User's Manual of CNCRouter Engraving CCD Integrated System

Chengdu multidimensional robotization technology company

The manual is a printed product, so there is a certain lag, resulting in some new content can not be timely and new. Please understand the customer, if you need to know the latest content, please go to our official website to download.

Foreword

How to use

If you are using this type of system for the first time, please read this manual carefully; if you have relevant experience, please quickly find the information you need through the catalogue.

This manual is divided into 7 parts:

1. Preface, an overview of the contents of this manual. And explain to the customer the use of the system and other related information. Please read carefully.
2. In the introduction section, a brief introduction to the system, including the system's use environment, application industry, system features, etc.
3. Installation and connection, this section mainly introduces the system installation and hardware wiring. Including software installation and machine wiring, as well as system upgrades.
4. Interface introduction, this part introduces the main interface of the software briefly, introduces the functions of each module, and is convenient for users to search for functions.
5. Basic operation flow, this part introduces the simplest use of the system to help users get started quickly and process.
6. Detailed software description, this part describes the various parts of the software in detail, introduces the interface structure of each function, explains each action that can be completed and explains the corresponding operation of the software. Users can view all the operations supported by this system in this section.
7. Parameters and equipment debugging, this section mainly explains the meaning of various parameters, and describes how to set the correct parameters according to the mechanical structure.

Precautions

1. Storage and transportation
   1) Need to pay attention to moisture during storage and transportation;
   2) Be careful not to over-squeeze during transportation;
   3) Storage stacking must not exceed the specified upper limit;
   4) Do not stack heavy objects on the product to prevent pressure loss.
2. Unpacking inspection
   1) First, please confirm whether it is the product you purchased;
   2) Check the list to see if the equipment is complete;
   3) Check if the product is damaged during transportation;
   4) If there is any damage, please contact us.
3. Installation
   1) This product needs to be installed in a professional electrical cabinet;
   2) The electrical cabinet needs to be convected for ensuring heat dissipation and ventilation;
   3) If using a fan to dissipate heat, install a dustproof device;
   4) The installation should leave the position for other devices according to the regulations;
   5) The installation of each device of this product should be firm, and it is not possible to strike with force during installation. After installation, the product can not be shaken;

6）Reduce electromagnetic interference, use professional power supply equipment for power supply, and the power supply should keep a distance of more than 100mm from the equipment;

7）When installing, consider installing the product in a convenient service place.

4. Line connection

1）Wiring must be correct and firm;

2）The connecting wire must not be damaged, not squeezed, or screwed, otherwise short circuit or open circuit may occur;

3）Do not plug and unplug the power plug;

4）Keep hands dry before plugging in and out to prevent safety accidents;

5）Participating wiring personnel must have the corresponding capabilities。

5. Debugging

1）After the assembly is completed, the correct parameters must be set before operation, otherwise the machine will perform the wrong action and even damage the mechanical structure;

2）Modify the parameters within the scope allowed by the software, otherwise it may return the machine to operate incorrectly or even damage the machine structure.

6. Others

1）At the beginning of the design, electromagnetic compatibility should be fully considered. Interference reduction by filter components;

2）When the goods are delivered, the payment guarantee power supply;

3）【Restrictions】 For instructions related to 【Function】, please refer to the manual issued by the machine tool builder;

4）Refer to the machine manufacturer's instruction manual for parameter settings;

5）For machine description, please refer to the machine manufacturer's instruction manual.

Contact us

You can get our services and technical support in the following ways:

Company Name: Chengdu multidimensional robotization technologycompany

Company Address: No. 406, 4th Floor, Unit 4, Building 1, No. 333, Jiqing 3rd Road, Chengdu High-tech Zone, China (Sichuan) Pilot Free Trade Zone

Phone: 028-64844144

Zip code: 610041

Official Website: http://www.mdrobotization.com

Mailbox: 365800795@qq.com

Wechat public account: 成都兴多维科技有限公司

Directory

Chapter One Introduction

CNC-Couter CNC system is a CCD patrol and engraving motion control system independently designed and independently developed by Chengdu multidimensional robotization technology company.

Based on the Windows operating system, CNC-Couter makes the application scope wider, the user interface is more concise, the operation is simpler, easier to learn, and the processing is more reliable.

The system can directly support JPG, DXF, AI, PLT and other files. By adding Mark points to the file, the positioning and offset calculation of the machining graphics can be performed during the machining process, and the micro-repair can be improved. quality. The system can generate G code files from the appeal file and view, simulate and process them. The system can simulate, view, dynamically track display, view file line number, line number processing, power off and other functions for G code files, which is convenient for users.

In addition to manual control, automatic machining, and mechanical zero point, the system also has the functions of milling bottom, multiple workpiece coordinate system switching, real-time control feed rate, and automatic tool setting. Users can also customize the input port and output port 6 according to their needs.

The system is used with a variety of three engraving machines, suitable for a variety of complex advertising cutting, woodworking and other industries.

Chapter Two System installation and connection

This section provides a detailed description of the system installation, including the wiring instructions for each port of the control board and the detailed steps for software installation. The precautions will also be explained to the user.

2.1 CNC-Couter configuration requirements

32, 64-bit operating system for Windows 7 and above.

Memory 2G.

2.2 CNC-Router System installation

If you have an older version of CNC-Router installed on your computer, you can install the new software directly, which will overwrite the old one with the new one. It should be noted that performing such an operation will overwrite all previous data, please pay attention to the backup of the data.

The CNC-Router system consists of two parts, the computer software and the motion control board. Therefore, the installation of the system is divided into two parts: the installation of the computer software and the installation of the motion control board.

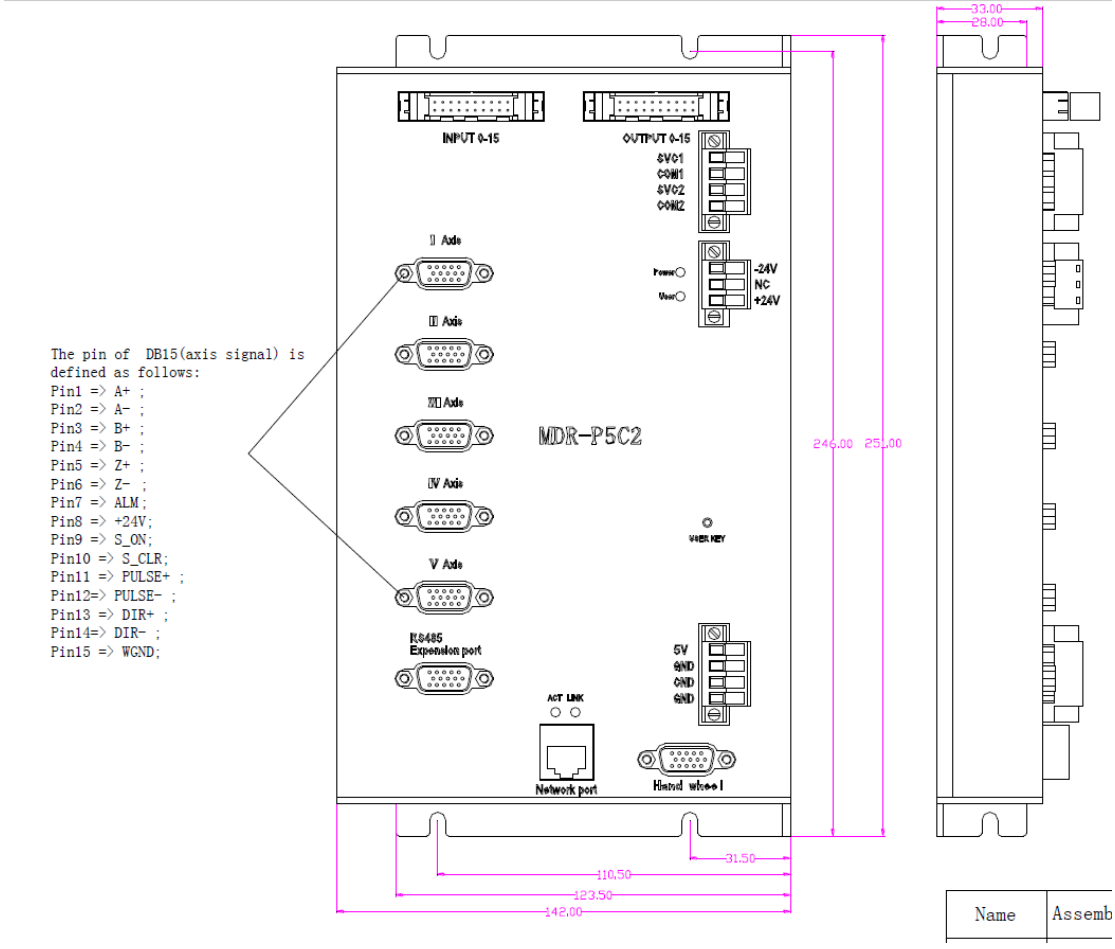
2.2.1 Hardware installation

After the CNC-Couter control system computer software is installed, you need to install the motion control board. The motion control board of the CNC-Couter system is connected to the computer through the network cable to realize communication and achieve the purpose of controlling the movement of the machine.

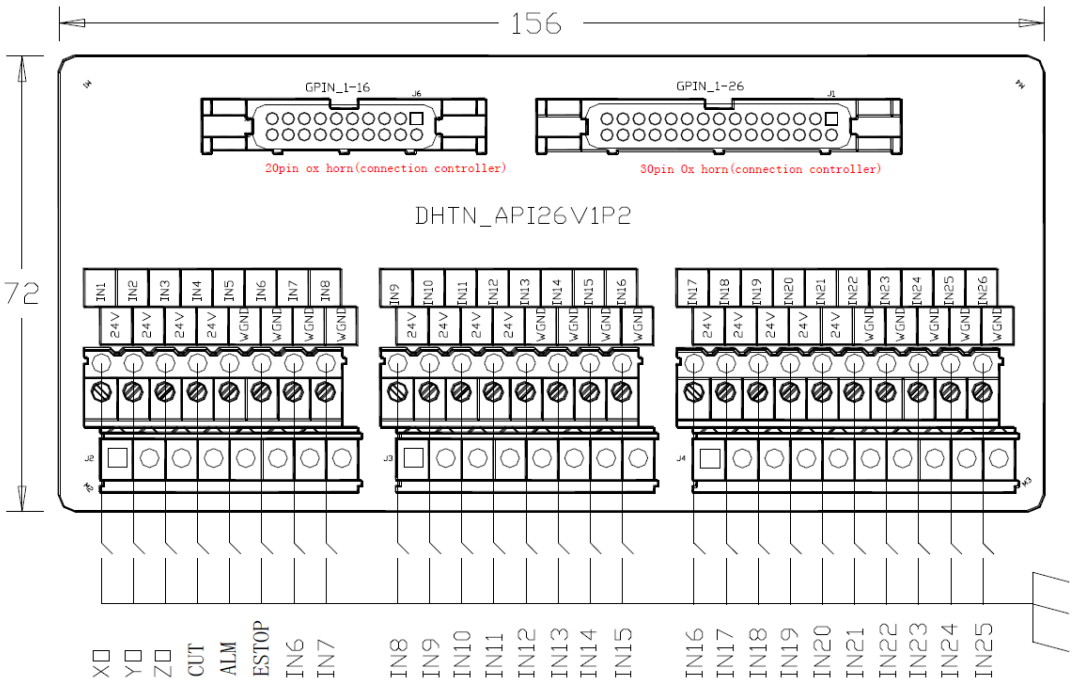
Connect the signal wires of the board to the corresponding positions of the motor and the machine, pay attention to the connection quality and prevent the phenomenon of poor contact.

The following two controllers are available for users to choose from.

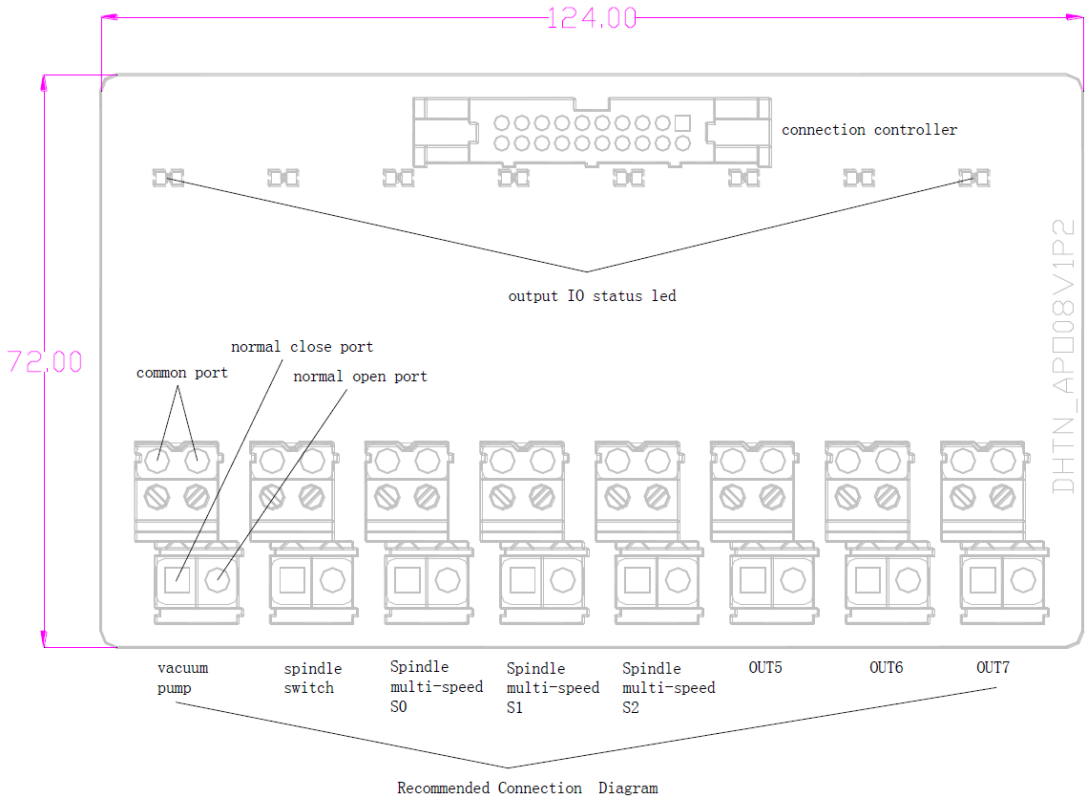
CV1P1 Model recommended wiring diagram:



Motherboard

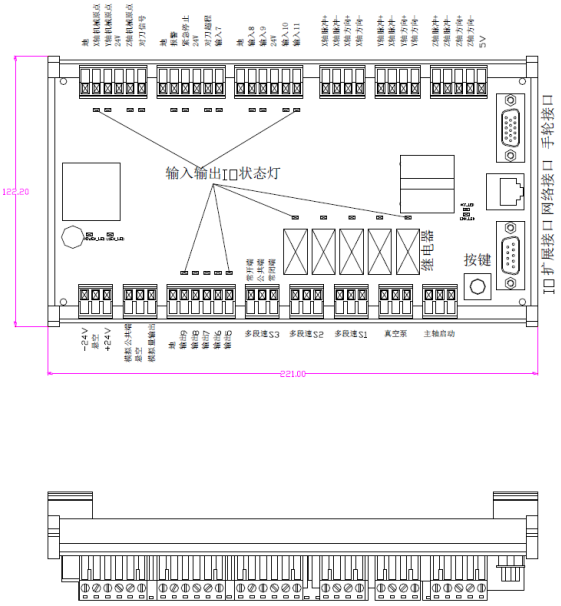


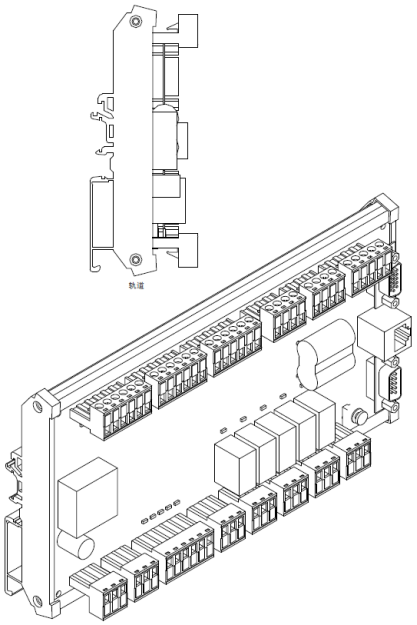
Input board



Output board

SV2P2 Model recommended wiring diagram:





After the control board is installed, the user must first fix the camera, fix the camera to the bracket, and then fix the bracket to the bed. The installation should be firm and reliable to ensure that it does not shake. If the installation is not strong enough, the position of the camera may be offset due to vibration during use, resulting in processing deviation. And the end users of such problems do not have the processing capability themselves, and the maintenance cost is high in the later stage. Please ensure that they are secure during installation.

Once the camera is securely attached, step into the slot and connect the camera's power cord to the camera's flexible cable and connect it to the camera. The connection should also ensure good and reliable contact with the camera.

