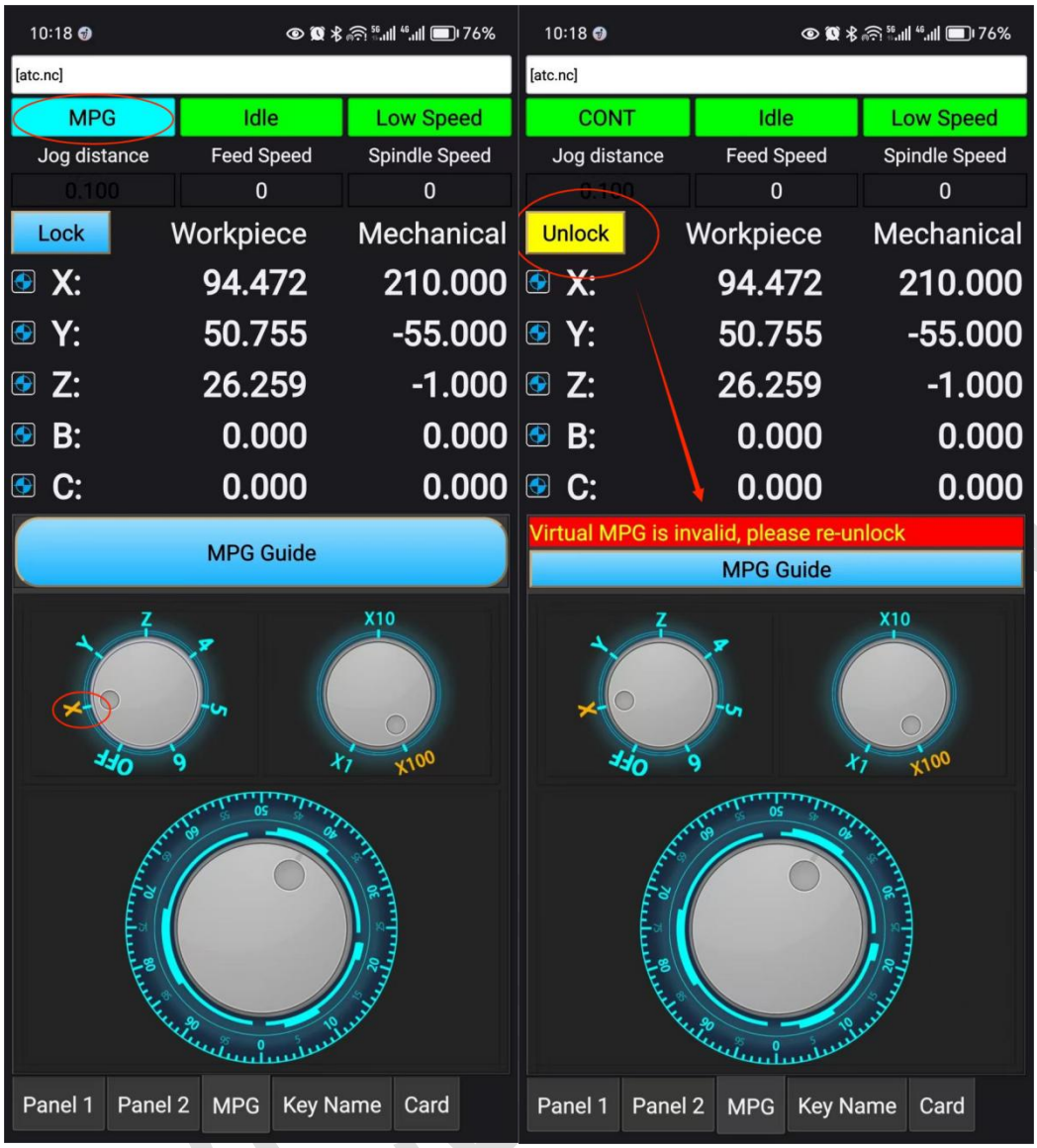
for instance:

- When the phone is in a non-locked state but out of the hotspot range, control is still on the virtual hand wheel. The host reconnects the card (on the control card page-click the device in the device list) to restore the control of the physical handwheel.
- When the mobile phone terminal is opened first, and then the all-in-one machine opens, the virtual hand wheel is invalid, and the physical hand wheel is valid. However, the mobile phone panel manual is still valid (when it is not locked state).



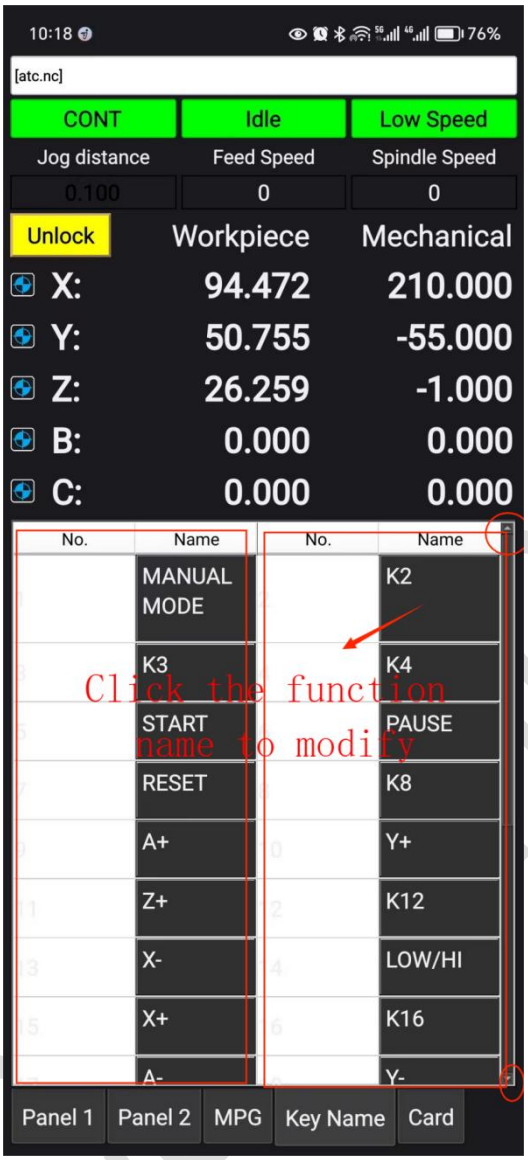
When the axis is selected as OFF, the state is continuous, and when the axis is selected as other items, the state is hand wheel. See below.

When the phone is locked, a prompt appears: the virtual wheel is invalid, please unlock it again to get the control of the virtual wheel. See below



4. Key name

The key names of both panels 1 and 2 can be changed here.



5. Control Card

Control card status, synchronization with the all-in-one machine. When the mobile phone is not online, the running time is stopped by timing.

Language selection can switch between Chinese / English.


10:19
76%
[atc.nc]
CONT
Idle
Low Speed
Jog distance
Feed Speed
Spindle Speed
0.100
0
0
Lock
Workpiece
Mechanical
X:
94.472
210.000
Y:
50.755
-55.000
Z:
26.259
-1.000
B:
0.000
0.000
C:
0.000
0.000
Production serial number
Product model
CREC6_V1.8.07
Hardware ID
99:3a:1f:a2:15:74
Firmware Version
0925
CPU frequency
144MHz
CPU utilization
4%
Memory utilization
46%
Run time
00:05:56
IP address
192.168.1.100
Subnet mask
255.255.255.0
Gateway address
192.168.1.1
Device List
Language:
English
Search Device
Panel 1
Panel 2
MPG
Key Name
Card

Appendix Methods of macro macro

The system is compatible with the FANUC macro program class B macro.

If the user uses the macro programming method (such as writing subprograms, using conditional jump statement, loop statement, etc.), the NC file name must start with "macro" (example: macroxxxxx.nc,macro_xxxx.nc), and the parser will automatically resolve the user program by macro resolution method.

Appendix IV G instruction set

1. G00 rapid positioning

Format: G00 X..Y..Z..A..B..C...
Description: Run at G0 speed.

2. G01 straight-line cutting

Format: G01X..Y..Z..A..B..C...
Description: Run to the specified location with the default feed speed or file specified F value;

3. G02 clockwise circular interpolation

Radius method: G02X..Y..Z..A..B..C..R...
Center method: G02X..Y..Z..A..B..C..I..J..K..
explain:

- Related parameters: [centrifugal acceleration of circular arc segment], [velocity ratio of circular arc segment], [high error of circular arc split string], configured on [automatic parameter] page;
- [Arc centrifugal acceleration]: When this parameter is 0, linear acceleration is used as centrifugal acceleration;
- [Arc velocity ratio]: * as the given velocity of arc segment;
- [Arc split chord height error]: used to control the arc machining error;

4. G03 counterclockwise

Radius method: G03X..Y..Z..A..B..C..R...
Center method: G03X..Y..Z..A..B..C..I..J..K..
explain:

- Related parameters: [centrifugal acceleration of circular arc segment], [velocity ratio of circular arc segment], [high error of circular arc split string], configured on [automatic parameter] page;
- [Arc centrifugal acceleration]: When this parameter is 0, linear acceleration is used as centrifugal acceleration;
- [Arc velocity ratio]: * as the given velocity of arc segment;
- [Arc split chord height error]: used to control the arc machining error;

5. G04 pause the delay command

Format: G04 P...

