Catalogue

Chapter I Overview

Welcome to use the computer engraving machine control system produced by our company. This manual describes the features of the engraving machine control system and the operation of each function in details, and is illustrated with a large number of examples and diagrams. Please read the operating instructions carefully before using the engraving machine to ensure proper use of the machine to prevent accidents. Please keep this manual in a safe place so that you can check it out at any time

This system is a professional 3+1 axis motion controller based on embedded platform. It does not need to be configured with PC and runs independently. The system uses an embedded operating system and is not infected with computer viruses. The system adopts advanced adaptive speed forward-looking control algorithm and spline interpolation, which has the characteristics of high processing efficiency and good surface quality. The operation is simple, easy to learn and understand, easy to install, small in size, suitable for various plate engraving machines, engraving and milling machines, cutting machines.

1.1 Precautions

1.1.1 Engraving Machine Installation Environment

- Solid ground
- Avoid direct sunlight
- Leave some space for maintenance
- Space temperature: 5-40 ° C
- Relative humidity: 30-95%
- Equipment should be installed horizontally
- Ventilation should be good.

1.1.2 Engraving Machine Safety Precautions

- Do not use this product in a strong interference or strong magnetic environment.
- Do not plug or unplug the cable box while power on
- Pay attention to waterproof, dustproof and fireproof
- Prevent conductive substances such as metals from entering the shell
- Unauthorized disassembly is strictly prohibited, and there are no user-repairable

parts inside.

- Use a moderate force when plugging and unplugging U disk and other connections
- Do not use for a long time, please pay attention to power off, and save it properly
- The engraving tool is very sharp, and it is forbidden to touch it by hand during operation to prevent injury. Do not use handkerchief or silk scarf to touch it to prevent injury or damage to the equipment;
- The power must be turned off when servicing and adjusting the machine
- Operation and maintenance personnel must be trained

1.2 System Features

- Compatible with standard G code data format. Support mainstream CAD/CAM software, such as ArtCam, MasterCam, ProE, etc.
- Maximum number of control axes: four axes. 2-3 axis linear interpolation, arbitrary 2-axis circular interpolation, 4th axis disc cutter control;
- Spline interpolation function, fitting and interpolating small line segments under the condition of satisfying the spline. Improving the surface quality of the machining;
- The user interacts with the external file through the U disk, and works completely offline;
- Multi-stage pre-processing, adaptive speed forward-looking control of process path, fast processing speed, high precision and good processing continuity;
- Small line segments are continuously processed at high speed, and the most efficient algorithm is automatically selected among various small line segment control algorithms;
- Standard 4G data storage space, support for large-capacity file processing;
- 3D view of the machining path, real-time graphic display during machining;
- MDI function (user input G code online);
- Jump Execution function is processed according to the specified processing line number;
- With Reverse Backlash Compensation, Screw Rod Deviation Compensation, Tool Compensation;
- With Breakpoint Memory, Power Failure Automatic Protection function;
- Machine Fault Diagnosis function, System Log function;
- With Automatic Return to Origin, Automatic Tool Setting, Ruturn to Reference

Point function;

- Built-in Process File Editor Manager: Users can manage, edit and modify files at any time without affecting the current processing status;
- Simulation Function: It can quickly simulate the machining program in a very short time, which is convenient for checking whether the machining program is wrong and whether the machining result is satisfactory.

Chapter II System Display Interface

2.1 System Interface

The entire system interface consists of a title bar, a menu bar, a toolbar, a status bar, a machining track window, and some function windows. As shown in Figure 2-1 below

Figure 2-1 Operation Interface

USB Area: Connect the USB flash drive and copy the processed files into the system.

LCD Display: Contains multiple interfaces, representing six main types of interfaces, “Status”, “File”, “Parameter”, “Offset”, “Information”, “System”. You can switch to the corresponding interface by pressing the button on the right.

Main Menu Button: “Status”, “File”, “Parameter”, “Offset”, “Information”, “System” menu selection on the corresponding interface

Keyboard Area: used to input letters when editing letter program, as a shortcut in a specific state; numeric keys can enter numbers.

Sub-menu Button: The F menu function on the corresponding interface has different functions in different states;

Function Button: used for mode switching, override adjustment, manual motor control, start/stop control, custom buttons of "Continuous", "Automatic", "Handwheel", "Origin", "Handwheel Guidance", "Stepping" functions.

2.2 LCD Display

Figure 2.2 LCD Interface

2.3 Function Button

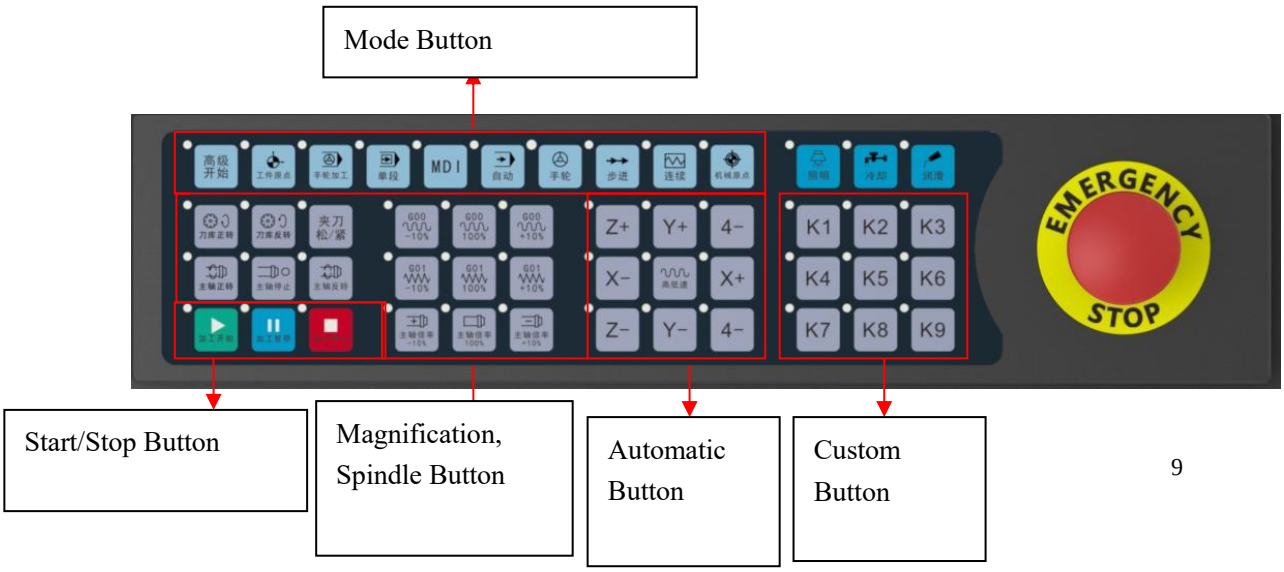


Figure 2.3 Function Button

2.4 Keyboard Button

The buttons of function keyboard can be divided into three parts, alphabet buttons, number buttons, and operation buttons.

Figure 2.4 Button

The letter button has two modes:

One: as a shortcut function button

Each button will correspond to different functions under different function interfaces. For details, see the display area prompt; for example, the N button in the machining interface can enter the jog distance, and in the parameter interface, can enter the manufacturer parameter setting;

Second: input characters during file editing; Shift key is used to switch the upper and lower position enable during file editing, such as 

key. When the shift key is

pressed, the key outputs the U key value.

Number Key:

One: used to control the motion of the motor under the machining interface;

Two: used to enter numbers in file editing

Arrow Key: This is the most used button. In addition to the machining interface, the arrow keys are used to turn pages and switch the cursor position function.

Chapter III How to Input Machining File

There are two ways to input machining file: 1. Import from the U disk, 2. Manually write in the system. The first method is generally applicable to machining files that are more complicated. It is necessary to use CAD/CAM software to assist in generating the machining path and import it into the system through the U disk. The second one is for simpler machining files.

3.1 Import Machining File from U Disk

When machining a new file from a USB flash drive, it must be input into memory of system to start engraving. It is not possible to read the file from the USB flash drive and engrave directly.

In the idling status, press [File] on the main menu to enter the program management interface, and then press the shortcut N button to enter the "U Disk File" menu item. After the system recognizes the USB flash drive, the system will display all the folders and supported file names. Use the arrow keys on the keyboard to select the required processing files in the USB flash drive. Press the [F1 Import] button, the system will process the USB flash drive. The file is imported into system memory. During the import process, there will be a progress bar display to prompt user the import progress. After the import is complete, the progress bar will disappear automatically. If you select [F2 Import and Load], the system will automatically load the imported file after importing the file into the system. Users can also delete and rename U disk files under this window.

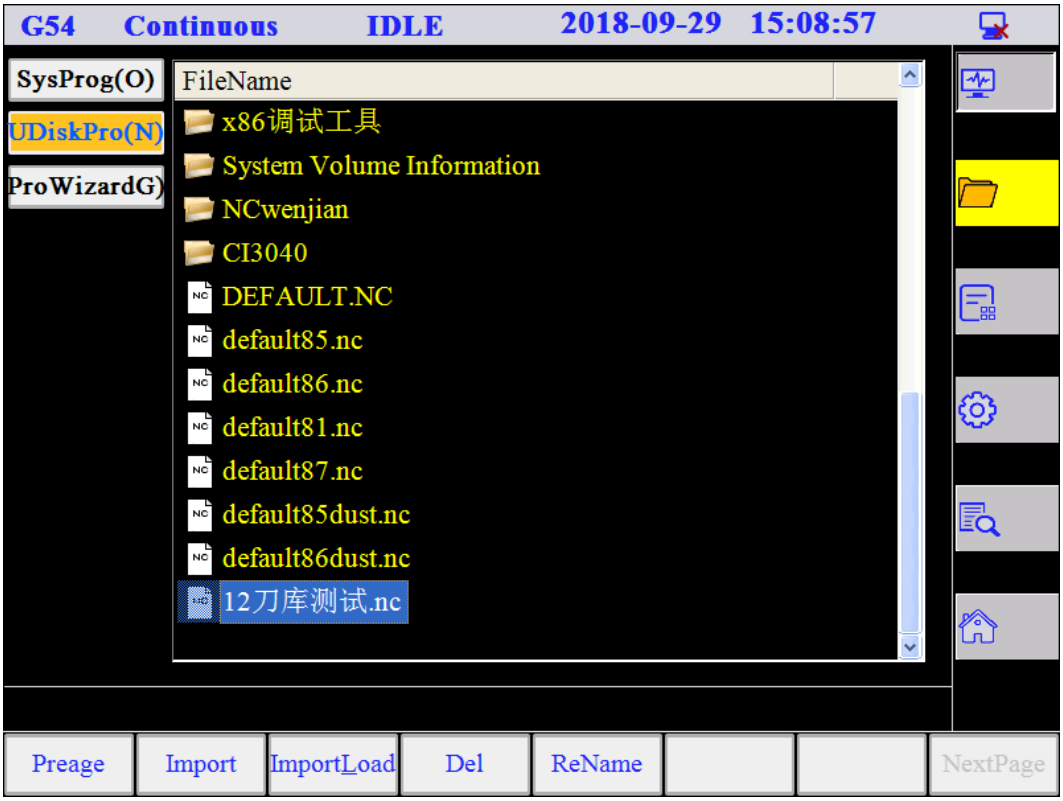


Figure 3-1 "U Disk File" Window

If the U disk is abnormal or the U disk is not found, a prompt box is displayed:

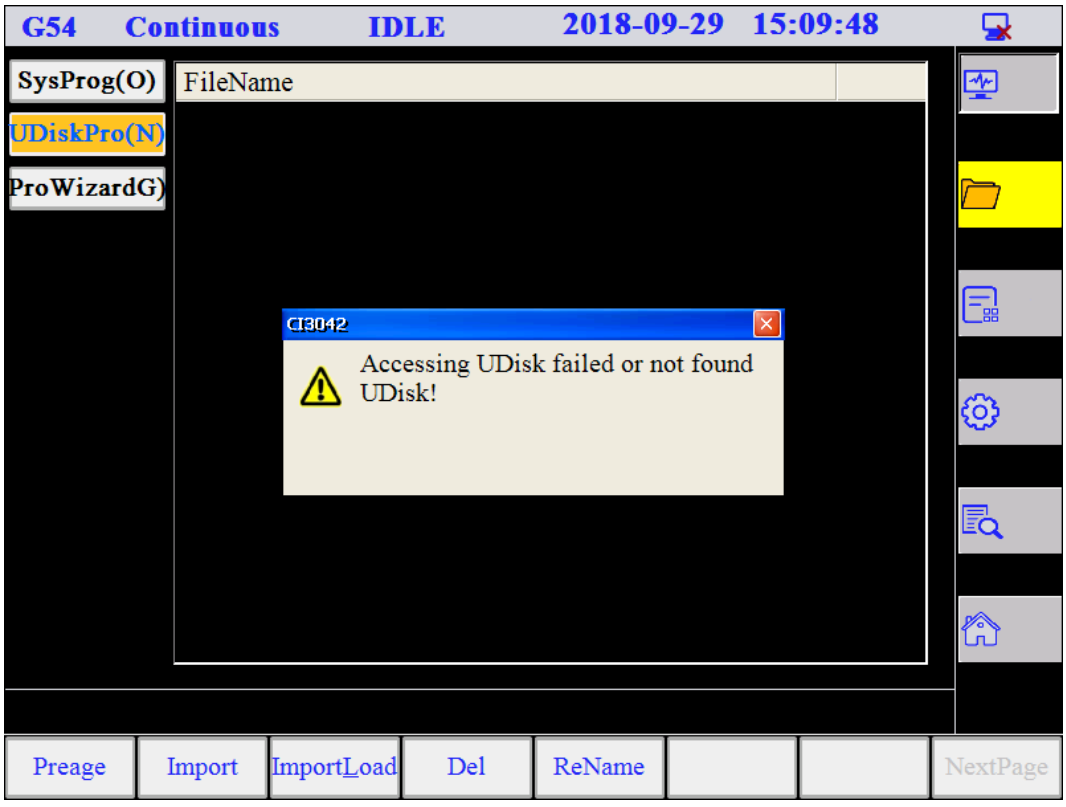


Figure 3-2 U Disk File Prompt Box

Tip: If you do not check the U disk, you can not do other operations while the status prompt area is prompted to read the U disk, until the system prompts that there is noU disk;

3.2 Manually Write Machining File

In addition to input files from the U disk, the user can also write machining files online. First press the [File] button to enter the [System Program (O)] menu item. Then press F5 [New] button, in the window it will create a new empty file named by default, the user can click the [Rename] button to rename the default named empty file. As shown below:

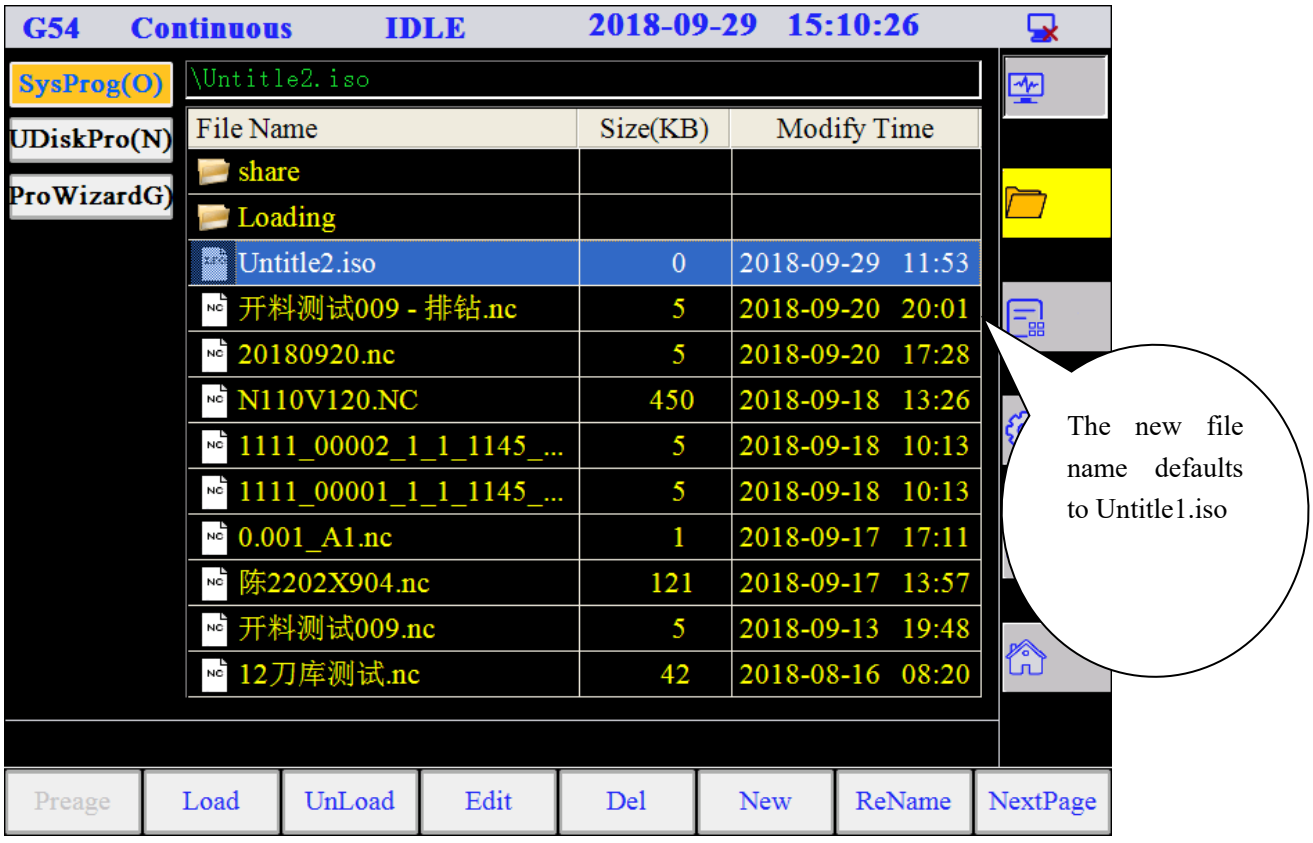


Figure 3-3 New Machining File

Press the F3 Edit button to manually write the G code. When writing the G code, F3 [Save], F4 [Save and Load], F6 [Return to Previous Level]. As shown below:

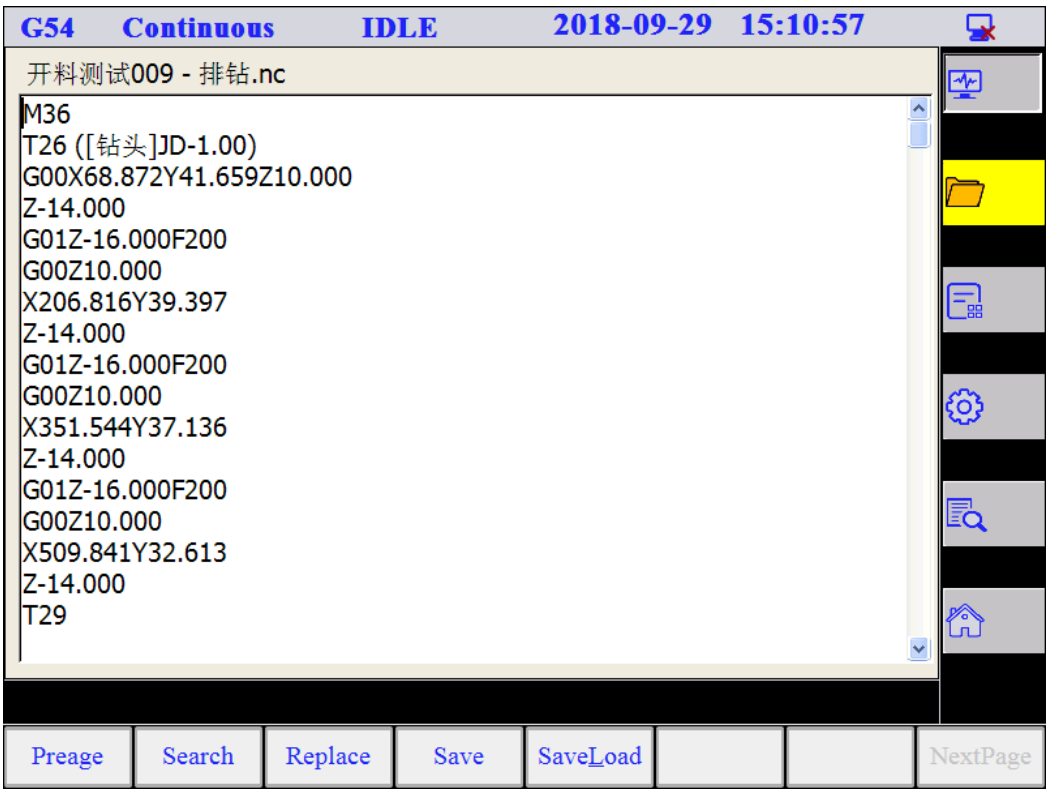


Figure 3-4 Program Editing

Note:

This editing window can edit programs larger than 5 megabytes in size. If the file size is larger than 5 megabytes, please edit it on a PC using a dedicated editor.

In the editing window, the G code can be input by the user. (At present, the system only supports editing function of G code.) The writing of G code must conform to the programming specification established by our company (see Part II for details), otherwise the system reports an error. After the input is completed, the system will automatically perform a grammar checking to ensure that the machine will not execute the wrong command and cause damage to the machine.

Special Note: If you do not exit the editing state with F6 key, the contents of the system file editing will not be saved.

Chapter IV How to Manually Machine

Manual Machining means that the machine tool manually processes the machining program according to the parameters set by the user. There are three ways to manually operate the machine: Handwheel Pulse Mode, Continuous Jog Mode, and Custom Step Size Mode.

The user can select the manual operation mode to process the program file: under the machining interface, through the mode selection area button 

to switch, you can perform the corresponding manual operation under the machining interface. There are eight manual buttons in the manual button area, which correspond to the positive and negative directions of X, Y, Z, and 4. Manual buttons provide an interactive operating environment for the user to manually manipulate the machine.

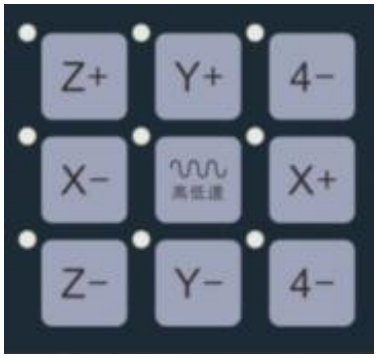


Figure 4-1 Manual Button Window

Note: You can move the machine in [Continuous], [Stepping], [Handwheel], [Idle].

4.1 Handwheel Pulse Mode

In the machining interface, the user can select the Handwheel Pulse Mode for continuous machining. Press the “Handwheel” button to switch the processing interface to the handwheel state. The motion of the machine is determined by the input of the handwheel.

1. Pulse multiplier selection on the handwheel: X1, X10, X100, respectively, indicating different pulse multiples of the handwheel.

2. Axis selection on the handwheel: the user can select X, Y, Z axes that need to be fed.

3. Stepping direction selection on the handwheel: each axis has positive and negative

directions. After selecting the stepping axis on the handwheel, the handwheel can be shaken in positive and negative directions, corresponding to the "+/-" arrow on the direction knob of the handwheel.

4. When the handwheel in X1, X10, X100 gear position, the corresponding distance of each handwheel can be set by parameters

5. Handwheel acceleration can be set separately

6. The handwheel supports two modes, 1 strict pulse count and 2 non-strict pulse count. In the case of strict pulse count, the distance traveled by the machine tool is strictly equal to the number of hand pulses. (Note: In this mode, if the hand pulse is too fast, it may cause too much buffering pulse. When the handwheel is stopped, the machine will go a long distance.) In the non-strict pulse count mode, the number of hand pulses and the machine running distance is not strictly equal. When the handwheel stops, the machine starts to slow down and stop.

Note: Check that the external handwheel device is properly connected before performing the operation. This mode is mainly used for rapidly positioning of machine tool.

4.2 Continuous Jog Mode

Press the "Continuous" button and the "Continuous" indicator lights up to enter the Continuous Jog Mode. In this mode, press and hold the corresponding axis control key on the keyboard. When the corresponding button is pressed, the machine moves; when the keyboard is released, the machine stops.

When performing Jog action, the track display window displays the relevant machining path.

4.3 Incremental Stepping

Similar to the Continuous Jog Mode, the Incremental Stepping Mode (Incremental Mode) is another manual operation mode. Unlike the Continuous Jog Mode, the Incremental Stepping Mode can accurately control the feeding distance of machine

moving axis

Before using this mode, select the mode  and it will automatically pop up the window to set the appropriate step size.