2.2.2 Computer software installation

After installing the control board and camera

(1) First obtain the latest version of the computer software of the CNC-Router system from the manufacturer or agent, and copy the installation file to the computer to be installed;

(2) Find the computer software installation package of the CNC-Router system in the target folder of the copy CNCRouter1.0.19.exe . The following numbers are version codes. If you have multiple versions of the installation package, please install the latest version;

(3) Double-click the icon to start installing the computer software, and the password verification window will pop up, password is 1992. As shown in Figure 2.2-1:

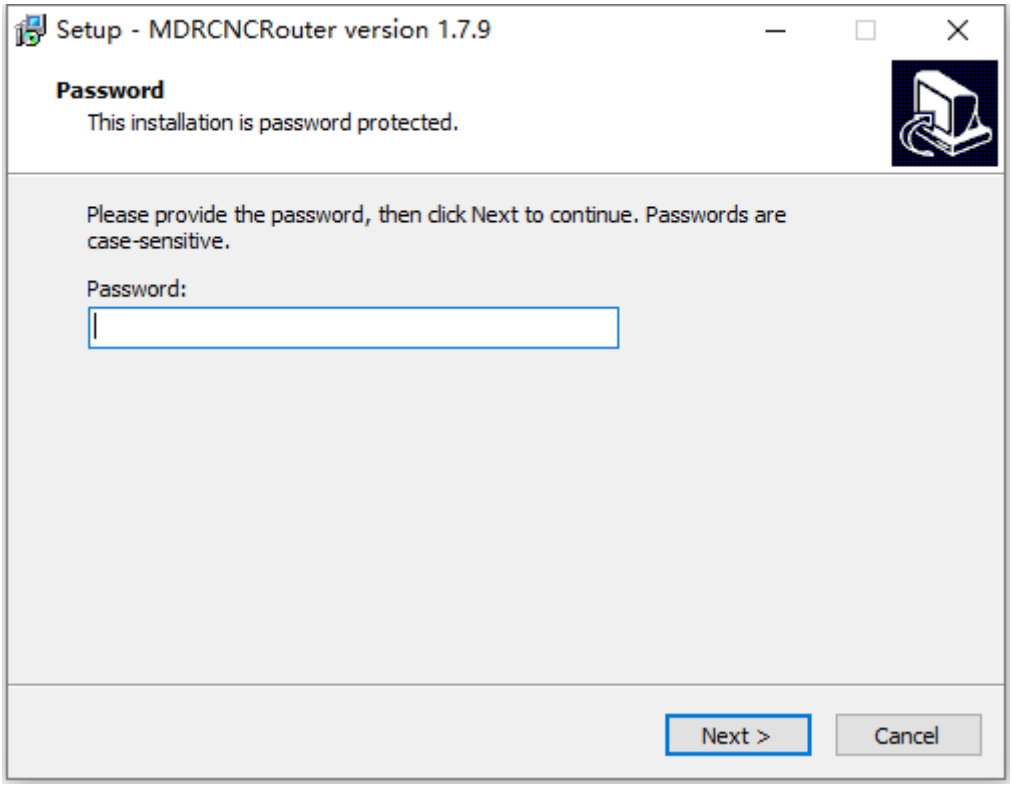


Figure 2.2-1

(4) Enter your password and click Next. The installer will pop up the installation target location prompt dialog box。

(5) Click Next. The installer enters the Select Additional Tasks dialog box. In this dialog box, you can choose whether to create a desktop shortcut. The installer is created by default. Users can click to uncheck and do not create shortcuts. As shown in Figure 2.2-3:

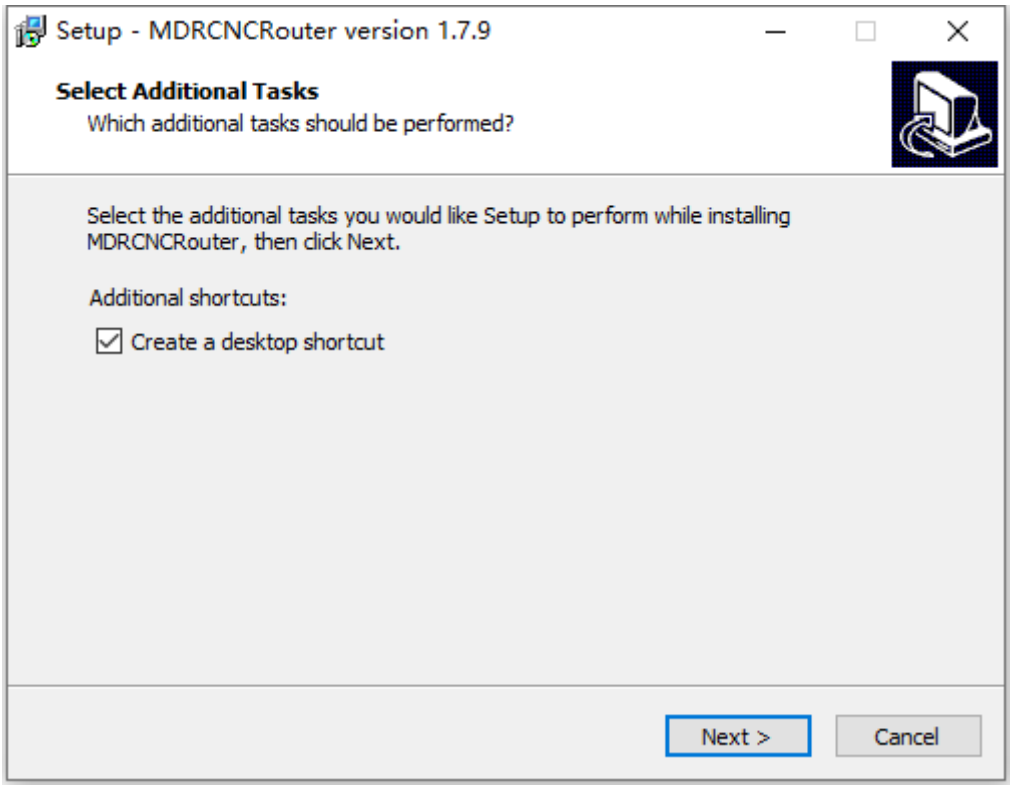


Figure 2.2-3

(6) Click Next, the program is ready to install, and a dialog box pops up, as shown in Figure 2.2-4:

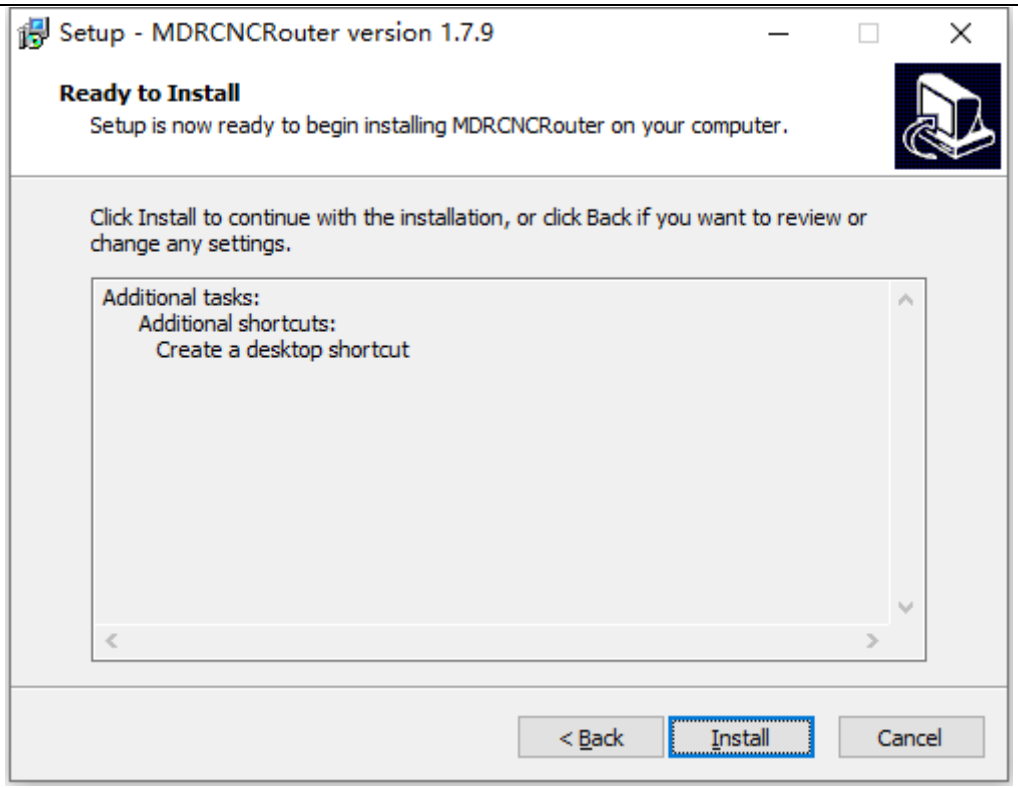


Figure 2.2-4

(7) Click Install, the CNC-Router system computer software will start installation, the installation progress will be displayed during the installation process, if you need to cancel the installation, click Cancel. As shown in Figure 2.2-5:

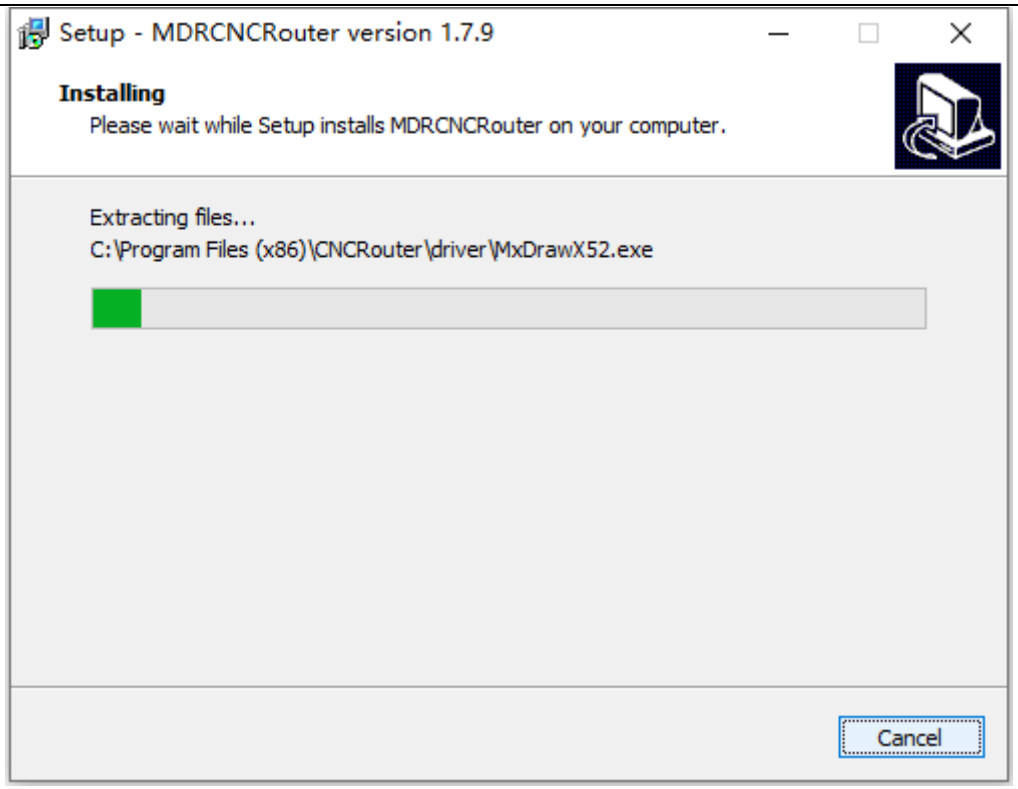


Figure 2.2-5

(8) Wait for the installation to complete, a prompt will appear to install the runtime environment dialog box, which is used to install the necessary operating environment of the software, as shown in Figure 2.2-6:

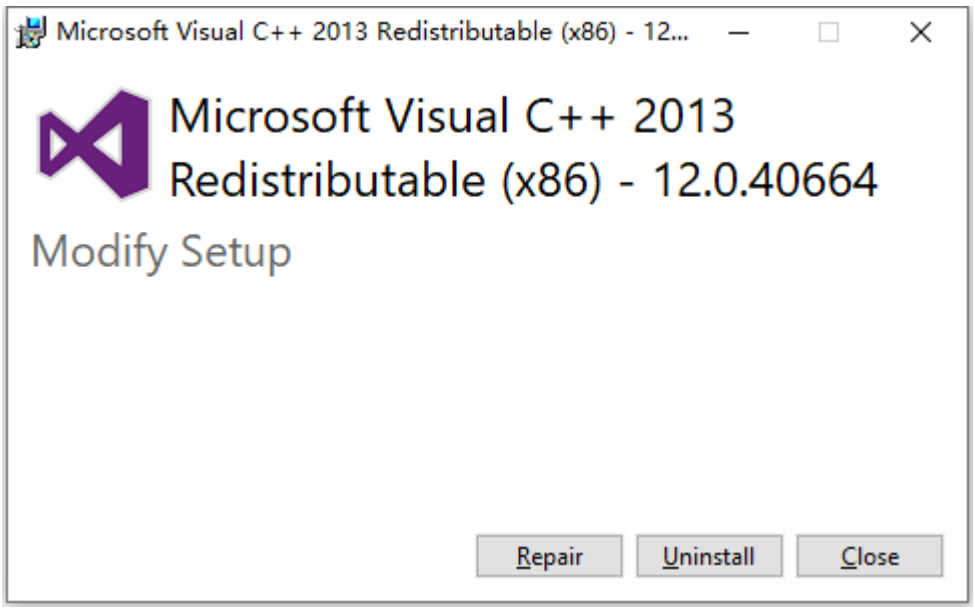


Figure 2.2-6

(9) Check "I agree to the license terms and conditions" and click Install. It can be installed. As shown in Figure 2.2-7:

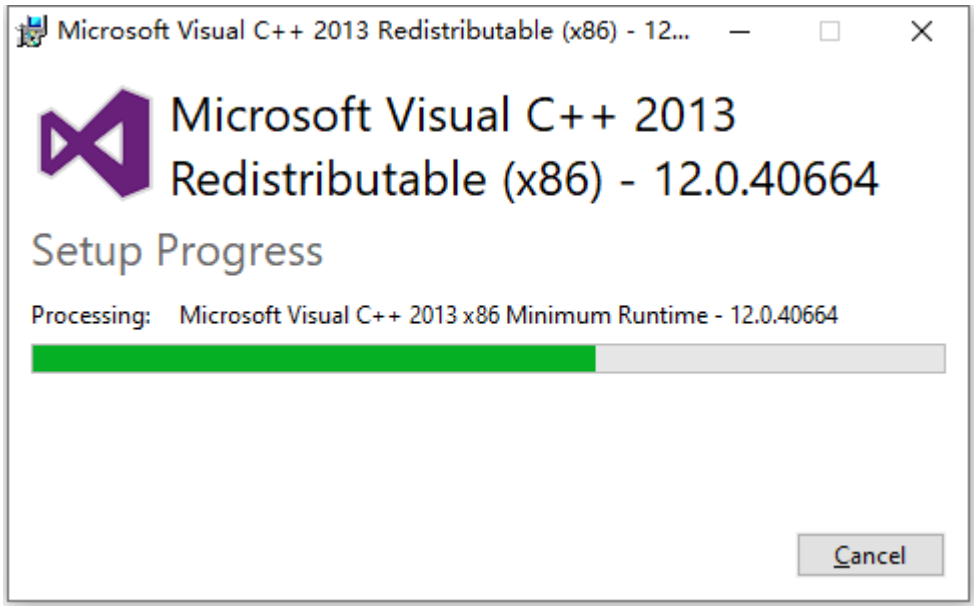


Figure 2.2-7

(10) When the runtime installation is complete, the dialog box for completing the installation will pop up. Click Finish to complete the installation of the runtime library. As shown in Figure 2.2-8:

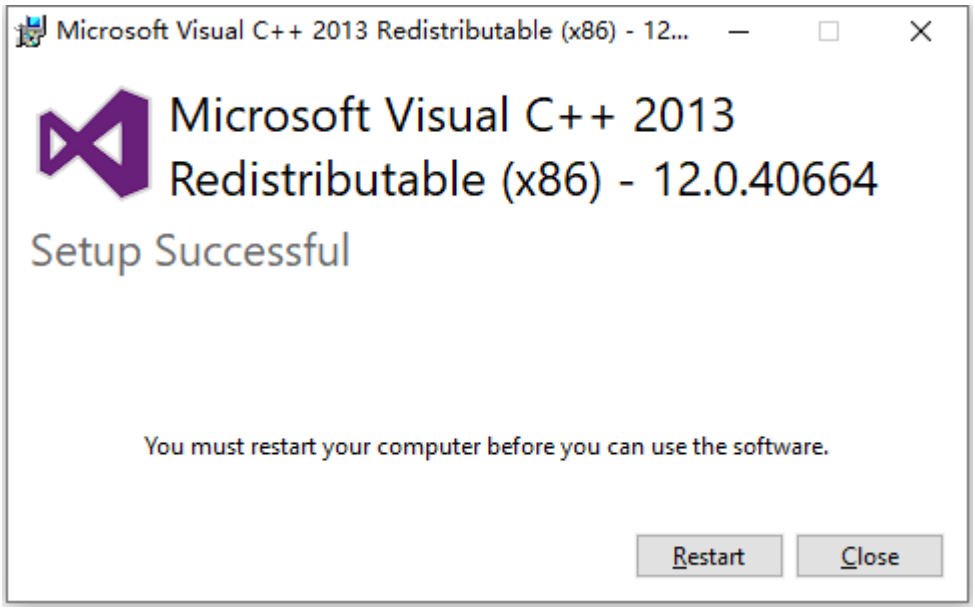


Figure 2.2-8

(11) After that, the software automatically enters the CAD component installation phase, and the dialog box shown in Figure 2.2-9 pops up. Click OK to proceed to the next step.

Figure 2.2-9

(12) The installation dialog pops up, click Install to start the installation. As shown in Figure 2.2-10:

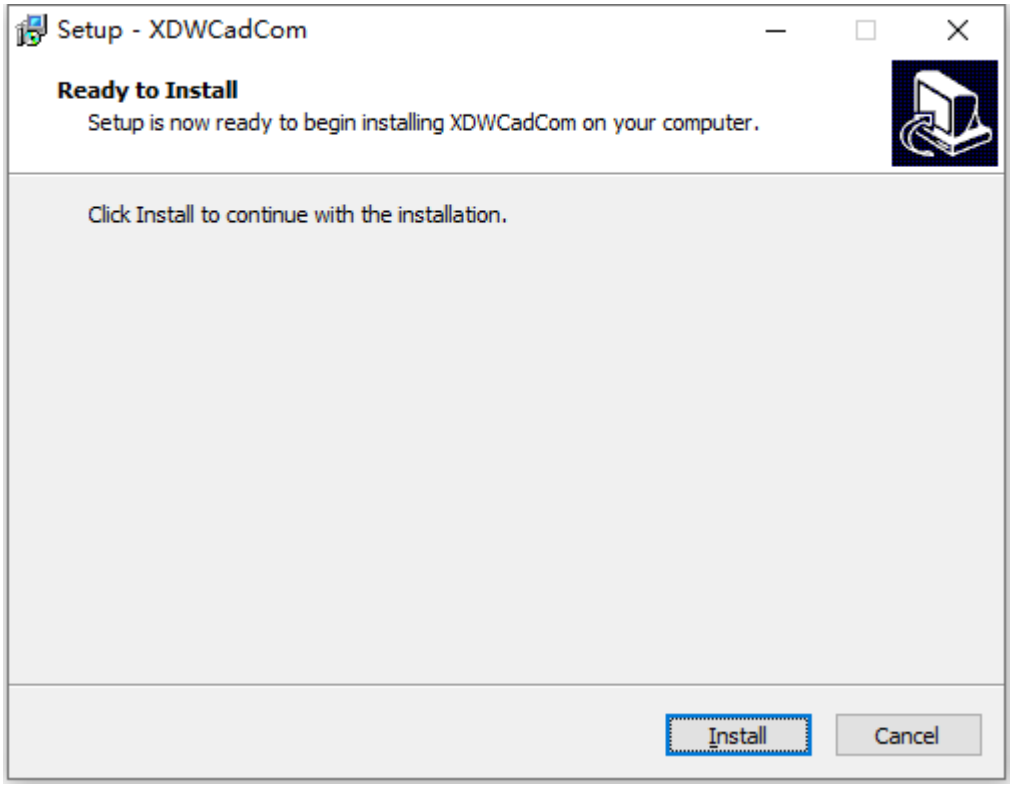


Figure2.2-10

(13) Wait for the installation to complete, click "Finish". Complete the CAD component installation. As shown in Figure 2.2-11:

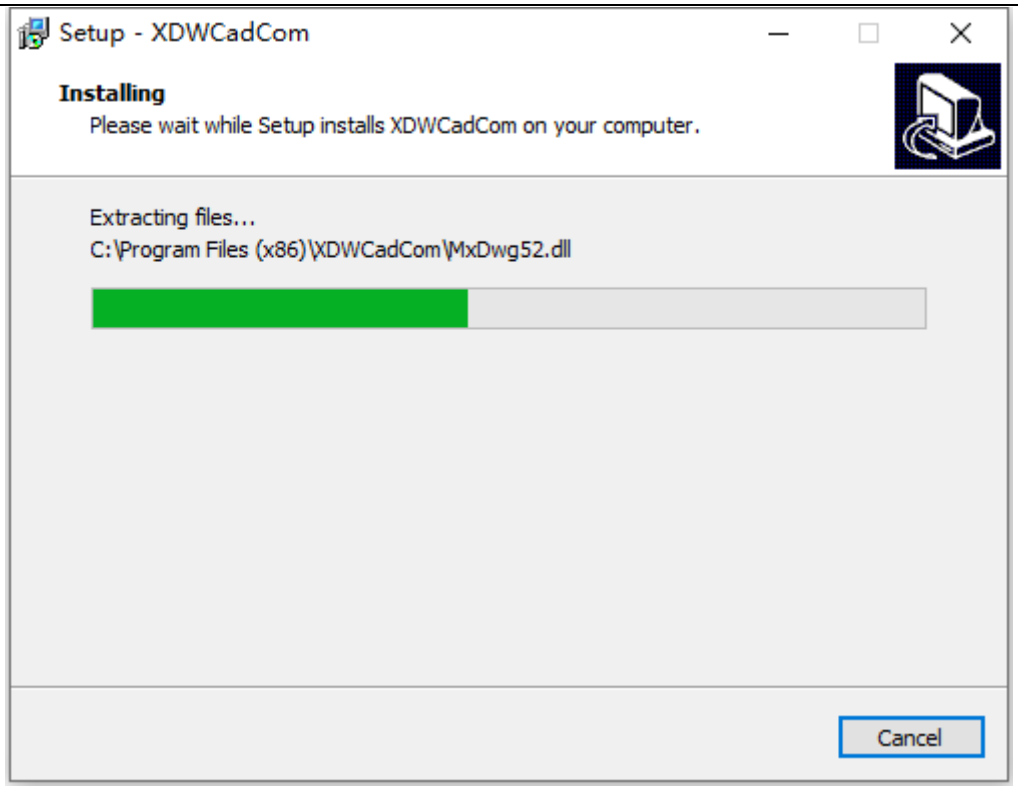


Figure2.2-11

(14) Click CNCRouter to complete the installation and complete the installation. As shown in Figure 2.2-12:

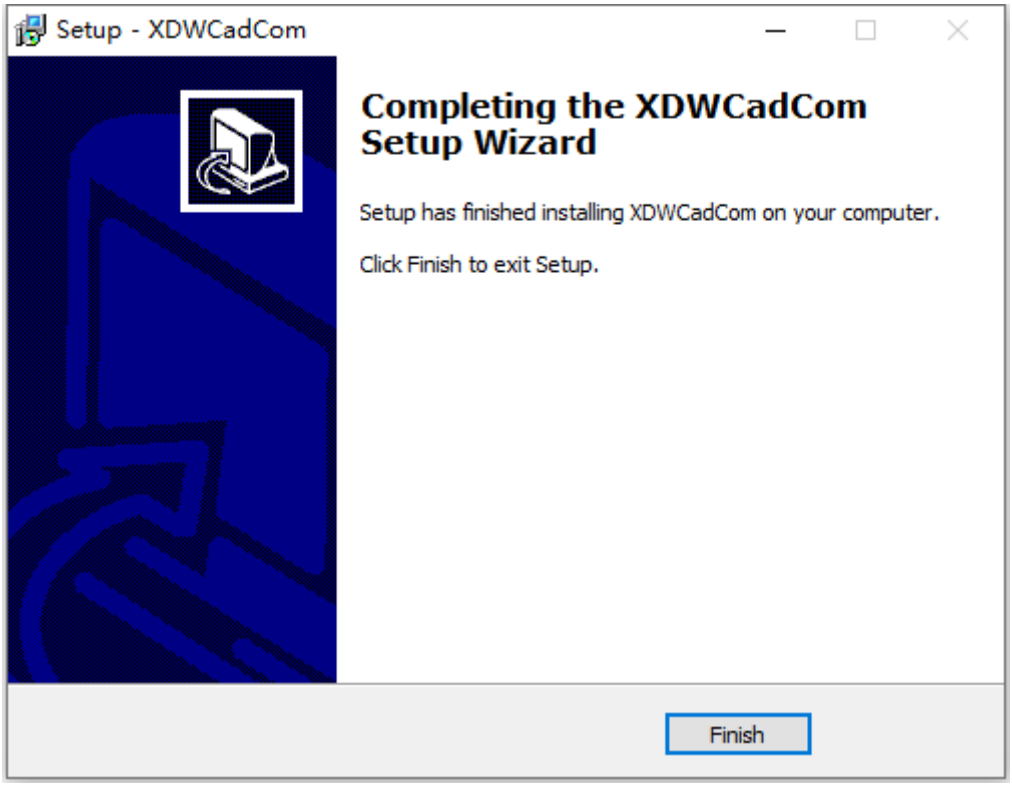


Figure 2.2-12

(15) The system supports CCD processing, you need to install the camera driver, select the appropriate driver installation according to the camera brand, open the installation package. Now take Daheng camera as an example for explanation. Open the installation package, 

Popup installation wizard。As shown in Figure 2.2-13:

Figure 2.2-13

(16) Click “下一步”continue the installation and start the camera driver installation. As shown in Figure 2.2-14:

Figure 2.2-14

(17) Click “下一步” continue the installation, select the camera driver installation directory, it is recommended to use the

default directory. As shown in Figure 2.2-15:

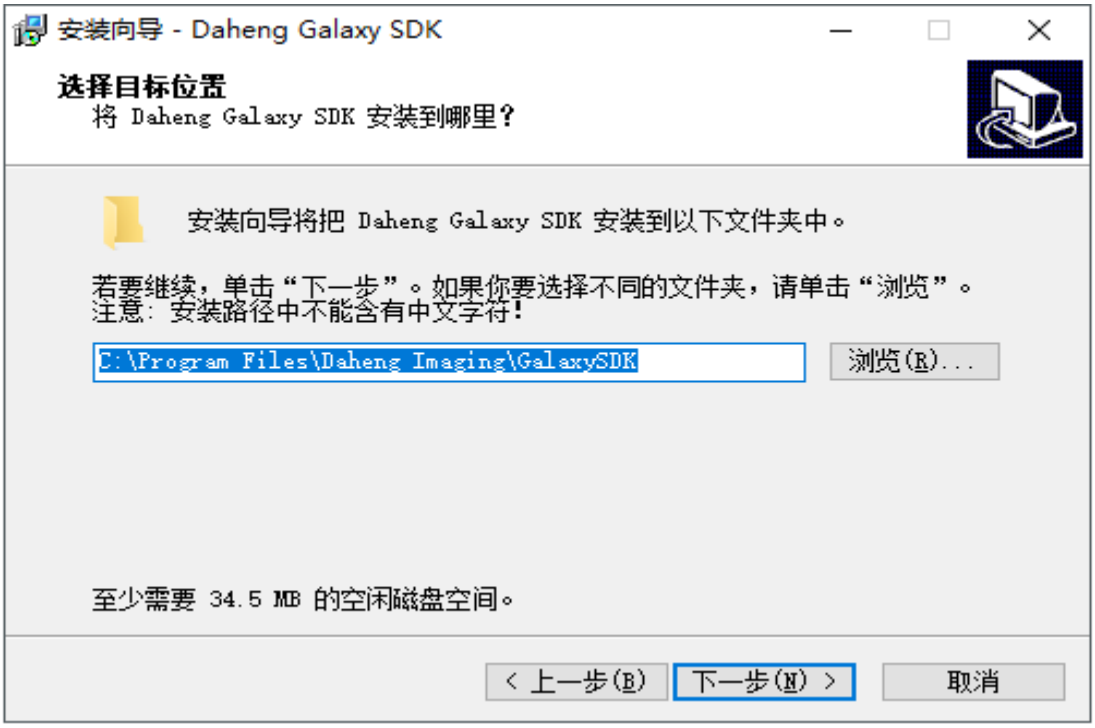


Figure 2.2-15

(18) Click “下一步” proceed with the installation and select the components you need to install. As shown in Figure 2.2-16:

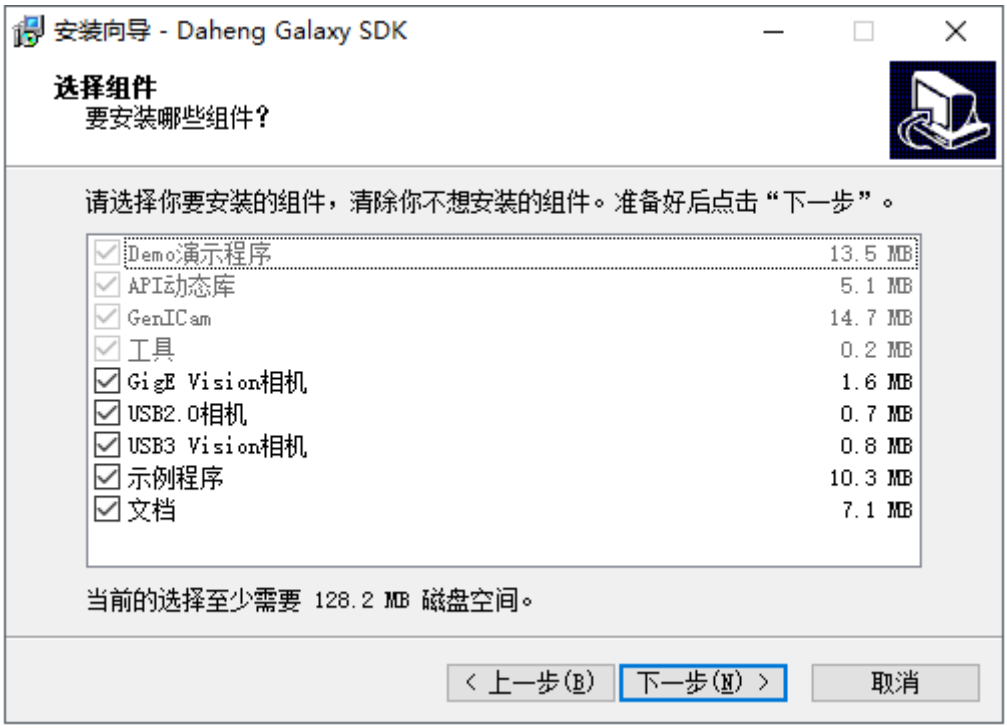


Figure 2.2-16

(19) Click “下一步” and select the installation directory for the program shortcut. As shown in Figure 2.2-17:

Figure 2.2-17

(20) Click “下一步” select whether to create a desktop shortcut for the driver, as shown in Figure 2.2-18:

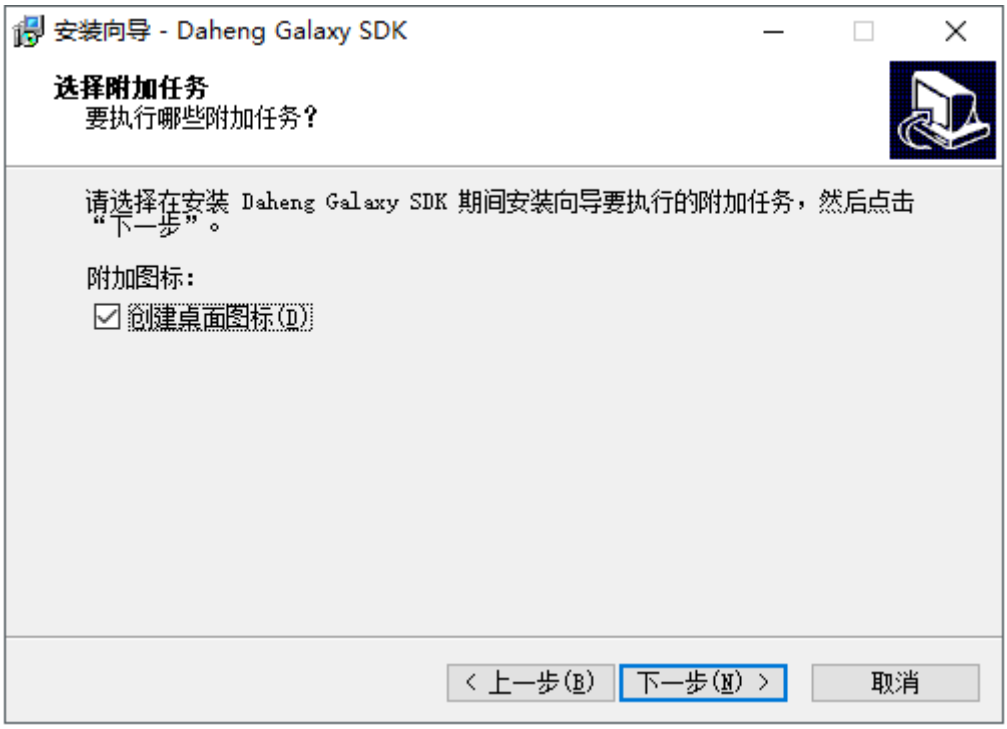


Figure 2.2-18

(21) Click “下一步” prompt the installation. As shown in Figure 2.2-19:

Figure 2.2-19

(22) Click 安装 to get an idea of the progress of the installation based on the progress bar. As shown in Figure 2.2-20:

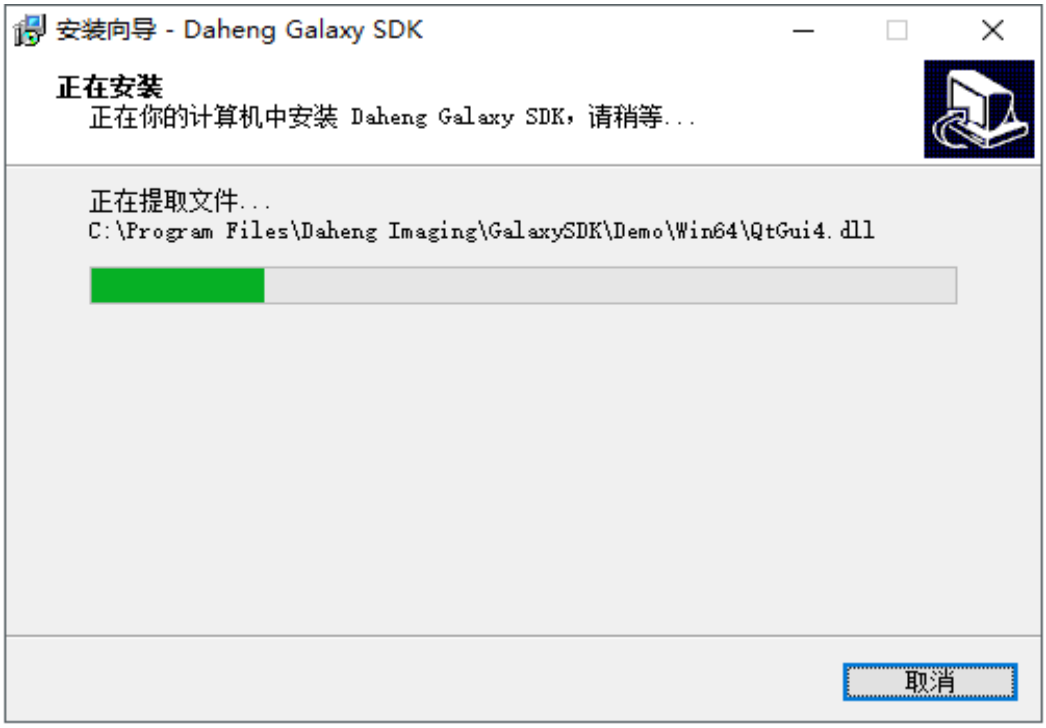


Figure 2.2-20

(23) Wait for the installation to complete, after the installation is complete, a dialog box will pop up, click “完成” to complete the installation of the camera driver. As shown in Figure 2.2-21:

Figure 2.2-21

(24) After the driver installation is complete, you will be prompted to restart your computer. After the installation is complete, it is recommended that the customer restart the computer to ensure that all functions are normal. Check "是", click "完成", the computer will restart. As shown in Figure 2.2-22:

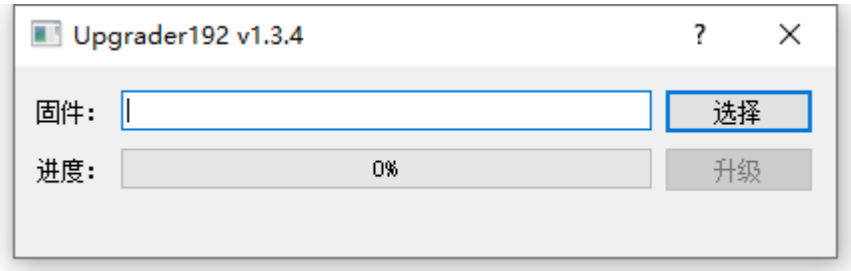
Figure 2.2-22

2.2.3 Motion Control Board Driver Upgrade

The sports board is upgraded to a relatively professional operation. When the board needs to be upgraded, please contact the manufacturer first, obtain the upgrade tool from the manufacturer, and obtain the latest driver of the control board. The driver is the .mfw file. During the upgrade process, upgrade. The tool recognizes such files by default.

The board driver upgrade is the same as the control, and the data transmission is performed through the network cable. First make sure that the network cable is connected properly, power off the control board, find the black button on the board, keep the button in the

button state and re-energize the card. Open the upgrade tool as shown below:



Click [Select] to select the file, and the file selection dialog will pop up, and select the latest version of the sports board driver obtained from the manufacturer. The file path and the file name will be displayed in the front text box, and the user can check whether the file is the target file.

After the file is selected, the [Upgrade] button below will become interactive. At this point, click the [Upgrade] button and the upgrade will be performed. During the upgrade process, the user checks the upgrade progress on the left side and waits for the upgrade to complete.

After the upgrade is complete, the tool will remind the user that the upgrade is complete. Upgrade operation is now complete.

Re-energize the sports board to use the new version.