Where P instruction word specifies the delay time, unit ms, if P is-1, the program is suspended;

6. G12 clockwise round cutting

Format: G12 I__

Where the programming word I specifies the circle radius

Action description:

1) The X axis moves the I programming value at the programming speed F;
2) With the initial point as the center of the circle, clockwise and complete circular cutting;
3) The X-axis moves back to the initial point at the programming speed F;

example:

G00Z5
X10Y10
G01Z-3
G12I5; Take (10,10) as the center
G12I8; Take (10,10) as the center, clockwise the circle of radius 8

7. G13 counterclockwise circular cutting

Format: G13 I__

Where the programming word I specifies the circle radius

Action description:

1.X axis moves I programming value at the programming speed F;
2. Take the initial point as the center of the circle, counterclockwise circular cutting;
3. Move the X-axis back to the initial point at the programming speed F;

example:

G00Z5
X10Y10
G01Z-3
G13I5; (10,10), counterclockwise circle of radius 5
G13I8; Take (10,10) as the circle of radius 8

8. The G15 cancels the polar coordinate programming mode

Format: G15

9. The G16 turns on the polar coordinate programming mode

Format: G16 X..Y...

The X instruction specifies the polar diameter, the Y instruction word specifies the pole Angle [degree], and the origin of the polar coordinate system is the origin of the current coordinate system;

example:

G0X0Y0Z3; Rapid positioning of tool to workpiece coordinate system (0,0,3)
G52X10Y10; Create a temporary coordinate system, and the origin position is the workpiece coordinate system (10,10,3)
G99G16G81X10Y0Z-5R3; start the polar coordinate programming mode and move the tool to the work piece coordinate system (20,10,3) drilling
Y30; borehole
Y60; borehole
Y90; borehole
Y120; borehole
Y150; borehole
Y180; drilling hole
Y210; borehole
Y240; borehole
Y270; borehole
Y300; borehole
Y330; borehole
G80; exit hole loop
G15G52X0Y0; Cancel the polar coordinate coding mode and cancel the temporary coordinate system
G0X0Y0; Rapid positioning of tool to workpiece coordinate system (0,0,3)

10. G17 selects the [XY] plane as the arc insertion plane and the radius compensation plane

Format: G17
explain:

When specified after the G17:

- The arc interpolation instruction (G2 / G3) will interpolate in the [XY] plane;
- The tool radius compensation instruction (G41 / G42) will compensate for the tool radius of the [XY] plane contour;

11. G18 selects the [ZX] plane as the arc interpolation plane and the radius compensation plane

Format: G18
explain:

When specified after the G18:

- Arc interpolation instruction (G2 / G3) will perform arc interpolation in the [ZX] plane;
- The tool radius compensation instruction (G41 / G42) will compensate the tool radius for the [ZX] plane contour;

12. G19 selects the [ZY] plane as the arc insertion plane and the radius compensation plane

Format: G19
Explain:

When specified after the G19:

- The arc interpolation instruction (G2 / G3) will interpolate in the [ZY] plane;
- The tool radius compensation instruction (G41 / G42) will compensate the tool radius for the [ZY] plane contour;

13. G20 r input

Format: G20
Explain:

- Programming size word (X / Y / Z) input in inches
- When the [automatic parameter-path planning] page [line axis default programming unit] is configured as an inch, the unit selection instruction is G20

14. G21 Metric input

Format: G21
Explain:

- The programming size word (X / Y / Z) is input in mm
- When the [automatic parameter-path planning] page [line axis default programming unit] is configured as metric mm, the unit selection instruction is G21

15. G28 passes back through the midpoint to the G28 reference point

Format: G28 X..Y..Z..A..B..C...
Explain:

- If the zero point signal is not configured with the corresponding axis, then only move to the middle point;
- If each shaft has completed the mechanical return to zero operation, first move to the middle point, and then return to the G28 reference point;
- If the mechanical return to zero operation is not completed, move to the middle point and then perform the mechanical return to zero operation;

point out:

- The G28 reference point is set under the [parameter setting] -> [axis parameter] -> [reference point] page;
- To ensure that the G28 reference point is still located without the mechanical zero operation, please execute the instruction again in G91 mode;

Example 1:

G90G28X0Y0Z0A0; Move to the work coordinates (0,0,0,0) and then return to reference point 1
G91G28X0Y0Z0A0; Ensure the final positioning to the G28 reference point without mechanical zero operation

Example 2:

The G91G28Z0; Z axis returns from the current position to the G28 reference point
G91G28Z0; Call again to ensure that the Z-axis is finally located to the G28 reference point without performing the mechanical return to zero operation

16. The G30 passes back through the middle point to the G30 reference point P

Format: G30 X..Y..Z..A..B..C..P...

The P programming word is used to specify the G30 reference point number, the effective value domain is [1,3], and the P value is default or overbound, select the G30P1 reference point
explain:

- If the zero point signal is not configured with the corresponding axis, then only move to the middle point;
- If each shaft has completed the mechanical zero operation, move to the middle point and then return to the G30P [1,3] reference point;

- If the mechanical return to zero operation is not completed, move to the middle point and then perform the mechanical return to zero operation;

point out:

- The G30P [1,3] reference point is set under the [parameter setting] -> [axis parameter] -> [reference point] page;
- To ensure that the G30P [1,3] reference point is still finally located without completing the mechanical return to zero operation, please execute the command again in the G91 mode;

Example 1:

G90G30X0Y0Z0A0P1; Move to the work coordinates (0,0,0,0) and then return to the G30P1 reference point

G91G30X0Y0Z0A0P1; Ensure that the final positioning to the G30P1 reference point without the mechanical return zero operation

Example 2:

The G91G30Z0P3; Z axis returns from the current position to the G30P3 reference point

G91G30Z0P3; Call again to ensure that the Z-axis is finally located to the G30P3 reference point without performing the mechanical return to zero operation

17. The G31 probe instruction

Format: G31 X..Y..Z..A..B..C..P..L..Q..K..F...

Where the [XYZABC] programming word specifies the detection stroke of each axis of the probe instruction;

[P] programming word specifies the detection signal source, default or 0, use the preset detection signal source;

[L] The programming word specifies the effective level of the detection signal;

[K] programming word specifies whether the hard limit in the probe is closed 0-off 1-not closed;

[Q] programming words specify the stop mode after the detection signal appears 0-deceleration stop 1-stop immediately;

[F] The programming word specifies the detection speed;

Explain:

During the command operation, if the detection signal is found, the movement stops;

point out:

- During the operation of G31 instruction, the detection signal of any axis with the motion stroke is changed to an effective level, and the detection is stopped;

- If the hard limit is set to close [K=0], the probe axis hard limit is disabled during detection;
- If the hard limit is set to not closed [K=1], the hard limit policy of the detection axis is not changed during the detection period (note that the hard limit is not forced to open). If the hard limit is enabled and the hard limit is triggered in the detection, the movement is stopped and the system is reset;
- G31 instruction motion acceleration is controlled by [line acceleration] in [automatic parameter-acceleration related] page;

Example 1:

G91G31Z-1000L0Q1K0F100

The Z axis detects 1000 mm downward at 100 speed, the effective level of the detection signal is low, the hard limit is closed in the detection, and the detection signal of the Z axis stops moving immediately after the appearance;

Example 2:

G0X0Y0

G90G31X100Y100L1Q0K1

Detection the XY axis (0,0) to (100,100), the detection effective level is high, the hard limit is opened in the detection, and the X axis (or Y axis) detection signal appears;

Advanced applications:

The probe signal source of the axis can be changed using the macro variable, where:

1010.item[37] Specifying the detection signal source for the X programming axis;

1011.item[37] The probe signal source for assigning the Y programming axis;

1012.item[37] Detection signal source for assigning the Z programming axis;

1013.item[37] The probe signal source for assigning the A programming axis;

1014.item[37] Specifying the detection signal source for the B programming axis;

1015.item[37] The probe signal source for assigning the C programming axis;

Example 3:

1010.item[37] =6; the X-axis probe signal source is set to port 6

G91G31X-1000L0Q0K0F3000; probe

1010.item[37] =8; The X-axis probe signal source is set to port # 8

G91G31X100L0Q1K0F100; Detection

18. The G40 cancels the tool radius compensation

Format: G40

19. G41 left-bias tool radius compensation

Format: G41 X..Y..Z..A..B..C..D...