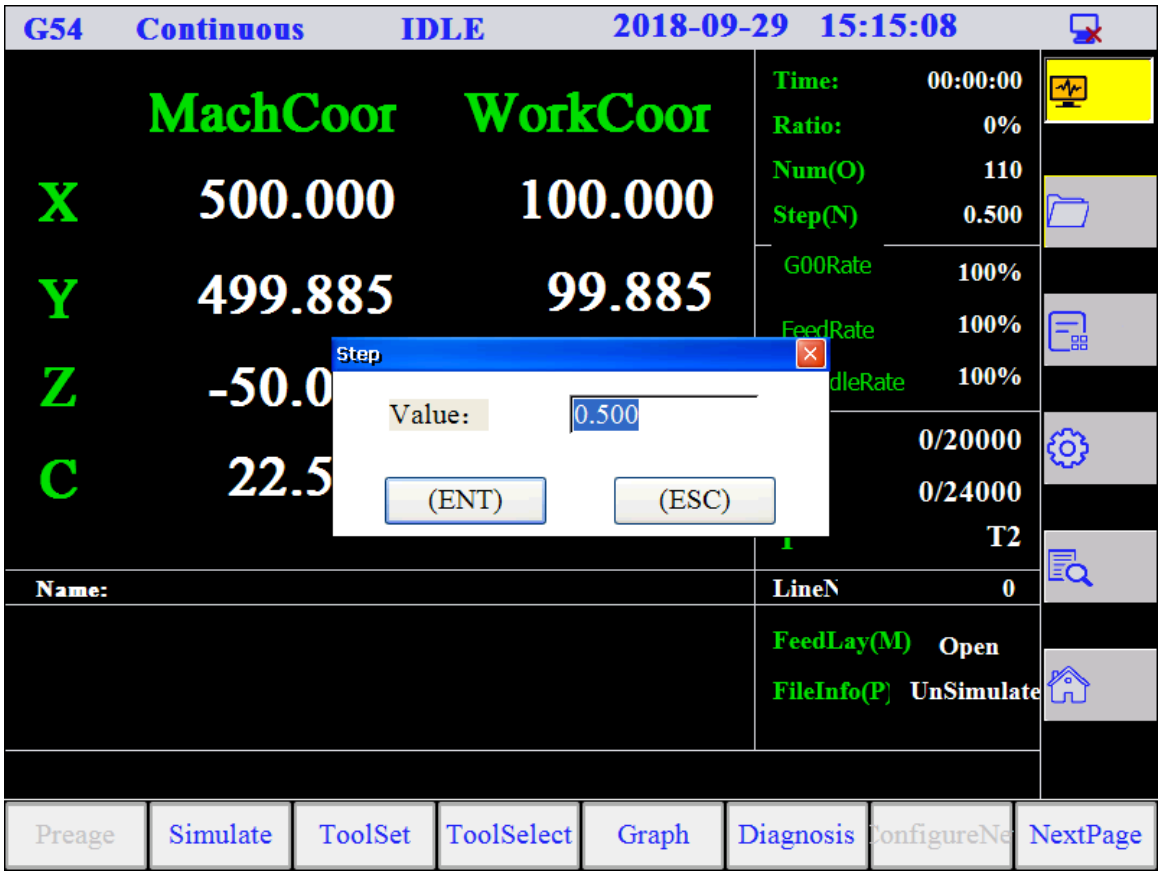


Figure 4-2 Step Size Setting Window

Set the feeding distance for each jog by modifying the jog step size. The user can view the step size distance in the status interface.

Figure 4-3 Jog Display

The unit for XYZ axis corresponds to mm, and for A axis, the unit of the step size is degree.

In the machining interface and stepping status, increase or decrease the jog step size by pressing the corresponding number key. Each time the number key is pressed, the corresponding axis moves for a given step size.

Note: Avoid setting the jog step size in the Z direction too large to avoid damage to the machine due to misuse.

Chapter V Workpiece Coordinate System Setting

The workpiece coordinate system is used by the programmer during programming. The programmer selects a known point on the workpiece as the origin (also called the program origin) and creates a new coordinate system called the workpiece coordinate system. The origin of the workpiece coordinate system (ie the workpiece origin) is determined relative to a point on the workpiece and is floating relative to the mechanical coordinate origin. The selection of workpiece coordinate system origin should satisfy the conditions of simple programming, simple size conversion, and small machining error.

The workpiece offset corresponds to the coordinate system G54, G55, G56, G57, G58, G59. When the system is turned on, the default coordinate system is G54. The relationship between the workpiece coordinate system and the workpiece origin is as follows

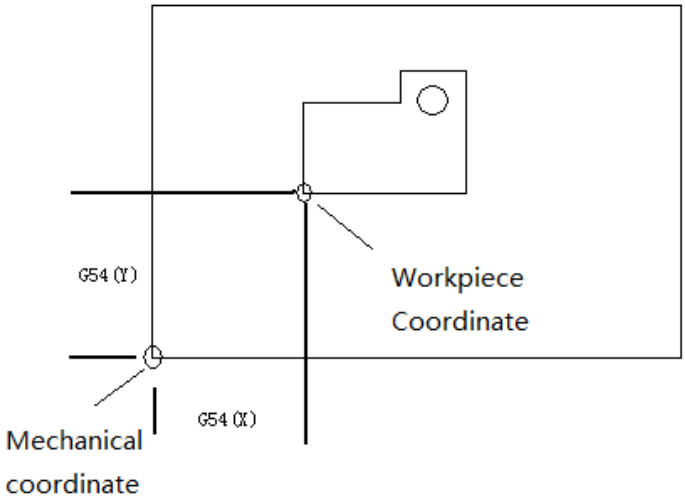


Figure 5-1 Coordinate System

Before machining the file, the user adjusts the position of the tool and workpiece by manual operation to start machining from a predetermined position of the workpiece. If the machining file does not specify a coordinate system, the current set coordinate system is used as the origin. The common offset is used to adjust the workpiece origin of the XYZ axis for all coordinate systems. This value does not change the offset value of G54 to G59. The workpiece offset, tool offset, and common offset satisfy the

following formula:

Workpiece Coordinate = Mechanical Coordinate - Workpiece Offset - Tool Offset - Common Offset

Note:

The offset value for each axis in the common offset is the result of the accumulation of several offset settings. The reason for this is that some workpieces have a deep machining depth, and the machine tool cannot be completely processed in one machining process, and the machining is required in several steps. For example, a workpiece has a machining depth of 4.5 mm, but the tool has a machining depth of 1.5 mm each time. This requires three offset settings to complete the machining, and each machining offset is 1.5 mm.

Since the origin of workpiece may be at the center of the workpiece while the path is being made, once the workpiece is clamped, the tool cannot reach the origin of the workpiece. This can be set by a common offset, such as with an X-axis rotary machine. The Z-axis center usually uses a fixed point above the center of Z axis coordinate origin as the tool setting point in the center of rotary axis. The position of this point is fixed from the center of the axis of rotation. For example, the distance is 50mm. You need to enter the distance from the current position to the center of rotation in the common offset -50mm. After the Z axis finishing tool setting in this position, similar to X and Y, click the button to set.

5.1 Manual Setting of Coordinate System

In the [Continuous] mode, manually move the X axis, Y axis, and Z axis to the predetermined machining position and switch to the Offset Interface: as shown below

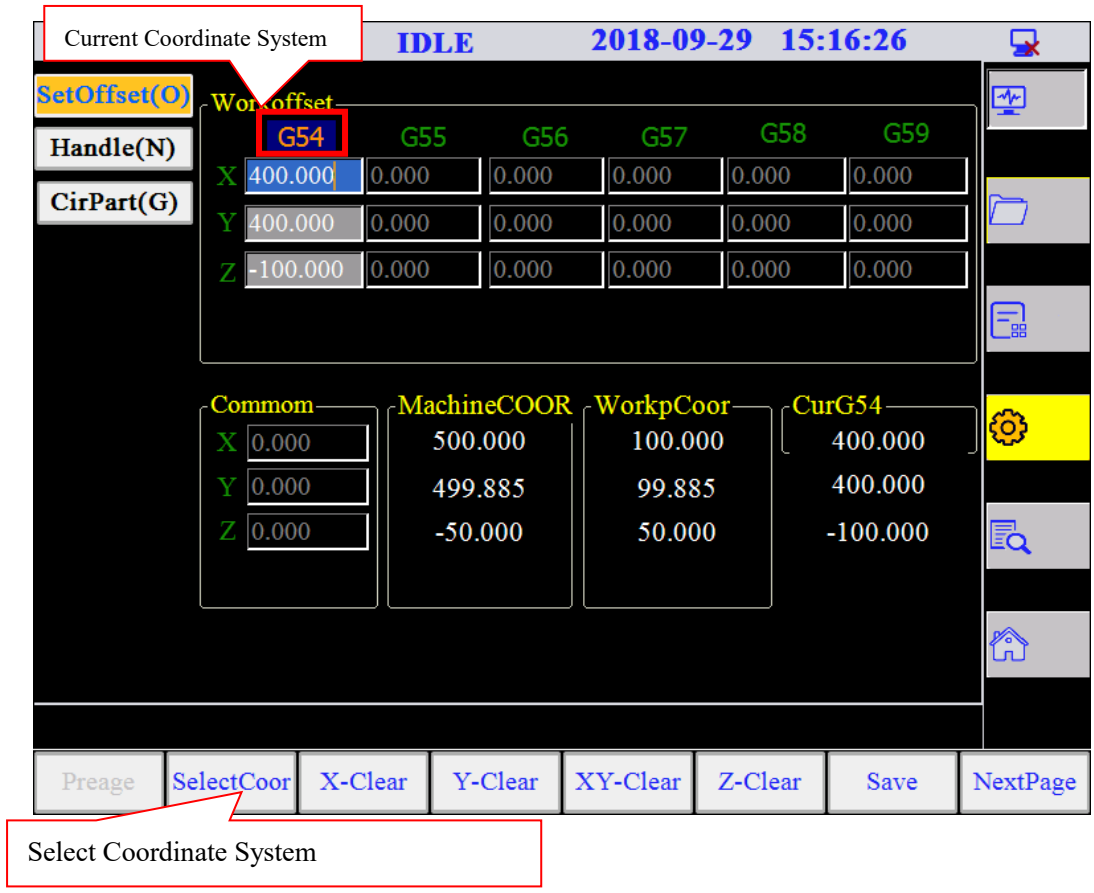
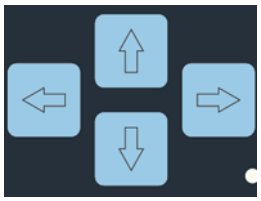


Figure 5-2 Offset Interface

Use the button  to view each coordinate system, and directly change the common offset value when the cursor moves to the common offset; in the state of T1, the Reset Menu can directly set the current coordinate value to the corresponding value of the current coordinate system.

Reset Menu, as shown below

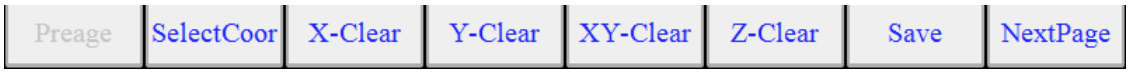


Figure 5-3 Reset Submenu

Reset the X-axis, Y-axis, and Z-axis coordinate values of the current position according to the dialog prompt. As shown below:

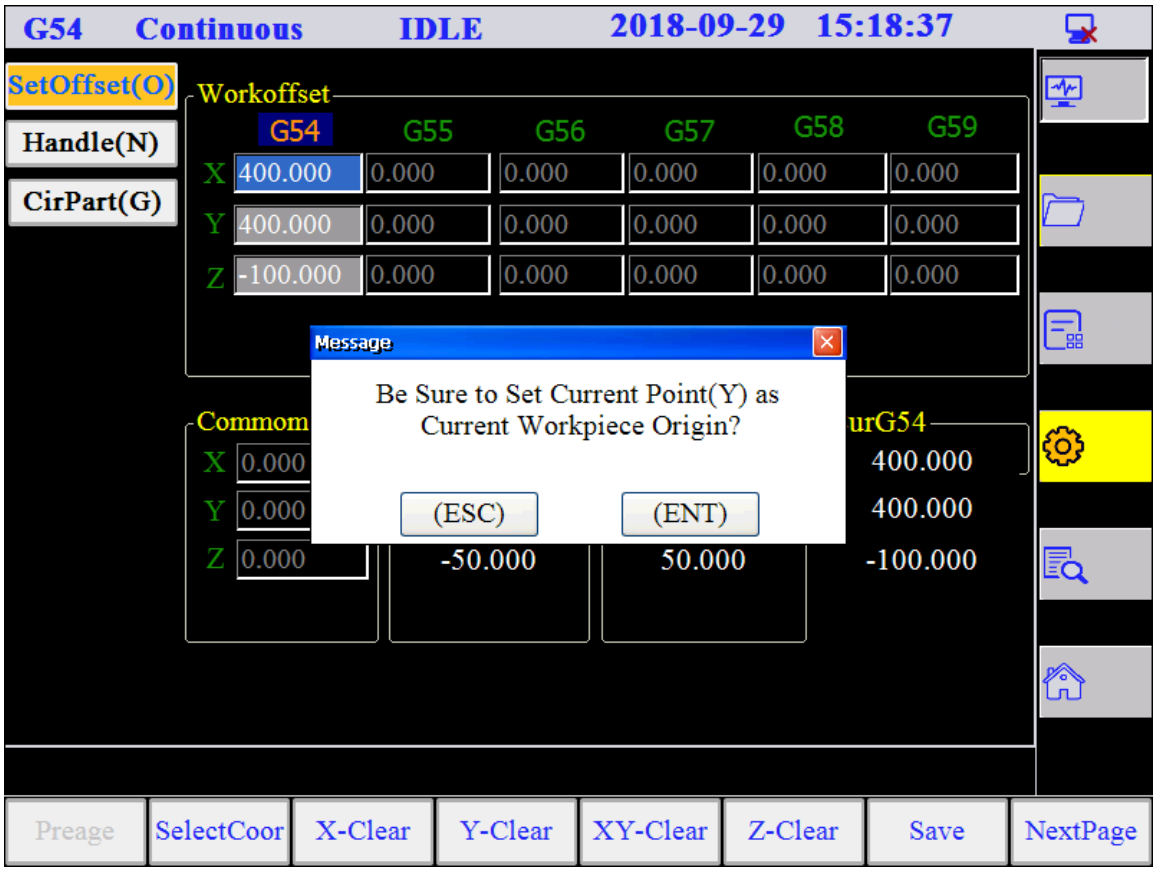


Figure 5-4 Setting Workpiece Origin

The [F1 Select Coordinate] button sets the coordinate system of the current editing to the current coordinate system.

5.2 Center Parting Setting

In addition to setting coordinate system directly, the XY workpiece origin can be determined by the center parting function - in the rectangular workpiece, two points to center part.

[Offset] → [Manually Center Parting X] into the center parting interface, as follows

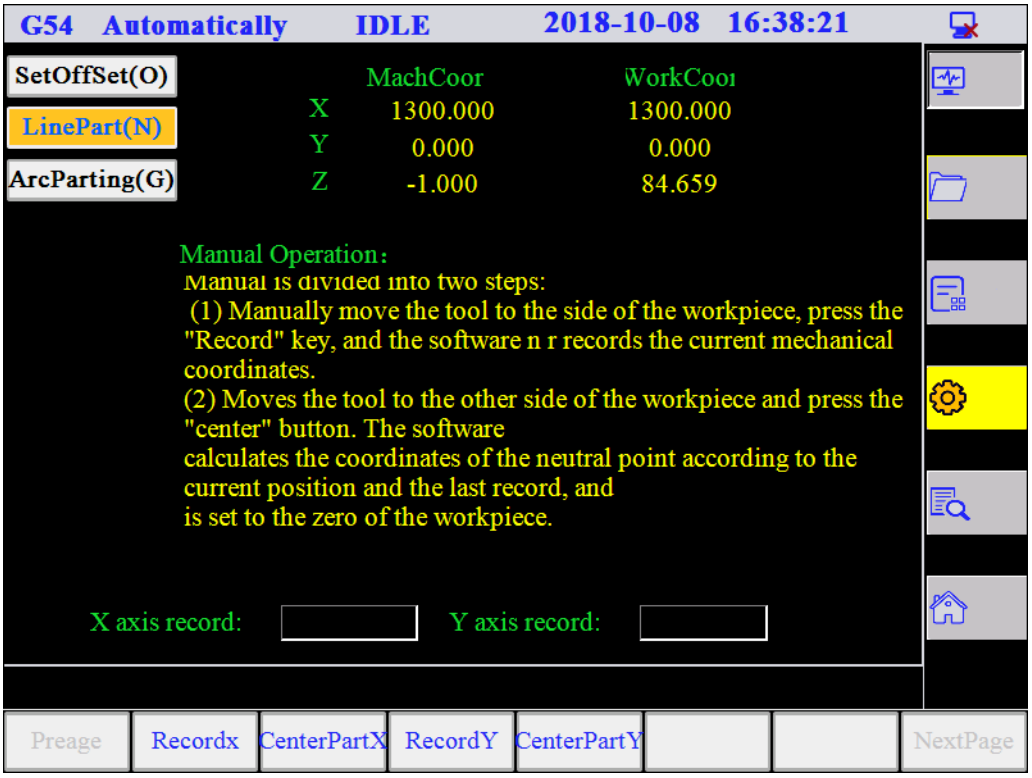


Figure 5-5 Center Parting

Lift the spindle in the direction of the Z axis;

Move to the side of the workpiece in the X direction until the tool hits the edge of the workpiece

Press [F1 and F3] to record the mechanical X, Y coordinates, then lift the spindle in the Z-axis direction

Press the direction key of each axis to move the spindle to the other side of the workpiece until the tool touches the edge of the workpiece;

Press [F2 and F4] to automatically take the coordinate center point on both sides as the workpiece origin.

5.3 Tool Setting Operation

In the G54 coordinate system, Z axis has two settings: 1. Floating Tool Setting and 2 Fixed Tool Setting.

1. Floating Tool Setting: The floating tool setting allows the user to easily determine the surface height of the workpiece and set the Z-axis workpiece origin. Similar to manual tool setting, since the workpiece origin of the Z axis is usually at the center of the rotary axis, it is necessary to set the distance from the position of tool block

to the center of the rotary axis to the common offset. (Or add the distance from the tool block to the center of the rotary axis to the thickness of the tool block.) The specific operation is as follows: Place the tool block on the surface of the workpiece and move the tool tip to the top of the workpiece by manual operation. Click the “Floating Tool Setting” button and a dialog box will pop upby the system and ask if the position of the tool block is correct and click "OK". The machine tool will perform the tool setting action. After the tool tip touches the tool block, it will automatically lift 10mm, and then add the thickness of the tool block to determine the Z axis coordinate.

The Floating Tool Setting corresponds to the toolbar [Machining Interface] → [F2 Tool Setting] → [F1 Floating Tool Setting].

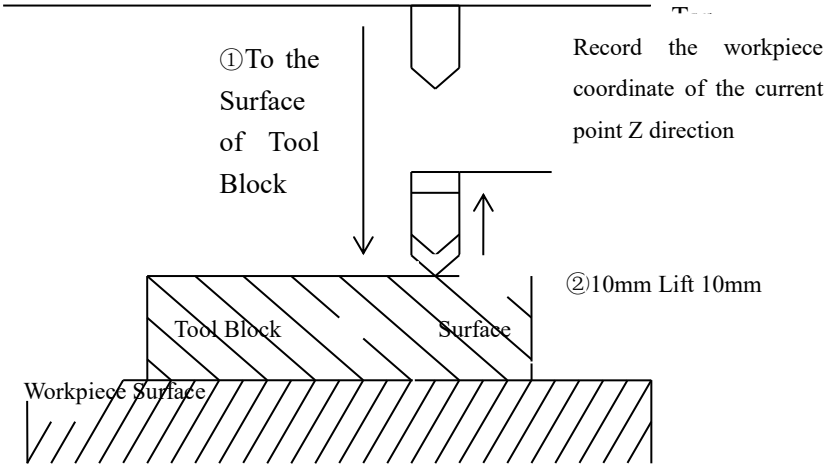


Figure 5-6 Floating Tool Setting Sketch Map

Note:

（1）Before tool setting, the user must ensure that the tool tip is located above the tool block. That is when the tool is down, the tool tip can touch the tool block. Otherwise, the tool will continue to go down, resulting in damage to the tool bit and the workpiece.

（2）The thickness of the tool block can be set in the manufacturer parameters. The workpiece origin coordinate in the Z-axis direction will be automatically compensated after tool setting.

（3）Due to the certain gap between the workpiece surface and the tool block, the gap can be compensated in setting the workpiece offset after the tool setting is completed.

The compensation difference depends on the gap size. If the surface of the workpiece is rough, the compensation difference can be larger. If the surface of the workpiece is smoother and the clearance to the tool block is smaller, the compensation difference can be set smaller. In general, 0.1mm can be used.

（4）Regarding the tool setting speed, it is performed in the parameter setting. The tool setting speed is in the range of 60-1000mm/min. If the tool setting speed exceeds the maximum value set by the parameter, the tool bit will be worn or the tool block will be damaged. It is recommended that the tool setting speed is less than 300mm/min.

2.Fixed Tool Setting: First, you need to set the mechanical coordinates of the tool setter in the parameter management. When the fixed tool setting action is performed, the system will automatically move to the corresponding mechanical coordinates of X and Y axis, and then start the Z axis tool setting. The Z axis tool setting action is similar to the floating tool setting action.

When perform fixed tool setting, in order to protect the tool, the speed segmentation method is divided into two types: fixed tool setting fast speed and tool setting speed. The speed can be set by parameters. When the tool tip is close to the tool setter, the fast tool setting speed is adopted. The normal tool setting speed is used after the tool tip is in contact with the tool setter.

Note: It is necessary to clear the surface of the T1 workpiece before tool setting! Fixed tool setting is completed under G54!

Installed → Connected Tool Setter → Test Tool Setter Signal → Set Tool Parameter → Set Fixed Tool Setting Parameter → T1 to Clear Workpiece Surface → Tool Setting after Tool Changing/ Automatic Tool Setting

Chapter VI How to Select File for Machining

6.1 Loading File

First, open the [File] management window, then select the file to be processed this time, and then press the [F1 Load] button at the bottom of the window. After the loading is completed, it will automatically jump to the [Status] interface, and the [Status] interface bar will display the loaded file name.

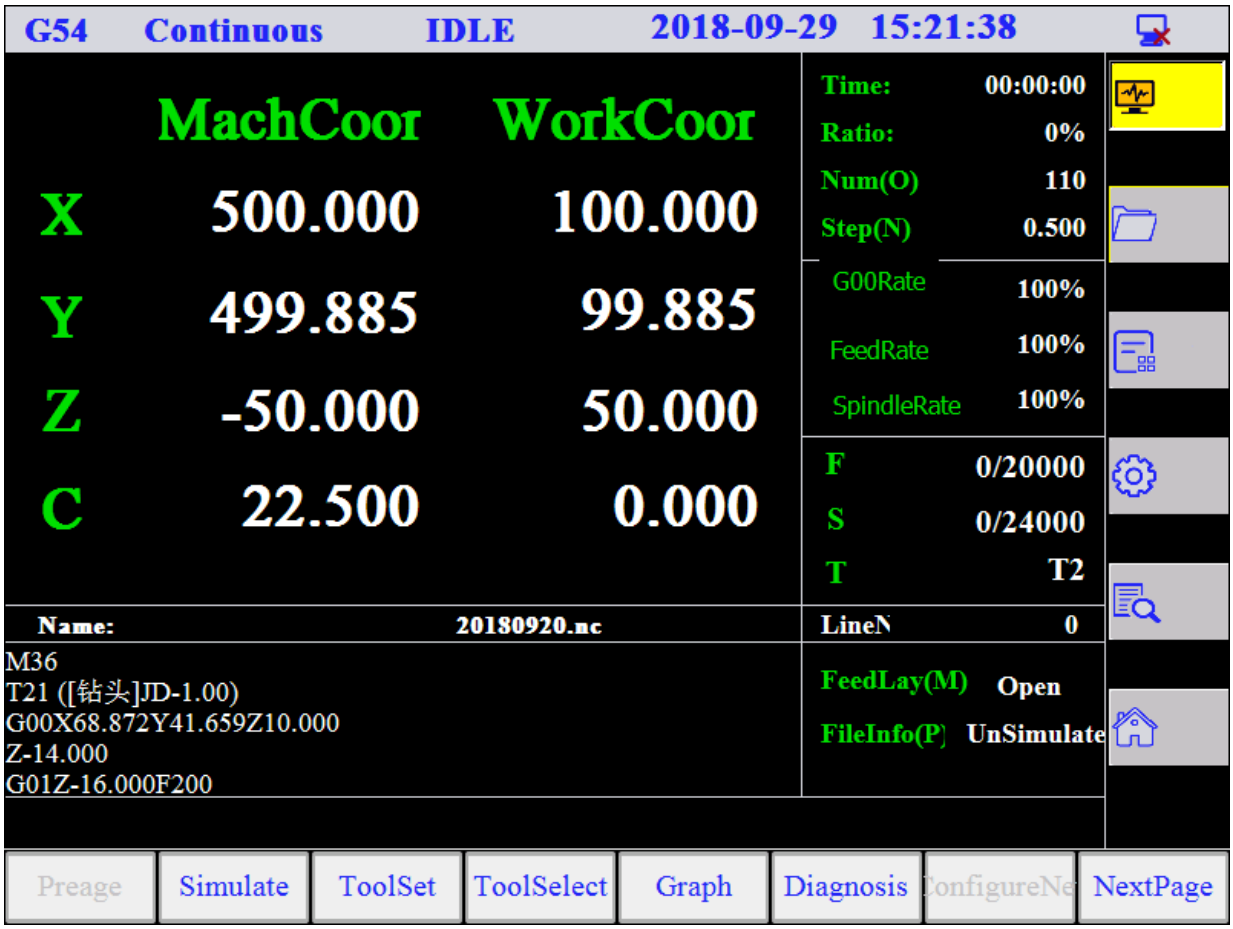


Figure 6-1 "Loaded" Machining File

6.2 Set Workpiece Offset

See Chapter V for details. If the workpiece offset is already set, there is no need to reset it.