2.2.4 Computer and control system connection

Make sure that the power supply between the controller and the switch is normal. Connect the computer, controller, and camera to the switch using a network cable. Wiring is shown in Figure 2.2-20:

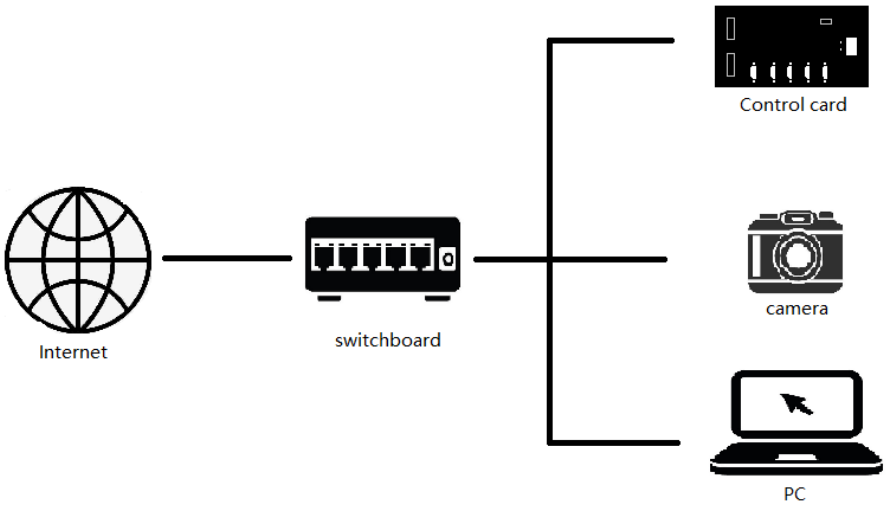


Figure 2.2-20

Open the Control Panel, open [Network and Internet], select the [Network and Sharing Center] sub-bar, click [Ethernet], and the Ethernet status bar will pop up. Click the [Properties] option below, and select [Internet Protocol Version 4 (TCP/IPv4)] in the pop-up dialog box and open it, as shown in Figure 2.2-21:

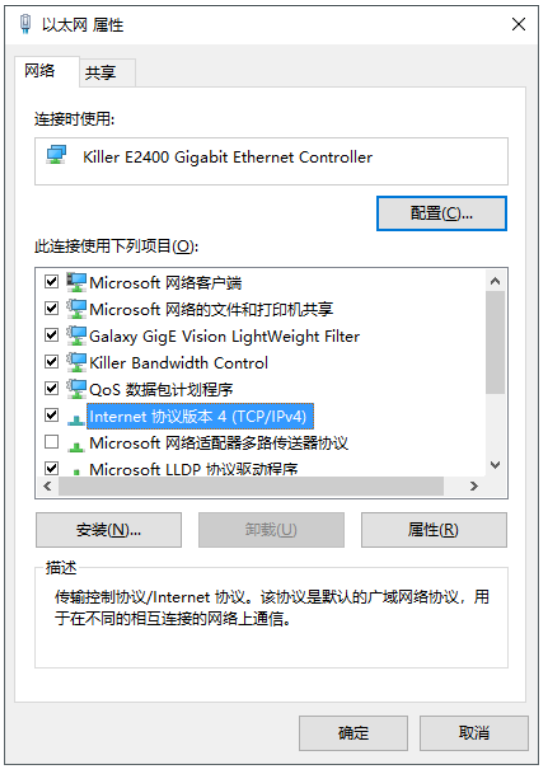


Figure 2.2-21

Open [Internet Protocol Version 4 (TCP/IPv4)] and pop up the pop-up property box to change the IP address and subnet mask, as shown in Figure 2.2-22:

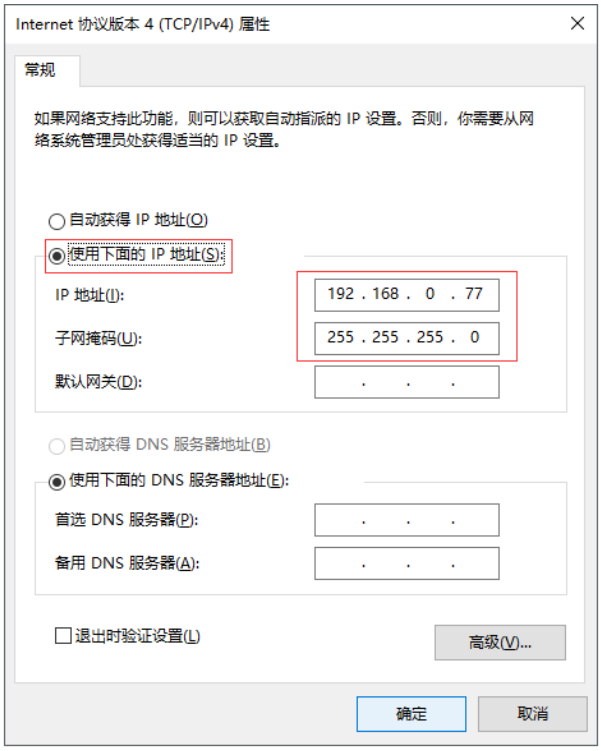


Figure 2.2-22

After selecting [Use the following IP address], set the IP address. Set the IP address to 192.168.0.xxx, and the last three digits can be set to any number from 0-255. However, it should be noted that it cannot be repeated with the IP address of the controller and the lens. The IP of the controller defaults to 100, and the camera IP is user-defined. That is to avoid 100 and user-defined numbers. The subnet mask uses 255.255.255.0. After setting, click OK to save the IP settings.

Open the GigE IP Configurator IP Configuration Tool to set the camera's IP address. Click the desktop shortcut to open the Network Configuration Tool. The interface is shown in Figure 2.2-23:

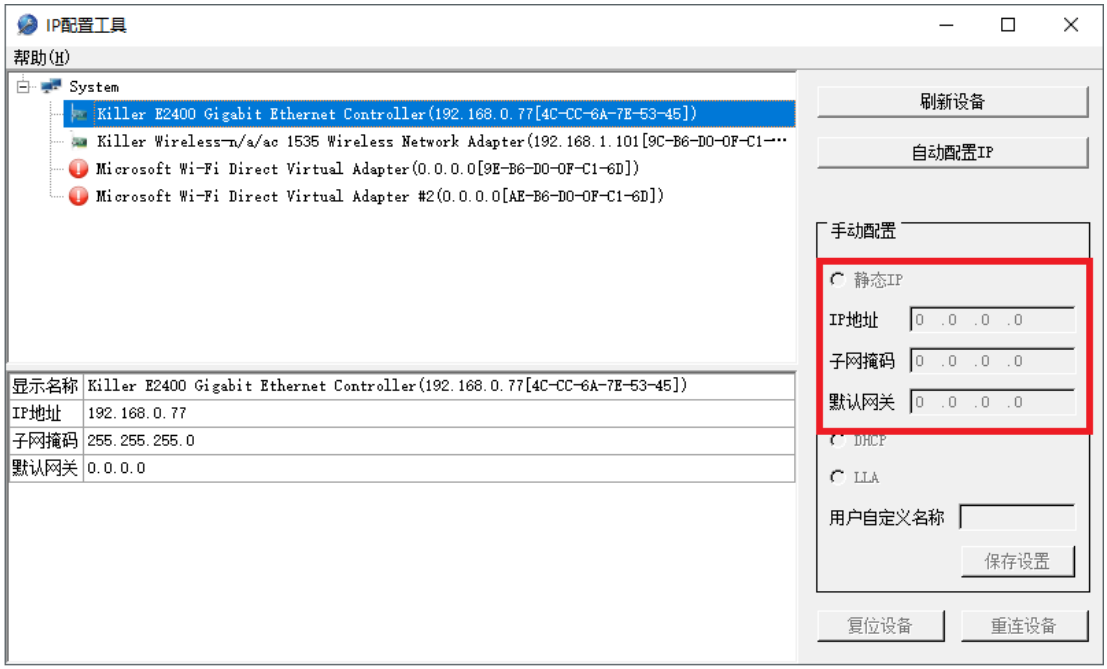


Figure 2.2-23

When the camera is powered on and the network cable is connected properly, click [Refresh Device]. After the device is refreshed, select the camera on the left. Then click [静态 IP] in the manual configuration bar to set the camera IP. Set the IP address to 192.168.0.xxx, taking care not to conflict with the controller and computer IP. Set the subnet mask to 255.255.255.0. Click Save Settings and the camera is set up.

2.2.5 CNC-Couter Control system unloading

Open the control panel, find CNCRouter, select click to uninstall.

Chapter Three Introduction

This section provides a brief description of the CNC-Router software interface, including the names of the various sections and the functions they can perform. The user will refer to the concept of mechanical coordinate system and workpiece coordinate system during use, and this chapter will explain.

3.1 Coordinate System

Including the mechanical coordinate system in the workpiece coordinate system, for the sake of uniformity, the standard coordinate system is a coordinate system using the rule of hand. As shown in Figure 3.1-1:

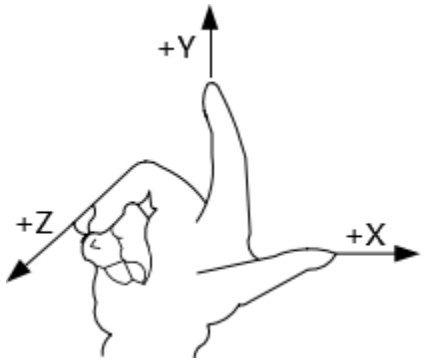


Figure3.1-1

Z axis: Coincides with the spindle axis, the direction of the tool away from the workpiece is positive (+Z).

X axis: Vertical to the Z axis.

Y axis: Together with the X and Z axes, a coordinate system that follows the right-hand rule.

3.1.1 Mechanical coordinate system

The mechanical coordinate system is a fixed right-handed coordinate system whose origin is always relative to a fixed position on the machine. At any time, each point on the machine can be

uniquely determined using a mechanical coordinate system. Every time the power is turned off and restarted, the system needs to return the machine to the mechanical zero point after the system stops.

3.1.2 Workpiece coordinate system

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 workpiece coordinate system is also a set of right-handed coordinate system. The origin of the workpiece coordinate system (ie the workpiece origin) is determined relative to a certain point on the workpiece, and can be floated relative to the mechanical coordinate origin. The origin selection of the workpiece coordinate system should be as simple as possible, such as 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 workpiece offset and mechanical coordinate system is shown in Figure 3.1-2.:

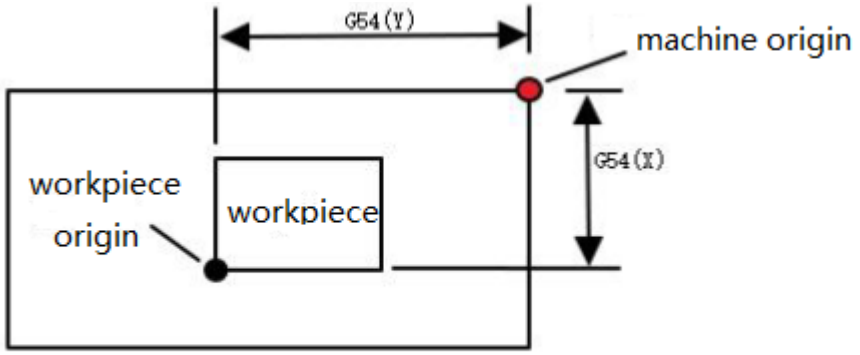


Figure 3.1-2

One or more public offsets can be used during machining, as shown in Figure 23.1-3. If three workpieces are installed on the workbench, each workpiece has a workpiece origin corresponding to the workpiece coordinate system G code. Drill a line on each of the three workpieces on the way, the machine selection depth is Z-5, the programming example is as follows:

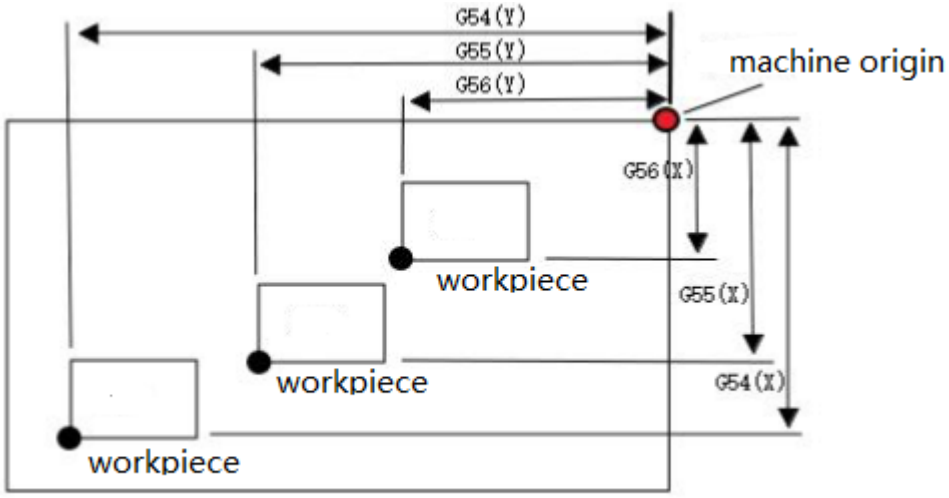


Figure3.1-3

The block N3~N8 is related to the first workpiece in the G54 workpiece coordinate system; the block N9~N15 will drill the straight line of the second workpiece in the same batch installation in the G55 workpiece coordinate system, the block N16~22 will Drill the third straight line in the same batch installation in the G56 workpiece coordinate system. 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~G59.

3.2 Software interface

The CNC-Couter software consists of a title bar, a menu bar, a status bar, a function selection bar, a drawing toolbar, a display area, a file information area, a display area, and a machine control bar. As shown in Figure 3.2-1:

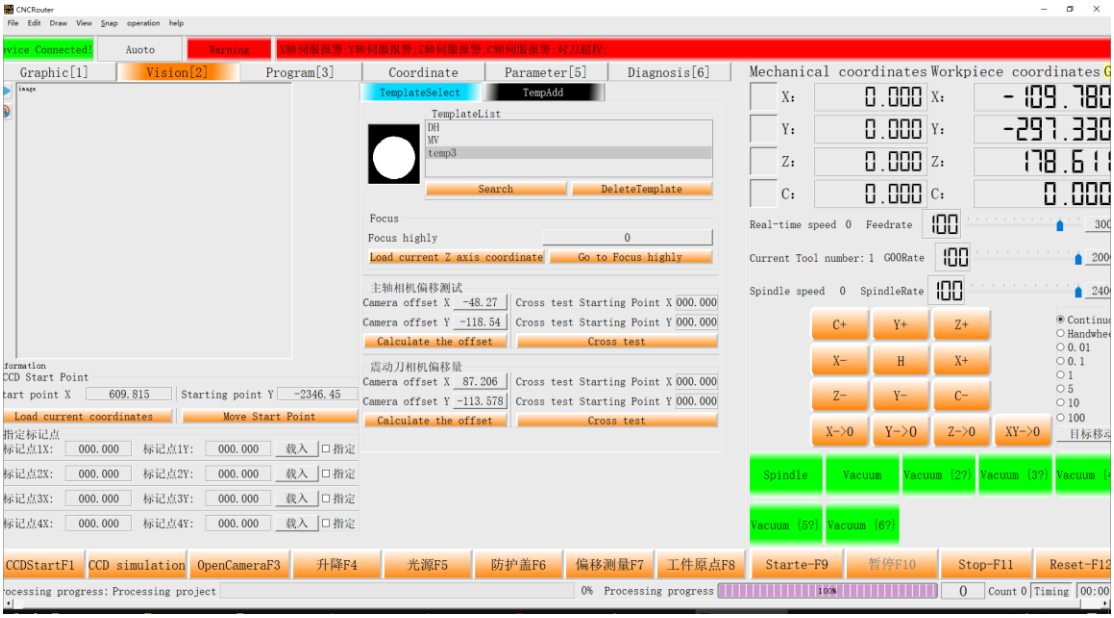


Figure 3.2-1

Menu bar: including file, edit, draw, view, capture, action, help options to accomplish different tasks.

Status bar: used to display the current status of the machine, including network connection status, machining status.

Window selection: Contains graphical display, vision, program, offset, parameter, diagnostic, automatic, manual, reference point function interface selection. Choose different windows depending on the operation.

Drawing Tools: After loading a graphics file (JPG.DXF), you can use the toolbar tool to make simple image modifications. Includes drawing of lines, arcs, circles, polylines, and rectangles.

Layer selection: A file can be defined into multiple layers, and the entire layer of the file can be uniformly modified. This area also contains the selection of Mark points.

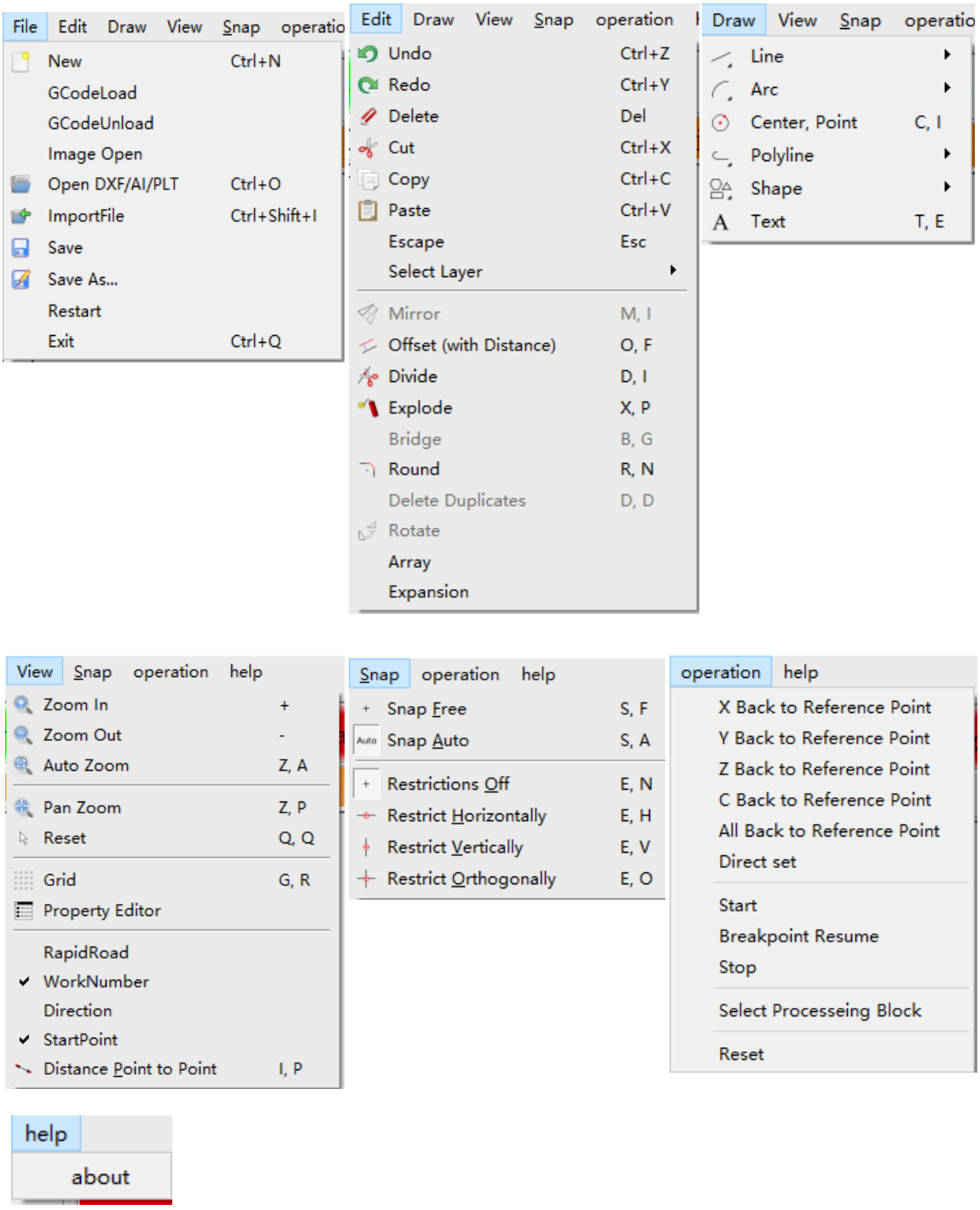
Processing options: including sorting, starting point, compensation options.