Where the D programming word specifies the tool diameter compensation number

Point Out:

- The radius compensation plane is determined by the instruction G17 / G18 / G19;
- Enter the tool diameter or diameter wear on the [Tool Management-Tool Parameters] page for compensation instruction call;
- The compensation establishment section or cancellation section must be a straight line section, and there is an amount of exercise in the compensation plane, otherwise the system gives the [radius compensation establishment or cancellation error] warning, and the nc program analysis ends;
- After the compensation is established, if the subsequent compensation code has no motion path in the compensation plane, the system gives a warning of [radius compensation establishment or cancellation error], and the nc program resolution ends;
- In the process of compensation calculation, if the interference is found, the system gives the [radius compensation error] warning, and the nc program analysis ends;
- The length of the compensation path segment is recommended to be greater than the tool diameter, otherwise the knife path may be overcut or appear [radius compensation error] warning after the compensation;
- Reasonable design of compensation establishment and cancellation section, and simulation, determined after the physical processing;

Example: (unopened tool radius compensation machining work profile)

G90G00X-20Y-20
Z-3
G01X0Y0
Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0
X45Y10
X0Y0
X-20Y-20

(Open left side tool radius compensation machining piece contour)
G01G41X0Y0D1; D1 tool diameter of 10

```
Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0
X45Y10
X0Y0
G40X-20Y-20
```

20.G42 right-bias tool radius compensation

Format: G42 X..Y..Z..A..B..C..D...

Where the D programming word specifies the tool diameter compensation number

Point Out:

- The radius compensation plane is determined by the instruction G17 / G18 / G19;
- Enter the tool diameter or diameter wear on the [Tool Management-Tool Parameters] page for compensation instruction call;
- The compensation establishment section or cancellation section must be a straight line section, and there is an amount of exercise in the compensation plane, otherwise the system gives the [radius compensation establishment or cancellation error] warning, and the nc program analysis ends;
- After the compensation is established, if the subsequent compensation code has no motion path in the compensation plane, the system gives a warning of [radius compensation establishment or cancellation error], and the nc program resolution ends;
- In the process of compensation calculation, if the interference is found, the system gives the [radius compensation error] warning, and the nc program analysis ends;
- The length of the compensation path segment is recommended to be greater than the tool diameter, otherwise the knife path may be overcut or appear [radius compensation error] warning after the compensation;
- Reasonable design of compensation establishment and cancellation section, and simulation, determined after the physical processing;

Example: (unopened tool radius compensation machining work profile)

```
G90G00X45Y35
Z-3
G01X45Y10
X0Y0
```

Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0
X45Y10
X45Y35

(Open the right tool radius to offset the machining piece outline)
G01G42X45Y10D1
X0Y0
Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0
X45Y10
G40X45Y35

21. G43 tool bias forward correction instruction

Format: G43 H...

Where the H programming word specifies the tool length compensation number

Point Out:

- Enter the length compensation value or length wear amount on the [Tool Management-tool parameters] page for the compensation instruction call;
- To cancel the tool offset correction, pass the G49 or H00 instruction; (G49: knife length offset. H00 indicates a bias value of zero)

Example: (set the tool length compensation value of no. 1 to be 5, and the coordinate of X / Y / Z axis workpiece is 0 / 0 / 0 when the tool offset compensation is not opened)

G43H1; X / Y / Z coordinate coordinates 0 / 0 / -5
G43H0; X / Y / Z axis workpiece coordinates are displayed as 0 / 0 / 0

22. G43.4 Tool Center Point Control Forward Correction Order (RTCP)

Format: G43.4H...

Where the H programming word specifies the tool length compensation number
Explain:

After opening the tool center point control function, the programming instruction word is no longer the exercise amount of each axis, but is the workpiece coordinate value of the workpiece model;

Example: (Example contains two identical codes, the first RTCP is not on and the second RTCP is on)

G49
X0Y-5Z5A0F1000
Y5Z5
A-90
Y5Z-5
A-180
Y-5Z-5
A-270
Y-5Z5
A-360

G43.4H1
X0Y-5Z5A0F1000
Y5Z5
A-90
Y5Z-5
A-180
Y-5Z-5
A-270
Y-5Z5
A-360
G49

23. G44 tool bias negative correction instruction

Format: G44 H...

Where the H programming word specifies the tool length compensation number
Point Out:

- Enter the length compensation value or length wear amount on the [Tool Management-tool parameters] page for the compensation instruction call;
- To cancel the tool offset correction, pass the G49 or H00 instruction; (G49: knife length offset. H00 indicates a bias value of zero)

Example: (set the tool length compensation value of no. 1 to be 5, and the coordinate of X / Y / Z axis workpiece is 0 / 0 / 0 when the tool offset compensation is not opened)

G44H1; X / Y / Z coordinate coordinates 0 / 0 / 5

G44H0; X / Y / Z axis workpiece coordinates are displayed as 0 / 0 / 0

24. G44.4 Tool Center Point Control Forward Correction Order (RTCP)

Format: G44.4H...

Where the H programming word specifies the tool length compensation number

Explain:

After opening the center point control function of the tool, the programming instruction word is no longer the exercise amount of each axis, but the workpiece coordinate value of the workpiece model;

Example: (Example contains two identical codes, the first RTCP is not on and the second RTCP is on)

G49
X0Y-5Z5A0F1000
Y5Z5
A-90
Y5Z-5
A-180
Y-5Z-5
A-270
Y-5Z5
A-360

G44.4H1
X0Y-5Z5A0F1000
Y5Z5
A-90
Y5Z-5
A-180
Y-5Z-5
A-270
Y-5Z5
A-360
G49

25. The G49 cancels the tool offset compensation instruction

Format: G49

26.G52 local coordinate system establishment instruction

Format: G52 X..Y..Z..A..B..C...

The XYZABC programming word is the offset of the local coordinate system to be established and the origin of the current artifact coordinate system

Point Out:

- The G52 instruction can arbitrarily set the coordinate system to be processed without changing the work piece coordinate system (G54~G59);
- The G52 instruction is specified and remains valid until the next G52 instruction is specified;
- The machine tool does not move when the G52 instruction is specified;
- G52X0Y0Z0A0B0C0 Local coordinate system can be cancelled;
- At the end of the processing program, the local coordinate system is automatically cancelled;

Example: (drilling with local coordinate system using G52)

G0X0Y0Z3; Quick positioning of the tool to the workpiece coordinate system (0,0,3)

G52X10Y10; Create a local coordinate system, and the origin position is the workpiece coordinate system (10,10,3)

G99G16G81X10Y0Z-5R3; start the polar coordinate programming mode and move the tool to the work piece coordinate system (20,10,3) drilling

Y30; borehole

Y60; borehole

Y90; borehole

Y120; borehole

Y150; borehole

Y180; borehole

Y210; borehole

Y240; borehole

Y270; borehole

Y300; borehole

Y330; borehole

G80; exit hole loop

G15G52X0Y0; Cancel the polar coordinate encoding mode and cancel the local coordinate system

G0X0Y0; Rapid positioning of tool to workpiece coordinate system (0,0,3)

27.G53 machine tool coordinate system moving instruction

Format: G53 X..Y..Z..A..B..C...

Description: Run to the specified machine tool location

point out:

- The corresponding configuration shaft must have completed the mechanical return to zero operation, otherwise the program alarm exits

Example:

G53X10Y10; XY axis is moved to the machine coordinates (10,10)

28. G53.1 Axial control instructions

Format: G53.1

Description: Using the instruction will make the tool axis perpendicular to the XY plane of the characteristic coordinate system specified by the G68.2 instruction

point out:

- G53.1 needs to be invoked in G68.2 mode, generally followed by G68.2;
- G53.1 must be used alone and cannot appear in the same segment as the other G code

29. G54 artifact coordinate system selection instruction

Format: G54

Point Out:

- Select the coordinate system in the upper left corner on the home page. G54

30. The G55 artifact coordinate system selection instruction

Format: G55

Point Out:

- Select the coordinate system in the upper left corner on the home page. G54

31. The G56 artifact coordinate system selection instruction

Format: G56

Point Out:

- Select the coordinate system in the upper left corner on the home page. G54

32. G57 artifact coordinate system selection instruction

Format: G57

Point Out:

- Select the coordinate system at the upper left corner on the home page. G54

33. The G58 artifact coordinate system selection instruction

Format: G58
Point Out:

- Select the coordinate system at the upper left corner on the home page. G54

34. The G59 artifact coordinate system selection instruction

Format: G59
Point Out:

- Select the coordinate system at the upper left corner on the home page. G54

35. G68 opens the coordinate system rotation instruction

Format: G68 X..Y..Z..R...

Among:

[XYZ] specifies the coordinate value of the center of rotation (can be any two of X, Y, Z, determined by one of the current plane selection instructions G17 / G18 / G19). In default, use the current position as the center of rotation;

[R] programming word is the rotation angle, counterclockwise rotation is defined as the positive direction, clockwise rotation is defined as the negative direction;

Example:

G68X45Y35R45; Take (45,35) as the center of rotation and rotate 45 degrees counterclockwise

(Tool tool radius not opened)
G90G00X45Y35
Z-3
G01X45Y10
X0Y0
Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0

```
X45Y10
X45Y35

(Open the right tool radius to offset the machining piece contour)
G01G42X45Y10D1
X0Y0
Y35
X20
G03X25Y60R65
G02X65R-25
G03X70Y35R65
G01X90
Y0
X45Y10
G40X45Y35

G69; Cancel the coordinate system rotation
```

36. G68.2 Open the inclined surface processing mode instruction

Format: G68.2 X..Y..Z..I..J..K...