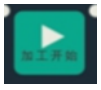
6.3 Machining Interface

Figure 6-2 Machining Interface

Origin

The mechanical origin and the workpiece origin are separated. The mechanical origin is controlled by the origin switch, and the workpiece origin is artificially set by the workpiece position.

6.3.2 Start

After the user selects the machining file, press the key  in the function button, and the machine will automatically machine from the first line according to the selected machining file. In the [Status] interface, press [F4 Graphic Window] to see that the machining track window displays the corresponding machining path according to the movement of the tool; in the [Status] window, you can see that the program is machinced line by line, the machining line number displays the current execution code line number, and the machining instruction keeps scrolling down. The user can view the current machining program code information through this window.

Note: If the parameter is set to <Return to Mechanical Origin before Machining>, the

system will prompt to return to the machine origin first. The automatic machining instruction cannot be executed without returning to the machine origin.

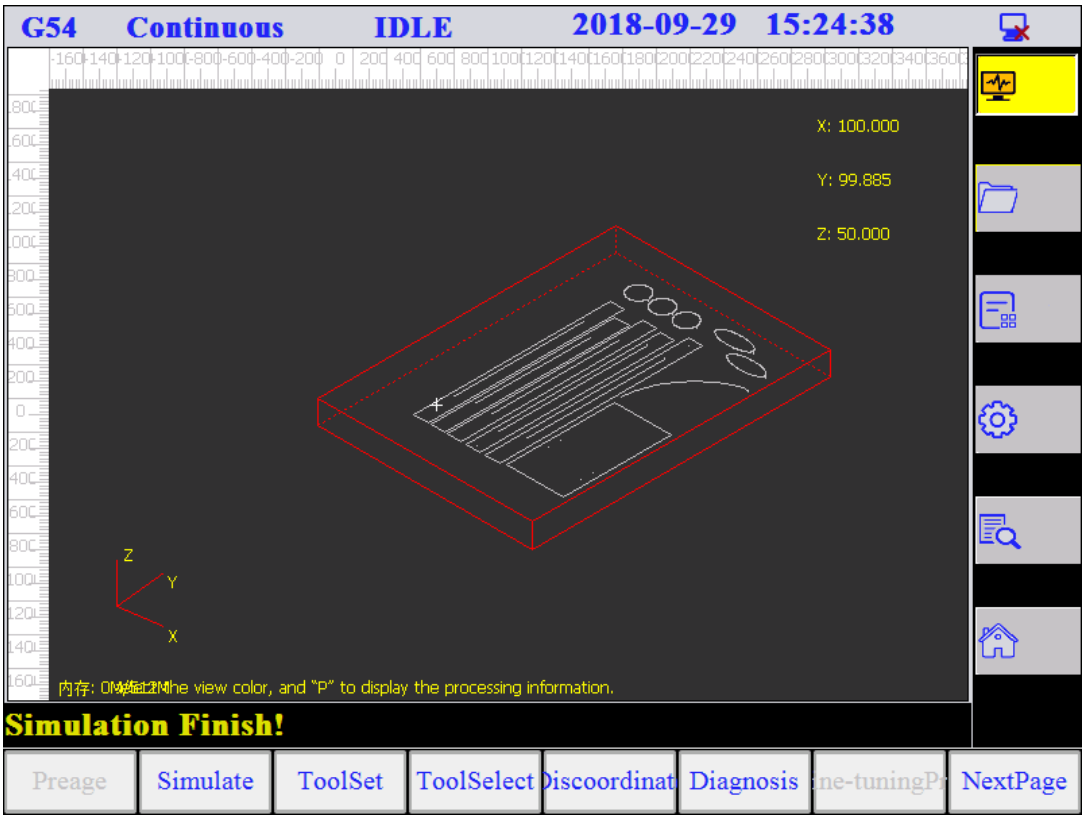


Figure 6-3 Graphic Display Window

Note:

The system will perform a grammar checking of the auto-machining file while machining, and the grammar checking will be performed earlier than the automatic machining (ie, the grammar checking has the "forward-looking" function). If the system checks that the syntax of a line program in the machining file is wrong, the error statement is highlighted and alarmed in the automatic machining window, and the automatic machining stops. The user can perform syntax, semantic checking and modifying, editing on the error statement. Click Save after modifying and editing, then press the “Advanced Start” button to select the starting line and continue machining.

New machining files cannot be loaded during automatic machining. In the status bar "Machining Information Window", the machined time of current file and the tool number information currently being used is displayed. It is convenient for users to

check the operation of automatic machining.

The feedrate can be adjusted by adjusting the feedrate magnification. The spindle speed can be adjusted by setting the spindle magnification. The value will be effective immediately after changing.

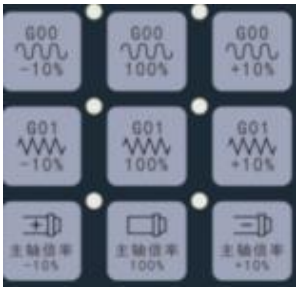


Figure 6-4 Magnification Setting

The start and end information for automatic machining is saved in the system log file. The system log records the important operations and events of the user. The user can not only view the log information that has occurred since the start of this operation in system log window, but also to review the history log information. This feature can help you with system analysis and diagnostics if the system fails.

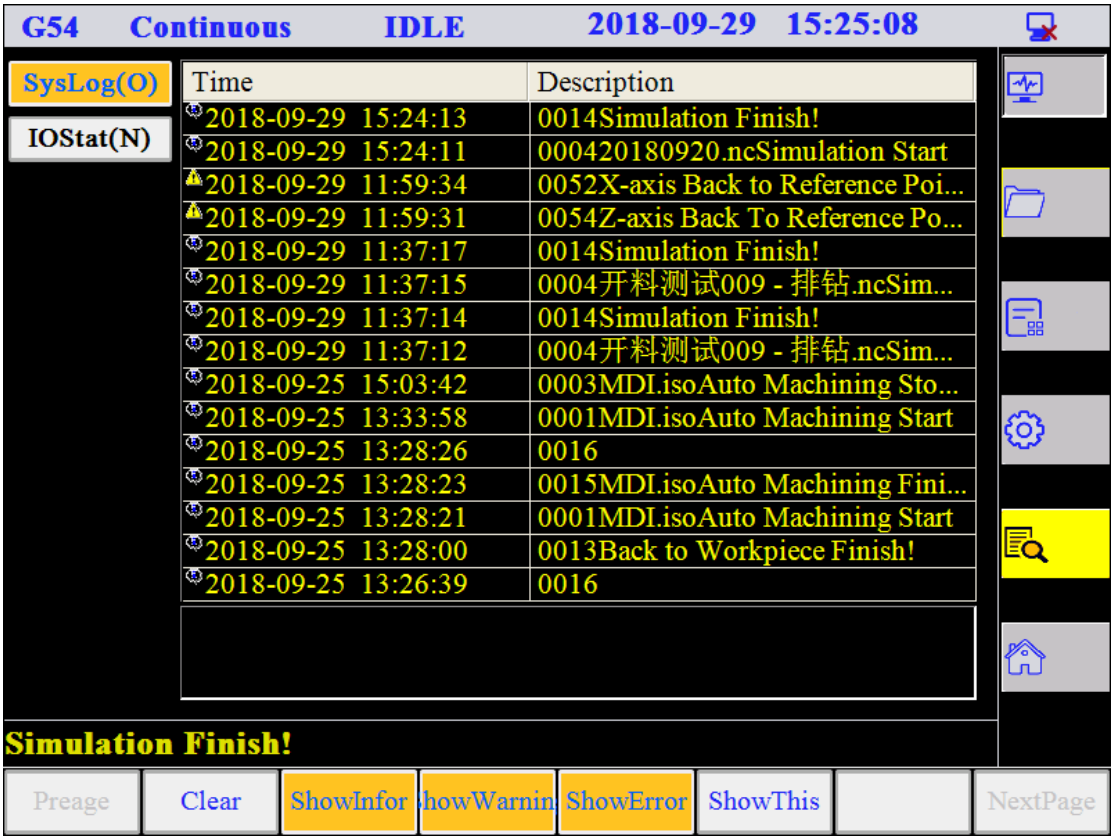


Figure 6-5 System Log Function

The log information currently recorded by the system includes:

(1) Automatic machining start and end information;

(2)Workpiece coordinate change;

(3) System alarm information;

(4) Machining completion information of the file;

(5) Some other system information.

Note:

Users need to periodically clear the system log. The system log is too large and can cause the system to run slowly.

6.3.3 Pause

The start button and the pause button are a reuse button. After the "Machining Start" is automatically processed, if you want to pause the machining, just press the machining pause button again, the machine will decelerate from the current speed until the speed is zero.

6.3.4 Stop

After the "Start" automatic machining, if the user wants to abort the machining file, press the "Stop" button: the machine will decelerate from the current speed until the speed is zero and the tool is lifted. The breakpoint is automatically saved when the system is stopped.

During automatic machining, if the system is in the simulation status, press the “Stop” button, the system stops the simulation, but does not exit the simulation status, at which time the user can analyze the simulation results.

6.3.5 Advanced Start

Sometimes the user does not need to process the entire file. It only needs to start from a certain line specified in the machining file and end the machining to the specified line number. This is the "Jump Execution" in automatic machining.

Press the [Advanced Start] button in the [Mode Button]. This function implements the function of program jump execution. Select this function, system will pop up a dialog box “Select for Machining”, as shown in Figure 6-7.

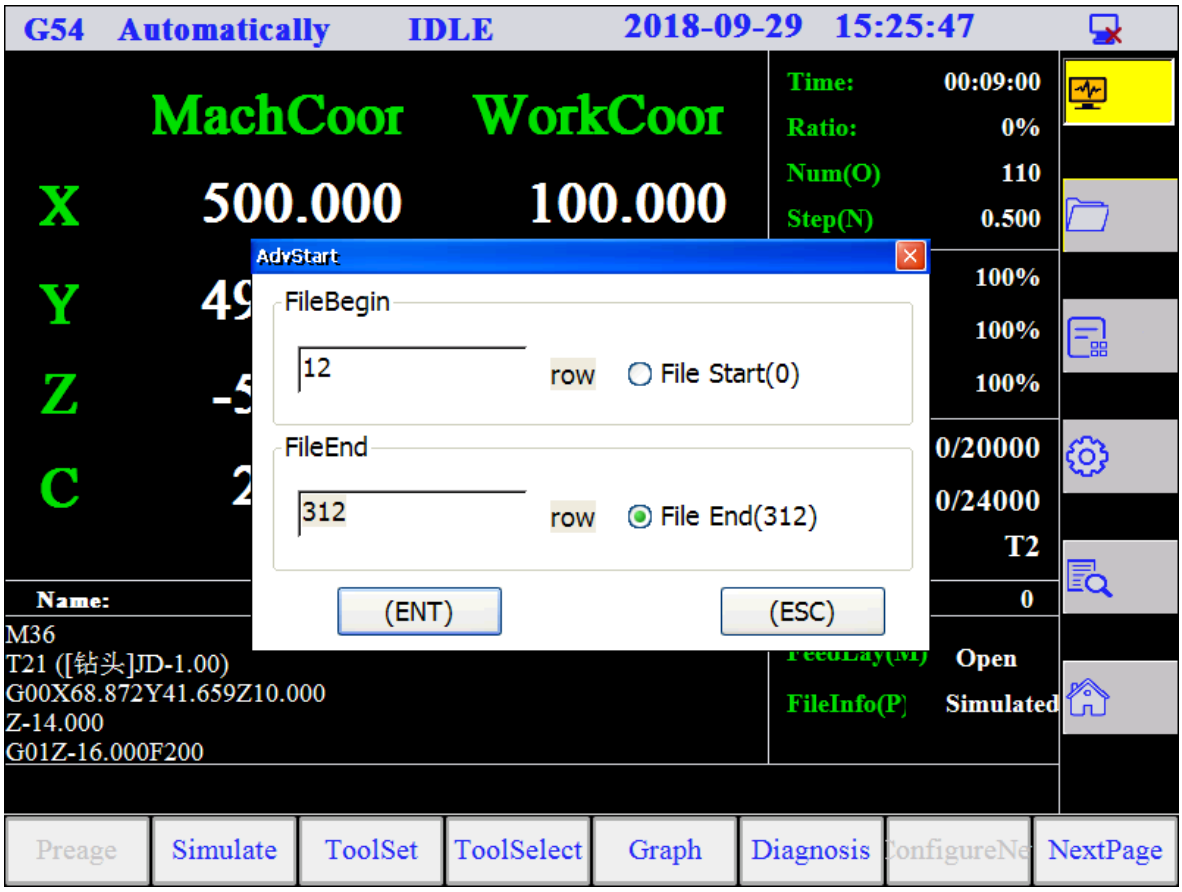


Figure 6-6 Select for Machining

The user sets the file start position line number and end position line number in the dialog box, and then clicks the "OK" button. The machine will execute only the specified program segment in the entire machining program according to your requirements.

Note:

If you choose to start from the beginning till the end of the file, the entire program file is processed. At this point, it can be considered as the maximum range of jump execution.

The "Select for Machining" function allows the user to easily process the interested program segment in the machining file or to check that whether a certain program in the machining file is correct.

Chapter VII How to Check Machining File

When the machining file is loaded and the current system status is “Idle”, the user can select [F1 Simulation] option in the submenu to perform high-speed simulation on the loaded machining file.

The simulation provides the user with a fast and realistic simulation machining environment. After the simulation starts, the system no longer emits pulses to drive the machine movement, just the high-speed tracking of the actual effect after the tool is machined in the window. Through simulation, the user can know the movement and machining effect of the machine in advance, prevent the machine from being damaged due to mistakes in editing the machining program, and also know some other additional information. Once the simulation process begins, press the [F1 Simulation] button and the simulation will terminate immediately. Tip: Simulation information includes

(1) When the parameter setting simulation limit is valid, the system will check if it will overstroke during the simulation. If the overstroke is prompted during the simulation, the actual machining will be overstroke without changing workpiece origin.

(2) Whether NC checks the syntax error when simulating by the parameter N15012 during the simulation; whether to check the NC file during the simulation.

(3) The system calculates the required machining time during the simulation. The simulation time is the same as the actual machining time without changing the machining parameters and machining magnification.

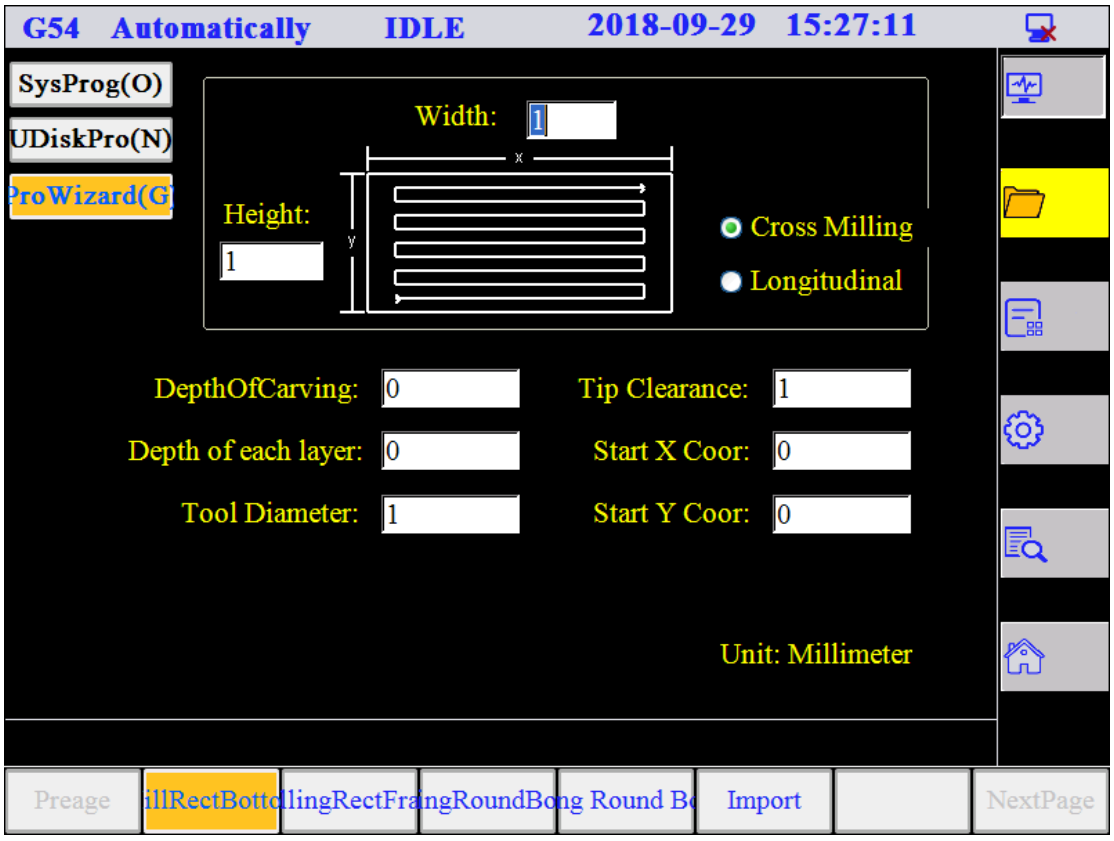
Figure 7-1 Simulation Status Coordinate Display

Chapter VIII How to Perform Milling and Milling Frame Operation

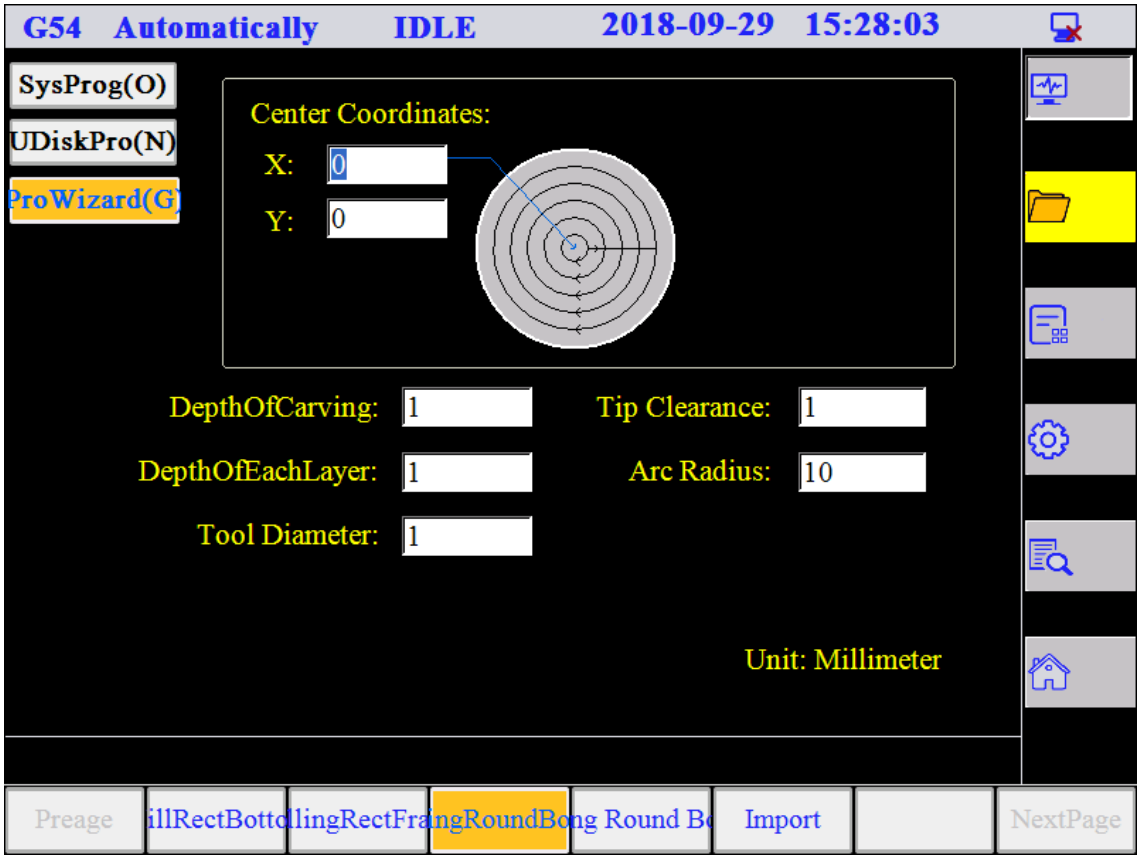
When the user needs to perform simple milling and milling of the outer frame, it is not necessary to manually write the G code or use the CAM software to generate the machining file. The system provides the execution of machining instruction function, and only needs to input a few parameters to complete.

The execution of machining instructions includes Rectangle Base Mill, Round Base Mill, Mill Rectangle Frame, and Mill Round Frame folding function window.