Function selection area: Selecting different windows will have different functions, click to operate.

File information: Display the basic information of the file. When loading, etc., you can understand the progress of the work according to the progress bar.

Machine control bar: This interface can display machine coordinates and workpiece coordinates, real-time machining speed, current tool number and spindle speed. The user can control the corresponding speed in real time by the feedrate override, G00 override (empty stroke), and spindle override slide slider. Click the button corresponding to each axis to manually control the movement of each axis and manually open the spindle and vacuum pump. And the beginning of machining and suspension of the machine。

3.3 Menu List


File Edit Draw View Snap operatio
New Ctrl+N
GCodeLoad
GCodeUnload
Image Open
Open DXF/AI/PLT Ctrl+O
ImportFile Ctrl+Shift+I
Save
Save As...
Restart
Exit Ctrl+Q
Edit Draw View Snap operation
Undo Ctrl+Z
Redo Ctrl+Y
Delete Del
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Escape Esc
Select Layer
Mirror M, I
Offset (with Distance) O, F
Divide D, I
Explode X, P
Bridge B, G
Round R, N
Delete Duplicates D, D
Rotate
Array
Expansion
Draw View Snap operatio
Line
Arc
Center, Point C, I
Polyline
Shape
Text T, E
View Snap operation help
Zoom In +
Zoom Out -
Auto Zoom Z, A
Pan Zoom Z, P
Reset Q, Q
Grid G, R
Property Editor
RapidRoad
WorkNumber
Direction
StartPoint
Distance Point to Point I, P
Snap operation help
Snap Free S, F
Snap Auto S, A
Restrictions Off E, N
Restrict Horizontally E, H
Restrict Vertically E, V
Restrict Orthogonally E, O
operation help
X Back to Reference Point
Y Back to Reference Point
Z Back to Reference Point
C Back to Reference Point
All Back to Reference Point
Direct set
Start
Breakpoint Resume
Stop
Select Processeing Block
Reset
help
about

Chapter Four Commissioning Process

Ensure that the control board is connected to all the signal lines such as the driver, inverter, in-position signal, and output signal. Doing a good job of debugging can ensure safe, stable and efficient work.

The debugging process includes axis parameter setting, machine movement range setting, spindle setting, tool magazine setting, vibrating knife zero setting, lens setting.

4.1 Axis parameter setting

After opening the software, first move each axis separately in low speed mode, and do this in the machine control bar. Click X+, The nose will move to the X axis; Click X-, The nose will move negatively towards the X axis. The remaining axes are the same here.

观 Check whether the movement direction of the machine head is the same as the operation. If the actual movement direction of the axis is opposite to the control direction, open the parameter page and click [Manufacturer Parameter]. Manufacturer F3 View the manufacturer parameters, click on the right axis parameter Axis classification, Find the corresponding [Axis Direction] parameter of the opposite axis of motion and change the parameters. If the current parameter is 1, the motion is reversed, then change this parameter to -1.

Set the pulse equivalent of each axis. If the parameter is wrong, the size will not match the file size during processing. Please make sure the parameters are correct.

After the change is complete, restart the software. The parameters take effect. The axis parameters are set.

4.2 Machine movement range setting

According to the position limit switch of the machine tool, select the zero return direction of the mechanical zero return. The zero return direction of each axis needs to be set separately. After confirming the zero return direction, change the [rough positioning direction] parameter and set the zero return direction. The same as the positive direction of the axis is the positive direction.

Switch the interface to the diagnostic interface and trigger the limit switch to ensure that each axis limit signal can be triggered normally. Click [All Zero] in the [Action] menu bar. The machine starts to return to zero.

Wait for the zero return to complete, manually move each axis to the maximum value of the forward and negative travel, and record. The software limits the machine movement stroke by limiting the upper and lower limits of the axis motion, and fills the mechanical coordinates of the recorded maximum stroke into the [stroke upper limit] and [stroke lower limit] parameters respectively. When the machine is equipped with a tool magazine, please make sure that the tool magazine is within the stroke range, otherwise the tool change operation will not be completed. After setting the parameters, complete the machine movement range setting.

4.3 Spindle setting

The system supports multi-speed and analog two spindle speed control modes. Change parameters according to machine design 20008

spindle speed control mode, Select the control mode corresponding to the machine.

Match the maximum speed of the software with the inverter. If the maximum speed of the inverter parameter setting is 36000R/Min, the software parameters **20001 spindle maximum speed**Also set to 36000R/Min.

Multi-speed mode: The machine frequency converter is designed to control at multiple speeds, **Spindle speed control mode** The parameter is set to 2, parameter **20010-20017 The Xth speed**, The setting should be completely the same as the speed of the inverter, otherwise the actual speed will be inconsistent with the specified speed, which will affect the machining effect.

Analog mode: The machine frequency converter is designed to control the analog quantity, **Spindle speed control mode** The parameter is set to 1, parameter **2006 analog minimum voltage value and 20007 analog maximum voltage value,** Set the same parameters in the inverter to ensure that the specified speed is consistent with the actual speed.

4.4 Tool magazine setting

This system supports the machine tool loading tool magazine. Before using the tool change function, the tool magazine position setting and tool setting position setting are required.

First, set the tool setting position, move the spindle to the top of the tool setting tool. On the [Tooling] page under the [Coordinate System] page, click [Load Current X-axis Coordinate] in the [Tool Setting] column. `Load current X machine coordinates` In [Load current Y-axis coordinates] `Load current Y machine coordinates`, At this point,

the X and Y axis coordinate setting of the tool setting tool is completed. Manually drop the Z axis, move to the appropriate position, click [Load current Z coordinate]载入当前Z轴坐标, Set the Z axis setting position. When changing the tool, when it arrives in place, it will first drop to the Z-axis coordinate at a faster speed, and then touch the tool setting tool at a slow speed. Therefore, the position should be raised appropriately, leaving a margin that is not too small, otherwise the tool setting device may be damaged.

Set the magazine capacity according to the machine design. Setting parameters **30001：Tool magazine capacity**, The value is the number of supported tools。

View the tool magazine design position, the tool holder is placed parallel to the X axis or parallel to the Y axis. Change parameter：**30004 Straight row tool magazine installation type.** Select the corresponding type to complete the magazine type setting。

According to the type of straight-row tool magazine, fill in the parameter tool change front point. If the straight-row tool magazine is parallel to the X-axis, fill in the parameters. **30007 Tool change Y axis front point**; If the straight row magazine is parallel to the Y axis, fill in the parameters **30006 Tool change X-axis front point**。The parameters to be filled in are the front point machine coordinates. Please ensure that the tool holder will not touch the tool holder and damage the tool magazine.

Now set the magazine position, manually move the machine, place the holder on the holder, and make sure the position is correct. The selected tool number is the number of the current magazine clamp. Click [Set as current tool number] to complete the tool number setting, and click [Load current X machine coordinate]

Load current X machine coordinates, [Load current Y machine coordinates] Load current Y machine coordinates,[Load current Z machine coordinates]载入当前Z机械坐标. The first tool magazine location setting is completed, and the remaining magazine positions are set to repeat the appeal step. After all the positions are set, the magazine location is set.

4.5 Vibration knife zero setting

Unlike a spindle cutter, a vibrating knife requires a zero angle to cut the blade. A deviation in the zero angle setting will cause burrs on the edge of the machined product and damage the tool; and if the error is too large, it cannot be processed.

During the commissioning, it is necessary to ensure that the angle between the cutting edge and the positive X-axis is zero degrees after the rotating shaft of the vibrating knife is returned to zero. The specific operation method is to click [C axis return to zero] in [Operation] in the menu bar. After the zero return is completed, check the positive edge angle between the blade and the X axis. Change parameters:11506 C Retreat distance,,Change the positive angle of the blade to the X axis after the zero return is completed. Constantly adjust this parameter until it is roughly zero degrees.

At this point, open the vibrating knife and use the manual mode to machine a line segment parallel to the X axis to see if the line segment is jagged. If there is jagged, fine-tune the parameters. Repeat the above steps until the line is drawn without serrations.

4.6 Camera settings

If the machine supports the CCD positioning function, the camera will be mounted next to the spindle. First adjust the camera exposure and focus. The observation lens reveals two adjustable rings, the upper ring is used to adjust the lens exposure, that is, the brightness of the picture is adjusted; the lower lens is used to adjust the lens focal length, that is, to adjust the picture clarity.

First adjust the exposure, the camera itself can automatically set the exposure, so you only need to set the approximate exposure, try to increase the lens exposure, so that the picture displayed during the movement will be more consistent. Rotate the adjustment ring, wait for the camera to automatically set the exposure, observe the software camera display window, and the brightness is moderate. Tighten the screws to lock the adjustment ring.

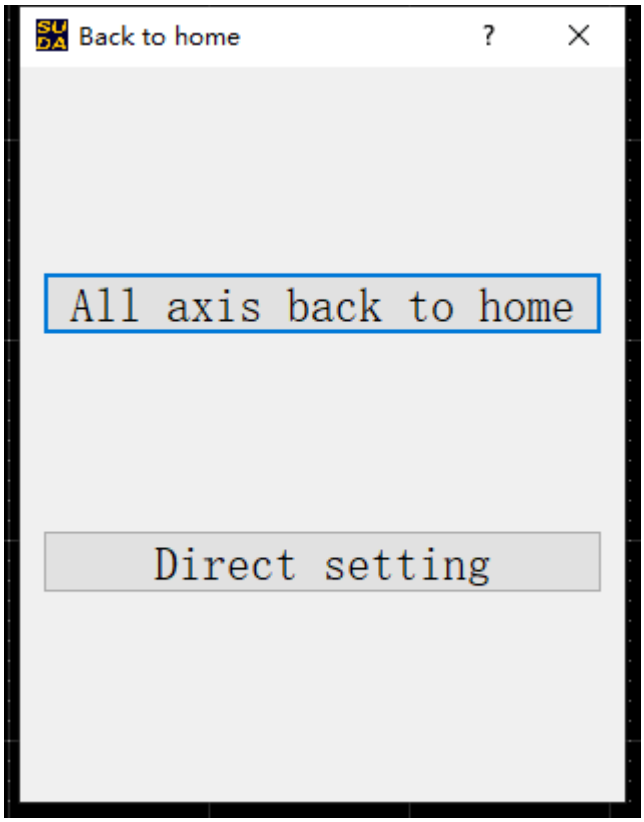
After the exposure adjustment is completed, set the focus. Place a piece of paper under the camera, adjust the ring below the lens and view the software image display window to adjust the lens focal length to the clearest point. Tighten the screws to lock the adjustment ring. Complete the lens adjustment.

Check the software camera page display window to see if the image is upright. If the image display is rotated, adjust the rotation angle under Camera Settings. After rotating the erect, check whether the image needs horizontal or vertical mirroring, and check the corresponding options below to set the screen. Complete camera settings。

Chapter Five Basic Operation Process

This chapter will explain the CCD patrol processing and G code file processing, and inform the user of the basic operation process. The purpose is to let the user get started quickly, master how to process, and understand the action and significance of each step of the machine.

When the user opens the software again, if the motion control board is working normally, the zero return operation dialog box will pop up. As shown below:



This interface allows the user to choose what to do. If the software is started for the first time after the software is turned on for the board, the mechanical zero return operation is required. Click [All Zeros] on this screen, the machine will immediately start

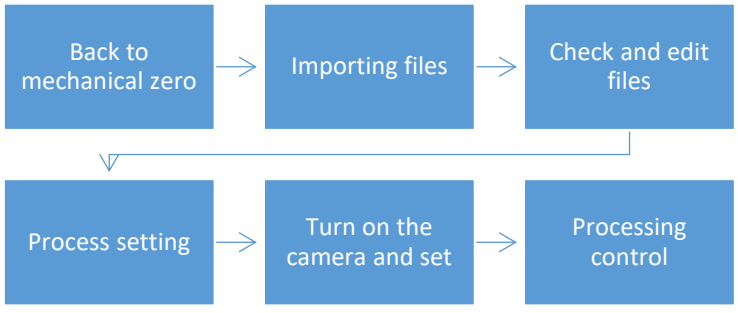
zero return operation. After the zero return is completed, the machining operation can be performed.

If the motion board is not powered off when the software is turned on, and the mechanical zero return operation has been performed when the file was last opened. The user can click [Direct Settings] to set the mechanical zero coordinate data stored on the motion board directly to the mechanical zero point. Reduce zero return operation and improve processing efficiency.

If the user does not need to set the mechanical zero point at this moment, click to close this window.

Warning: Do not click [All Zeros] when the user opens the software for the first time. Because the parameters are not set, the parameters in the zero return direction may be wrong, resulting in wrong mechanical zero return direction. Safety accidents such as broken tools or damage to the bed. The parameter setting should be completed before using this function.

5.1 CCD Machining



5.1.1 Back to mechanical zero

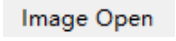
To return to the mechanical zero point, you need to set the axis direction, pulse equivalent and coarse positioning direction first. Refer to 6.1 axis parameters for details.

Open the software, open the [Operation] column of the software menu bar, and then click [All Zeros], the machine starts the mechanical zero return action, waiting for the zero return. After confirming that the machine origin is successful, the mechanical zero return completion flag will appear in the front axle return status bar of the corresponding axis  .

When the user ensures that the mechanical coordinate of the current position is consistent with the actual machine coordinate of the machine tool, for example, the machine tool has not been closed, and no abnormal situation such as sudden stop occurs, click [Direct Settings] of [Operation] in the menu bar to directly use the current Mechanical coordinate system. When this operation is completed, the mechanical zero return status flag will also appear on the machine control bar mechanical zero return status bar  .

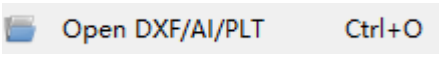
5.1.2 Importing files

The CNC_Couter system supports image editing and can import .JPG and .TIF files directly. After importing the file, the software will automatically extract the outline of the graphic and display it in the display interface.

Click [File] on the menu bar  , Click [Image Open] in the pop-up dialog box.  . After that, the file selection interface will pop up, open the folder location of the file, double-click the

file, and the software automatically recognizes the graphic outline, that is, the processing path, and projects it in the display window.

In addition to being able to import JPG files, users can also open edited files to improve productivity. Click [Open]



A dialog box pops up and selects a folder. Select the file that was edited and saved in the past, and the content of the file will be displayed in the window, and the user will adjust it and then process it.

If the user has generated contours, the exported file is in DXF or PLT format. You can use DXF or PLT format files for processing. Click [Open DXF/AI/PLT] in the [File] menu bar to open this type of file. The file can be displayed in the software by opening the file in the pop-up dialog box, and the user can edit and process the file.

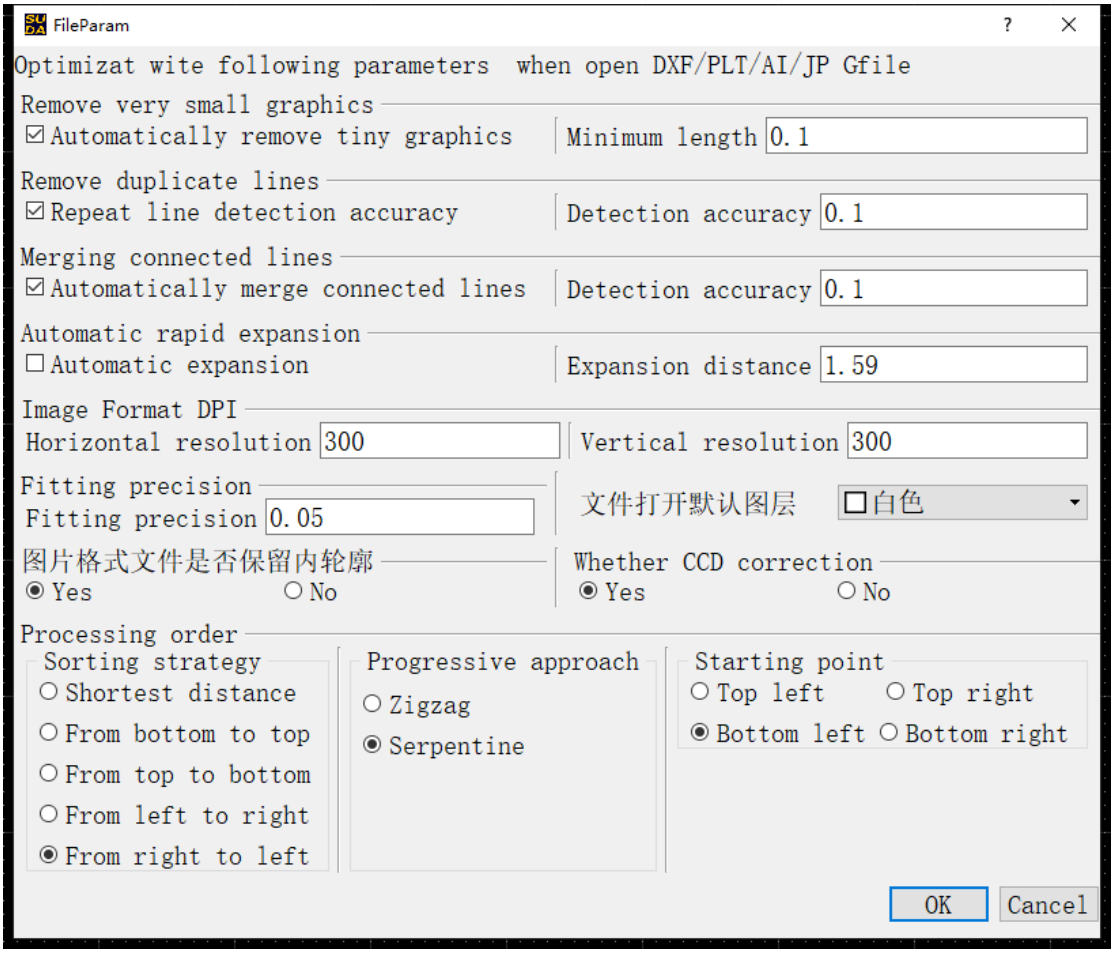
5.1.3 Check and edit files

This part mainly involves three tools: drawing tool, layer selection interface, and processing selection. Drawing tools can be used to draw straight lines  , arc  、circle  , multi-end line  、rectangle  ; The layer selection interface can adjust the processing parameters of each layer to meet the requirements of processing multiple different parameters at one time. CNC-Couter provides a total of 7 layers for users to choose. Mark points are selected in the graphic display window. When using the CCD patrol edge processing function, the user needs to mark the Mark point before it can be processed normally. The user can select an image outline in the display window, and then click the layer selection

window on the left side to set the layer to which the element belongs.