Among:

[XYZ] The programming word specifies the origin coordinates of the feature coordinate system;

[IJK] programming word specifies the Euler angle of the direction of the feature coordinate system (rotation order: first around the Z axis, then around the X axis, and then around the Z axis);

Point Out:

- X, Y and Z are specified by the absolute value of the original coordinate system. If X, Y and Z are omitted, the origin of the original coordinate system will become the origin of the characteristic coordinate system;
- If I or J or K is omitted, the omitted programming word is considered 0 specified; if IJK is omitted, the system automatically assigns the angle with the current tool axis;
- G68.2 only establishes the characteristic coordinate system, which does not make the machine tool move. Generally, follow the G53.1 instruction to adjust the tool axis to be perpendicular to the characteristic coordinate system XY plane.

37. G69 undo the coordinate system rotation or tilt plane machining mode instruction

Format: G69

38. G73 high-speed peck type deep hole drill cycle

Format: G73 X..Y..Z..A..B..C..R..Q..K..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91 mode, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;

The [R] programming word specifies the Z coordinate of the R point;

[Z] The programming word specifies the Z coordinate of the hole bottom;

[Q] The programming word specifies the depth of each pecking hole;

[K] programming word specifies the peck hole back knife distance;

[L] Specifies the number of boreholes;

Action description:

- Tool X / Y / A / B / C axis is quickly positioned to the hole position;
- The tool moves quickly in the Z direction to the point R;
- The tool drills the Q distance at the specified speed F, and then quickly lifts the K distance;
- Thereafter, the tool drill Q + K distance at the specified speed F, then lifts the K distance quickly; repeat this action until drilling reaches the bottom Z;
- The tool is quickly raised to point R;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-5 until all the hole position processing is completed;
- When using the G98 instruction, the tool Z-axis quickly returns to the initial plane position.

Example:

G90G0Z3

G16G1X5Y90; Open the polar coordinate programming mode and move the tool to (0,5,3)

G99G91G73Y30Z-7R3Q5K1.5L12F100; G73 drilling in incremental programming, 12, polar programming, the pole Angle increases by 30 degrees, R point coordinate 3, hole bottom coordinate-7, each hole depth is 5, the hole back knife distance is 1.5, the overall effect is equivalent to drilling every 30 degrees on the circle with a radius of 5

G80; exit hole loop
G90X8Y90; Move the tool to (0,8,3)
G99G91G73Y30Z-7R3Q5K1.5L12F100; G73 drilling under incremental programming mode, 12, polar coordinate programming mode, polar Angle increases by 30 degrees, R point coordinate 3, hole bottom coordinate-7, each hole depth 5, hole back knife distance 1.5, the overall effect is equivalent to every 30 degrees on the circle with a radius of 8
G80; exit hole loop
G90G15 Turn off the polar coordinate programming mode;

39. G74 left-tapping thread machining cycle instruction

Format: G74 X..Y..Z..A..B..C..R..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91 mode, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;

The [R] programming word specifies the Z coordinate of the R point;

[Z] The programming word specifies the Z coordinate of the hole bottom;

[L] Specifies the number of boreholes;

Action description:

- The wire cone X / Y / A / B / C axis is quickly positioned to the hole position of the thread processing;
- The cone moves quickly in the Z direction to point R;
- The wire cone is drilled to the bottom of the hole Z at the specified speed F, and the main axis is reversed and moves together with the Z axis;
- The wire cone is raised to point R at the specified speed F, and the main axis is positive and moves in combination with the Z axis;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-4 until all the hole position processing is completed;
- When using the G98 instruction, the wire cone quickly returns to the initial plane.

pay attention to:

Use this instruction to configure the spindle interface type as servo (direction / pulse) on the [external device] -> [spindle] page;

Example:

G0Z10
M29S200; Enter the rigid tapping mode. In the rigid tapping mode, the Z axis

and the servo spindle are inserted together
G99G74X0Y0Z-5R3F500; rigid tapping at (0,0,3), hole bottom Z coordinate-5, the pitch of the screw is F/S=500/200=2.5mm
G91G74X5L4F500; Rigid tapping at (5,0,3), in turn, (10,0,3), (15,0,3), (20,0,3), Z coordinate-5 of hole bottom, the pitch of screw is F/S=500/200=2.5mm
G80
M5

40. The G80 cancels the fixed loop instruction

Format: G80

Point Out:

- This command cancels the fixed cycle mode and the machine returns to the normal operation state. The hole processing data, including R, Z, etc., are cancelled, but the movement rate F will continue to be valid;
- To cancel the fixed loop mode, in addition to the G80 command, can also use G code movement group command G00, G01, G02, G03 any command;

41. G81 borehole cycle instruction

Format: G81 X..Y..Z..A..B..C..R..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91 mode, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;
The [R] programming word specifies the Z coordinate of the R point;
[Z] The programming word specifies the Z coordinate of the hole bottom;
[L] Specifies the number of boreholes;

Action description:

- Tool X / Y / A / B / C axis is quickly positioned to the hole position;
- The tool moves quickly in the Z direction to the point R;
- The tool is drilled to the bottom Z at the specified speed F;
- The tool is quickly raised to point R;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-4 until all the hole position processing is completed;
- When using the G98 instruction, the tool Z-axis quickly returns to the initial plane position.

Example:

G90G0Z3
G16G1X5Y90; Open the polar coordinate programming mode and move the tool to (0,5,3)
G99G91G81Y30Z-7R3L12F100; G81 in incremental programming, 12 holes and polar programming, the polar angle increases by 30 degrees, 3 R point coordinate and bottom coordinate-7, the overall effect is equivalent to drilling every 30 degrees in a radius of 5
G80; exit hole loop
G90X8Y90; Move the tool to (0,8,3)
G99G91G81Y30Z-7R3L12F100; G81 drilling in incremental programming, 12 holes, polar programming, the polar angle increases by 30 degrees, 3 R point coordinate and hole bottom coordinate-7, the overall effect is equivalent to drilling every 30 degrees in a radius of 8
G80; exit hole loop
G90G15 Turn off the polar coordinate programming mode;

42.G82 drill hole cycle instruction

Format: G82 X..Y..Z..A..B..C..R..P..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;

The [R] programming word specifies the Z coordinate of the R point;

[Z] The programming word specifies the Z coordinate of the hole bottom;

[P] hole bottom residence time, in milliseconds;

[L] Specifies the number of boreholes;

Action description:

- Tool X / Y / A / B / C axis is quickly positioned to the hole position;
- The tool moves quickly in the Z direction to the point R;
- The tool drills to the bottom Z at the specified speed F and stays for P milliseconds;
- The tool is quickly raised to point R;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-4 until all the hole position processing is completed;
- When using the G98 instruction, the tool Z-axis quickly returns to the initial plane position.

Example:

G90G0Z3

G16G1X5Y90; Open polar programming and move the tool to (0,5,3)
G99G91G82Y30Z-7R3P300L12F100; G82,12, polar coordinate programming, the pole angle increases by 30 degrees, 3 R point coordinate, hole bottom coordinate-7, hole bottom residence time 300 ms, the overall effect is equivalent to drilling every 30 degrees on the radius of 5
G80; exit hole loop
G90X8Y90; Move the tool to (0,8,3)
G99G91G82Y30Z-7R3P300L12F100; G82,12, polar coordinate programming, the pole Angle increases by 30 degrees, 3 R point coordinate, hole bottom coordinate-7, hole bottom residence time 300 ms, the overall effect is equivalent to drilling every 30 degrees on the radius of 8
G80; exit hole loop
G90G15 Turn off the polar coordinate programming mode;

43. G83 pecking type deep hole drilling cycle

Format: G83 X..Y..Z..A..B..C..R..Q..K..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91 mode, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;

The [R] programming word specifies the Z coordinate of the R point;

[Z] The programming word specifies the Z coordinate of the hole bottom;

[Q] The programming word specifies the depth of each pecking hole;

[K] programming word specifies the peck hole gap;

[L] Specifies the number of boreholes;

Action description:

- Tool X / Y / A / B / C axis is quickly positioned to the hole position;
- The tool moves quickly in the Z direction to the point R;
- The tool drills the Q distance at the specified speed F, and then quickly lifts to the R point;
- Thereafter, the tool quickly drops to the top of the previous drilling position (the distance is the hole gap K), and then press the specified speed F drilling (Q + K) distance, and then quickly lift to the R point; repeat this action until drilling to the bottom of the hole, Z;
- The tool is quickly raised to point R;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-5 until all the hole position processing is completed;
- When using the G98 instruction, the tool Z-axis quickly returns to the initial plane position.

example:

G90G0Z3

G16G1X5Y90; Open the polar coordinate programming mode and move the tool to (0,5,3)

G99G91G83Y30Z-7R3Q5K1.5L12F100; G83 drilling, 12, polar coordinate programming, polar Angle increases by 30 degrees, R point coordinate 3, hole bottom coordinate-7, each hole depth 5, hole back knife distance 1.5, the overall effect is equivalent to drilling every 30 degrees on the circle with a radius of 5

G80; exit hole loop

G90X8Y90; Move the tool to (0,8,3)

G99G91G83Y30Z-7R3Q5K1.5L12F100; G83 drilling, 12, polar programming, the pole Angle increases by 30 degrees, R point coordinate 3, hole bottom coordinate-7, each hole depth 5, hole back knife distance 1.5, the overall effect is equivalent to every 30 degrees on the circle with a radius of 8

G80; exit hole loop

G90G15 Turn off the polar coordinate programming mode;

44. G84 right-hand tapping thread machining cycle instruction

Format: G84 X..Y..Z..A..B..C..R..L..F...