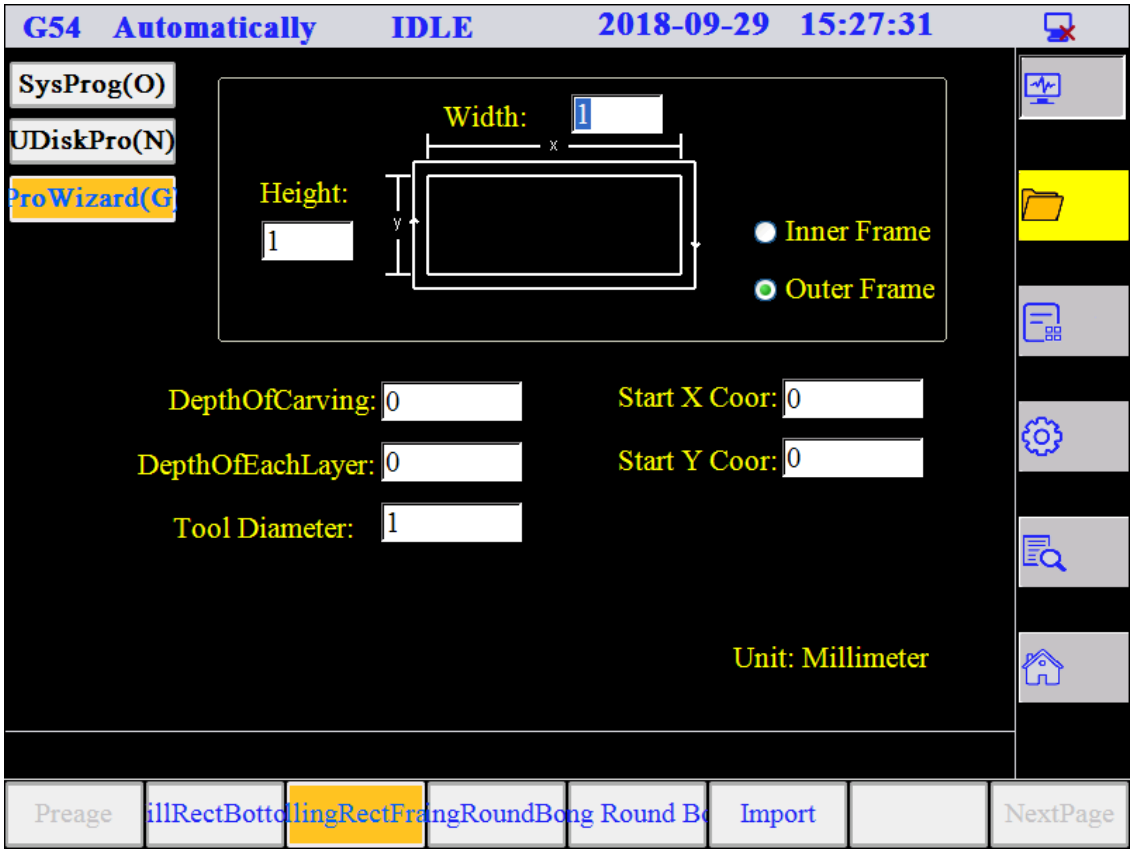
Rectangle Base Mill window:



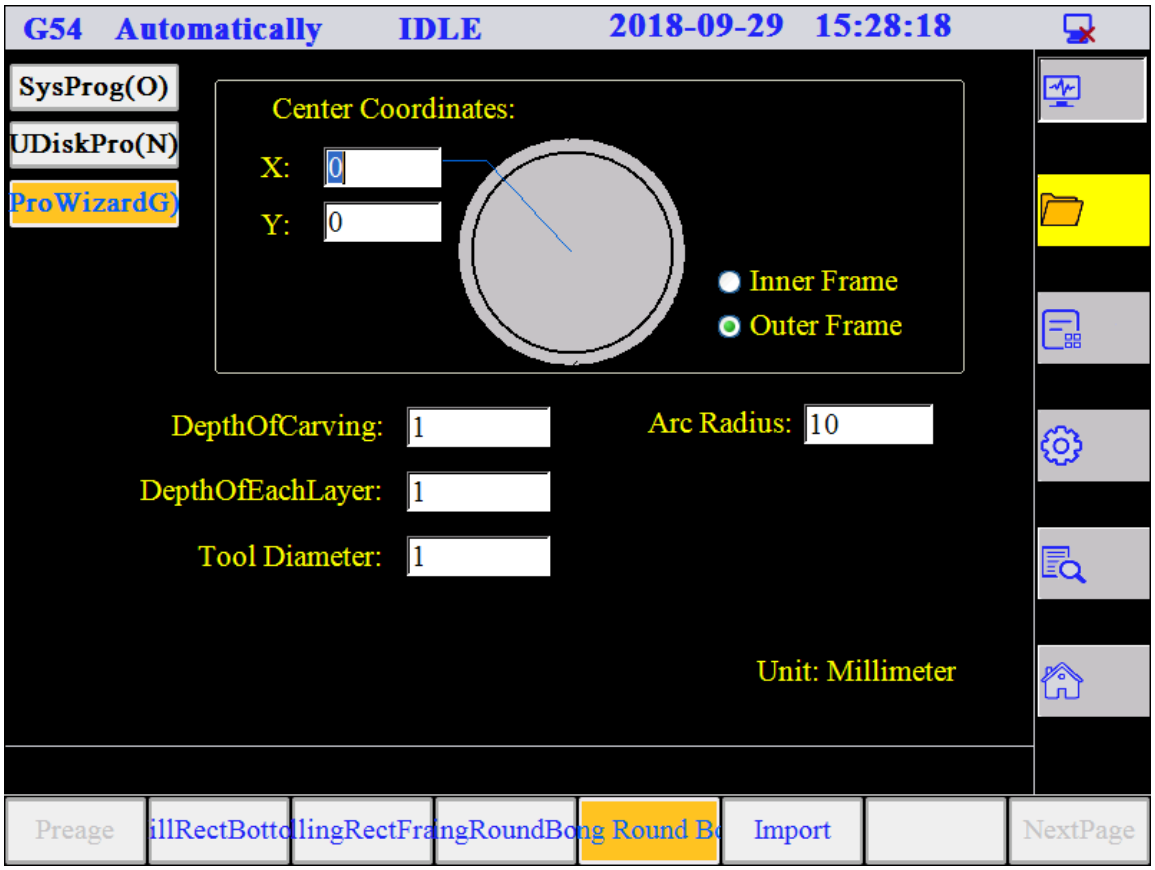
Round Base Mill window:



Mill Rectangle Frame window:



Mill Round Frame window:



Chapter IX Return to Mechanical Origin

The mechanical origin is a fixed position of the machine tool, which is determined by the mechanical switch and the electrical system, and is the zero point of the machine coordinate system. Executing the “Return to Mechanical Origin” function requires the machine itself to have an origin switch. If the machine does not have the relevant hardware support, you need to disable this function. For details, see “Origin Parameter” in Chapter XI Parameter Management. Since the mechanical origin is the reference for the entire machine, the important role of this function is to correct the current point coordinates. To prevent power failure or to cause the current position to be incorrect, perform a mechanical origin return operation after the program is started or an emergency stop occurs.

After the system is powered on, it will enter the mechanical origin status, click the submenu button, the corresponding axis will automatically return to the mechanical origin, and the system coordinates will be corrected. Before X or Y axis returning to the origin, return the Z axis to the mechanical origin first. Worktable stroke inspections only work after returning to the mechanical origin.

Press the status bar  button, the system submenu will pop up the origin dialog box as shown below:

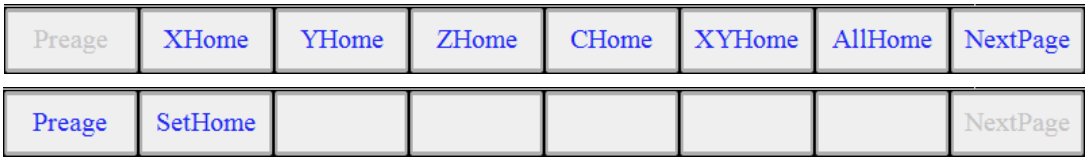


Figure 9-1 Return to Mechanical Origin Function Screen

The single-axis return to its own mechanical origin includes:

X axis returns to mechanical origin

Select this command, X axis returns to mechanical origin.

Y axis returns to mechanical origin

Select this command, Y axis returns to mechanical origin.

Z axis returns to mechanical origin

Select this command, Z axis returns to mechanical origin.

C axis returns to mechanical origin

Select this command, C axis returns to mechanical origin

XY axis returns to mechanical origin

Select this command, XY axis returns to mechanical origin.

All axes return to mechanical origin

Select this command, all axes return to mechanical origin. When perform all axes returning to mechanical origin, the Z axis will return first, then X and Y will return to mechanical origin at the same time.

Note:

If the machine don't return to mechanical origin, please manually raise the Z axis as much as possible during manual operation to ensure that the tool head does not collide with the workpiece to be machined.

When the system exits, the current coordinate information is automatically saved. If there is a sudden power failure during the automatic machining process, the system will automatically save the relevant information before the power failure to the breakpoint protection file (The breakpoint protection file saves the breakpoint information, file name, etc. into the system memory when the power is turned off. The same machining file corresponds to only one breakpoint protection file). After the power is restored, the system will pop up a prompt box to prompt the user for power failure of a certain machining file. The user must manually perform Return to Mechanical Origin operation, and then continue to process the powered down file at last time, or re-select the new machining file:

1. If the user wants to continue machining the file that was last lost last time, press the "Advanced Start" button, select the starting line number, and click "ENTER". The machine will continue to seamlessly process the files that have not been processed before the power is turned off from the breakpoint.
2. If a new file is selected for machining, after the machining is finished, the user can

continue to process the file that was previously powered down, and the machine will continue seamlessly machining from the breakpoint of the corresponding file.

3. The system supports the encoder zero position auxiliary to find the zero position function. Turning on this feature can make the position finding more accurate, and the position of the power-down recovery memory is more accurate. To enable this function, you need to set the encoder zero position in the origin parameter to set the valid parameter to Valid. Each time the zero position is found, the system records the distance value of the origin switch from the zero position of the encoder and writes the value to the system log. The user can adjust the relative position of the origin switch and the encoder zero position according to the distance value. Because if the distance value is too small, it’s easy to cause the problem of overstroke.

Chapter X Program Management

Select the [Program] menu item window to switch to the [Program Management] window. As the picture shows:

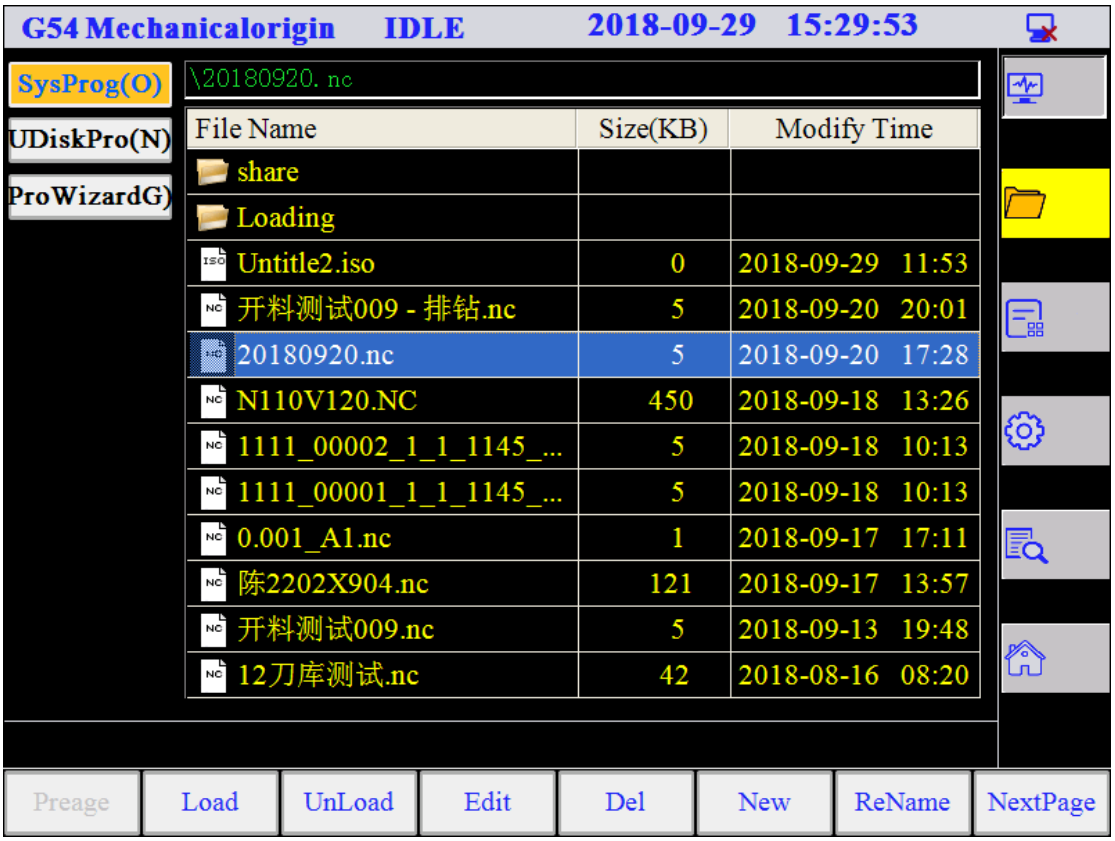


Figure 10-1 Program Management Menu

In the program management function, you can do the following:

10.1 New

Click the [F5 New] button at the bottom of the window, and a new empty file named by default will be created in the window, as shown below:

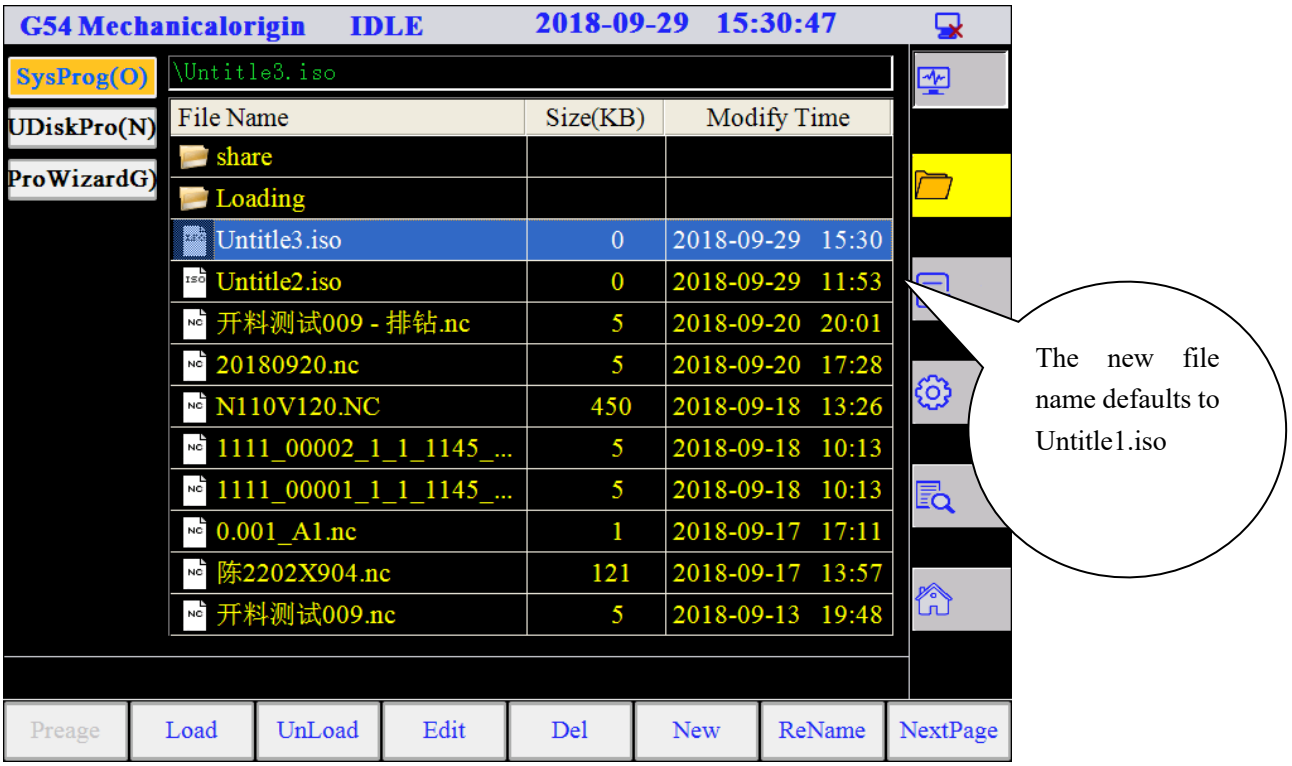


Figure 10-2 Program Management Window

The user can click on the [Rename] button to rename the default named empty file. Rename and click the [Edit] button to edit the generated new file (Note: The system only supports the writing and editing of standard G code). The user can implement manual data input (MDI) through the keyboard, execute and display it. MDI is characterized by simple input, quick verification and calibration syntax, and easy modification. It is suitable for parts with simple shapes and short programs. After the user has finished editing, click [Save] button below to save the file.

10.2 Edit

Select [Program Edit] menu item in the [Window] menu or switch the status bar window at the bottom right of the main interface to the [Program Management] screen. You can edit and modify the file by selecting the file to be edited in the window and clicking the "Edit" button at the bottom of the window. After the user has finished editing, click the "Save" button below to save the file. The system must be shut down after saving. As shown below:

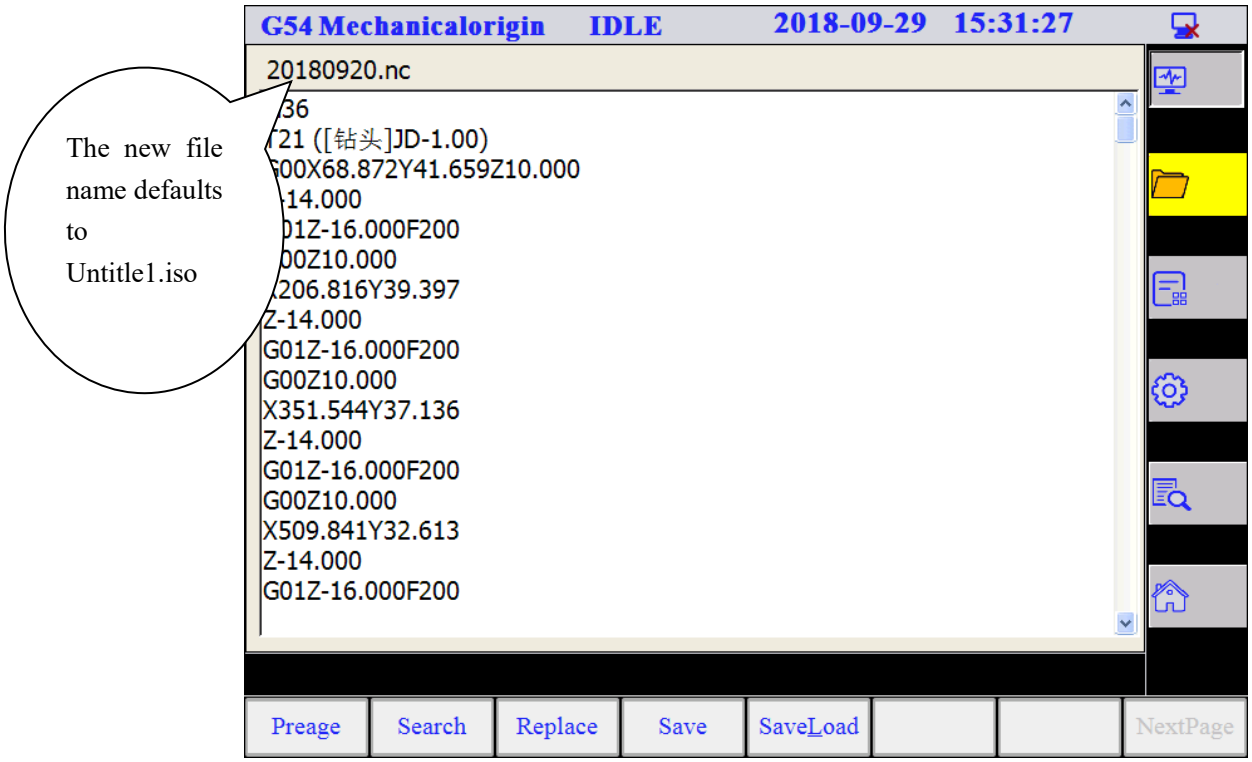


Figure 10-3 Edit

For newly created or edited files, the system will automatically check the syntax before saving. The user must edit the file according to our programming specifications, otherwise the system will report an error. See Part II for details.

Note:

This editing window can edit programs smaller than 10 megabytes in size. If the file size is larger than 10 megabytes, please edit it on a PC using a dedicated editor.

Any text can be input by the user in the editing window. After the input is completed, the system will automatically perform grammar checking to ensure that the machine will not execute the wrong command and cause damage to the machine.

10.3 Delete

Select a file by pressing the up and down buttons on the operation panel and click "Delete".

10.4 Rename

Rename files in the system.

10.5 Output to U Disk

Output the files in the system to the USB flash drive.

10.6 Array Machining

The user can perform array machining on the file, click [Program], first select the machining file to be arrayed, and select [F7 Array Machining] in the sub-menu on the program management page to pop up the array machining dialog box. Enter the array row number, array column number, array row spacing, array column spacing. After inputting the parameters, click Generate Machining File to generate the array file in the program management directory. The file name after the array is named by the following rules: Original File Name Array Row Number X Array Column Number

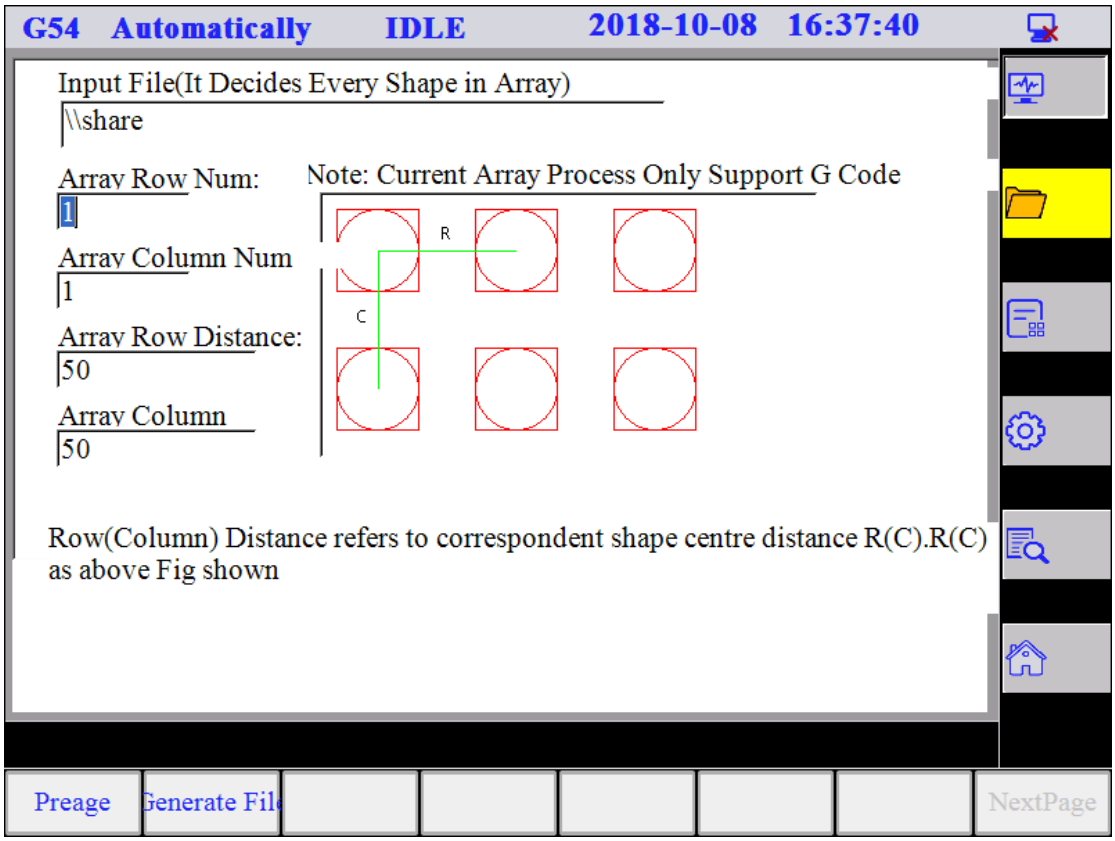


Figure 10-4 Matrix Function

Chapter XI Parameter Management

[Parameter Management] the interface contains the items related to the parameters. Press the [Parameter] button in the main menu to enter the [Parameter Management] interface:

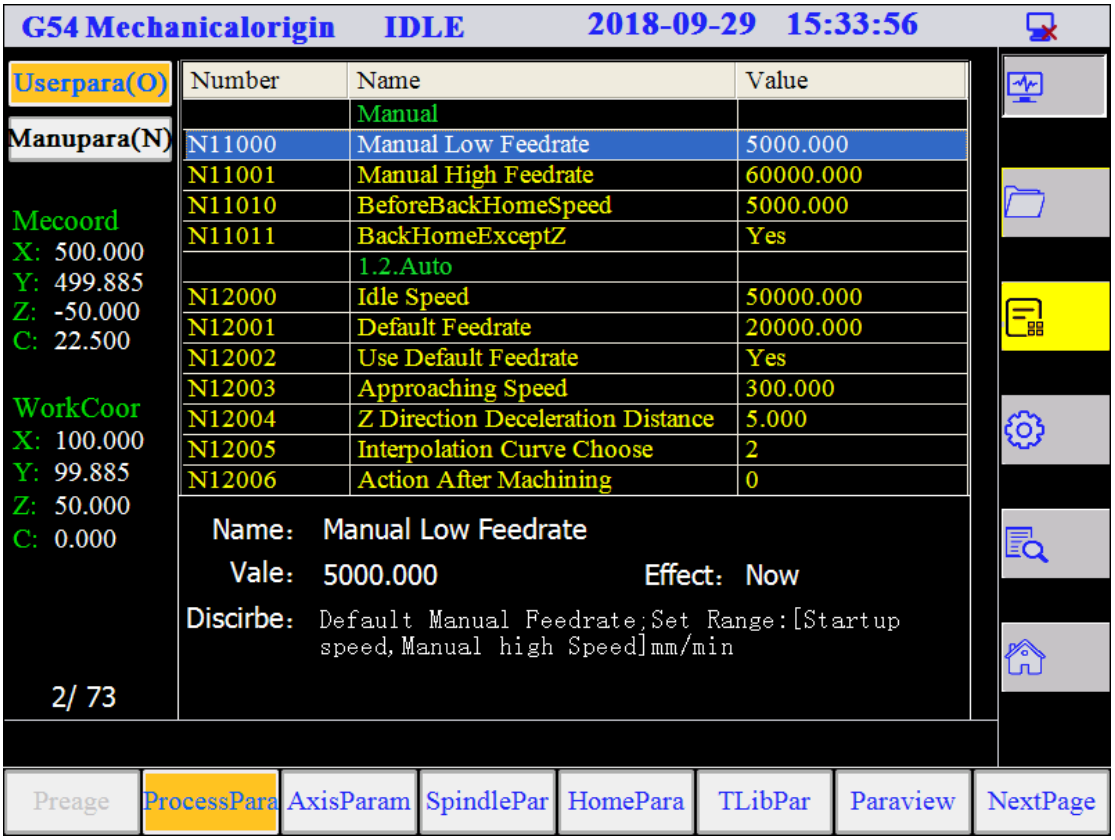


Figure 11-1 "Parameter Management" Interface

Set Parameter: This function is used to open the parameter window for parameter setting, which are divided into two parts: the user parameter and the manufacturer parameter.