5.1.4 Process setting

To perform CCD processing, you need to turn on the CCD correction function. Click [File Parameters] at the bottom of the [Graphics Display] page, and the dialog box shown below will pop up:



Click [Whether CCD correction] is [Yes], at this time, CCD correction will be performed before processing.

When processing multiple products, the software will automatically sort after the user has set the layer to be processed. If the user needs to change the order of processing, you can click [Manual Sort]. After clicking, you will have one more circled

number symbol next to the mouse. The starting number is 1, then click on the element you want to process the first. At this point, the workpiece becomes the first workpiece to be machined, and the number in the circle next to the mouse will change to 2. At this point, click on the next element to be processed. The processing order of the element is the second one. By analogy, the ordering of all primitive processing sequences can be completed. During the reordering process, the software will automatically plan the machining path according to the rules set by the user, and indicate the machining empty travel with white lines.

Click [Start Point Selection] in the processing selection column.  , You can customize the workpiece from that point to meet the needs of more situations.

After selecting the workpiece, click [Compensation] in the processing selection column.  , It can set the expansion and contraction and perform tool compensation. Set the frame as shown in Figure 5.1.3-1:

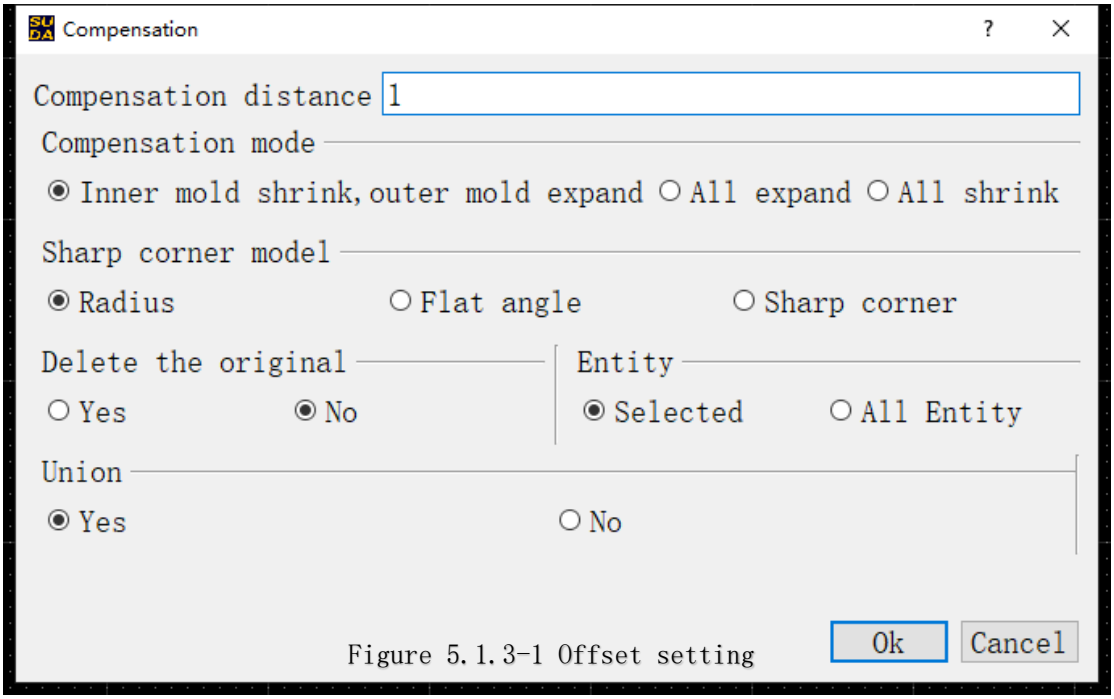


Figure 5.1.3-1 Offset setting

In this window, you can set the offset distance, offset mode, sharp angle mode, whether to delete the original image, etc., and the user can set according to the actual situation.

5.1.5 Turn on the camera and set

In the CCD processing mode, the user needs to set the camera. Click on the visual bar in the window selection bar. The position is shown in Figure 5.1.5-1:

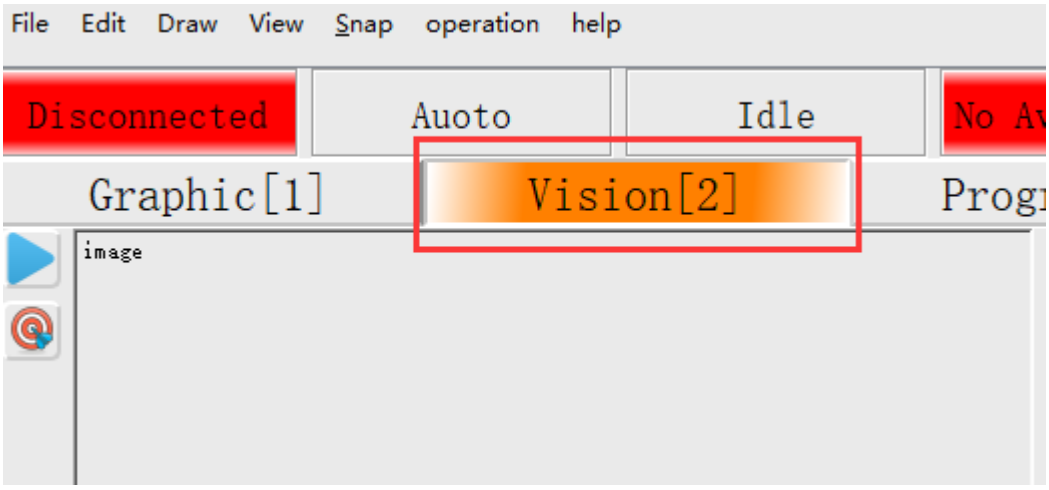


Figure 5.1.5-1 Visual optio

Click [Open Camera] OpenCameraF3, The camera starts up and the image will be displayed to the display window. The first use of the camera screen may be rotated, the screen display moving direction does not match the actual moving direction, etc., and the screen is added to rotate and flip the screen. Click [Template Add]; as shown in Figure 5.1.5-3:

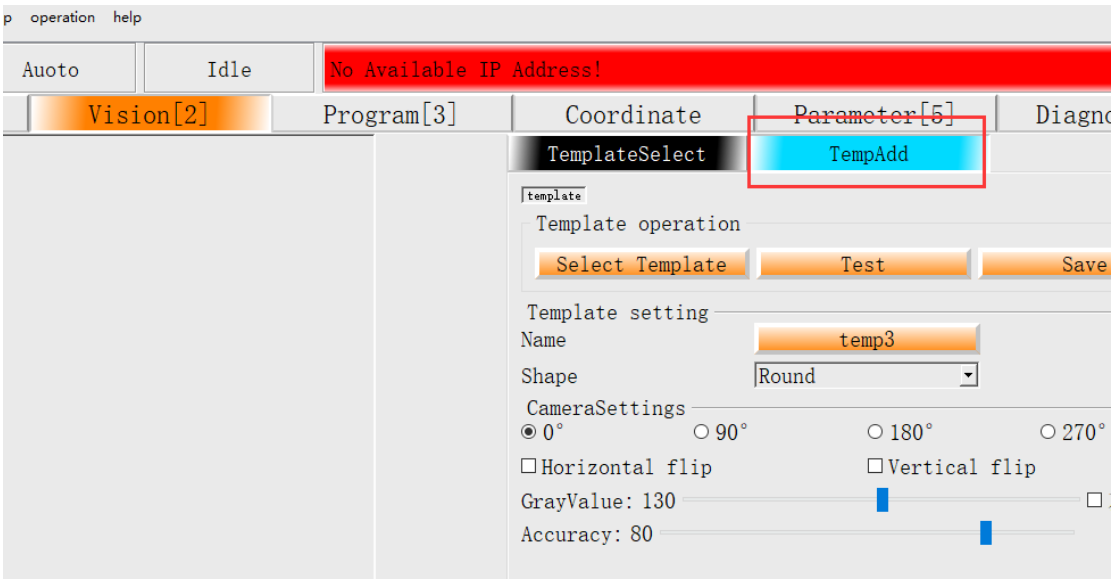


Figure 5.1.5-3

The user makes the display normal by selecting the rotation angle under the [Camera Settings] column. By checking whether the horizontal flip and the vertical flip, the problem that the moving direction of the screen display does not match the actual direction during the movement process is solved.

After the camera screen is set, use the manual movement window of the machine console, and the user can also use the numeric keyboard shortcut keys corresponding to each button to move. As shown in Figure 5.1.5-2:

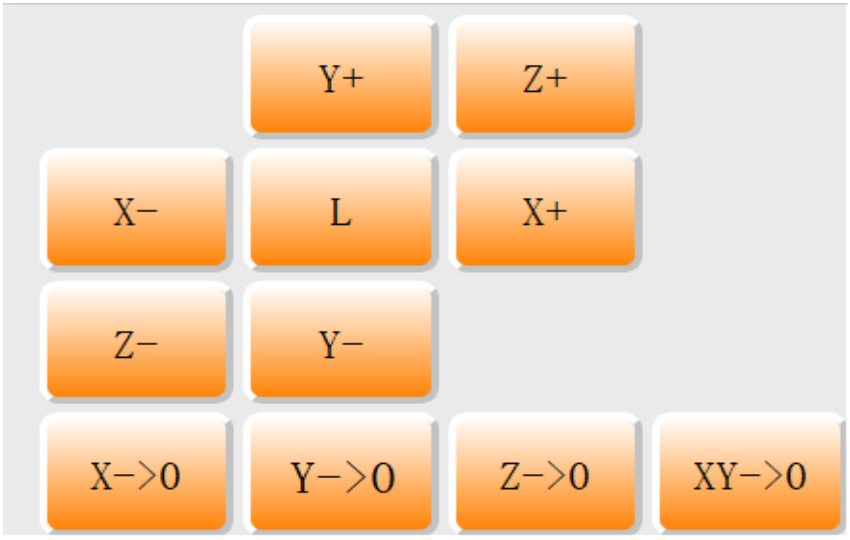


Figure5.1.5-2

To move the X and Y axes, the user needs to point the camera at the Mark point in the lower left corner of the processing file (the system defaults to the lower left corner, the user can adjust according to the requirements), and move the camera position so that the Mark point appears in the display window. Click on the bottom right corner of the page [CCD Processing] CCDStartF1, At this point, the system starts automatic patrol positioning, and starts processing after positioning is completed.

The appeal method uses the Mark point identification template as the system's own template. The CNC-Router system provides users with autonomous template addition function, and users can add templates themselves to increase adaptability. Click [Select Template] Select Template The template add operation page will appear, and the template will be selected. The template can then be framed. The template setting interface awakens the shape selection and name setting, and can set the gray value and the precision value as needed.

Then test, click [test] Test,Perform a template test. Check to see if the camera can find Mark points based on the template we selected. If you can find the Mark point to indicate that the template is available, click [Save] Save, Save the template for direct use in the next processing.

After the first processing, the software will automatically record the first Mark point position as the recognition starting point. If the processing material position has not moved, the user can click [Move Start Point]. Move Start Point Quickly return

to the recognition starting point, the system automatically moves the camera to the starting Mark point position, without the need for the user to manually move the camera again, improving processing efficiency.

5.1.6 Processing control

The CNC-Couter control system can view the position information of each axis during the machining process and control the feedrate override and spindle speed in real time to view the real-time speed of the motion and the spindle speed. Use the understanding of the processing status, so that users can respond to sudden changes in real time.

Below the user can view the processing progress and present it to the user in the form of a progress bar. The control interface is shown in Figure 5.1.6-1.:

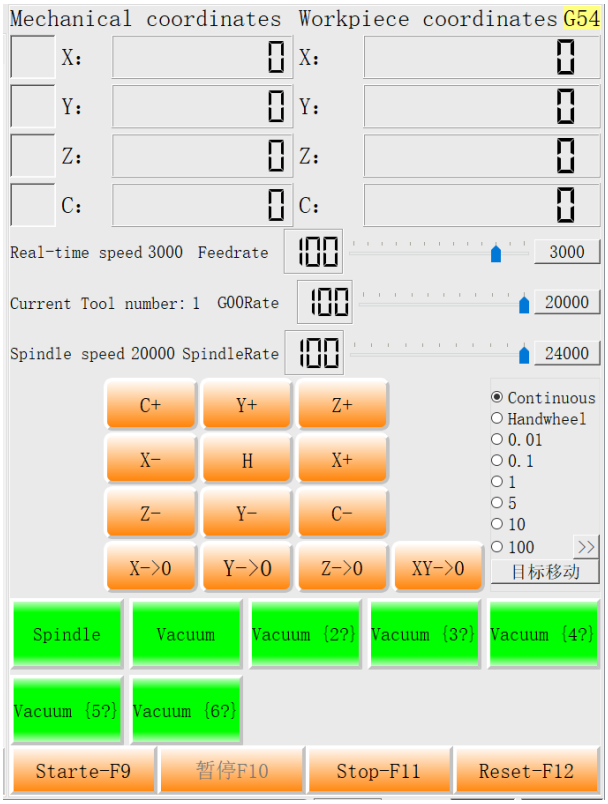
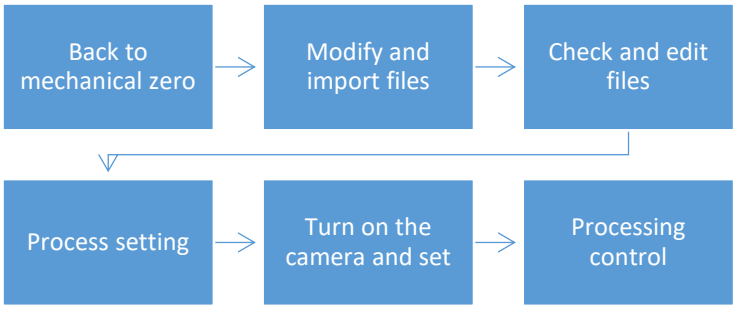


Figure 5.1.6-1

You can view the current speed, spindle speed and tool number on this screen. Move the slider to adjust the feedrate override, G00 override and spindle override. The machining rate can also be controlled in real time by the sliding feedrate override slider during machining. The progress of the process can be viewed in the progress bar. Click [Start] Starte-F9 The file will be processed from the beginning; click [Pause] 暂停F10 Start machining from where the machining was stopped last time; click [Stop] Stop-F11 The software immediately stops the current processing; when an alarm occurs and the problem is solved, click [Reset] Reset-F12 Able to sign.

5.2 G code file CCD patrol cutting



5.2.1 Back to mechanical zero

To return to the mechanical zero point, you need to set the axis direction, pulse equivalent and coarse positioning direction first. Refer to 6.1 axis parameters for details.

Open the software, open the [Operation] column of the software menu bar, and then click [All Zeros], the machine starts the mechanical zero return action, waiting for the zero return. After confirming that the machine origin is successful, the mechanical zero return completion flag will appear in the front axle return status bar of the corresponding axis. .

When the user ensures that the mechanical coordinate of the current position is consistent with the actual machine coordinate of the machine tool, for example, the machine tool has not been closed, and no abnormal situation such as sudden stop occurs, click [Direct Settings] of [Operation] in the menu bar to directly use the current Mechanical coordinate system. When this operation is completed, the mechanical zero return status flag will also appear on the machine control bar mechanical zero return status bar.

5.2.2 Modify and import files

The user first needs to add a file header of the specified format to the G code file, as shown in the following figure:

```
////////G code file patrol file header

#20000=4    //Mark point identification number

#20002=0    //The X coordinate of the first Mark point
#20003=0    //The Y coordinate of the first Mark point
#20004=1094 //The X coordinate of the second Mark point
#20005=0    //The Y coordinate of the second Mark point
#20006=1094 //The X coordinate of the third Mark point
#20007=2184.9   //The Y coordinate of the third Mark point
#20008=0    //The X coordinate of the fourth Mark point
#20009=2184.9   //The Y coordinate of the fourth Mark point

#20030=5    //X width of the first Mark point
#20031=5    //Y width of the first Mark point
#20032=5    //X width of the second Mark point
#20033=5    //Y width of the second Mark point
#20034=5    //X width of the third Mark point
#20035=5    //Y width of the third Mark point
#20036=5    //X width of the fourth Mark point
#20037=5    //Y width of the fourth Mark point
FUNC"CCD"   //Recognition function
M03     //Open the spindle

//////////////////


G00 X-0.001 Y5.495 Z10.000
```

Figure 5.1.7-1

The text format is as follows:

#20000=4 // Mark point identification number

#20002=0 // The X coordinate of the first Mark point

#20003=0 // The Y coordinate of the first Mark point

#20004=1094 // The X coordinate of the second Mark point

#20005=0 //The Y coordinate of the second Mark point

#20006=1094 // The X coordinate of the third Mark point

#20007=2184.9// The Y coordinate of the third Mark point

```
#20008=0  // The X coordinate of the fourth Mark point
#20009=2184.9// The Y coordinate of the fourth Mark point

#20030=5  // X width of the first Mark point
#20031=5  // Y width of the first Mark point
#20032=5  // X width of the second Mark point
#20033=5  // Y width of the second Mark point
#20034=5  // X width of the third Mark point
#20035=5  // Y width of the third Mark point
#20036=5  // X width of the fourth Mark point
#20037=5  // Y width of the fourth Mark point
FUNC"CCD" // Recognition function
M03       // Open the spindle
```

Explanation:

#20000 Fill in the number in the back to use the number of Mark points, that is, if you use 3 Mark points to fill in 3 here, use 4 Mark points to fill in 4 here;

#20002 The rear fill-in number is the X-axis coordinate of the first Mark point, which is the X-axis coordinate of the center position of the origin viewed in the CAM software. The remaining Mark points have the same X-axis coordinates;

#20003 The rear fill-in number is the Y-axis coordinate of the first Mark point, which is the Y-axis coordinate of the center position of the origin viewed in the CAM software. The remaining Mark points have the same Y-axis coordinates;

#20030 The rear fill-in number is the X-direction width of the first Mark point, which is the X-axis width of the Mark point in the print file. The remaining Mark points have the same X width;

#20030 The rear fill-in number is the X-direction width of the first Mark point, which is the X-axis width of the Mark point in the print file. The remaining Mark points have the same X width;

#20031 The trailing number is the width of the first Mark point in the Y direction. This value is the Mark point Y-axis width in the print file. The remaining Y points of the Mark point are the same;

The FUNC "CCD" change code is the CCD patrol start code. You need to add this code software to perform the patrol action.

The spindle start command in the M03 G code, the spindle will be rotated if the command is added, otherwise the spindle will not open for machining, causing the cutter to be broken.

After adding the header of the appeal file, the file is set and the file is ready for processing.