Among:

[X / Y / A / B / C] The programming word is used to determine the hole position. In the incremental programming G91, the programming word is the coordinate increment; when the repeat number L is greater than 1, the hole position is increased according to the difference between the first hole position and the initial position;

The [R] programming word specifies the Z coordinate of the R point;

[Z] The programming word specifies the Z coordinate of the hole bottom;

[L] Specifies the number of boreholes;

Action description:

- X / Y / A / B / C axis quickly positioned to the hole position of thread processing;
- The cone moves quickly in the Z direction to point R;
- The wire cone is drilled to the bottom of the hole Z at the specified speed F, and the main axis is positive and moves together with the Z axis;
- The wire cone is raised to the R point at the specified speed F, and the main axis reverses and moves together with the Z axis;
- If the repeat number is greater than 1, calculate the next hole position according to the difference between the first hole position and the initial position, repeat steps 1-4 until all the hole position processing is completed;
- When using the G98 instruction, the cone quickly returns to the initial plane.

pay attention to:

Use this instruction to configure the spindle interface type as servo (direction / pulse) on the [external device] -> [spindle] page;

example:

G0Z10

M29S200; Enter the rigid tapping mode. In the rigid tapping mode, the Z axis and the servo spindle are inserted together

G99G84X0Y0Z-5R3F500; rigid tapping at (0,0,3), hole bottom Z coordinate-5, the pitch of wire cone is F/S=500/200=2.5mm

G91G84X5L4F500; Rigid tapping at (5,0,3), (10,10,0,3), (15,0,3), (20,0,3), Z coordinate-5 of the hole bottom, and the pitch of the screw is F/S=500/200=2.5mm

G80

M5

45. G90 absolute coordinate instruction

Format: G90

Explain:

When programming with the absolute coordinate instruction G90, the [X / Y / Z / A / B / C] size number in the program segment is the absolute coordinate value, that is, the coordinate value of all the track points of the tool, which is based on the origin of the program.

46. G91 Relative coordinate instruction

Format: G91

Explain:

When the relative coordinate instruction G91 is programmed, the [X / Y / Z / A / B / C] size number in the program segment is the incremental coordinate value, that is, the coordinate value of the current point of the tool, and the coordinate of the previous point is the reference.

47. G92 artifact coordinate system setting

Format: G92 X..Y..Z..A..B..C...

Description: Adjust the origin bias of the current coordinate system (G54 / G55 / G56 / G57 / G58 / G59);

point out:

- The G92 instruction only adjusts the origin bias and does not produce actual

movement;

- In G90 absolute coordinate programming mode, adjust the origin offset value of the current coordinate system to change the current tool work coordinate into the value specified by [X / Y / Z / A / B / C] size word. If the size word is missing, then the origin bias of the axis is maintained;
- In G91 relative programming mode, subtract the value specified by [X / Y / Z / A / B / C] size word from the previous coordinate frame origin bias as the current origin bias. If the size word is missing, then the origin bias of the axis is maintained;

Example:

G00X0Y0; Move the tool to the workpiece coordinate (0,0) position
G90G92X5Y10; After execution, the current tool coordinates changes to (5,10)
G91G92X-5Y-10; after execution, the current tool work coordinate changes to (0,0)

48. G93 inverse time feed instruction

Format: G93 F...

Where F specifies the running speed of the instruction segment in an inverse time feed mode;

Point Out:

The G93 program segment must specify the [F] programming word, the desired speed of the command segment length * F (programming unit / minute), or can be interpreted as the instruction segment is expected to be completed within 1 / F (minutes);

Example:

G0X0
G93G1X10F60; In G93 mode, the expected movement completion time of the command segment is 1 / 60 min =1 second
X0F30; in G93 mode, the desired movement completion time is 1 / 30 minutes =2 seconds

49. The G94 feeds the instructions every minute

Format: G94 F...

Where F specifies the running speed of the command segment with the amount of movement per minute;

50. The G98 fixed bore cycle instruction is returned to the initial plane

Format: G98
Description: Use to specify that the Z axis returns to the initial plane at the end of the fixed drill cycle command

51. The G99 fixed drill ring loop command will return to the R plane

Format: G99
Description: Use to specify the Z axis back to the R plane at the end of the fixed drill hole cycle instruction

52. The G128 comprehensive zero-finding instruction

Format: G128 X..Y..Z..A..B..C...

Where the [X / Y / Z / A / B / C] size word is set to 1, the corresponding axis is changed

Point Out:

If the axis is not configured with zero signal, there is no action when the instruction is executed;

Example:

G128Z1; Z-axis open zero-finding

G128X1Y1A1B1C1; XYABC axis simultaneously on zero simultaneously

Appendix V. M, instruction set

1. M0 program pause instruction

Format: M0

2. M1 program shutdown instruction

Format: M1
Note: When the shutdown is valid, the program will be suspended when the M1 instruction is executed

3. The M2 program is over

Format: M2
Description: it will not change the working state of the spindle, coolant and lubricating fluid;

4. The M3 spindle is turning positively

Format: M3 S...
Where the S programming word specifies the spindle speed

5. M4 spindle reversal

Format: M4 S...
Where the S programming word specifies the spindle speed

6. The M5 spindle stops

Format: M5

7. M6 change knife instructions

Format: M6
explain:
1） Software factory status M6 without any action;
2） You can write the O10006 number subroutine in env/slib-m.nc to define the behavior of the M6 instructions.
3） Description of O10006 (subroutine M6):
- # 00 X programming words, if not set, the value is 0
- # 01 Y programming word, if not set, the value is 0

- # 02 Z programming word if not set, the value is 0
- # 03 A programming word if not set, the value is 0
- # 04 B programming word if not set, the value is 0
- # 05 C programming word if not set, the value is 0
- # 06 I programming word if not set, the value is the most recently set value
- # 07 J If not set, the value is the most recently set value
- # 08 K programming word if not set, the value is the last set value
- # 09 R programming word if not set, the value is the most recently set value
- # 10 L programming word if not set, the value is the most recently set value
- # 11 H If not set, the value is the most recently set value
- # 12 P programming word, if not set, the value is the most recently set value
- # 13 Q If not set, the value is the last set value
- # 14 D If not set, the value is the most recently set value
- # 15 F programming word if not set, the value is the most recently set value
- # 16 S. If not set, the value is the most recently set value
- # 17 T programming word If not set, the value is the most recently set value
- # 18 U programming word If not set, the value is the most recently set value
- # 19 V If not set, the value is the last set value
- # 20 W programming word if not set, the value is the most recently set value
- # 21 E If the programming word is not set, the value is the most recently set value
- # 22 X set to 1, otherwise 0
- # 23 Y set to 1, otherwise 0
- # 24 Z set to 1, otherwise 0
- # 25 A set to 1, otherwise 0
- # 26 B set to 1, otherwise 0
- # 27 C set to 1, otherwise 0
- # 28 I set to 1, otherwise 0
- # 29 J programming set the word as set to 1, otherwise 0
- # 30 K programming setting word as set to 1, otherwise 0
- # 31 R set to 1, otherwise 0
- # 32 L programming setting words as set to 1, otherwise 0
- # 33 H set 1, otherwise 0
- # 34 P set 1, otherwise 0
- # 35 Q set to 1, otherwise 0
- # 36 D set to 1, otherwise 0
- # 37 F set 1, otherwise 0
- # 38 S set to 1, otherwise 0
- # 39 T set to 1, otherwise 0
- # 40 U set to 1, otherwise 0
- # 41 V set to 1, otherwise 0
- # 42 W set to 1, otherwise 0
- # 43 E set 1, otherwise 0
- # 44 X-axis current working coordinates (G20 / G21 mode conversion)

- # 45 Y-axis current working coordinate (G20 / G21 mode conversion)
- # 46 Current working coordinates of Z axis (G20 / G21 mode conversion)
- # 47 Current working coordinates of axis A (G20 / G21 mode conversion)
- # 48 Current working coordinates of axis B (G20 / G21 mode conversion)
- # 49 Current working coordinates of C axis (G20 / G21 mode conversion)
- # 50 X-axis current programming artifact coordinates (G90 / G91, G20 / G21 mode conversion)
- # 51 Y-axis current programming artifact coordinates (G90 / G91, G20 / G21 mode conversion)
- # 52 Z axis current programming work piece coordinates (G90 / G91, G20 / G21 mode conversion)
- Current programming artifact coordinates of the # 53 A axis (G90 / G91, G20 / G21 mode conversion was performed)
- # 54 B (G90 / G91, G20 / G21 mode conversion)
- # 55 C axis current programming work piece coordinates (G90 / G91, G20 / G21 mode conversion)
- # 56 G90 / G91 mode, 1 G90 mode and 0 G91 mode
- # 57 Programming unit proportional coefficient (25.4 in G20 mode, 1.0 in G21 mode)

4） Generally, M6 instruction is used together with T programming word, such as M6T2, we can obtain the tool number T through # 17 variable. In addition, at the end of M6, we need to return to the position when executing M6 instruction, which can be obtained through # 44- # 49 variable.