Parameter Backup: This function is used for parameters backup for later use.

Parameter Backup to U Disk: This function is used to output system parameters to the U disk as a file.

Parameter Recovery: This function restores the parameters to the last set value.

Restore Parameter from U Disk: This function is used to restore the parameters saved by the U dish to the system.

Change Password: This function can effectively protect the security of parameter setting. The parameter modification requires permission, that is, password, you need to

enter a new password. Under normal circumstances, the parameter part shows the user parameters for general processing. If you need to modify the machine performance related parameters, such as pulse equivalent, spindle maximum speed and other parameters, you must enter the password and open manufacturer parameter to modify.

The manufacturer enters the original password of the parameter, that is, the power to modify the parameters related to the machine performance. Once you have entered the system, you need to change your password immediately. Manufacturer Password in System→ Auxiliary Function → Password Management Modification

11.1 Set Parameter

Select [Set Parameter] menu item in the [Parameter Management] menu, and the following window will pop up. This function is used to set the parameters under the parameter permissions, which are divided into two parts: user parameter and manufacturer parameter.

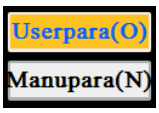


Figure 11-2 Parameter Selection

"User Parameter" setting: The system enters the user parameter by default, and sets the operation parameter, feeding axis parameter, spindle parameter, origin parameter, tool library parameter and parameter overview in the parameter category. After the setup is complete, all user parameters will take effect.

Note:

Under normal circumstances (the default state), the parameter part shows the user parameters for general processing. If you need to modify the machine performance related parameters, such as pulse equivalent, spindle maximum speed and other parameters, you must enter the password and open the manufacturer parameters to modify.

"Manufacturer Parameter" setting: Press 

to enter into "Manufacturer Parameter" setting, and then set the operation parameters, feeding axis parameters, spindle parameters, origin parameters, compensation parameters and tool parameters in the parameter category. After the setup is complete, all manufacturer parameters will take effect. If it is the first time to enter into the manufacturer parameters after booting, you need to enter the manufacturer parameter password. A password box will pop up, as shown below

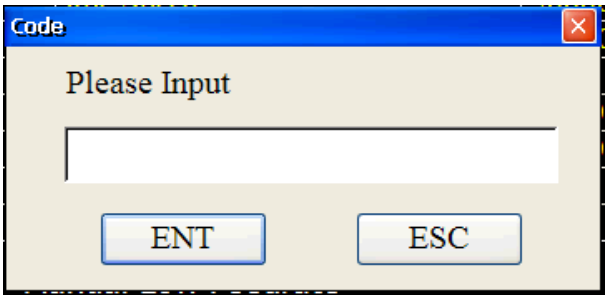


Figure 11-3 Password Input

11.2 Parameter Backup

This function is used to save all the set parameters for backup by the operator. In the parameter interface, select [F7 Parameter Backup] to pop up the backup file saving location, as shown in the following window.

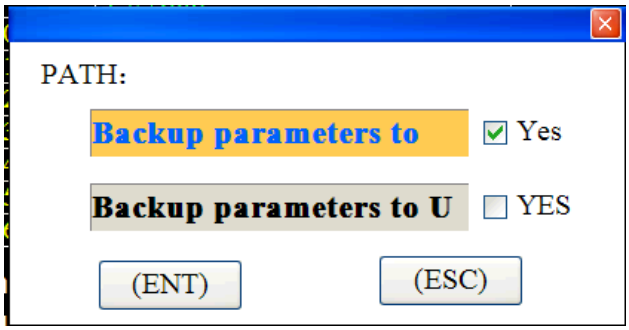


Figure 11-4 Backup System

Enter the backup file name in the window and click the "OK" button, all the parameter values will be saved.

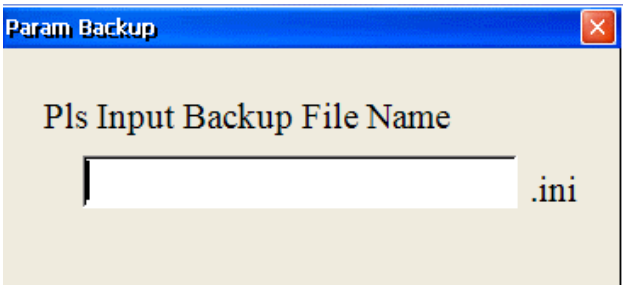
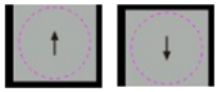


Figure 11-5 Parameter Backup Name

11.3 Parameter Recovery

Select the [Next Page] → [F2 Parameter Recovery] menu item in the submenu, and the following window will pop up. Using  to select to restore the parameters from the system or from the U disk.

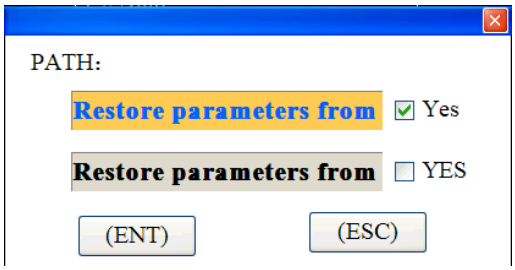


Figure 11-6 Parameter Recovery Selection

Then press 

to enter the backup parameter selection box:

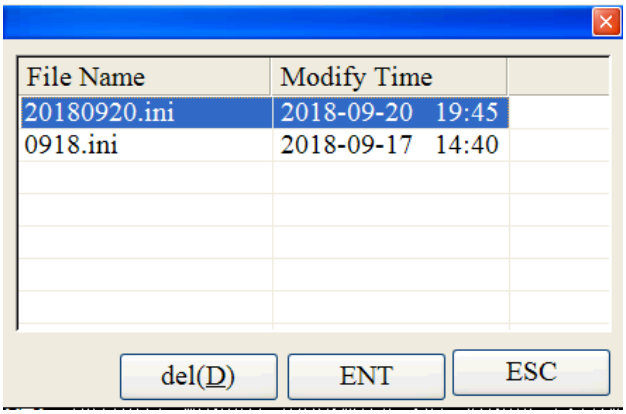
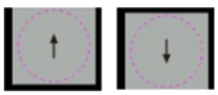


Figure 11-7 Parameter Recovery

Also use the button  to select parameter file, press 

to restore the parameters, the system automatically restarts.

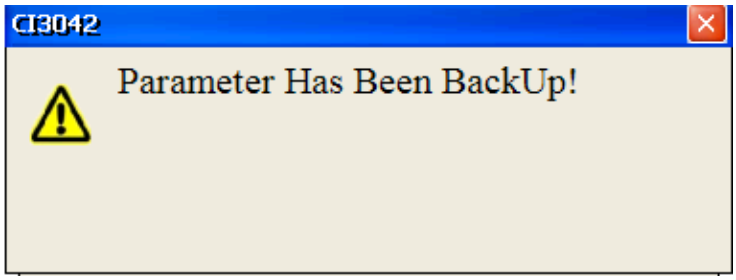


Figure 11-8 Parameter Recovery Successfully Prompt

11.4 Parameter Modification Permission

Under normal circumstances, the parameter part shows the user parameters for general processing. If you need to modify the machine performance related parameters, such as pulse equivalent, spindle maximum speed and other parameters, you must enter the password and open the manufacturer parameters to modify.

After the manufacturer parameter original password is entered, it has the right to modify the parameters related to the machine performance. Once you have entered the system, you need to change your password immediately.

11.5 Parameter Modification Method

To modify the parameters, press the up and down arrow keys to move to the parameter to be changed, press the Enter key, and then enter the value in the parameter input area; or use the mouse to directly double-click the line of the parameter and enter the value in the parameter area.

For "Yes\No" type parameter, input "1" for "yes", input "0" for no, or "yes" or "no" for direct input.

Note:

All parameters cannot be modified in the machining state and must be modified in the idle state.

11.6 User Parameter List

11.6.1 Operating Parameter

Number	Name	Value	Effective Time	Discription	Category
11000	Manual Low Speed	[Startup Feedrate - Manual High	Effective Immediatel	Default speed in manual mode	User Parameter

		Speed]	y		
11001	Manual High Speed	[Startup Feedrate - Maximum Speed of Each Axis]	Effective Immediately	High speed in manual mode	User Parameter
11010	Speed before returning to mechanical origin	[200,5000]	Effective Immediately	Default speed under manual mode before returning to mechanical origin	User Parameter
11011	Exclude the Z axis when operating working origin	[0,1]	Effective Immediately	Whether to exclude the Z axis when operating working origin	User Parameter
11006 To 11009	Manual Direction	1:Positive -1:Negative	Effective Immediately	Set the manual moving direction of axis	Manufacturer Parameter
12000	Idling Speed	[Startup Feedrate - Maximum Speed of Each Axis]	Effective Immediately	Speed when the tool is idling	User Parameter
12001	Default Feedrate	[Startup Feedrate - Maximum Speed of Each Axis]	Effective Immediately	System default feedrate	User Parameter
12002	Use default feedrate	1 Use the default feedrate 0 Use the specified speed in the file	Effective Immediately	If the default feedrate is used, the speed specified in the file is invalid	User Parameter
12003	Approaching Speed	[0, mini value of the max speed of each axis]	Effective Immediately	The working feedrate which the tool almost approaches during positioning	User Parameter
12004	Z-direction slowly deceleration distance	[0,6000]	Effective Immediately	The working distance which the tool almost approaches during positioning	User Parameter
12005	Interpolation algorithm selection	[0,2]	Effective Immediately	Interpolation algorithm selection, 0: ladder algorithm; 1:S algorithm, 2: mixed addition and subtraction	User Parameter

12006	Action after machining	0 (Keep still) 1 (Back to fixed point) 2 (Return to workpiece origin)	Effective Immediately	Tool action after machining	User Parameter
12036	Whether to close the cylinder at the end of machining	1: close cylinder and lift it to tool changing point 0: does not close cylinder at the end of machining	Effective Immediately	Whether to close the cylinder at the end of machining	User Parameter
12007	Safe Height	[5,500]	Effective Immediately	Relative to workpiece coordinate origin calculation, the system considers horizontal motion to be safe at this height. Used when performing origin return operation and breakpoint continuous operation	User Parameter
12037	Lifting height when idling	[1,1000]	Effective Immediately	The lifting height while idling relative to the workpiece coordinate origin point calculation.	User Parameter
12008 To 12010	Fixed point mechanical coordinate	[Worktable Lower Limit - Worktable Upper Limit]	Effective Immediately	The mechanical coordinates of the fixed point when the tool returns to a fixed point	User Parameter
12012	Z-axis Down Feedrate	[200,6000]	Effective Immediately	Feedrate at lowering tool in the Z direction	User Parameter
12013	Z-axis Up Feedrate	[200,6000]	Effective Immediately	Feedrate at lifting tool in the Z direction	User Parameter
12033	Idling Acceleration	[0,5000]	Effective Immediately	Acceleration when idling	User Parameter
12034	Idling Acceleration Jerk	[0,20000]	Effective Immediately	Acceleration Jerk when idling	User Parameter
12014	Startup Feedrate	[0, mini value of the max speed of each axis]	Effective Immediately	Minimum machnining speed during machining	Manufacturer Parameter

12015	Uniaxial acceleration	[0.01, 100000]	Effective Immediately		Manufacturer Parameter
12016	Turning Acceleration	[0. 10000]	Effective Immediately	Feeding motion occurs at the maximum acceleration on adjacent axes , the recommended value is 1~2 time uniaxial acceleration	Manufacturer Parameter
12017	Rate of Acceleration	[0.01,300000]	Effective Immediately	Speed change rate of feeding axis	Manufacturer Parameter
12018	Angular Tolerance	[0. 0.1]	Effective Immediately	In order to improve overall smoothness of the workpiece, the tool does not necessarily accurately run to the position specified by the program at the intersection of the two segments. When the actual position of the tool is different from the specified position, the system considers that this segment is processed normally. Not exceeding the error tolerance	Manufacturer Parameter
12019	Smoothing Time	[0, 0.06]	Effective Immediately	The larger the setting, the smoother the curve, but the local details will be weakened.	Manufacturer Parameter
12020 To 12023	Axis Max Feedrate	[0, 120000]	Effective Immediately	The maximum speed of each feeding axis	Manufacturer Parameter
12024	Arc Machining Min Speed	[Startup feedrate, reference circle speed]	Effective Immediately	The minimum speed of arc machining	Manufacturer Parameter
12025	Arc Machining Max Speed	[Startup feedrate, reference circle speed]	Effective Immediately	The maximum speed of arc machining	Manufacturer Parameter
12035	Whether arc speed limit is	0: Invalid 1: Valid	Effective Immediatel	Whether arc speed limit is valid	Manufacturer

	valid		y		Parameter
12027	Chordal deviation tolerance at arc machining	[0, 0.1]	Effective Immediately	Chordal deviation tolerance at arc machining	Manufacturer Parameter
12509	Arc Radius Tolerance	[0, 10]	Effective Immediately	Arc Radius Tolerance	Manufacturer Parameter
12029	Reference Circle Radius	[0, 10000]	Effective Immediately	Reference Circle Radius	Manufacturer Parameter
12030	Reference Circle Speed	[0, 6000]	Effective Immediately	Reference Circle Speed	Manufacturer Parameter
13000	Action parameter selection when paused	[0,1,2]	Effective Immediately	0: keep still 1: lift to safe height 2: lift to set lift height	User Parameter
13001	Z-axis lifting height when paused	[1,1000]	Effective Immediately	The height at which the tool moves up in the Z direction when the tool is paused	User Parameter
13002	Action parameter selection when stopped	[0,1,2]	Effective Immediately	0: keep still 1: lift to safe height 2: lift to set lift height	User Parameter
13003	Z-axis lifting height when stopped	[1,1000]	Effective Immediately	The height at which the tool moves up in the Z direction when the tool is stopped	User Parameter
14000	Floating tool block thickness	[0,500]	Effective Immediately	The thickness of floating tool setter	Manufacturer Parameter
14001	Function selection of tool setter	0：Floating tool setting 1：Fixed tool setting	Effective Immediately	Function selection of tool setter	Manufacturer Parameter
14002 to 14004	Fixed tool setter position	[Worktable stroke lower limit - Worktable stroke upper limit]	Effective Immediately	Position of the fixed tool setter in the machine coordinate system, X, Y, Z axis	Manufacturer Parameter
14006		[0.1]	Effective	Whether return to	Manufactur

	Whether return to workpiece origin after fixed tool setting completed		Immediately	workpiece origin after fixed tool setting completed 0: does not return to workpiece origin 1: return to workpiece origin	er Parameter
14008	Tool Setting Rate	[Startup feedrate -1000]	Effective Immediately	The speed which is approaching the speed of tool setter during tool setting process	Manufacturer Parameter
14009	Fast speed of fixed tool setting	[0, Z-axis down feedrate]		The speed at which the tool is moved from the highest point to the starting height of the tool when fixed tool setting	Manufacturer Parameter
15005	Whether G28 command is valid	[0,1]	Effective Immediately	Whether G28 command is valid, 0 is invalid, and 1 is valid.	User Parameter
15007	Whether NC check the syntax errors during machining	[0,1]	Effective Immediately	Whether NC check the syntax errors during machining, 0 for no, 1 for yes	User Parameter
15012	Whether NC check the syntax errors during simulation	[0,1]	Effective Immediately	Whether NC check the syntax errors during simulation, 0 for no, 1 for yes	User Parameter
15008	Whether to support the scanner	[0,1]	Effective Immediately	Whether to support the scanner, 0 for no, 1 for yes	User Parameter
15009	Whether always display the scanner	[0,1]	Effective Immediately	Whether always display the scanner, 0 for no, 1 for yes	User Parameter
16000	Whether lubrication pump activates automatically on a regular basis valid	[0,1]	Effective Immediately	Whether lubrication pump activates automatically on a regular basis 1:activate on regular basis 0: not activate on regular basis	Manufacturer Parameter
16001	Whether lubricate only	0 (No): Invalid	Effective Immediate	Lubricate only while machine working	Manufacturer

	while machine working valid	1 (Yes): Valid	y		Parameter
16002	Start lubrication pump time interval	[0, 3600000]	Effective Immediately	Every once in a while, the lubricant pump starts	Manufacturer Parameter
16003	Lubrication Pump Start Time	[0, 300]	Effective Immediately		Manufacturer Parameter
17001	Feeding Function Enabled	0: Not support feeding 1: Support feeding	Effective Immediately	Whether support feeding	User Parameter
17002	Pushing Function Enable	0: Not support pushing 1: Support pushing	Effective Immediately	Whether support pushing	User Parameter
17003	Feeding Startup Position	[Y Worktable stroke lower limit -, Y Worktable stroke upper limit -]	Effective Immediately	Feeding start point Y coordinate	User Parameter
17004	Pushing Startup Position	[Y Worktable stroke lower limit -, Y Worktable stroke upper limit -]	Effective Immediately	Laying-off start point Y coordinate	User Parameter
17005	Feeding End Position	[Y Worktable stroke lower limit -, Y Worktable stroke upper limit -]	Effective Immediately	Feeding end point Y coordinate	User Parameter
17006	Pushing End Position	[Y Worktable stroke lower limit -, Y Worktable stroke upper limit -]	Effective Immediately	Laying-off end point Y coordinate	User Parameter
17007	Feeding Speed	[0,50000]	Effective Immediately	Feeding Speed	User Parameter
17008	Pushing Speed	[0,50000]	Effective	Laying-off Speed	User

			Immediately		Parameter
17009	Feeding Delay M140	[0,300]	Effective Immediately	Feeding Cylinder Delay	User Parameter
17010	Adsorption Delay M141	[0,300]	Effective Immediately	Feeding Adsorption Delay	User Parameter
17011	Pushing Cylinder Delay M142	[0,300]	Effective Immediately	Laying-off Cylinder Delay	User Parameter
17012	Auxiliary Delay M144	[0,300]	Effective Immediately	Auxiliary Cylinder Delay	User Parameter
17013	Vacuum Pump Delay M143	[0,300]	Effective Immediately	Vacuum Delay	User Parameter
17014	Right Pushing Position	[Y Worktable stroke lower limit , Y Worktable stroke upper limit]	Effective Immediately	Right Pushing Position	User Parameter
17015	Right Pushing Speed	[0,Y axis max speed]	Effective Immediately	Right Pushing Speed	User Parameter
17016	Speed of current position to the starting point of feeding and laying-off	[0,Y axis max speed]	Effective Immediately	Speed of current position to the starting point of feeding and laying-off	User Parameter
17017	Feeding and laying-off auxiliary delay 1	[0,300]	Effective Immediately	Feeding and laying-off auxiliary delay 1	User Parameter
17018	Feeding and laying-off auxiliary delay 2	[0,300]	Effective Immediately	Feeding and laying-off auxiliary delay 2	User Parameter
17019	Feeding and laying-off auxiliary delay 3	[0,300]	Effective Immediately	Feeding and laying-off auxiliary delay 3	User Parameter
18021	Whether to use double station	0：No 1：Yes	Effective Immediately	Whether to use double station	User Parameter
18022	Whether to use	0：No	Effective		User

	clamping roller	1：Yes	Immediately	Whether to use clamping roller	Parameter
18023	Distance of Clamping Roller	[0,2000]	Effective Immediately	The distance between two clamping roller	User Parameter
18024	G54 Stroke Upper Limit	[Y Worktable stroke lower limit , Y Worktable stroke upper limit]	Effective Immediately	The clamping roller terminates the Y coordinate within the G54 stroke range	User Parameter
18025	G54 Stroke Lower Limit	[Y Worktable stroke lower limit , Y Worktable stroke upper limit]	Effective Immediately	The clamping roller starts the Y coordinate within the G54 stroke range	User Parameter
18026	G55 Stroke Upper Limit	[Y Worktable stroke lower limit , Y Worktable stroke upper limit]	Effective Immediately	The clamping roller terminates the Y coordinate within the G55 stroke range	User Parameter
18027	G55 Stroke Lower Limit	[Y Worktable stroke lower limit , Y Worktable stroke upper limit]	Effective Immediately	The clamping roller starts the Y coordinate within the G55 stroke range	User Parameter
19001	K1 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	

19002	K2 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying-off action; 41: perform fixed tool setting with one button.	
19003	K3 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	
19004	K4 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed	

				tool setting with one button.	
19005	K5 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	
19006	K6 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	
19007	K7 Ouput Control	[0,41]	Effective Immediately	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off	

				action; 41: perform fixed tool setting with one button.	
19008	K8 Ouput Control	[0,41]	Effective Immediatel y	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	
19009	K9 Ouput Control	[0,41]	Effective Immediatel y	Controlled output IO port. 0: Invalid; 1~32: corresponding OUT output port; 33: back to fixed point; 34~37: T1~T4 tool selection when library type 4; 38: execute M85 feeding action; 39: execute M86 laying-off action; 40: perform M81 automatic feeding and laying off action; 41: perform fixed tool setting with one button.	