In the menu bar [File] click [G code file loading]. The Windows file selection dialog will pop up, find the path to save the G code file, open the file to complete the loading.

[Load file] at the bottom of the program interface Load-F1, It is also possible to perform a file loading operation, and the subsequent operations are the same as above.

After the file is processed, if you do not need to perform secondary processing, click [G code file unload] in the menu bar [File]. Will uninstall the currently loaded file.

5.2.3 Check file

The G code file can be simulated on the [3D] interface under the [Program] page. When the loading is completed, click [Simulation] SimulationF3, Ability to simulate loading G code files. The user can view the line number corresponding to the G code file in the display interface.

You can view the G code on the [G Code] screen under the [Program] page. If you need to edit the file, click [Edit File]. EditFileF5, Ability to edit files in the pop-up action box.

5.2.4 Set the workpiece origin

The workpiece origin is set according to the G code file, and the Z axis is set to the material surface or the material bottom surface. On the [Manual] page, manually move the X-axis and Y-axis, and set the original X and Y-axis coordinates of the workpiece according to the position of the Mark point to ensure that when the workpiece is moved to the first Mark point, the camera is in the field of view. Can see the first Mark point. Click [X, Y to clear] XY->0, Complete the workpiece origin X and Y axis coordinate settings. Manually move the Z axis to perform the tool setting. After reducing the workpiece origin coordinate, click [Z Clear]. Z->0 Complete the workpiece origin setting. If the machine has an automatic tool setting tool, use the automatic tool setting. For details, please refer to 6.4.3.

The user can select the workpiece origin autonomously and select it on the [Workpiece Coordinates] screen under the [Coordinate System] page. 6 groups of 60 for users to choose to use.

5.2.5 Processing control

The CNC-Couter control system can view the position information of each axis and perform simple control during machining. The control interface is shown in Figure 5.2.5-1:

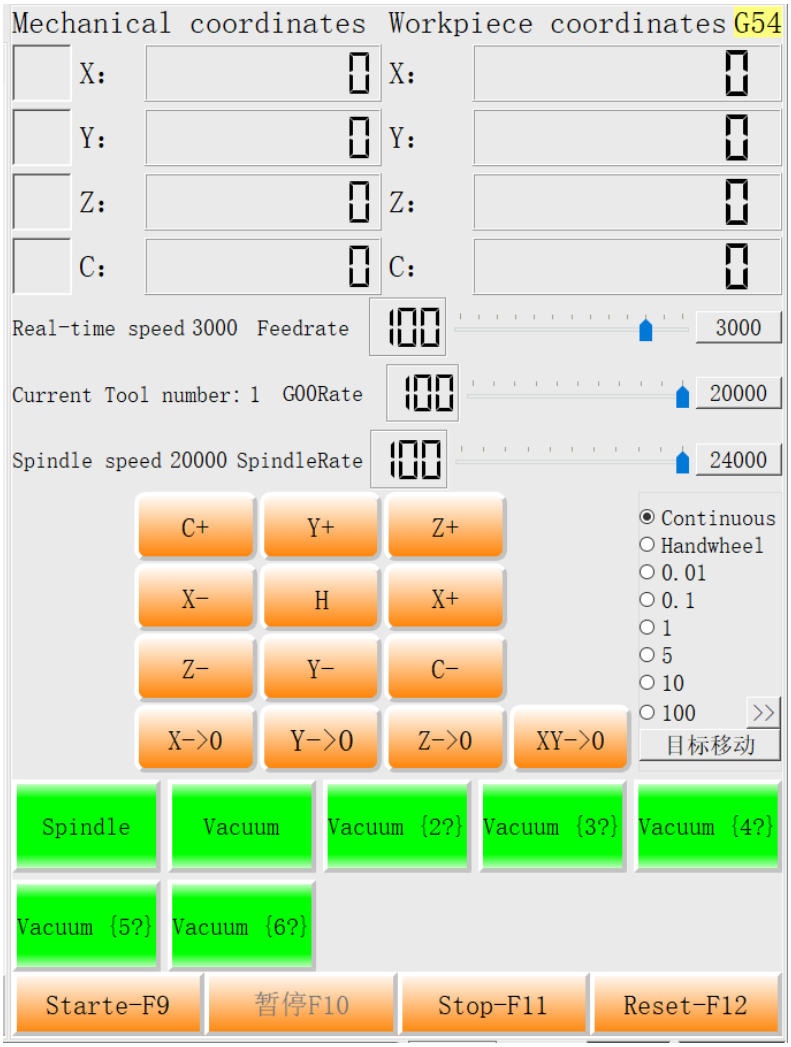


Figure 5.2.5-1

You can view the current speed, spindle speed and tool number on this screen. Move the slider to adjust the feedrate override, G00 override and spindle override. The machining rate can also be controlled in real time by the sliding feedrate override slider during machining. The progress of the process can be viewed in the progress bar. Click [Start] 

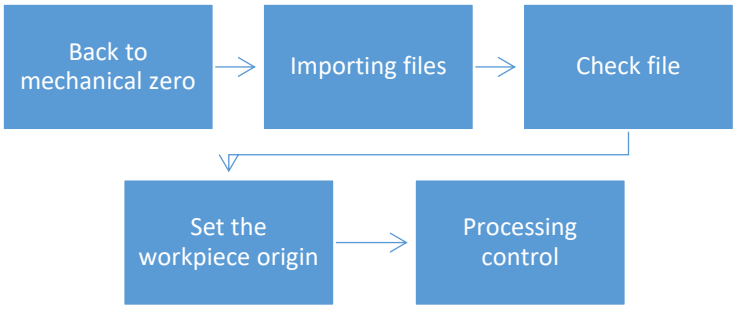
The file will be processed from the beginning; click [Pause] 

Start machining from where the machining was stopped last time; click [Stop] 

The software immediately stops the current processing; when an alarm occurs and the problem is solved, click [Reset] 

Able to clear the alarm flag.

5.3 G code file processing



5.3.1 Back to mechanical zero

To return to the mechanical zero point, you need to set the axis direction, pulse equivalent and coarse positioning direction first. For details, refer to 6.1 axis parameters.

Open the software, open the [Operation] column of the software menu bar, and then click the machine [all zeros], the machine starts mechanical zero return, waiting for the zero return. After confirming that the machine origin is successful, the mechanical zero return completion flag will appear in the front axle return status bar of the corresponding axis. .

When the user guarantees that the mechanical coordinate of the current position is consistent with the actual machine coordinate of the machine tool, for example, the machine tool has not been closed, and no abnormal situation such as sudden stop occurs, click [Direct Settings] in [Operation] in the menu bar to directly use Current mechanical coordinate system. When this operation is completed, the mechanical zero return status flag will also appear on the machine control bar mechanical zero return status bar. .

When the vibrating knife is used, it is necessary to ensure that the direction of the vibrating knife blade is in the same direction

as the positive direction of the X-axis. After the zero return operation, the angle between the blade and the horizontal is zero degrees. If the zero degree setting is not accurate, the edge of the finished product will be jagged, which greatly affects the processing effect. Therefore, please ensure that the zero setting is correct before use. Please refer to the debugging chapter for the specific setting method.

5.3.2 Importing files

In the menu bar [File] click [G code file loading]. The Windows file selection dialog will pop up, find the path to save the G code file, open the file to complete the loading.

[Load file] at the bottom of the program interface Load-F1, Can also perform file loading operations, subsequent operations are the same as above.

After the file is processed, if you do not need to perform secondary processing, click [G code file unload] in the menu bar [File]. Will uninstall the currently loaded file.

5.3.3 Check file

The G code file can be simulated on the [3D] interface under the [Program] page. When the loading is completed, click [Simulation] SimulationF3, Ability to simulate loading G code files. The user can view the line number corresponding to the G code file in the display interface.

You can view the G code on the [G Code] screen under the [Program] page. If you need to edit the file, click [Edit File]. EditFileF5, Ability to edit files in the pop-up action box.

5.3.4 Set the workpiece origin

Set the workpiece origin according to the G code file and tool the tool to the material surface or the material bottom surface. On the [Manual] page, manually move the X and Y axes to the target position. Click [X, Y to clear] XY->0, Complete the workpiece origin X and Y axis coordinate settings. Manually move the Z axis to perform the tool setting. After reducing the workpiece origin coordinate, click [Z Clear]. Z->0 Complete the workpiece origin setting. If the machine has an automatic tool setting tool, use the automatic tool setting. For details, please refer to 6.4.3.

The user can select the workpiece origin autonomously and select it on the [Workpiece Coordinates] screen under the [Coordinate System] page. 6 groups of 60 for users to choose to use.

5.3.5 Processing control

The CNC-Couter control system can view the position information of each axis and perform simple control during machining. The control interface is shown in Figure 5.2.5-1:

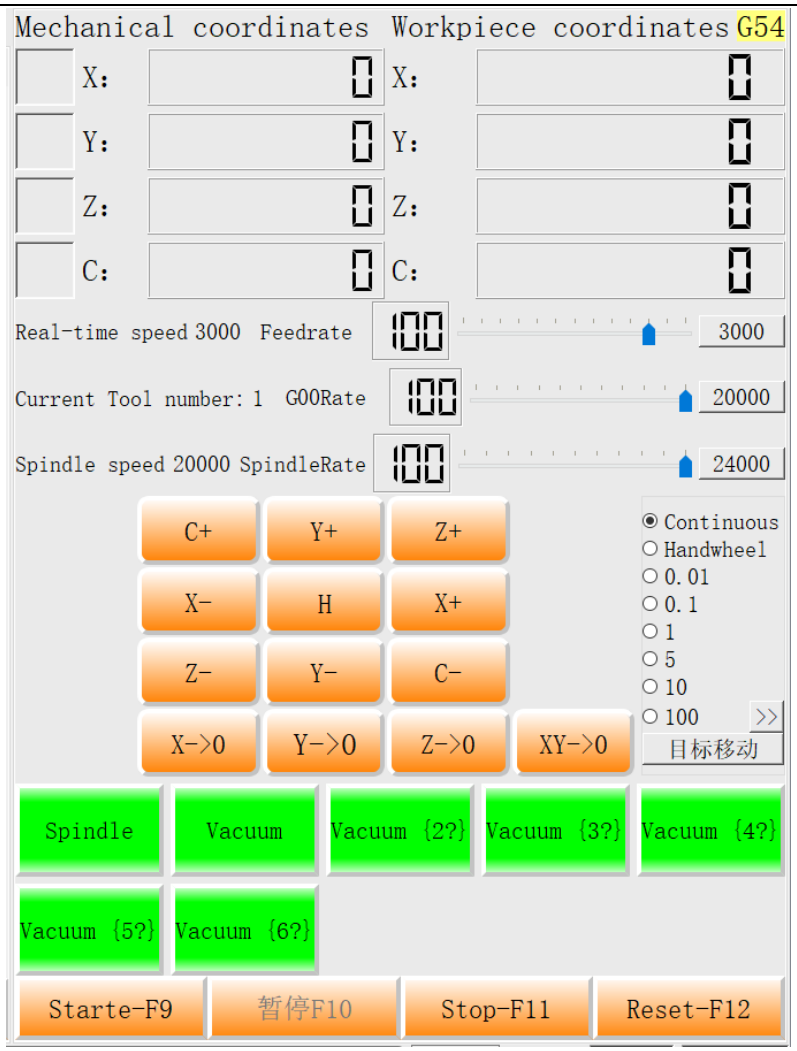


Figure 5.3.5-1

You can view the current speed, spindle speed and tool number on this screen. Move the slider to adjust the feedrate override, G00 override and spindle override. The machining rate can also be controlled in real time by the sliding feedrate override slider during machining. The progress of the process can be viewed in the progress bar. Click [Start] 

The file will be processed from the beginning; click [Pause] 

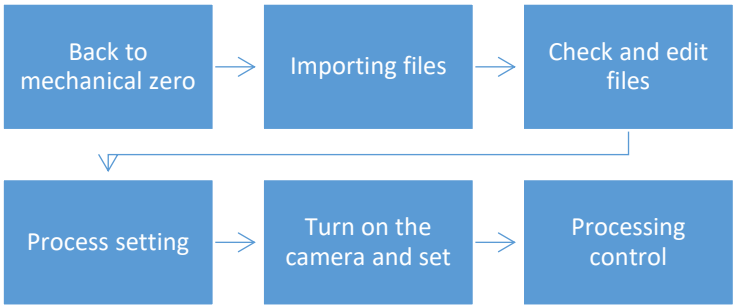
Start machining from where the machining was stopped last time; click [Stop] 

The software immediately stops the current processing; when an alarm

occurs and the problem is solved, click [Reset] Reset-F12 Ability to clear alarm flags.

5.4 *Vibrating knife processing*

CNC-Router software supports machine tool mounting vibration cutting equipment for cutting soft materials. Same as when using spindle cutting, choose whether to use CCD correction function according to your needs.



5.4.1 Back to mechanical zero

To return to the mechanical zero point, you need to set the axis direction, pulse equivalent and coarse positioning direction first. Refer to 6.1 axis parameters for details.

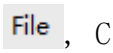
Open the software, open the [Operation] column of the software menu bar, and then click the machine [all zeros], the machine starts mechanical zero return, waiting for the zero return. After confirming that the machine origin is successful, the mechanical zero return completion flag will appear in the front axle return status bar of the corresponding axis.  .

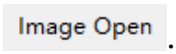
When the user guarantees that the mechanical coordinate of the current position is consistent with the actual machine coordinate of the machine tool, for example, the machine tool has not been closed, and no abnormal situation such as sudden stop occurs, click [Direct Settings] in [Operation] in the menu bar to directly use Current mechanical coordinate system. When this operation is completed, the

mechanical zero return status flag will also appear on the machine control bar mechanical zero return status bar.  .

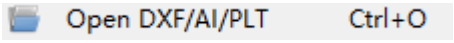
5.4.2 Importing files

The CNC_Couter system supports image editing and can import .JPG and .TIF files directly. After importing the file, the software will automatically extract the outline of the graphic and display it in the display interface.

Click [File] on the menu bar. 

, Click [Image Format Open] in the pop-up dialog box. 

. After that, the file selection interface will pop up, open the folder location of the file, double-click the file, and the software automatically recognizes the graphic outline, that is, the processing path, and projects it in the display window.

In addition to being able to import JPG files, users can also open edited files to improve productivity. Click [Open] 

A dialog box pops up and selects a folder. Select the file that was edited and saved in the past, and the content of the file will be displayed in the window, and the user will adjust it and then process it.

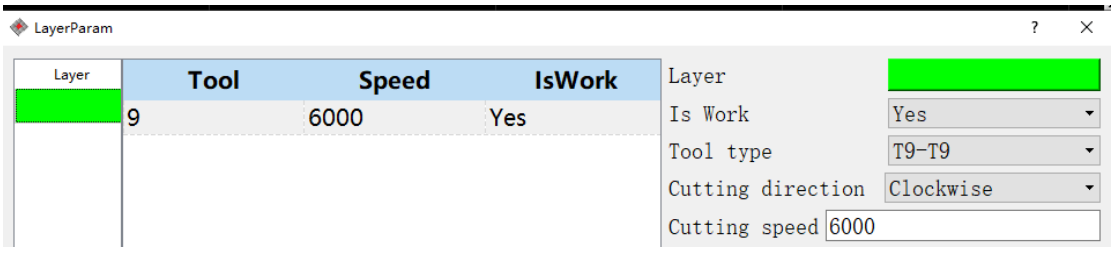
If the user has generated contours, the exported file is in DXF or PLT format. You can use DXF or PLT format files for processing. Click [Open DXF/AI/PLT] in the [File] menu bar to open this type of file. The file can be displayed in the software by opening the file in the pop-up dialog box, and the user can edit and process the file.

5.4.3 Check and edit files

This part mainly involves three tools: drawing tool, layer selection interface, and processing selection. Drawing tools can be used to draw straight lines , arc 、circle 、Polyline , rectangle ; The layer selection interface can adjust the processing parameters of each layer to meet the requirements of processing multiple different parameters at one time. CNC-Couter provides a total of 7 layers for users to choose. Mark points are selected in the graphic display window. When using the CCD patrol edge processing function, the user needs to mark the Mark point before it can be processed normally. The user can select an image outline in the display window, and then click the layer selection window on the left side to set the layer to which the element belongs.

5.4.4 Process setting

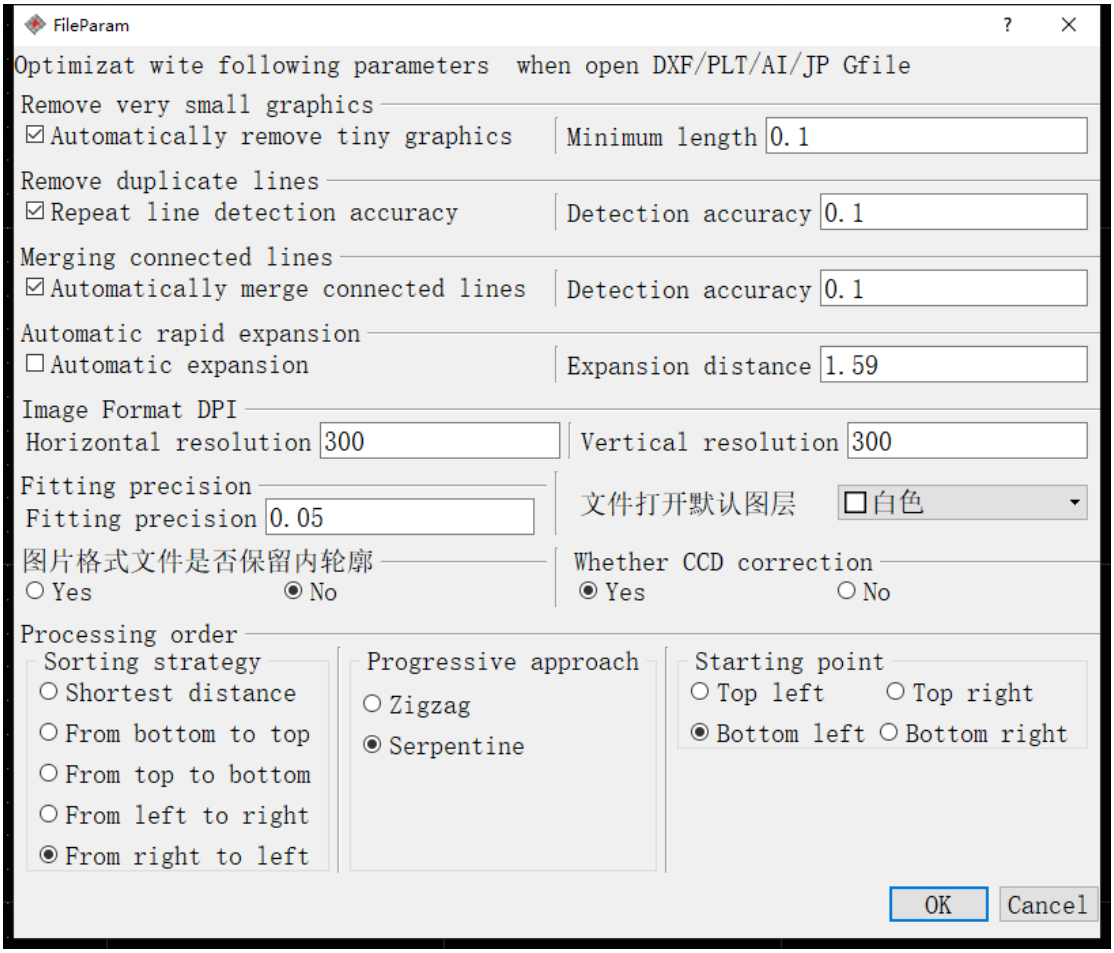
When the user needs to use the vibrating knife, adjust the [Tool Type] in [Layer Parameters], and click [Layer Parameters] to display the dialog box shown below:



Change the [tool type] to the tool number corresponding to the vibrating knife. In the manufacturer default parameter, the tool number corresponding to the vibrating knife is one bit larger than the maximum tool number of the spindle tool magazine. If the tool

magazine capacity of the machine is 4, the vibrating knife corresponds to the tool number. For T5.