Here is an example of manual procedure as follows:

- Close the coordinate system rotation function, close the tool length programming, and close the spindle
- Check the tool number, if the tool number is 0, end the M6 child
- Movement to reference point G30P2 (first back to XY, then back to Z, and finally back to rotation axis)
- Call the tool Length measurement configuration window in which sets the tool length measurement related parameters
- Press OK button to start tool length measurement; press Cancel button to finish processing;
- After the tool length measurement ends, open the current tool length compensation
- Return to the position before M6 instruction (motion order is first back to Z, then back to XY, and last back to the rotation axis)
- The manual knife procedure program ends and continue the subsequent program

O10006 (M6 subroutine)
G69

```
G49
M5

IF#17==0GOTO5

(Back to G30P2 reference point - tool change position)
G91G30X0Y0P2
G91G30X0Y0P2
G91G30Z0P2
G91G30Z0P2
G91G30A0B0C0P2
G91G30A0B0C0P2

#2500=2006

(#17 is Current measuring tool number T)
^2500.item[0]=#17

(The tool length measurement page pops up)
MarcoDialog "/env/macroParam.ui"

IF^2500.item[5]<1 THEN ^2500.item[5]=1
IF^2500.item[6]<0.5 THEN ^2500.item[6]=0.5
IF^2500.item[6]>1.0 THEN ^2500.item[6]=1.0

G0G53Z^2500.item[2]

IF^2500.item[1]==3GOTO12
IF^2500.item[1]==2GOTO11
IF^2500.item[1]==1GOTO10
G91G28X0Y0
G91G28X0Y0
G91G28Z0
G91G28Z0
GOTO1
N10
G91G30X0Y0P1
G91G30X0Y0P1
G91G30Z0P1
G91G30Z0P1
GOTO1
N11
G91G30X0Y0P2
G91G30X0Y0P2
```

```
G91G30Z0P2
G91G30Z0P2
GOTO1
N12
G91G30X0Y0P3
G91G30X0Y0P3
G91G30Z0P3
G91G30Z0P3
N1

G90G0A0B0C0

#1=0.0
#4=^2500.item[5]
#5=^2500.item[3]
WHILE #4>=1 DO1
#4=#4-1
G91G31Z-1000Q1K1F#5
#1=#1+#300.item[2]
G91G0Z^2500.item[4]
#5=#5*^2500.item[6]
END1

#1=#1/^2500.item[5]

IF#330.item[36]==1GOTO2
IF#330.item[36]==3GOTO2
IF#330.item[36]==5GOTO2
IF#330.item[36]==6GOTO2
IF#330.item[36]==7GOTO2
IF#330.item[36]==8GOTO2
IF#330.item[36]==9GOTO2
IF#330.item[36]==10GOTO2
IF#330.item[36]==11GOTO2
IF#330.item[36]==12GOTO2
IF#330.item[36]==13GOTO2
GOTO3
N2
#310.item[^2500.item[0]-1]=#1-#2505
#311.item[^2500.item[0]-1]=0
GOTO4
N3
#310.item[^2500.item[0]-1]=#1-#308.item[8]
#311.item[^2500.item[0]-1]=0
```

```
N4

G43H^2500.item[0]

(Return to the position before tool change, #44-#49 records the position information of X/Y/Z/A/B/C before tool change)
G90G0Z#46
X#44Y#45
A#47B#48C#49

N5
M99
```

8. The M8 coolant is turned on

Format: M8

9. The M9 coolant is closed

Format: M9

10. M10 lubrication on

Format: M10

11. M11 lubrication off

Format: M11

12. M29 tapping command

Format: M29S...

Where: [S] programming word for the tapping spindle speed;

Description: The spindle enters into the tapping mode;

Examples can be referred toG74/G84。

13. The M30 procedure was ended

Format: M30

Description: finish the procedure and close the spindle, coolant and lubrication fluid;

14. M31 rotation axis expanded

Format: M31
Note: After the instruction is executed, the A / B / C machine tool coordinates will be reset within the range of [0-360) with 360° as the module;

Note: The M31 is only valid when running in an online state

15. M32 coordinate system synchronization instruction

Format: M32
Description: When the machine coordinate is set through the macro variable, use this instruction to synchronize the coordinate system;

16. M47 for repeated processing

Format: M47

17. M50 switch quantity batch output instruction

Format: M50 X..Y..P...

among:

[X] The programming word is the output port mask, and a bit is 1, indicating that the corresponding output port of the bit will perform the output operation;

[Y] The programming word is the output value. When the mask corresponding to the output port is 1, the output port will be output in the state of the bit corresponding to the output value;

[P] The programming word is the output delay in microseconds;

example:

M50X7Y5P10; Output port 3 / 2 / 1 output logic 1 / 0 / 1, delay of 10 microseconds after output

18. M51 switch quantity single-port output instruction

Format: M51 X..Y..P...

Among:

[X] The programming word is the output port number, please select the corresponding number value according to the screen mark of the control box;

[Y] The programming word is the output value of the output port;

[P] The programming word is the output delay in microseconds;

Example:

M51X7Y1P1000000; Output port 7, output logic 1, with a delay of 1 second after output

19. M60 switch quantity input batch discrimination instruction

Format: M60 X..Y..P..E..Q...

Among:

[X] The programming word is the input port mask, the IO port input coding value and the mask value "with" operation, and the input expected value, when the two are equal, execute subsequent instructions;

[Y] The programming word is the input expected value;

[P] programming word is the discrimination time limit, the unit is microseconds, is 0 indicates the permanent discrimination waiting, until it meets the input expected value;

[E] The programming word is the filtering width, the unit is microseconds, IO input signal in the filtering width is stable without change will be compared to distinguish;

[Q] The programming word specifies the timeout processing policy, 0 means continuing after timeout, 1 means quitting the program after timeout (reset exit when MDI is running; suspended when processing is running);

Example:

M60X160Y0P3000000E20000Q1; Monitor input port 6 and port 8 (160=b1010 0000), input expected value is both 0, filter time is 20 ms, discrimination time is 3 seconds, and timeout policy is exit program

20. M61 switch quantity input single port discriminant instruction

Format: M61 X..Y..P..E..Q...

Among:

[X] programming word is the input port number, please select the corresponding number value according to the control box screen mark;

[Y] The programming word is the expected value of the input port;

[P] The programming word is the discrimination time limit, the unit is microseconds, and 0 indicates permanent discrimination waiting until the input port signal meets the input expected value;

[E] The programming word is the filter width, the unit is microseconds, the IO input port signal in the filter width is stable and no change will be compared to distinguish;

[Q] The programming word specifies the timeout processing policy, 0 means to continue running after timeout, 1 means quitting the program after timeout (reset exit when MDI is running; suspended exit when processing is running);

Example 1:

M61X6Y0P0E0; Monitor the input port 6 until the input port status is 0, without adopting the filtering strategy for port 6;

Example 2:

M61X8Y1P3000000E20000Q1; Monitor the 8 input port, filter time is 20 ms, when the input port status is 1; if the desired signal is not detected within 3 seconds, exit the program;

21. M98 subroutine call

Format: M98P..L... [parameter word...]

among:

[P] The programming word is the subprogram number to be invoked;

[L] The programming word is the number of calling subprograms and once in the default time;

[Parameter word..] can be all programming words except G / L / M / N / O / P, and the subroutine can be accessed by the variables corresponding to the letter number of the programming word, for example: A corresponds to # 1 variable, B corresponds to # 2 variable, and so on;

[Note]: If the user uses the macro programming method (such as writing subprograms, using conditional jump statement, loop statement, etc.,), the NC file name must start with "macro", at this time, the parser will automatically parse the user program by macro resolution mode;

Example:

Writing macrotest.nc is as follows:

O0000; Main program, the main program number shall be consistent with the [Main Program Number] parameter under the [Automatic Parameter] page

M98P0001L5A5F1000; Call 00015 times, incoming parameter A: 5 (# 1=5), F1000 (# 6=1000)

M30

O0001; 0001 subroutine

G01X # 1F # 6; # 1, # 6 for the caller incoming parameters, corresponding to A programming word and F programming word

G01X0

M99

22. The M99 subroutine ends

Format: M99

23. The M300 opens the axle

Format: M300P...

among:

[P] programming word is the programming axis number corresponding to the axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

Explain:

After the instruction is called, the corresponding programming axis will enter the axle mode, where the manual operation and command control cannot be performed.

24. The M301 closes the axle

Format: M301P...

among:

[P] programming word is the programming axis number corresponding to the axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

explain:

After calling this command, the corresponding programming shaft exits the axle mode.

25. M303 axle is turning

Format: M303S..P...

Among:

[S] programming word is axle speed, unit turn / minute;

[P] programming word is the programming axis number corresponding to the axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

Description: After calling the instruction, the corresponding programming shaft rotates at the specified speed.