11.6.2 Feeding Axis Parameter

Feeding parameters are divided into two categories: Regular (number 21xxx), Rotation (number 22xxx)

Number	Name	Value	Effective Time	Discription	Category
21000 to 21003	Pulse Equivalent	[0.0001,1]	Restart Effective	The drive sends a pulse corresponding to the distance moved by the	Manufact urer Parameter

				machine tool, X, Y, Z, A axis	
21004 To 21006	Worktable Stroke Range Checking Valid	0 (No): Invalid 1 (Yes): Valid	Effective Immediately	Whether to check the worktable stroke range before machining	Manufacturer Parameter
21007 To 21009	Worktable Stroke Lower Limit	[-9999, Worktable stroke upper limit]	Effective Immediately	Worktable mechanical coordinate lower limit, X, Y, Z axis	Manufacturer Parameter
21010 To 21012	Worktable Stroke Upper Limit	[0,9999]	Effective Immediately	Worktable mechanical coordinate upper limit, X, Y, Z axis	Manufacturer Parameter
21019 To 21021	Workpiece Coordinate Range Checking Valid	0 (No): Invalid 1 (Yes): Valid	Effective Immediately	Whether workpiece coordinate range valid	User Parameter
21022 To 21024	Workpiece Coordinate Lower Limit	[Worktable stroke lower limit - worktable stroke upper limit, Worktable stroke upper limit – worktable stroke lower limit]	Effective Immediately	Workpiece coordinate lower limit of worktable, X, Y, Z axis	User Parameter
21025 To 21027	Workpiece Coordinate Upper Limit	[0, Worktable Stroke Upper Limit - Worktable Stroke Lower Limit]	Effective Immediately	Workpiece coordinate upper limit of worktable, X, Y, Z axis	User Parameter
22000	Rotary Axis Programming Unit	[0，1]	Effective Immediately	0: Length. 1: Angle	User Parameter
22001 To 22003	Rotary Axis Control Radius	[0，9999]	Effective Immediately	The standard for calculating the length of the angle for XYZA axis	User Parameter
70000	Molecular	[0,1000]	Effective Immediately		User Parameter
70001	Denominator	[0,1000]	Effective Immediately		User Parameter

70002	Handwheel Acceleration	[0,6000]	Effective Immediately	The smaller the value, the smoother the speed	用户参数 User Parameter
70003	Handwheel Magnification X1 Position	[0,1000]	Effective Immediately	In the handwheel magnification X1 position, each turn to the handwheel 1 scale, the distance moved	User Parameter
70004	Handwheel Magnification X10 Position	[0,1000]	Effective Immediately	In the handwheel magnification X10 position, each turn to the handwheel 1 scale, the distance moved	User Parameter
70005	Handwheel Magnification X100 Position	[0,1000]	Effective Immediately	In the handwheel magnification X100 position, each turn to the handwheel 1 scale, the distance moved	User Parameter
70006	Strict Handwheel Pulse Count	0 (No): not strictly count; 1(Yes): strict count.	Effective Immediately	If strict handwheel counting is used, the system will move the distance specified by the handwheel; otherwise, the machine will only move when the handwheel is rocking	User Parameter

11.6.3 Spindle Parameter

Number	Name	Value	Effective Time	Discription	Category
30000	Whether use default spindle speed	0 (No): invalid 1 (Yes): valid	Effective Immediately	Use the system default spindle speed, the feedrate specified in the machining file will be invalid.	User Parameter
30001	Stop running when paused	0 (No): invalid 1 (Yes): valid	Effective Immediately	Whether the spindle stops running when stopped	User Parameter
30002	Stop running when stopped	0 (No): invalid 1 (Yes): valid	Effective Immediately	Whether the spindle stops running when paused	User Parameter

30003	Max Speed of Spindle	[0,100000]	Effective Immediately	Set the max speed of spindle	Manufacturer Parameter
30004	Default Speed	[0, maximum spindle speed]	Effective Immediately		Manufacturer Parameter
30005	Spindle Start Delay	[0.5,300]	Effective Immediately	The time required for the spindle to reach the speed set in the parameter from standstill	Manufacturer Parameter
30006	Spindle Stop Delay	[0.5,300]	Effective Immediately	The time required for the spindle from stop to speed down to zero	Manufacturer Parameter
30010	Whether spindle starts to wait for delay	0：wait 1：do not wait	Effective Immediately	Whether spindle starts to wait for delay	Manufacturer Parameter

11.6.4 Origin Parameter

Number	Name	Value	Effective Time	Discription	Category
40000	Return to mechanical origin before machining	0 (No): invalid 1 (Yes): valid	Effective Immediately	Set whether to return to mechanical origin before each machining	User Parameter
40050	Whether mandatory to return to the mechanical origin after power-on	0 (No): invalid 1 (Yes): valid	Effective Immediately	Whether mandatory to return to the mechanical origin after power-on	User Parameter
40051	Whether XYC axis return to mechanical origin together	0 (No): invalid 1 (Yes): valid	Effective Immediately	Whether XYC axis return to mechanical origin together	User Parameter
40002 To 40004	Origin Limit Effective Or Not	0 (No): invalid 1 (Yes): valid	Effective Immediately	Origin Limit Effective Or Not	Manufacturer Parameter
40006 To 40009	Rough Position Direction	-1: X, Y, C negative direction, Z positive direction; 1: X, Y, C positive	Effective Immediately	When the X, Y, Z and C axes return to the mechanical origin, the direction of rough positioning	Manufacturer Parameter

		direction, Z negative direction			
40010 To 40013	Rough Position Speed	[Startup feedrate - Maximum speed of each axis]	Effective Immediately	Movement speed of X, Y, Z and C axes during rough positioning	Manufacturer Parameter
40014 To 40017	Precise Position Direction	-1: X, Y, C negative direction, Z positive direction; 1: X, Y, C positive direction, Z negative direction	Effective Immediately	When the X, Y, Z and C axes return to the mechanical origin, the direction of precise positioning	Manufacturer Parameter
40018 To 40021	Precise Position Speed	[0.1, Rough position stage speed]	Effective Immediately	Movement speed of X, Y, Z and Caxes during precise positioning	Manufacturer Parameter
40022 To 40025	Back Distance	[-1000,1000]	Effective Immediately	Additional moving distance after the end of returning to mechanical origin precise positioning	Manufacturer Parameter
40026 To 40029	Allowed Direction At Origin Limit	0: Can move in both positive and negative directions 1: Can only move in the positive direction -1: Can only move in the negative direction	Effective Immediately	When the X, Y, Z and C axes are in the original position limit, the direction of each axis is allowed to move	Manufacturer Parameter
40030 To 40033	Whether the encoder origin valid	0：Invalid； 1：Valid.	Restart Effective	Whether the encoder origin valid	Manufacturer Parameter

11.6.5 Compensation Parameter

Number	Name	Value	Effective Time	Discription	Category
50000	Screw Rod Error Compensation Effective	0 (No): Invalid 1 (Yes): Valid	Effective Immediately	Screw Rod Error Compensation Effective	Manufacturer Parameter
50001	Reverse Backlash Compensation Effective	0 (No): Invalid 1 (Yes): Valid	Effective Immediately	Reverse Backlash Compensation Effective	Manufacturer Parameter
50002	Whether Tool Compensation Effective	0 (No): Invalid 1 (Yes): Valid	Effective Immediately	Whether Tool Compensation Effective	Manufacturer Parameter
50003 To 50005	Reverse Interval	[0，100mm]	Effective Immediately	When the gap between the screw rod is generated due to long-term using, the compensation XYZA is performed in order to achieve the set machining accuracy.	Manufacturer Parameter

11.6.6Tool Library Parameter

60016	Tool Library Type	[0,1,2,3,4,5]	Restart Effective	0. no tool library. 1. cylinder disk cutter. 2, fixed straight row. 3, the gantry frame is straight. 4, pneumatic tool changing with multi-drill. 5, fixed disk cutter.	Manufacturer Parameter
60017	Library Capacity	[1,20]	Effective Immediately	Library Capacity	Manufacturer Parameter
60001	Back to Working Piont Enable after Tool Changing	0: no return to the original position 1: return to the original position	Effective Immediately	Whether to return to the original position after tool changing	Manufacturer Parameter
60023	Tool Library Cylinder in Place Induction	0: invalid 1: valid	Effective Immediately	Check whether the tool library cylinder in-position induction is valid	Manufacturer Parameter

60024	Whether Pneumatic Tool Changing Parallel Transport	0: no parallel transport; 1: parallel transport	Effective Immediately	Whether XY moves according to the tool offset when pneumatically changing the tool	Manufacturer Parameter
60025	Tool changing Z axis position	[Z-axis worktable stroke lower limit, Z-axis worktable stroke upper limit]	Effective Immediately	Safe position of the Z-axis during tool changing and auxiliary action	Manufacturer Parameter
60026	Cylinder Lifting Delay	[0,999999]	Effective Immediately	Cylinder lifting delay when pneumatic tool changing	Manufacturer Parameter
60027	Drilling Cylinder Number	[0, Tool library capacity]	Effective Immediately	0: Do not use multi-drill; 1: Not available; Other values: cylinder number	Manufacturer Parameter
60028	Drilling tool capacity	[1,9]	Effective Immediately	Drilling tool capacity	Manufacturer Parameter
60029	Delay of tool cylinder in-position detection	[0,999999]	Effective Immediately	Delay time of tool cylinder in-position detection	Manufacturer Parameter
61007 To 61008	Tool 1XY offset	[-1000, worktable stroke upper limit]	Effective Immediately	Tool 1XY offset	Manufacturer Parameter
61016 To 61017	Tool 2XY offset	[-1000, worktable stroke upper limit]	Effective Immediately	Tool 2XY offset	Manufacturer Parameter
61025 To 61026	Tool 3XY offset	[-1000, worktable stroke upper limit]	Effective Immediately	Tool 3XY offset	Manufacturer Parameter
61034 To 61035	Tool 4XY offset	[-1000, worktable stroke upper limit]	Effective Immediately	Tool 4XY offset	Manufacturer Parameter
60003	The input speed of X-axis when changing tool	[50, X axis maximum speed]	Effective Immediately	The input speed of X-axis when changing tool	Manufacturer Parameter
60103	The output	[50, X axis	Effective	The output speed of X-	Manufacturer

	speed of X-axis when changing tool	maximum speed]	Immediately	axis when changing tool	Parameter
60100	Cutterhead Take-off Speed	[0, C axis maximum speed]	Effective Immediately	Cutterhead Take-off Speed	Manufacturer Parameter
60101	Cutterhead Acceleration	[0,6000]	Effective Immediately	Cutterhead Acceleration	Manufacturer Parameter
60004	Cutterhead tool changing speed	[0, C axis maximum speed]	Effective Immediately	Rotary speed of the cutterhead	Manufacturer Parameter
60005	Z-axis up and down speed when changing the tool	[0, Z axis maximum speed]	Effective Immediately	Z-axis up and down speed when changing the tool	Manufacturer Parameter
60006	Z-axis Grabing Speed	[0, Z axis maximum speed]	Effective Immediately	Z-axis Grabing Speed	Manufacturer Parameter
60007	Z-axis tool taken coordinate position when changing the tool.	[Tool changing start point Z axis coordinate, Z axis stroke upper limit]	Effective Immediately	Z-axis upper safety coordinate position when changing the tool.	Manufacturer Parameter
60008	Tool changing point Z coordinate	[Z axis stroke lower limit, tool change starting point Z coordinate]	Effective Immediately	Z-axis tool taken coordinate position when changing the tool.	Manufacturer Parameter
60009	Tool changing start point X coordinate	[0, X-axis stroke upper limit]	Effective Immediately	Tool changing startup coordinate of X axis	Manufacturer Parameter
60011	换刀点 X 坐标 Tool Changing Point X Coordinate:	[0, X-axis stroke upper limit]	Effective Immediately	Tool holder position coordinates of the X axis	Manufacturer Parameter
60050	Approaching the library speed when changing the tool	[50，3000]	Effective Immediately	Approaching the library speed when changing the tool	Manufacturer Parameter
60013	Whether the dust removal induction is	0：Invalid；1：Valid.	Effective Immediately	Whether the dust removal induction is valid	Manufacturer Parameter

	valid				
60102	Whether disk tool library in-position induction valid	0：Invalid； 1：Valid.	Effective Immediately	Whether disk tool library in-position induction valid	Manufacturer Parameter
60104	Delay time of detecting the disk tool library in-position induction timeout	[0,99999]	Effective Immediately	Delay time of detecting the disk tool library in-position induction timeout	Manufacturer Parameter
60003	Speed of XY axis by going to tool library while changing tool	[50,6000]	Effective Immediately	Speed of XY axis by going to tool library while changing tool	Manufacturer Parameter
60103	The output speed of XY axis when changing tool	[50，max speed of XY axis]	Effective Immediately	The output speed of XY axis when changing tool	Manufacturer Parameter
60005	Z-axis up and down speed	[0, max speed of Z axis]	Effective Immediately	Z-axis up and down speed	Manufacturer Parameter
60006	Z-axis Grabing Speed	[0, max speed of Z axis]	Effective Immediately	Z-axis Grabing Speed	Manufacturer Parameter
60007	Tool Changing Starting Point Z Coordinate	[Tool changing point Z coordinate, Z axis stroke upper limit]	Effective Immediately	Z-axis upper safety coordinate position when changing the tool.	Manufacturer Parameter
60010	Tool Changing Start Point Y Coordinate	[0，Y axis stroke upper limit]	Effective Immediately	Tool Changing Start Point Y Coordinate	Manufacturer Parameter
60050	Approaching the library speed when changing the tool	[50，3000]	Effective Immediately	Approaching the library speed when changing the tool	Manufacturer Parameter
60013	Whether the dust removal induction is effective	0：Invalid； 1：Valid.	Effective Immediately	Whether the dust removal induction is effective	Manufacturer Parameter
60105	Whether straight line tool library in-	0：Invalid； 1：Valid.	Effective Immediately	Whether straight line tool library in-position induction valid	Manufacturer Parameter

	position induction valid				
60106	Delay time of detecting straight line tool library in-position induction timeout	[0,99999]	Effective Immediately	Delay time of detecting straight line tool library in-position induction timeout	Manufacturer Parameter
61007 To 61010	Tool 1 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 1 Parameter	Manufacturer Parameter
61016 To 61019	Tool 2 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 2 Parameter	Manufacturer Parameter
61025 To 61028	Tool 3 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 3 Parameter	Manufacturer Parameter
61034 To 61037	Tool 4 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 4 Parameter	Manufacturer Parameter
61043 To 61046	Tool 5 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 5 Parameter	Manufacturer Parameter
61052 To 61055	Tool 6 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 6 Parameter	Manufacturer Parameter
61061 To 61064	Tool 7 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 7 Parameter	Manufacturer Parameter
61070 To 61073	Tool 8 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 8 Parameter	Manufacturer Parameter
61079 To 61082	Tool 9 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 9 Parameter	Manufacturer Parameter
61088 到 61091	Tool 10 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 10 Parameter	Manufacturer Parameter
61097 To 61100	Tool 11 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 11 Parameter	Manufacturer Parameter
61106 To 61109	Tool 12 Parameter	[0, worktable stroke upper	Effective Immediately	Tool 12 Parameter	Manufacturer Parameter

		limit]			
61115 To 61118	Tool 13 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 13 Parameter	Manufacturer Parameter
61124 To 61127	Tool 14 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 14 Parameter	Manufacturer Parameter
61133 To 61136	Tool 15 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 15 Parameter	Manufacturer Parameter
61142 To 61145	Tool 16 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 16 Parameter	Manufacturer Parameter
61151 To 61154	Tool 17 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 17 Parameter	Manufacturer Parameter
61160 To 61163	Tool 18 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 18 Parameter	Manufacturer Parameter
61169 To 61172	Tool 19 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 19 Parameter	Manufacturer Parameter
61178 To 61181	Tool 20 Parameter	[0, worktable stroke upper limit]	Effective Immediately	Tool 20 Parameter	Manufacturer Parameter

Chapter XII System Management

The [System] menu contains system related items: Software Registration, Language Selection, Software Upgrade, Auxiliary Function, Network Management, and Auxiliary File.

Figure 12-1 System Interface Submenu

12.1 Software Registration

Select the [F1 Software Registration] menu item in the [System] menu, which will show the current system software version and other related information, as well as the system encryption information.

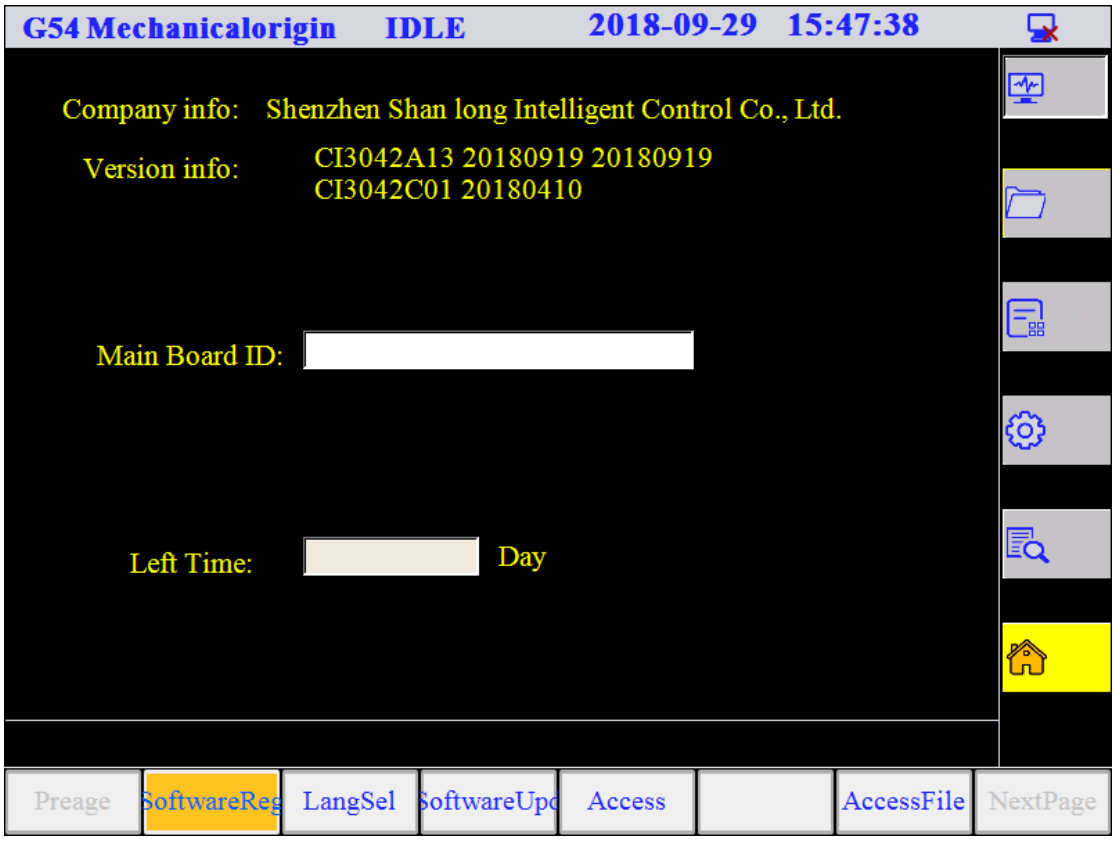


Figure 12-2 Software Registration & Version Information

12.2 Software Upgrade

When the software is updated or has a new version, select the [F3 Software Upgrade]

menu item in the [System] menu, and the following window will pop up. This function is used for system software upgrade. When you want to upgrade the software of the system, save the upgrade program to the USB flash drive, plug the USB flash drive into the system, and click the menu bar to upgrade the system.

Select the file to be upgraded with the suffix bin and press the "OK" button.

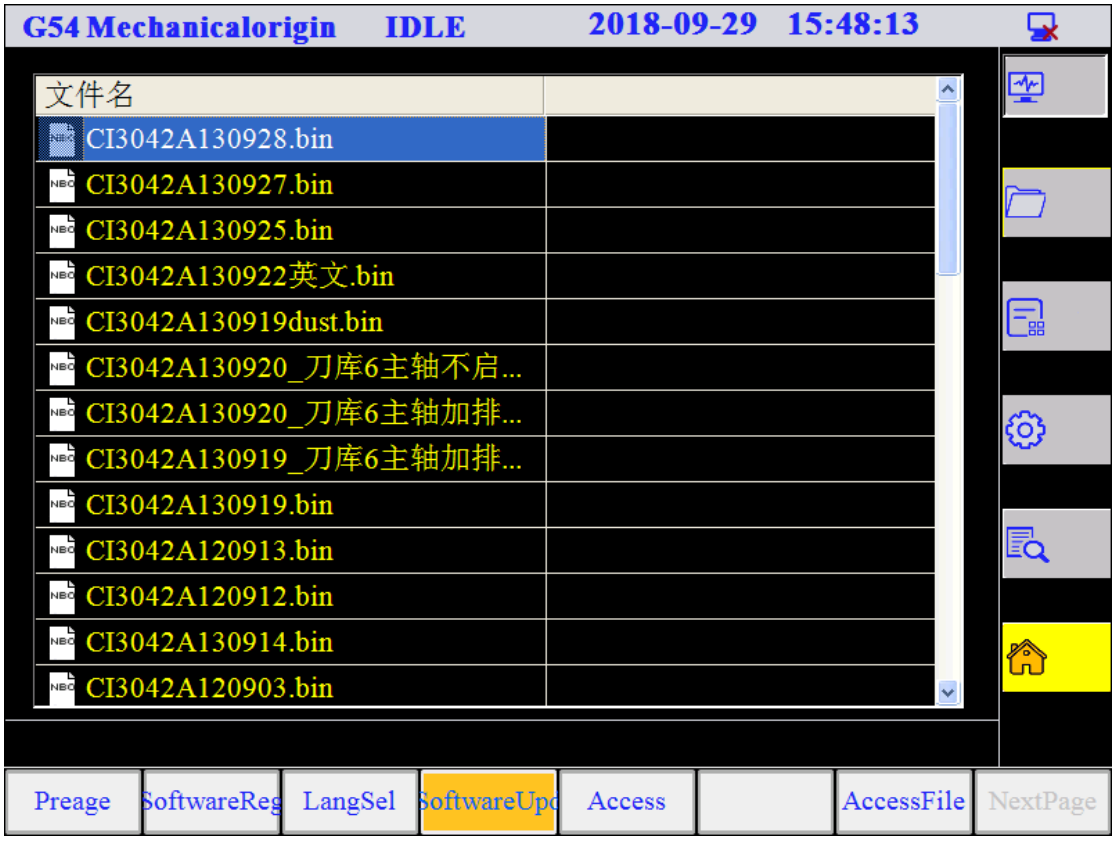


Figure 12-3 Software Upgrade

Note that you cannot turn off the power during the program upgrade!

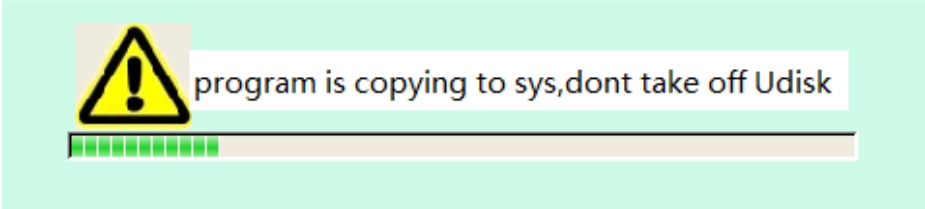
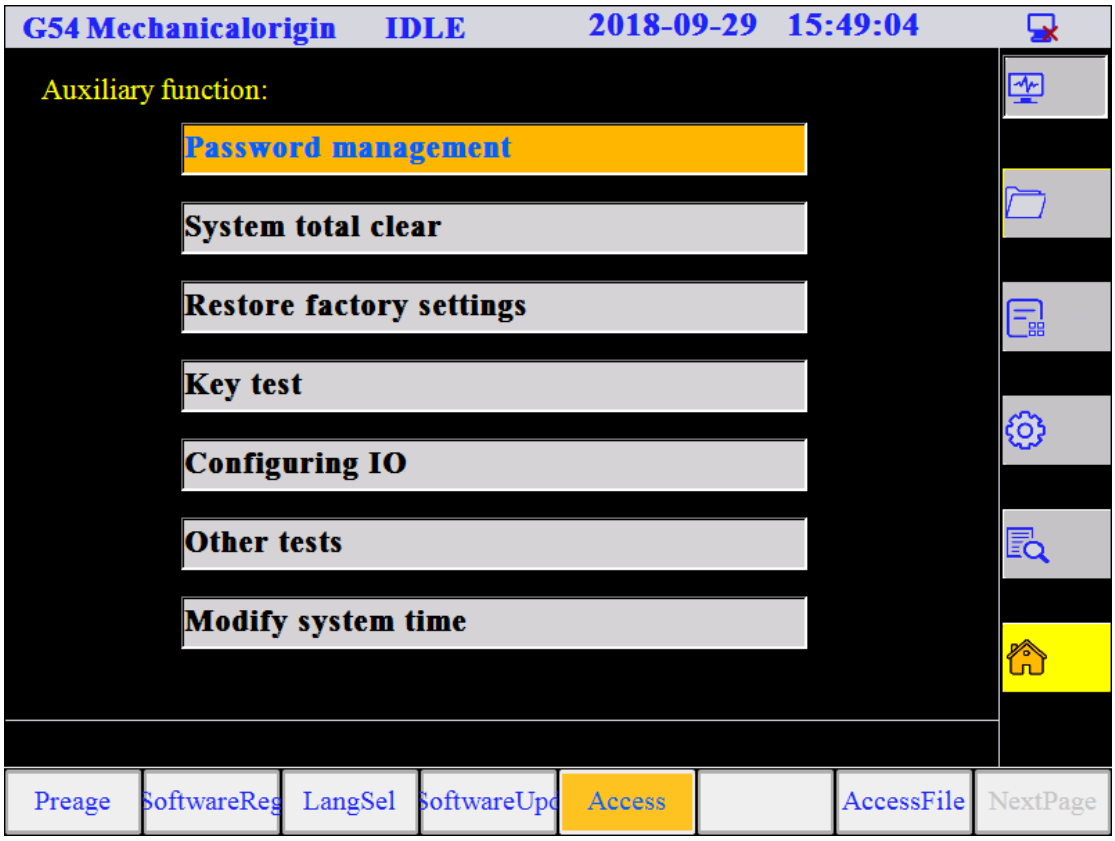


Figure 12-4 Software Upgrade Progress

12.3 Language Selection

The system only supports Chinese and English display for the time being.

12.4 Auxiliary Function



12.4.1 Change Password

Select the [System] → [Auxiliary Function] → [Password Management] menu item, and the following window pops up. This function is used to modify the user password and effectively protect the user's personal information, thus effectively protecting the security of the parameter setting.

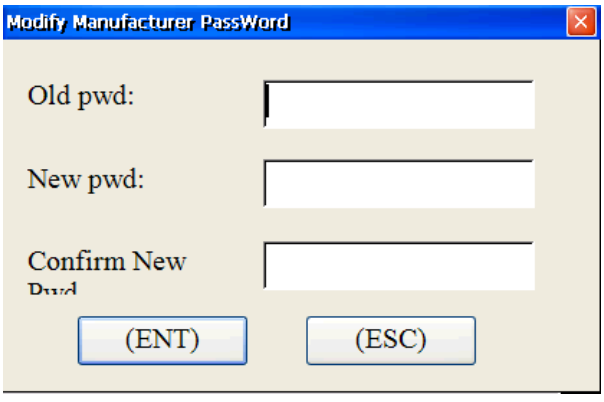


Figure 12-5 Change Password

There are many parameters involved in this system, and the parameters are divided into user parameters and manufacturer parameters. To modify and view a certain type of

parameter, you must have permission to view and modify this type of parameter.