Note: The programming shaft must be in axle mode before using this instruction.

26. M304 axle reversal

Format: M304S..P...

Among:

[S] programming word is axle speed, unit turn / minute;

[P] programming word is the programming axis number corresponding to the

axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

Description: After calling the command, the corresponding programming shaft is reversed at the specified rotation speed.

Note: The programming shaft must be in the axle mode before using this instruction.

27. M305 axle stop

Format: M305P...

Among:

[P] programming word is the programming axis number corresponding to the axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

Note: After the instruction is called, the corresponding programming axis will stop turning.

Note: The programming shaft must be in the axle mode before using this instruction.

28. M329 axle insertion

Format: M329P...

Among:

[P] programming word is the programming axis number corresponding to the axle, 0 is X axis, 1 is Y axis, 2 is Z axis, 3 is A axis, 4 is B axis, 5 is C axis, if the system has set axle programming axis, P programming word can be default;

explain:

- After the command is called, the corresponding programming axis stops turning and enters the interpolation mode. At this time, the programming axis can be controlled by the command.
- Use the M305 instruction to exit the axle plug state.

Note: The programming shaft must be in the axle mode before using this instruction.

29. M500 / M501 input port 1 waits for the on / off command

Format: M500 / M501

Description: Always wait for the on / off of port 1

30. M502 / M503 input port 2 waits for the on / off command

Format: M502 / M503

Description: Always wait for port 2 to turn on / off

31.M504 / M505 input port 3 waits for the on / off command

Format: M504 / M505
Description: Always wait for the on / off of port 3

32.M506 / M507 input port 4 waits for the on / off command

Format: M506 / M507
Description: Always wait for the on / off of port 4

33.M508 / M509 input port 5 waits for the on / off command

Format: M508 / M509
Description: Always wait for the on / off of port 5

34.M510 / M511 input port 6 waits for the on / off command

Format: M510 / M511
Description: Always wait for the on / off of port 6

35.M512 / M513 input port 7 waits for the on / off command

Format: M512 / M513
Description: Permanent wait for on / off port 7

36.M514 / M515 input port 8 waits for the on / off command

Format: M514 / M515
Description: Always wait for the on / off of port 8

37.M516 / M517 input port 9 waits for the on / off command

Format: M516 / M517
Description: Permanent wait for on / off port 9

38.M518 / M519 input port 10 waits for the on / off command

Format: M518 / M519
Description: Always wait for the on / off of port 10

39.M520 / M521 input port 11 waits for the on / off command

Format: M520 / M521
Description: Permanent wait for the on / off of port 11

40.M522 / M523 input port 12 waits for the on / off command

Format: M522 / M523
Description: Always wait for the on / off of port 12

41.M524 / M525 input port 13 waits for the on / off command

Format: M524 / M525
Description: Permanent wait for the on / off of port 13

42.M526 / M527 input port 14 waits for the on / off command

Format: M526 / M527
Description: Permanent wait for the on / off of port 14

43.M528 / M529 input port 15 waits for the on / off command

Format: M528 / M529
Description: Always wait for the on / off of port 15

44.M530 / M531 input port 16 waits for the on / off command

Format: M530 / M531
Description: Always wait for the on / off of port 16

45.M532 / M533 input port 17 waits for the on / off command

Format: M532 / M533
Description: Permanent wait for the on / off of port 17

46.M534 / M535 input port 18 waits for the on / off command

Format: M534 / M535
Description: Permanent wait for the on / off of port 18

47.M536 / M537 input port 19 waits for the on / off command

Format: M536 / M537

Description: Always wait for the on / off of port 19

48.M538 / M539 input port 20 waits for the on / off command

Format: M538 / M539
Description: Always wait for the on / off of port 20

49.M540 / M541 input port 21 waits for the on / off command

Format: M540 / M541
Description: Always wait for the on / off of port 21

50.M542 / M543 input port 22 waits for the on / off command

Format: M542 / M543
Description: Always wait for the on / off of port 22

51.M544 / M545 input port 23 waits for the on / off command

Format: M544 / M545
Description: Always wait for the on / off of port 23

52.M546 / M547 input port 24 waits for the on / off command

Format: M546 / M547
Description: Always wait for the on / off of port 24

53.M550 / M551 output port 1 on / off command

Format: M550 / M551
Description: Output on / off of output port 1

54.M552 / M553 output port 2 on / off command

Format: M552 / M553
Description: On / off of output port 2

55.M554 / M555 output port 3 on / off command

Format: M554 / M555
Description: On / off of output port 3

56. M556 / M557 output port 4 on / off command

Format: M556 / M557
Description: On / off of output port 4

57. M558 / M559 output port 5 on / off command

Format: M558 / M559
Description: On / off of output port 5

58. M560 / M561 output port 6 on / off command

Format: M560 / M561
Description: On / off of output port 6

59. M562 / M563 output port 7 on / off command

Format: M562 / M563
Description: put / off of output port 7

60. M564 / M565 output port 8 on / off command

Format: M564 / M565
Description: On / off of output port 8

61. M566 / M567 output port 9 on / off command

Format: M566 / M567
Description: Output on / off of output port 9

62. M568 / M569 output port 10 on / off command

Format: M568 / M569
Description: Turn-on / off of the output port 10

63. M570 / M571 output port 11 on / off command

Format: M570 / M571
Description: Turn-on / off of the output port 11

64. M572 / M573 output port 12 on / off command

Format: M572 / M573

Description: Turn-on / off of the output port 12

65. M574 / M575 output port 13 on / off command

Format: M574 / M575
Description: Turn-on / off of the output port 13

66. M576 / M577 output port 14 on / off command

Format: M576 / M577
Description: Turn-on / off of the output port 14

67. M578 / M579 output port 15 on / off command

Format: M578 / M579
Description: Turn-on / off of the output port 15

68. M580 / M581 output port 16 on / off command

Format: M580 / M581
Description: Turn-on / off of the output port 16

69. M582 / M583 output port 17 on / off command

Format: M582 / M583
Description: Turn-on / off of the output port 17

70. M584 / M585 output port 18 on / off command

Format: M584 / M585
Description: Turn-on / off of the output port 18

71. M800 / M801 MODBUS input port 1 waits for the on / off command

Format: M800 / M801
Description: Permanent wait for the on / off of MODBUS input port 1.

72. M802 / M803 MODBUS input port 2 waits for the on / off command

Format: M802 / M803
Description: Permanent wait for the on / off of MODBUS input port 2.

73. M804 / M805 MODBUS input port 3 waits for the on / off command

Format: M804 / M805
Description: Permanent wait for the on / off of MODBUS input port 3.

74. M806 / M807 MODBUS input port 4 waits for the on / off command

Format: M806 / M807
Description: Permanent wait for the on / off of MODBUS input port 4.

75. M808 / M809 MODBUS input port 5 waits for the on / off command

Format: M808 / M809
Description: Permanent wait for the on / off of MODBUS input port 5.

76. M810 / M811 MODBUS input port 6 waits for the on / off command

Format: M810 / M811
Description: Permanent wait for the on / off of MODBUS input port 6.

77. M812 / M813 MODBUS input port 7 waits for the on / off command

Format: M812 / M813
Description: Permanent wait for the on / off of MODBUS input port 7.

78. M814 / M815 MODBUS input port 8 waits for the on / off command

Format: M814 / M815
Description: Permanent wait for the on / off of MODBUS input port 8.

79. M816 / M817 MODBUS input port 9 waits for the on / off command

Format: M816 / M817
Description: Permanent wait for the on / off of MODBUS input port 9.

80. M818 / M819 MODBUS input port 10 waits for the on / off command

Format: M818 / M819
Description: Always wait for the on / off of MODBUS input port 10.

81. M820 / M821 MODBUS input port 11 waits for the on / off command

Format: M820 / M821
Description: Always wait for the on / off of MODBUS input port 11.

82. M822 / M823 MODBUS input port 12 waits for the on / off command

Format: M822 / M823
Description: Always wait for the on / off of MODBUS input port 12.

83. M824 / M825 MODBUS input port 13 waits for the on / off command

Format: M824 / M825
Description: Always wait for the on / off of MODBUS input port 13.

84. M826 / M827 MODBUS input port 14 waits for the on / off command

Format: M826 / M827
Description: Always wait for the on / off of MODBUS input port 14.

85. M828 / M829 MODBUS input port 15 waits for the on / off command

Format: M828 / M829
Description: Always wait for the on / off of MODBUS input port 15.

86. M830 / M831 MODBUS input port 16 waits for the on / off command

Format: M830 / M831
Description: Always wait for the on / off of MODBUS input port 16.

87. M832 / M833 MODBUS input port 17 waits for the on / off command

Format: M832 / M833
Description: Always wait for the on / off of MODBUS input port 17.

88. M834 / M835 MODBUS input port 18 waits for the on / off command

Format: M834 / M835
Description: Always wait for the on / off of MODBUS input port 18.

89. M836 / M837 MODBUS input port 19 waits for the on / off command

Format: M836 / M837
Description: Always wait for the on / off of MODBUS input port 19.

90. M838 / M839 MODBUS input port 20 waits for the on / off command

Format: M838 / M839
Description: Always wait for the on / off of MODBUS input port 20.

91. M840 / M841 MODBUS input port 21 waits for the on / off command

Format: M840 / M841
Description: Always wait for the on / off of MODBUS input port 21.

92. M842 / M843 MODBUS input port 22 waits for the on / off command

Format: M842 / M843
Description: Always wait for the on / off of MODBUS input port 22.

93. M844 / M845 MODBUS input port 23 waits for the on / off command

Format: M844 / M845
Description: Always wait for the on / off of MODBUS input port 23.

94. M846 / M847 MODBUS input port 24 waits for the on / off command

Format: M846 / M847
Description: Always wait for the on / off of MODBUS input port 24.

95. M848 / M849 MODBUS input port 25 waits for the on / off command

Format: M848 / M849
Description: Always wait for the on / off of MODBUS input port 25.

96. M850 / M851 MODBUS input port 26 waits for the on / off command

Format: M850 / M851
Description: Always wait for the on / off of MODBUS input port 26.

97. M852 / M853 MODBUS input port 27 waits for the on / off command

Format: M852 / M853
Description: Always wait for the on / off of MODBUS input port 27.

98. M854 / M855 MODBUS input port 28 waits for the on / off command

Format: M854 / M855
Description: Always wait for the on / off of MODBUS input port 28.

99. M856 / M857 MODBUS input port 29 waits for the on / off command

Format: M856 / M857
Description: Always wait for the on / off of MODBUS input port 29.

100. M858 / M859 MODBUS input port 30 waits for the on / off command

Format: M858 / M859
Description: Always wait for the on / off of MODBUS input port 30.

101. M860 / M861 MODBUS input port 31 waits for the on / off command

Format: M860 / M861
Description: Always wait for the on / off of MODBUS input port 31.

102. M862 / M863 MODBUS input port 32 waits for the on / off command

Format: M862 / M863
Description: Always wait for the on / off of MODBUS input port 32.

103. M900 / M901 MODBUS output port 1 on / off command

Format: M900 / M901
Description: MODBUS On / off of output port 1.

104. M902 / M903 MODBUS output port 2 on / off command

Format: M902 / M903
Description: MODBUS On / off of output port 2.

105. M904 / M905 MODBUS output port 3 on / off command

Format: M904 / M905
Description: MODBUS On / off of output port 3.

106. M906 / M907 MODBUS output port 4 on / off command

Format: M906 / M907
Description: MODBUS On / off of output port 4.

107. M908 / M909 MODBUS output port 5 on / off command

Format: M908 / M909
Description: MODBUS On / off of output port 5.

108. M910 / M911 MODBUS output port 6 on / off command

Format: M910 / M911
Description: MODBUS On / off of output port 6.

109. M912 / M913 MODBUS output port 7 on / off command

Format: M912 / M913
Description: MODBUS On / off of output port 7.

110. M914 / M915 MODBUS output port 8 on / off command

Format: M914 / M915
Description: MODBUS On / off of output port 8.

111. M916 / M917 MODBUS output port 9 on / off command

Format: M916 / M917
Description: MODBUS On / off of output port 9.

112. M918 / M919 MODBUS output port 10 on / off command

Format: M918 / M919
Description: Turn-on / off of the MODBUS output port 10.

113. M920 / M921 MODBUS output port 11 on / off command

Format: M920 / M921
Description: On / off of the MODBUS output port 11.

114. M922 / M923 MODBUS output port 12 on / off command

Format: M922 / M923

Description: On / off of MODBUS output port 12.

115. M924 / M925 MODBUS output port 13 on / off command

Format: M924 / M925
Note: On / off of MODBUS output port 13.

116. M926 / M927 MODBUS output port 14 on / off command

Format: M926 / M927
Note: On / off of the MODBUS output port 14.

117. M928 / M929 MODBUS output port 15 on / off command

Format: M928 / M929
Description: Turn-on / off of the MODBUS output port 15.

118. M930 / M931 MODBUS output port 16 on / off command

Format: M930 / M931
Description: On / off of the MODBUS output port 16.

119. M932 / M933 MODBUS output port 17 on / off command

Format: M932 / M933
Note: Turn-on / off of the MODBUS output port 17.

120. M934 / M935 MODBUS output port 18 on / off command

Format: M934 / M935
Note: On / off of the MODBUS output port 18.

121. M936 / M937 MODBUS output port 19 on / off command

Format: M936 / M937
Description: On / off of the MODBUS output port 19.

122. M938 / M939 MODBUS output port 20 on / off command

Format: M938 / M939
Description: On / off of the MODBUS output port 20.

123. M940 / M941 MODBUS output port 21 on / off command

Format: M940 / M941
Description: Turn-on / off of the MODBUS output port 21.

124. M942 / M943 MODBUS output port 22 on / off command

Format: M942 / M943
Description: Turn-on / off of the MODBUS output port 22.

125. M944 / M945 MODBUS output port 23 on / off command

Format: M944 / M945
Description: On / off of MODBUS output port 23.

126. M946 / M947 MODBUS output port 24 on / off command

Format: M946 / M947
Description: On / off of the MODBUS output port 24.

127. M948 / M949 MODBUS output port 25 on / off command

Format: M948 / M949
Description: Turn-on / off of the MODBUS output port 25.

128. M950 / M951 MODBUS output port 26 on / off command

Format: M950 / M951
Description: Turn-on / off of the MODBUS output port 26.

129. M952 / M953 MODBUS output port 27 on / off command

Format: M952 / M953
Description: On / off of the MODBUS output port 27.

130. M954 / M955 MODBUS output port 28 on / off command

Format: M954 / M955
Note: On / off of MODBUS output port 28.

131. M956 / M957 MODBUS output port 29 on / off command

Format: M956 / M957

Description: On / off of the MODBUS output port 29.

132. M958 / M959 MODBUS output port 30 on / off command

Format: M958 / M959
Description: On / off of the MODBUS output port 30.

133. M960 / M961 MODBUS output port 31 on / off command

Format: M960 / M961
Note: Turn-on / off of the MODBUS output port 31.

134. M962 / M963 MODBUS output port 32 on / off command

Format: M962 / M963
Description: On / off of the MODBUS output port 32.

Appendix 6

1. SIN sine function

Format: SIN [Parameter 1]

Where: parameter 1 is a given Angle, unit: °;

example:

1 = SIN [# 2 / 2]; calculate the sine of the # 2 / 2 angle and assign the # 1 variable.

2. COS Cosine function

Format: COS [Parameter 1]

Where: parameter 1 is a given Angle, unit: °;

example:

1= COS [# 2 / 2]; compute cosine of # 2 / 2 angle and assign # 1 variable.

3. TAN tangent function

Format: TAN [Parameter 1]

Where: parameter 1 is for a given Angle, in unit: °;

example:

1 = TAN [# 2 / 2]; Calculate the tangent value of the # 2 / 2 angle and assign the # 1 variable.

4. ATAN anyway cut function

Format: ATAN [parameter 1, parameter 2]

Where: parameter 1 is the Y-axis coordinate value;

Parameter 2 is the X-axis coordinate value;

Description: The returned value is the anyway cut value of parameter 1 / parameter 2, in unit: °;

example:

1 = ATAN [7, -7]; calculate cut cut of 7 / -7 and assign # 1 variable, # 1=135 °.

5. The ABS absolute value function

Format: ABS [parameter 1]

Description: returns the absolute value of the parameter 1.

example:

1= ABS [# 2]; assign the absolute value of # 2 variable to the # 1 variable.

6. Take the integer function on the FUP

Format: FUP [Parameter 1]
Description: Organize the parameter 1.
example:
1 = FUP [1.2]; # 1 value = 2
2 = FUP [-1.2]; the # 2 value is-2

7. Take the integral function under FIX

Format: FIX [Parameter 1]
Description: Complete the parameter 1.
example:
1 = FIX [1.2]; # 1 value is 1
2 = FIX [-1.2]; the # 2 value is-1

8. ROUND Rounding the function

Format: ROUND [Parameter 1, parameter 2]
Where: parameter 1 is the number to be rounded off;
Parameter 2 is rounded to the operation bit, that is, to the decimal point;
Note: The default time of parameter 2 indicates the rounding of the first bit after the decimal point.

example:
1= ROUND [12.6578]; # 1 value is 13
1= ROUND [12.6578,1]; # 1 value is 13
1= ROUND [12.6578,2]; # 1 value is 12.7
1= ROUND [12.6578,3]; after execution the # 1 value is 12.66
1= ROUND [12.6578,4]; after execution the # 1 value is 12.658

9. MOD mod

Format: MOD [Parameter 1, parameter 2]
Description: find the mode of parameter 1 to parameter 2.
example:
1 = MOD [7,4]; # 1 = 3
1 = MOD [-5,3]; # 1 value is-2

10. The SQRT square-root function

Format: SQRT [Parameter 1]

Note: To find the square root of the parameter 1.
example:
1 = SQRT [# 2]; calculates the square root of # 2 and assign # 1