12.4.2 System Total Clear

Function reservation.

12.4.3 Factory Reset

Select [Factory Reset], the password dialog box will pop up, enter a valid password to determine the recovery system parameters and restart the system. The system will be restored to the factory settings.

12.4.4 Button Test

The button test is used to test all the buttons on the panel and check if the button input is correct to facilitate the problem.

12.4.5 Modify System Time

If the time changes, click Modify System Time to change the software time.

12.4.6 Configuring IO and Other Test

Function reservation.

12.5 Network Management

The [Network Management] menu contains system related items: Network Configuring, Network Connection, Network Disconnecting, Automatically Obtaining, and Manually Configuring;

Configure machine IP address information and local IP address information to communicate between the two.

12.6 Auxiliary File

Insert the USB flash drive containing the feeding and laying-off file into the USB port, press the Auxiliary File menu, select the file to be imported, and press the Enter key to import.

Chapter XIII Multi-tool Machining

There are three types of tools: Disc Tool Library, Straight-row Tool Library, Pneumatic Tool Changing. They can be selected by tool library parameters

13.1 Tool Library Setting

1. There are two types of tool libraries, straight-row tool library and disk tool library;

The straight tool library is divided into fixed straight row and gantry frame straight row. The main difference between the two is that the fixed straight-row tool library is usually installed at the tail of the machine tool, and the tool library does not move when the tool is changed. The gantry frame straight row is mounted directly on the gantry and moves with the gantry during processing. The tool changing is realized by the cylinder movement of the tool library.

The disc library is also divided into a fixed disc cutter and a follow-up disc cutter, wherein 1 is a follow-up disc cutter, 5 is a fixed disc cutter. The tool library of follow-up disc cutter is mounted on the Z-axis, and the cylinder is pushed out and retracted. The library of fixed disc cutter is mounted in the positive direction of the X axis.

2. Select the tool library type as 1, 2, 3, 5, and enter the selection [Tool Selection] to test the movement of the tool library.

3.Spindle No.: Set the tool number of the spindle. After the manual tool changing or other operations, the system cannot record the current tool number and needs to be set manually. After pressing F1, the following dialog box will pop up, enter the tool number and press OK to complete the setting.

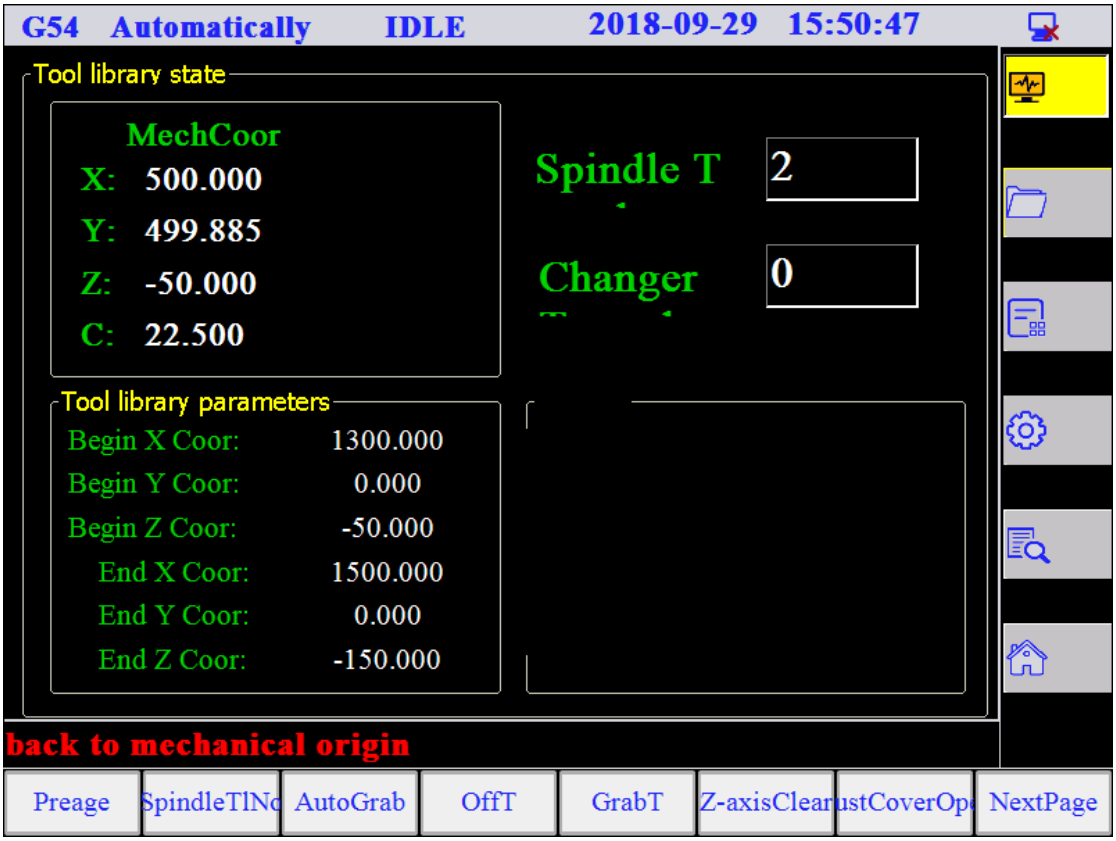


Figure 13-2 Spindle Tool Number Dialog Box

Tool Library Input: The test of the tool cylinder is only effective when is in gantry frame straight row and cylinder disc cutter. The library is switched between input or output.

Place Tool: Perform tool placing action. Press F3 and the following confirmation dialog pops up. Press OK to start the tool changing.

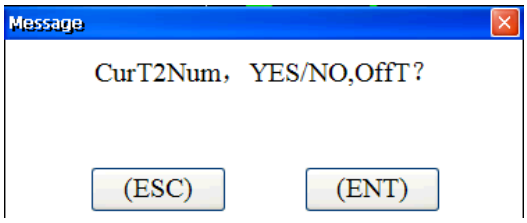


Figure 13-3 Tool Placing Confirmation Dialog Box

Grab Tool: Perform tool grabbing action. After pressing F4, the dialog box for selecting the tool number will pop up as follows:

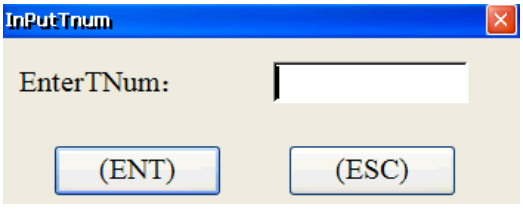


Figure 13-4 Tool Grabbing Number Dialog Box

After entering the tool changing number, press OK to pop up the Confirm Tool Changing dialog box:

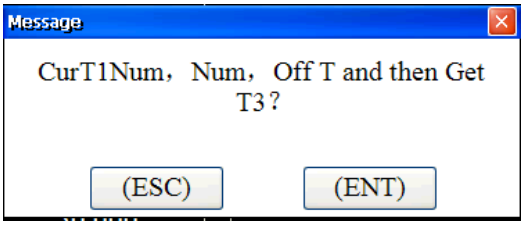


Figure 13-5 Tool Grabbing Number Dialog Box

The grabbing action is performed after pressing the Enter key.

Dust Cover Off: Dust cover cylinder test, switched between dust cover off and on.

Vacuum On: vacuum switch test, switched between vacuum on and off.

Save Tool Location: Set the current position to the library location of the current tool. After pressing F2, a confirmation dialog pops up:

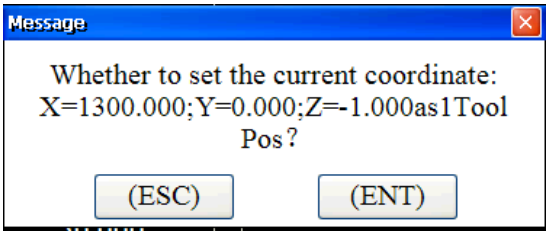


Figure 13-6 Tool Library Location Dialog Box

After pressing the OK key, the coordinates of the library location where the current tool is located are saved.

4. Set Library IO: Due to the limited resources of the IO port, the IO port needs to be reused. Generally, the ports that are commonly used should not be configured as much as possible. The special function IO like the tool library needs to be configured. Select "Information" - "IO Status" - move to several IOs related to the library (tool library cylinder, spindle loose clamping, dust collection cover, spindle clamping in place, spindle loosening tool in place, manually loosening clamping, library feeding tool in place, tool library receiving tool in place), press F2, the following dialog box pops up:

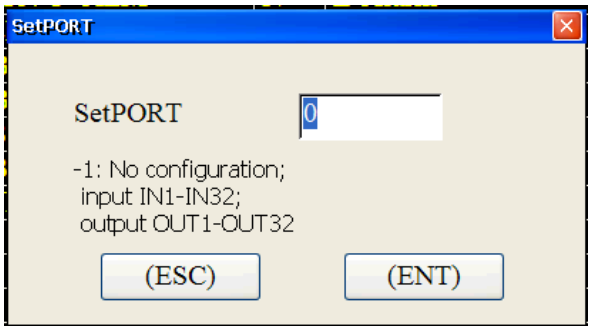
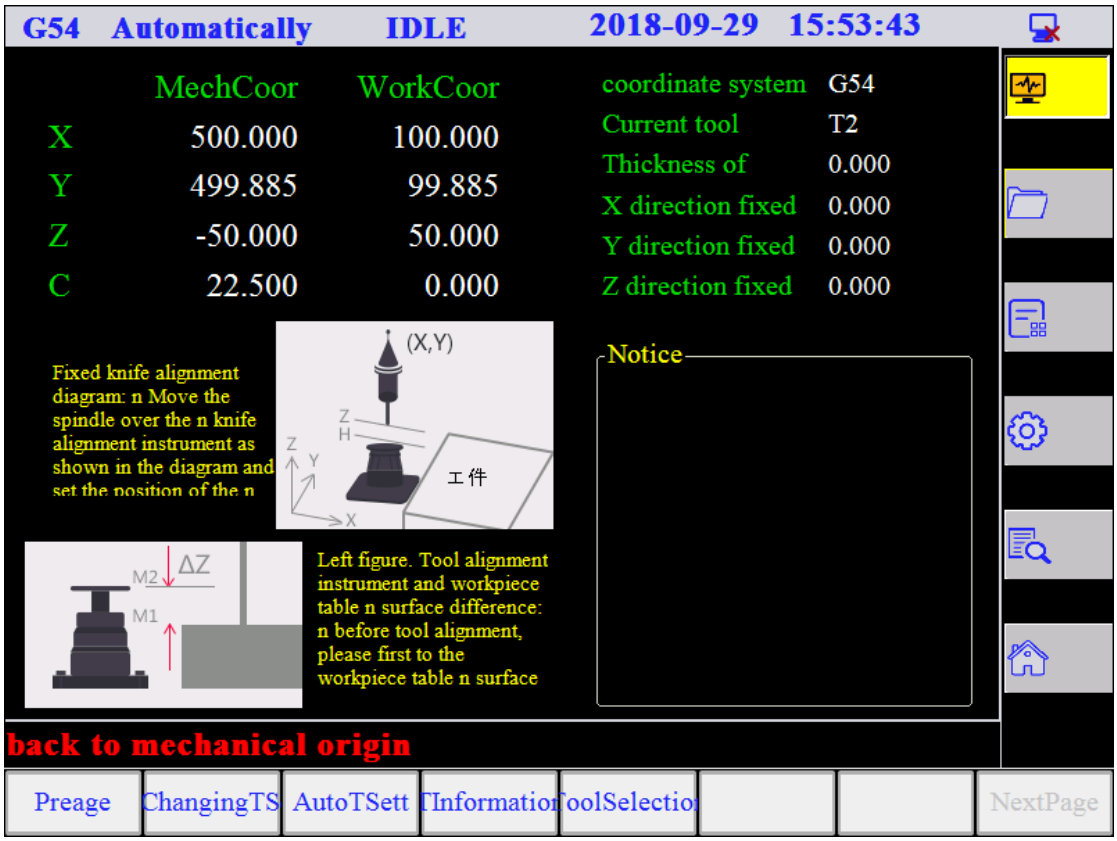


Figure 13-7 Configuring Port Dialog Box

Set the corresponding wiring port.

1. Offset Setting: perform tool setting first, press F3 for tool setting on the main interface of the machining,



Floating Tool Setting: this function can only be used if the floating tool setting is enabled. Press the next page and then press F1 to start the tool setting. The tool must be aligned on G54.

Tool Changing and Setting: perform fixed tool setting after tool changing. This function is valid only when the fixed tool setting is enabled. This function is used when the current tool is replaced after the first tool setting. It is also possible to select the current tool for tool setting in

automatic tool setting. The tool setting must be performed after the tool changing under G54, and the workpiece origin must be cleared first.

Automatic Tool Setting: One-time multi-tool setting, press F2 to pop up the following dialog box. The tool setting must be performed after the tool changing under G54, and the workpiece origin must be cleared first.

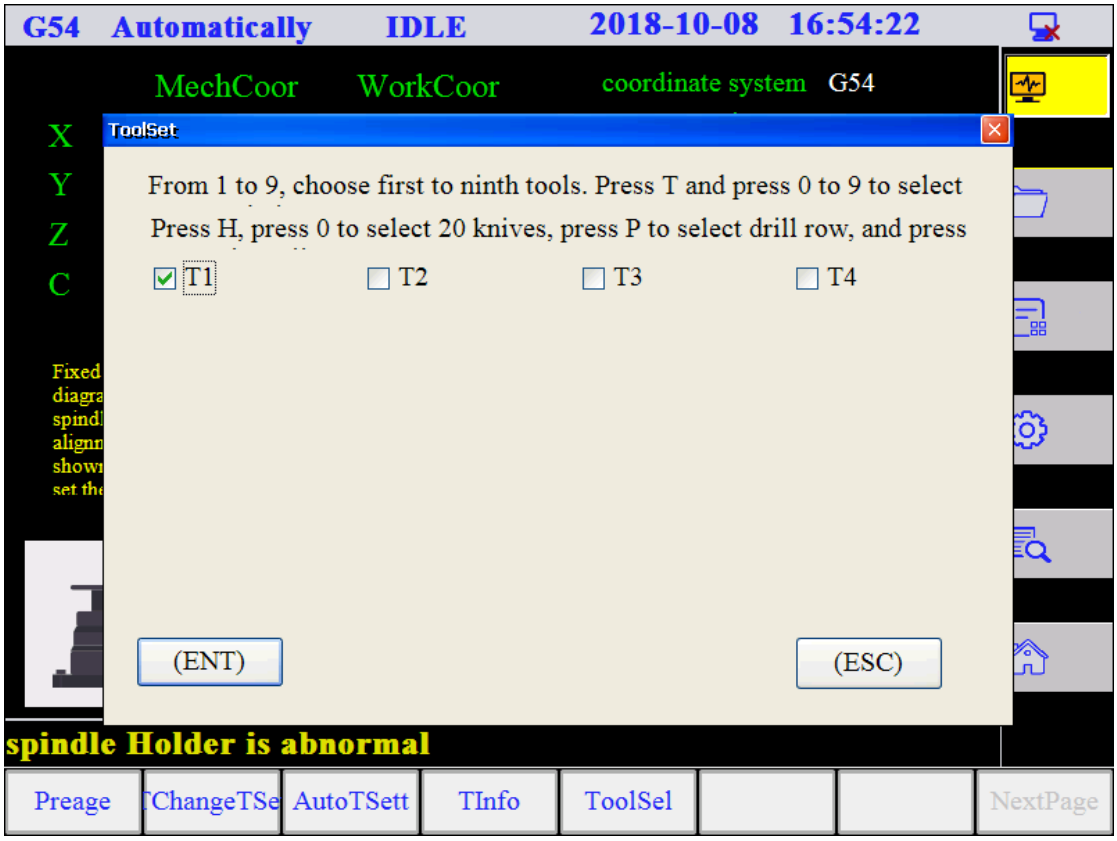


Figure 13-8 Automatic Tool Setting/Selecting Dialog Box

Select the tool you want, and press OK to perform tool setting for the corresponding tool.

Z-axis Clear: Manually perform tool setting for current tool, after selecting, the following interface pops up:

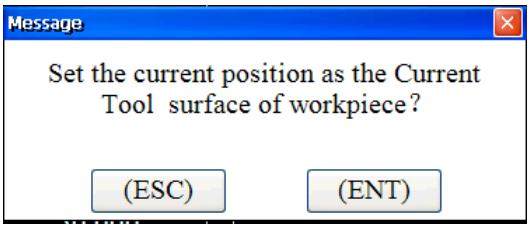


Figure 13-9 Automatic Tool Setting/Selecting Dialog Box

Confirm to set the offset of the current tool.

All tools offset setting: “Offset” – “Set Offset” – “Z Clear” will pop up the following interface:

Figure 13-10 Z Clear Confirmation Dialog

All tool offsets will change after confirmation.

The offset of XY can be set separately or set together. Press XY to clear, the following interface pops up:

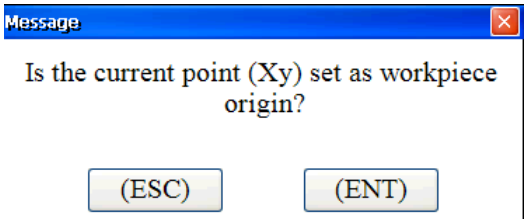


Figure 13-11 XY Clear Confirmation Dialog Box

Set the XY offset after confirmation. X offset, Y offset independent setting method is the same.

Z-axis offset fine-tuning method: Z-axis offset can be manually fine-tuned. All tool offsets can be fine-tuned. Press the Z-axis fine-tuning to pop up the following dialog

box. Press the left or right button to select the tool. Press the corresponding number key to deepen or raise the corresponding offset. After setting, press OK to save the offset setting.

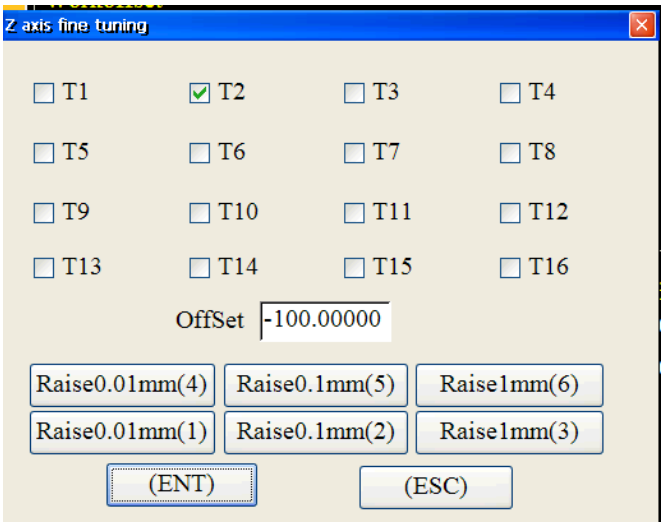


Figure 13-12 XY Clear Confirmation Dialog Box

The MDI test can also be used for the tool changing test. After selecting MDI, enter the corresponding tool number, press Execute 1, Execute 2, Execute 3, and Execute 4 to test the tool changing.

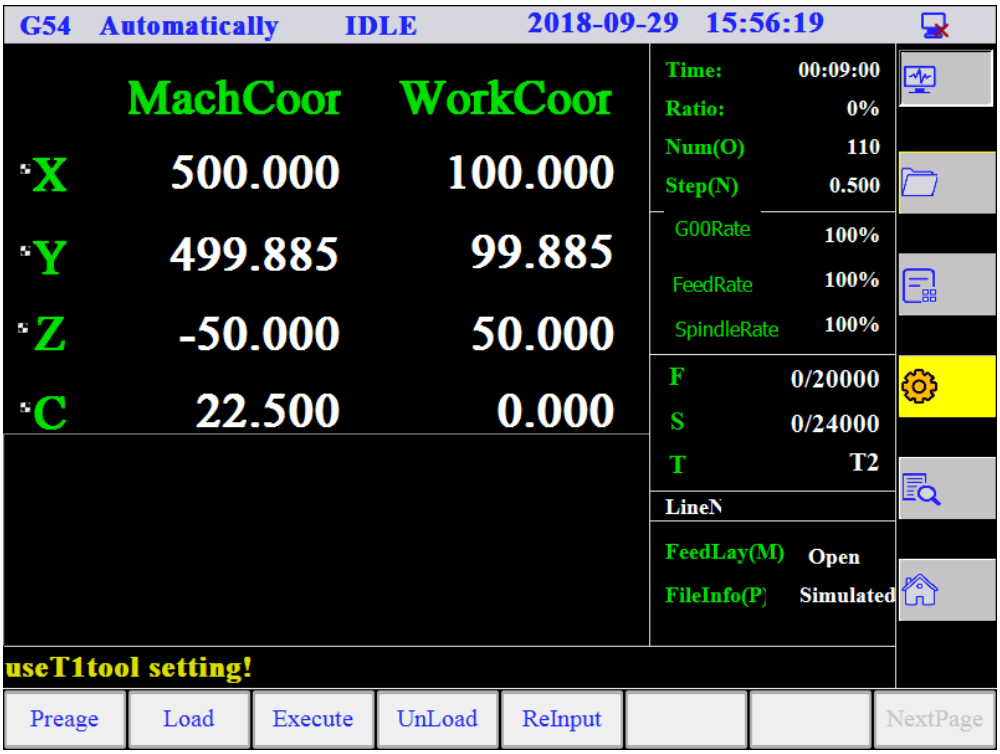


Figure 13-13 MDI Operation

At this point, the operation of tool library changing is completed.

13.2 Pneumatic Tool Changing

1. Pneumatic tool changing compared to tool library changing, the tool library capacity will be much less, generally within four. First see the previous section, set the library type to "4 Pneumatic Tool Changing";
2. The capacity of pneumatic tool changing is 1~4, and the multi-drill is on one of the cylinders. When the Drilling Cylinder No. is set to 0, it means that the multi-drill is not used (see the next section for multi-drill).
3. The Pneumatic Tool Changing IO configuration is shown in Figure 13-11. The multi-process cylinders 1~4 correspond to the 1~4 tool cylinders.

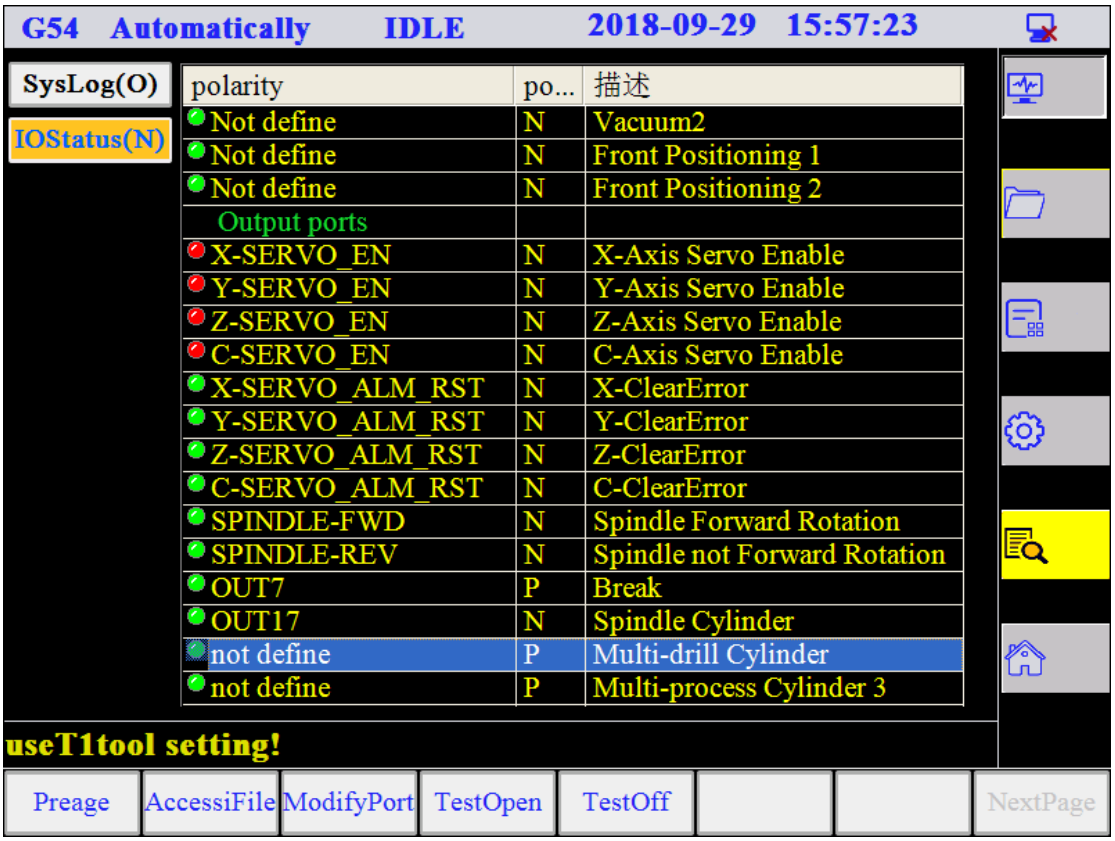


Figure 13-14 Pneumatic Tool Changing Output IO Configuration

4. Pneumatic Tool Changing Cylinder Configuration: If the upper and lower position detection is installed, the corresponding in-position signal IN port of the pneumatic tool 1~4 is configured in the interface; if not installed, the corresponding tool cylinder up and down position detection switch should be set to unconfigured. As shown below,

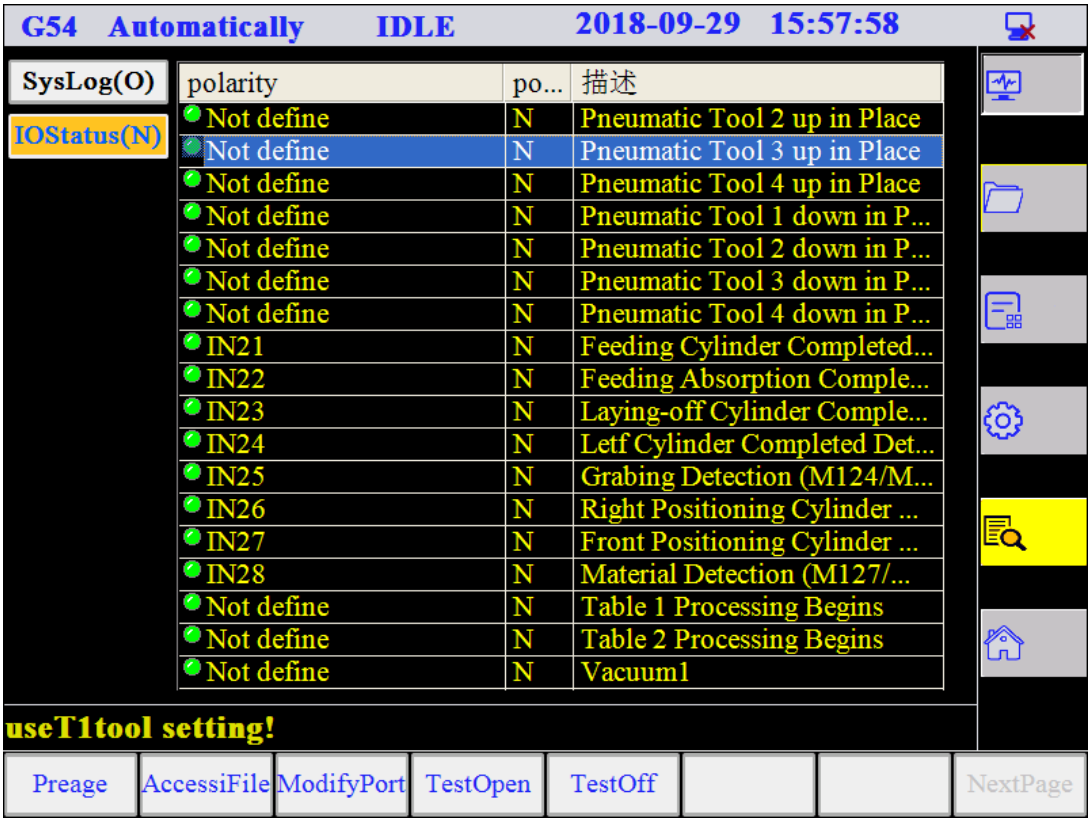


Figure 13-15 Switch Configuration of Tool Up and Down Position Induction

5. Pneumatic Tool Changing Offset Setting: After selecting the tool, you need to set the tool offset first;

1) Pneumatic Tool Changing XY offset should be set manually. In the spindle parameters, as shown below

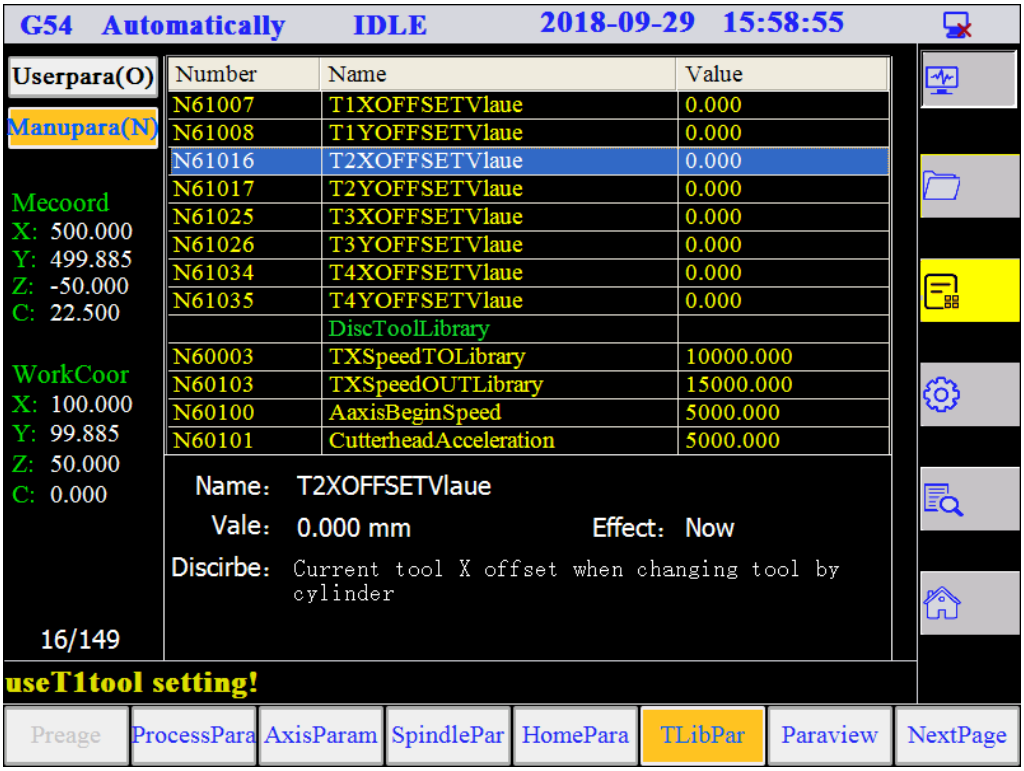


Figure 13-16 Cylinder Tool Changing XY Offset Setting

2) Z-axis offset setting method for cylinder tool changing: the tool zero point in tool selection can directly set as Z-axis offset of the current tool, as shown below:

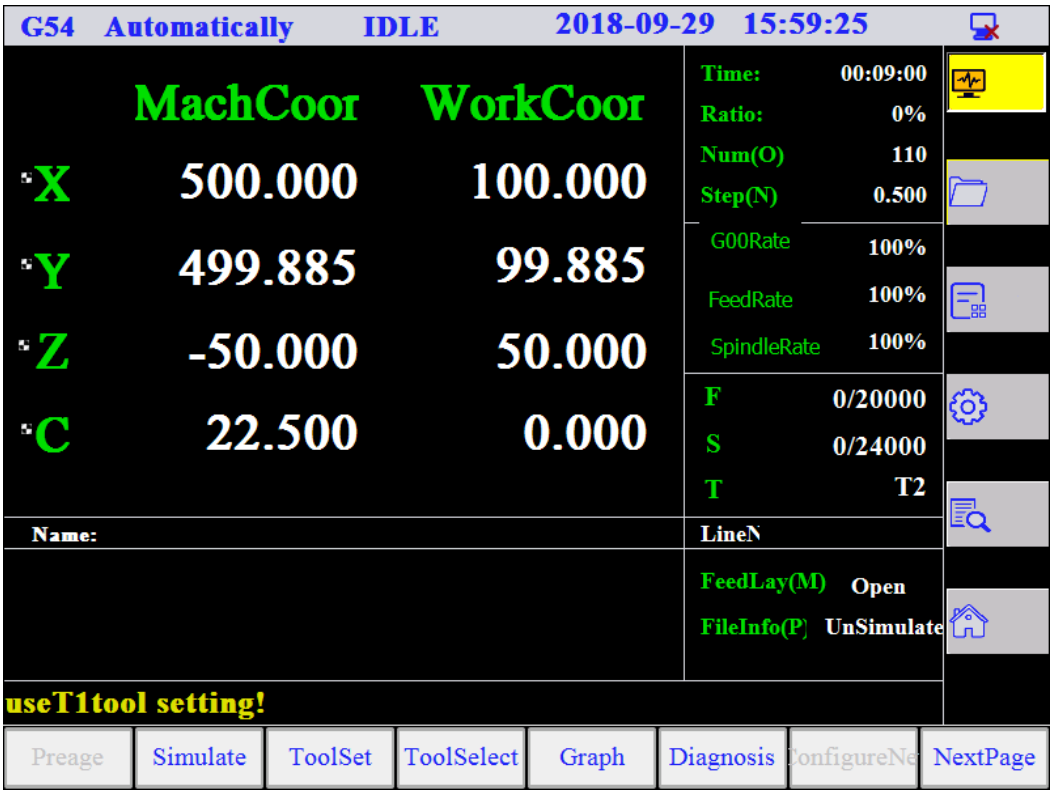


Figure 13-17 Pneumatic Tool Changing Z Setting

6. After selecting pneumatic tool changing, the tool selection on the main interface changes from gray to blue, indicating that it can be used. After pressing the tool selection, the tool selection box for the pneumatic tool changing appears. Before doing the tool changing operation, set the relevant parameters first. Set whether the pneumatic tool changing is parallel transport parameter, and determine whether the XY is corresponding to the offset during the tool changing. Set the offset of each tool.
7. As with straight-line tool changing, cylinder tool changing can also be switched under MDI (see previous section).

13.3Multi-drill

1. Multi-drill is a special tool for pneumatic tool changing. Multi-drill can punch quickly, which is common on the hole machine. The tool changing of multi-drill is quite fast, and the multi-drill and the pneumatic tool are used together to realize the first punching and then cutting to complete entire process. Thereby greatly save the cost. Multi-drill pneumatic port control, see the following figure

OUT19	N	Multi-drill1
OUT20	N	Multi-drill2
OUT21	N	Multi-drill3
OUT22	N	Multi-drill4
OUT23	N	Multi-drill5
OUT24	N	Multi-drill6
OUT25	N	Multi-drill7
OUT26	N	Multi-drill8
OUT27	N	Multi-drill9

Manually test the drilling cylinder first, then press [Multi-drill Selection] to select the drilling number. Then press the digital keyboard 1 to 9 to manually test the drilling tools from 1 to 9, as shown below:

Figure 13-18 Cylinder Tool Changing XY Offset Setting

Multi-drill Offset Setting: manually set the XYZ offset of multi-drill, manually input multi-drill offset, press OK to save.

Figure 13-19 Multi-drill Offset Setting

2. Since the Z-axis offset using frequency of multi-drill is relatively high, for the convenience of setting, after the drill is performed, the current coordinate can be set to multi-drill Z-axis zero point at the tool zero point. As shown below:

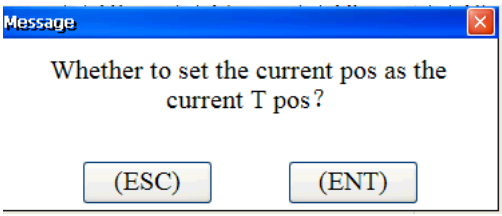


Figure 13-20 Multi-drill Tool Zero Point

Chapter XIV Automatic Feeding and Laying-off

At present, the workmanship of automatic feeding and laying-off machine is quite different. The system can realize automatic feeding, automatic laying-off function or automatic feeding and laying-off customization function through programmable method. Different configuration modes can adapt to different equipments.

14.1 Manual Operation

1. Press M on the machining interface to pop up the interface for manual operation of feeding and laying-off as follows:

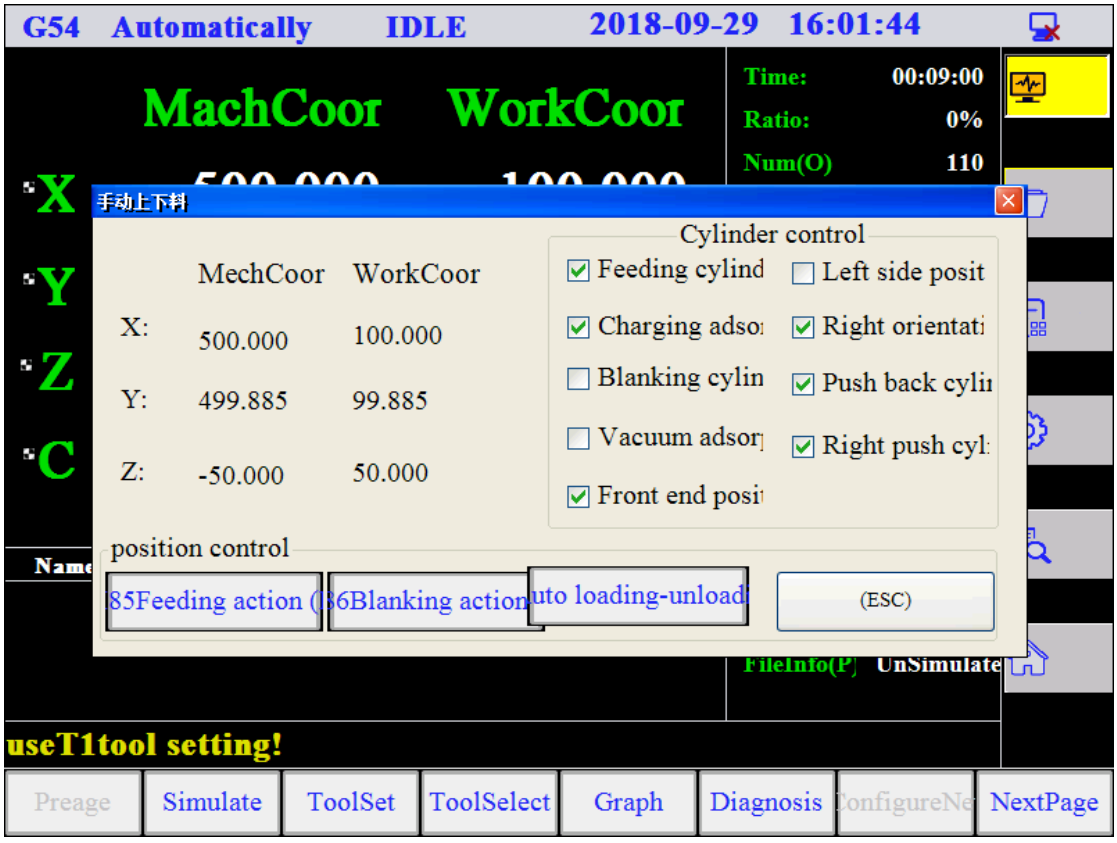


Figure 14-1 Manually Feeding and Laying-off

2. Press number kyes 1 to 9 to test material cylinder. Press the corresponding letter to test the position of the feeding and laying-off. If the position is wrong, you can set the corresponding position in the operating parameters as shown below. Press P to test the entire feeding and laying-off action.

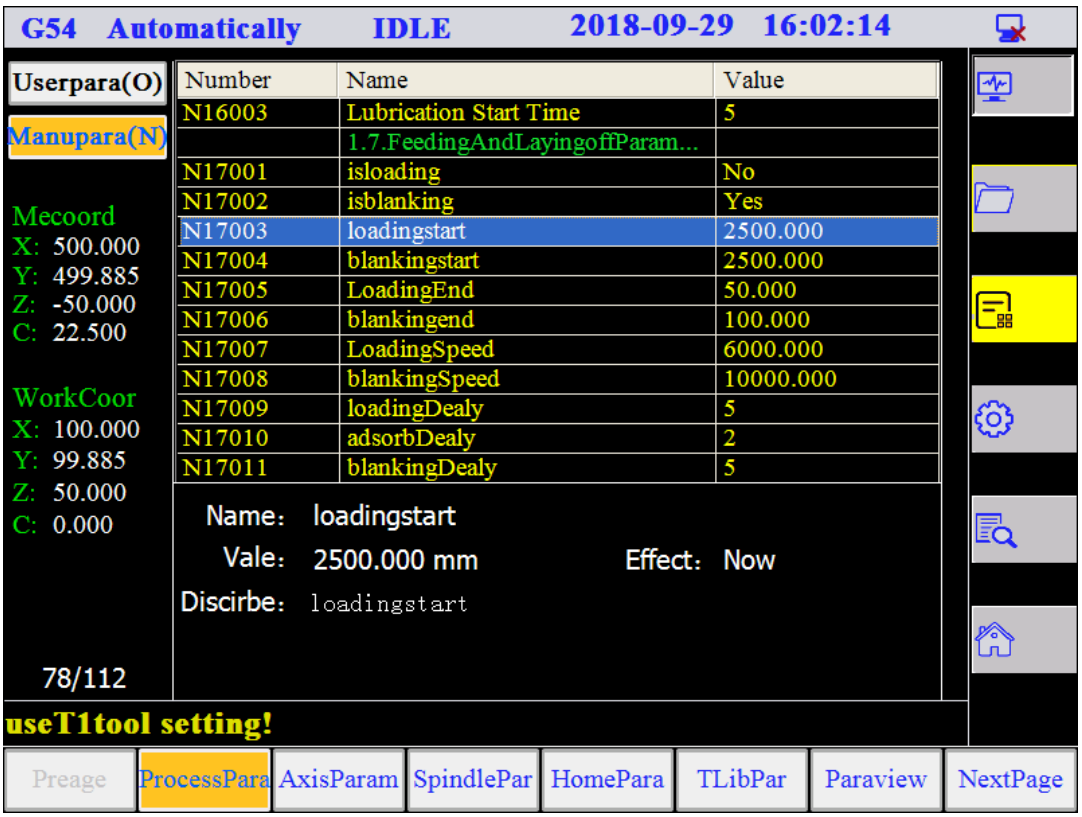


Figure 14-2 Feeding and Laying-off Parameter Setting

14.2 Automatic Machining

To achieve feeding and laying-off during automatic machining, you need to add M81 at the end of the file.

;////////////////

G17

G01X20

G01X40Y10

G01X60

M81

Chapter XV Code Scanner Load Specified File

The system supports to connect code scanner to scan the barcode and load the file with the same file name and barcode number. The code scanner only supports the USB connection. The specific usage is related to the setting of code scanner parameters. The specific operation is as follows.

15.1 Set Code Scanner Parameter

Enter Parameter Management. The parameter number 15008 in the user parameter interface is used to set code scanner. The default value of this parameter “0”, means the code scanner is not supported. When using code scanner, the parameter is should be changed to “1”, that is supporting code scanner.

15.2 Scan Code Loading

After changing the value of the parameter "Whether to support code scanner" to "1", when entering the file interface, a window asking for scanning code will pop up. The user only needs to connect the code scanner to scan the barcode. It will load the file which has the corresponding number of bar code.

Chapter XVI Network Connection

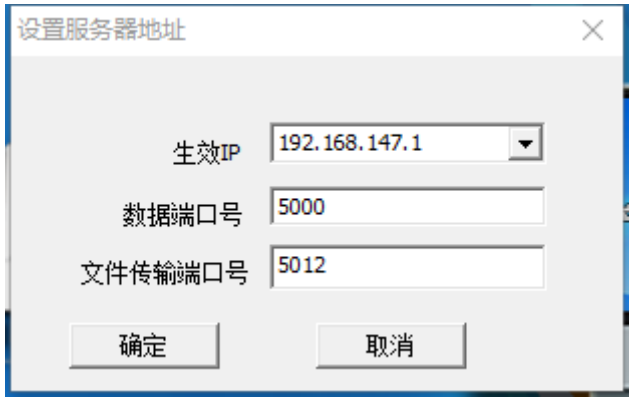
16.1 Pre-work

Use the straight-through cable or peer-to-peer cable to access the LAN port of the system and the LAN port of the computer (if the router is connected, the computer and the system are connected to the router). After the power is turned on, the network port will light yellow, and the network port and network cable are normal.

16.2 Computer Setting

Install "Shanlong Technology CNC Production Management System" plugin. (Contact any office for access)

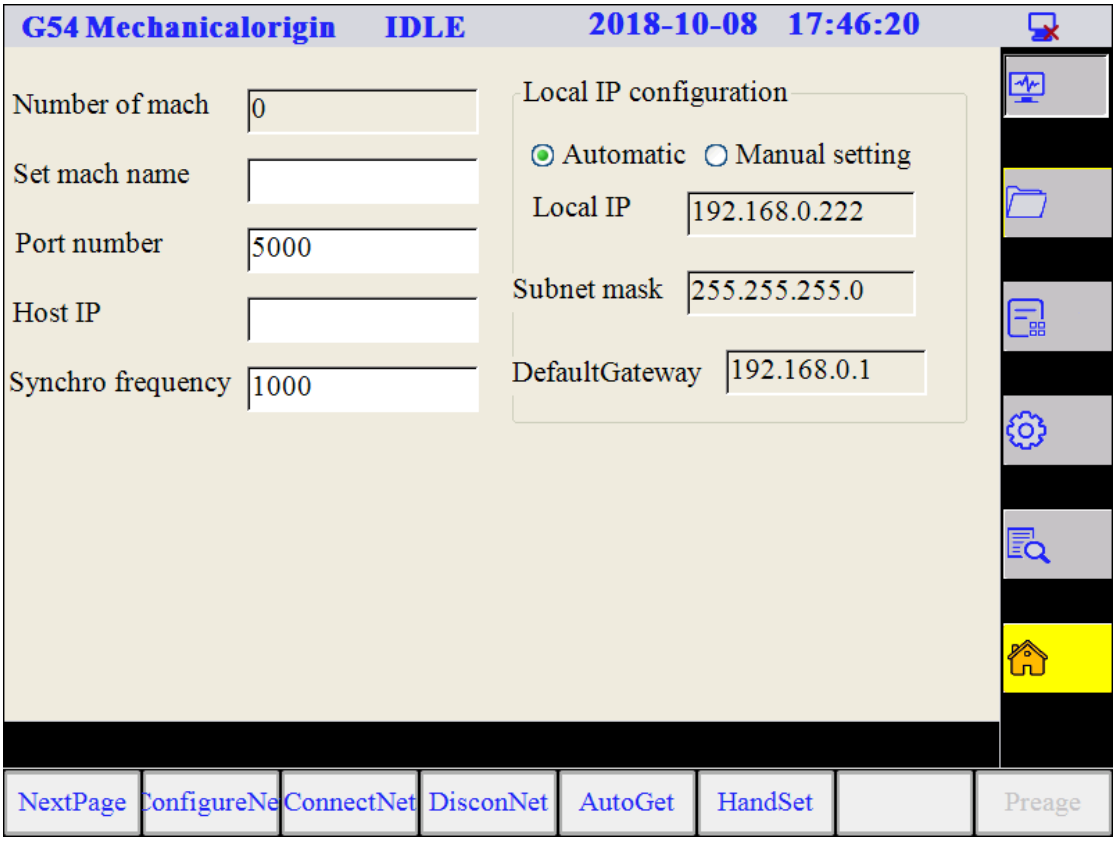
Enter the management system, select Machine Management > Network Connection, pop up the Set Server Address window.



Make a note of the valid IP and click OK.

16.3 System Setting

Click the "System" button on the right side of the system > F5 Network Management, the following interface pops up:



Set in order:

1: Machine Name: Connected when sharing a folder

2: Output Transmission Port: consistent with the data port of the management system

3: Host IP: IP address of the PC to be connected (that is, the valid IP recorded by the computer just now)

4: Synchronization Frequency: default 1000

5: Local IP Address Configuration: It is the IP address of the system. It must be in the same network segment as the PC (that is, the first three addresses must be the same, and the last address must be different). If you use the router, click F4 to obtain it automatically. After the network is configured, you must power on and restart.

After rebooting, return to the network management interface and click to connect to the network. If it indicates that the network is connected and successful. The red cross of the network logo in the upper right corner disappears.

16.4 File Transfer

1. On the computer side, "Shanlong Technology CNC Production Management System" selects Machine Management > Newly Supervised Machine Tool, which can supervise the machine status.

2. In File Management > Send File, you can select the file to be transferred and transfer it to the corresponding machine system.