To perform CCD processing, you need to turn on the CCD correction function. Click [File Parameters] at the bottom of the [Graphics Display] page, and the dialog box shown below will pop up:



Click [Do you want CCD correction] to [Yes]. At this time, CCD correction will be performed before processing; if the user does not need to perform CCD correction during processing, click [No].

When processing multiple products, the software will automatically sort after the user has set the layer to be processed. If the user needs to change the order of processing, you can click [Manual Sort]. After clicking, you will have one more circled number symbol next to the mouse. The starting number is 1, then click

on the element you want to process the first. At this point, the workpiece becomes the first workpiece to be machined, and the number in the circle next to the mouse will change to 2. At this point, click on the next element to be processed. The processing order of the element is the second. By analogy, the ordering of all primitive processing sequences can be completed. During the reordering process, the software will automatically plan the machining path according to the rules set by the user, and indicate the machining empty travel with white lines.

Click [Start Point Selection] in the processing selection column.  , You can customize the workpiece from that point to meet the needs of more situations.

5.4.5 Turn on the camera and set

Ignore this step if the user does not need to perform CCD correction during processing.

In the CCD processing mode, the user needs to set the camera. Click on the visual bar in the window selection bar. The position is shown in Figure 5.3.5-1:

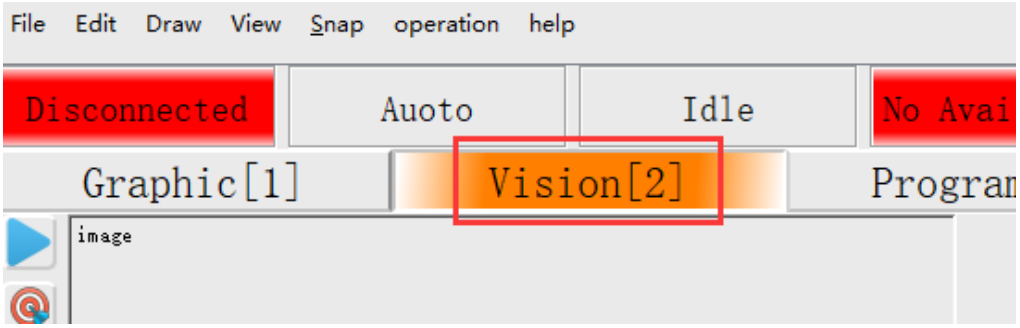


图 5.4.5-1

Click [Open Camera] OpenCameraF3, The camera starts up and the image will be displayed to the display window. The first use of the

camera screen may be rotated, the screen display moving direction does not match the actual moving direction, etc., and the screen is added to rotate and flip the screen. Click [Template Add]; as shown in Figure 5.3.5-3:

Figure 5.4.5-3

The user makes the display normal by selecting the rotation angle under the [Camera Settings] column. By checking whether the horizontal flip and the vertical flip, the problem that the moving direction of the screen display does not match the actual direction during the movement process is solved.

After the camera screen is set, use the manual movement window of the machine console, and the user can also use the numeric keyboard shortcut keys corresponding to each button to move. As shown in Figure 5.3.5-2：

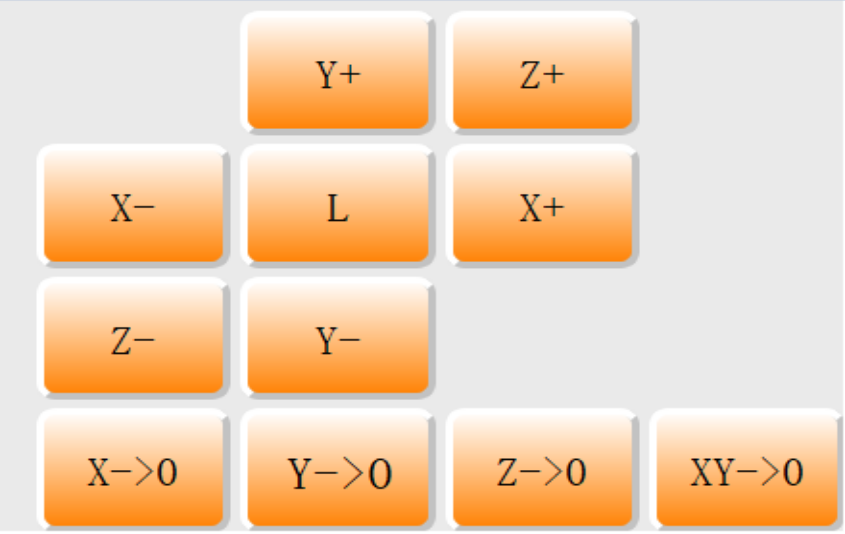


Figure5.3.5-2

To move the X and Y axes, the user needs to point the camera at the Mark point in the lower left corner of the processing file (the system defaults to the lower left corner, the user can adjust according to the requirements), and move the camera position so that the Mark point appears in the display window. Click on the bottom right corner of the page [CCD Processing] CCDStartF1, At this point, the system starts automatic patrol positioning, and starts processing after positioning is completed.

The appeal method uses the Mark point identification template as the system's own template. The CNC-Router system provides users with autonomous template addition function, and users can add templates themselves to increase adaptability. Click [Template Add] Select Template The template add operation page will appear, and the template will be selected. The template can then be framed. The template setting interface awakens the shape selection and name setting, and can set the gray value and the precision value as needed.

Then test, click [test] Test, Perform a template test. Check to see if the camera can find Mark points based on the template we selected. If you can find the Mark point to indicate that the template is available, click [Save] Save, Save the template for direct use in the next processing.

After the first processing, the software will automatically record the first Mark point position as the recognition starting point. If the processing material position has not moved, the user can click [Move Start Point]. Move Start Point Quickly return to the recognition starting point, the system automatically moves the camera to the starting Mark point position, without the user manually moving the camera again, improving processing efficiency.

5.4.6 Processing control

The CNC-Couter control system can view the position information of each axis during the machining process and perform the feedrate override in real time to view the real-time speed of the movement. Make the user understand the processing status, and facilitate the user to respond to sudden changes in real time.

Below the user can view the processing progress and present it to the user in the form of a progress bar. The control interface is shown in Figure 5.3.6-1.

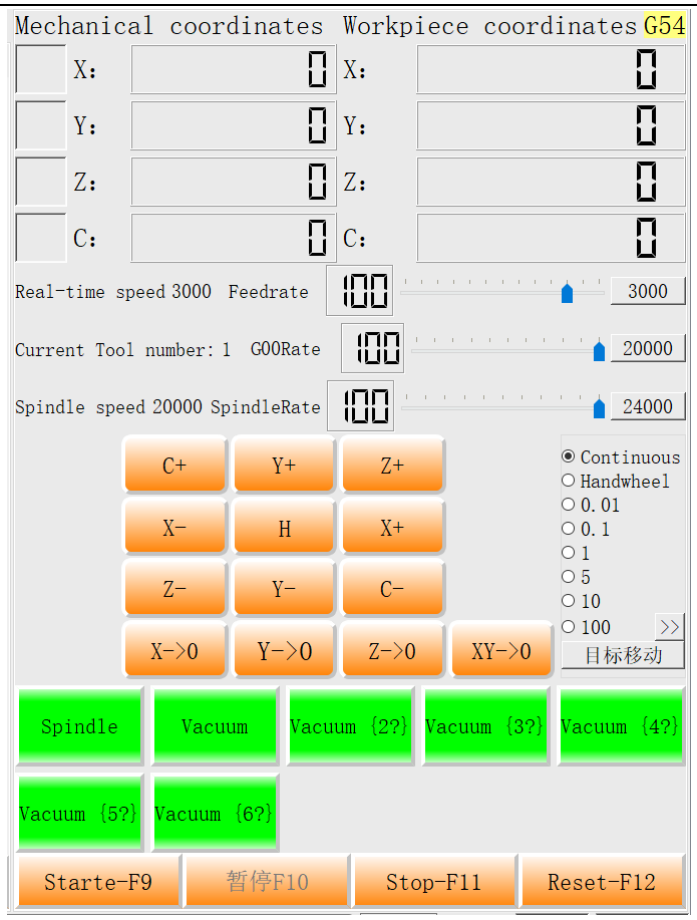


Figure 5.4.6-1

可 To view the current processing speed on this interface. Move the slider to adjust the feedrate override, G00 magnification. The machining rate can also be controlled in real time by the sliding feedrate override slider during machining. The progress of the process can be viewed in the progress bar. Click [Start] Starte-F9 The file will be processed from the beginning; click [Pause] 暂停F10 Start machining from where the machining was stopped last time; click [Stop] Stop-F11 The software immediately stops the current processing; when an alarm occurs and the problem is solved, click [Reset] Reset-F12 Able to sign.

Chapter Six Detailed softwareDetailed software

In this chapter, the modules of the CNC-Router software are explained in detail, and the corresponding functions that can be completed will also be explained.

6.1 Menu

The menu bar can use most of the functions of the software, edit files, set processing technology, draw graphics, machine control, etc..

6.1.1File

Click on the file in the menu bar and the file operation tab will appear, as shown in question 6.1-1:

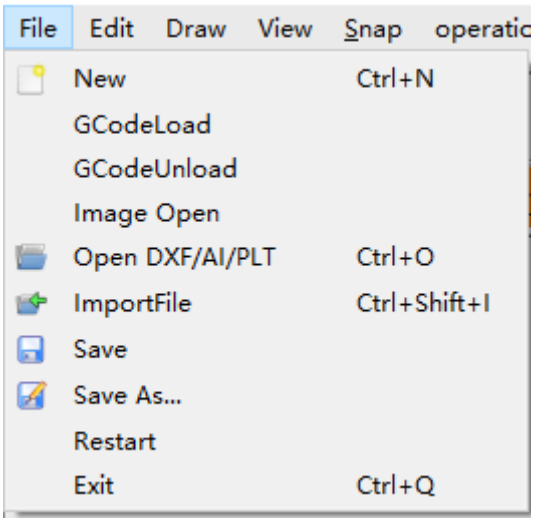


Figure 6.1-1

In this column, you can perform various operations on files.

6.1.1.1 G code file loading and unloading

CNC-Router software supports G code file processing, click [G code loading] GCodeLoad Options can load files. Then pop up the file selection interface, find the road strength where the G code

file is located, and double-click the file to load the file. You can view and edit the file in the program window as shown in Figure 6.1-2:

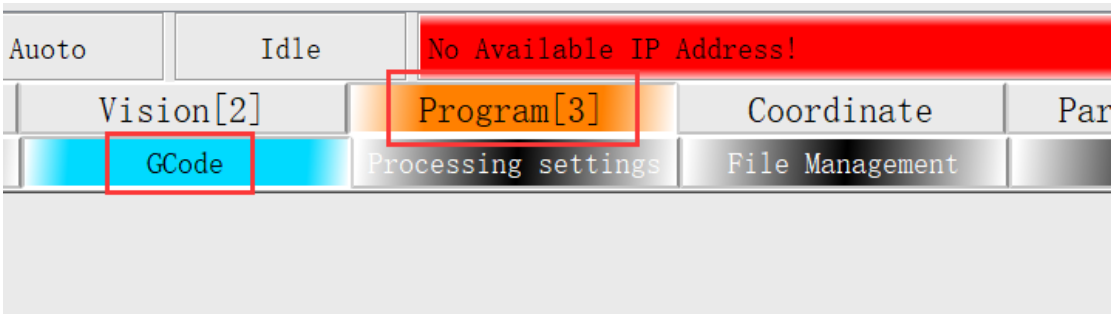


Figure 6.1-2

After processing, if you need to replace the G code file, you can click [G code file loading] GCodeLoad Overwrite the loaded file, click [Load File] at the bottom of the program window. Load-F1, Can also achieve this effect.

When you need to make a simple edit of the G code file, you can click [Edit] EditFileF5, The editing interface pops up and is edited in this window. Please pay attention to save after editing.

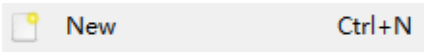
[G code file uninstall] GCodeUnload Can uninstall the currently loaded NC file, click this button to uninstall, or click [Uninstall File] at the bottom of the program window. UnLoad-F2, Can also achieve the purpose of unloading files.

6.1.1.2 Image format file opens

[Image open] Image Open The option is to open the image format file. Common include JPG, DIF files. The user can directly open the modified type file, and then click [image open] Image Open, The file selection dialog will pop up, find the image format file we need

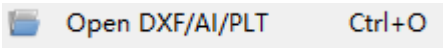
to open, and double-click to open it. In the open project, the soft drop will automatically extract the outline of the graph, and the progress of the file can be seen in the progress bar below. Waiting for loading, the extracted outline of the graphic will be displayed in the display area, allowing the user to edit and other operations..

6.1.1.3 New

[New] 

The option can recreate a blank file for the user to edit. Click to create a new file. This function can be implemented using the shortcut key Ctrl+N.

6.1.1.Open

[Open DXF/AI/PLT] 

The option is to open the .DXF file. This can be done using the shortcut Ctrl+O.

After clicking, the file selection box will pop up, find the file that needs to be opened, click, and wait for it to view the file in the preview area on the right. As shown in Figure 6.1-3:

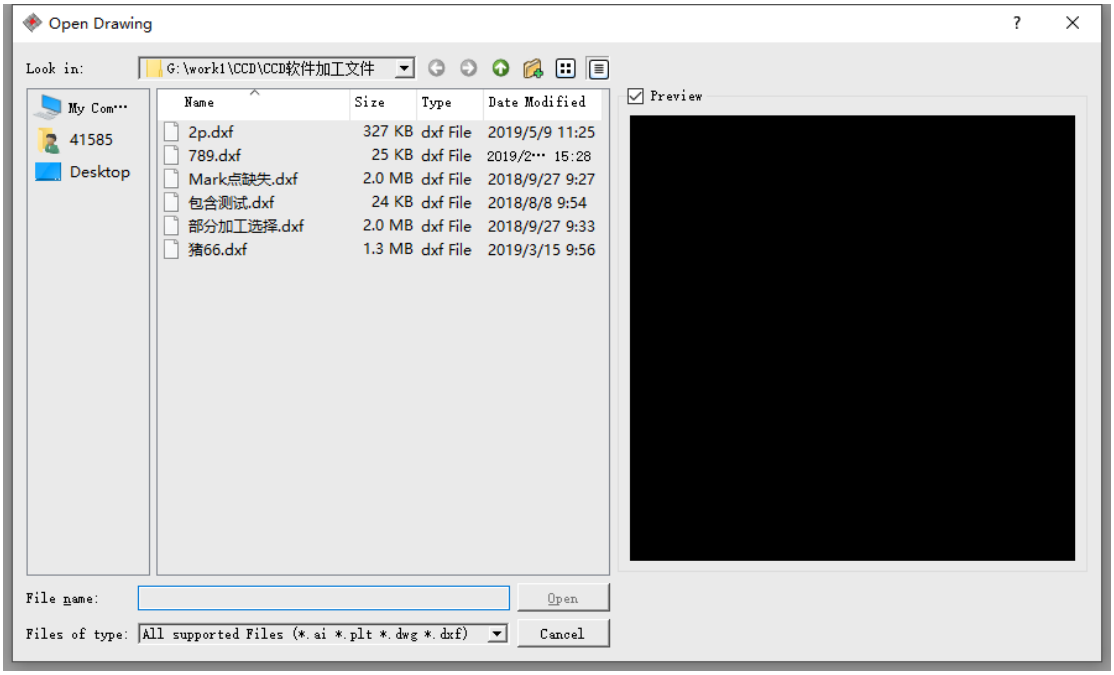
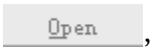


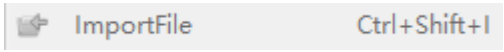
Figure6.1-3

After confirming that the file is correct, click [Open]



, Ability to load files into the drawing window for editing. In this option, you can open previously saved files, reduce the workload when re-processing the same file, and improve work efficiency.

6.1.1.5 Input

[Input] 

The option is to load the product information of other files into the current file to combine the two files and the purpose of one file. You can quickly use this feature with the shortcut Ctrl+Shift+I.

After clicking, the file selection dialog will pop up. The window style is similar to the window style when opening a file. Select the file to be imported, click, wait a little, preview the file in the preview display box on the right, confirm the file is correct and click [Open] below. After a short wait, the file was imported successfully.

Next, move the mouse to select the position you want to place, and you can see the position of the graph in real time during the move. After the location is selected, click the left mouse button to place it at the specified location. At this point, the image is still attached to the file mouse. If you need to continue importing the image, then select a new location to place the second set of images; if you do not need to continue to place. Click the right mouse button or use the keyboard ESC key to exit the file import mode.

6.1.1.6 File Save

[Save】 

and[Save as] 

Used to save a completed file that has been edited now. The saved format is DXF format.

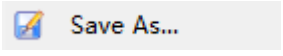
When the file to be saved is the first time edit, click [Save] 

, A file selection dialog will pop up, and the selected folder is the destination folder that the file wants to save. Look in this directory when you need to open the file later. At the bottom of the window is a file name window, in which you can customize the name of the file you want to save, you can name the file name by product name or code, which is convenient for future file management.

If the open file is not edited for the first time, click [Save] 

, The file will be saved. And the file name and save path will not change.

[Save as] 

Unlike saving, this function copies and saves the file that is currently being edited to a folder selected by another user without changing the file being operated. If you now add a picture element to the A file, click [Save As] 

, Save it in another folder and name it B, then close the software. There is this line in the B file at this time and there is no such line in the A file.

After clicking [Save As], the file selection dialog box will pop up to select the save path, and the file name can also be named in the file name window below. In the future, you need to find the file in the secondary file path.

6.1.1.7 Exit

【Exit】 Exit Ctrl+Q Can exit the software and click to launch. This can be done using the shortcut Ctrl+Q. If the edited file is not saved, a prompt dialog will pop up asking if you want to save it. As shown in Figure 6.1-4:

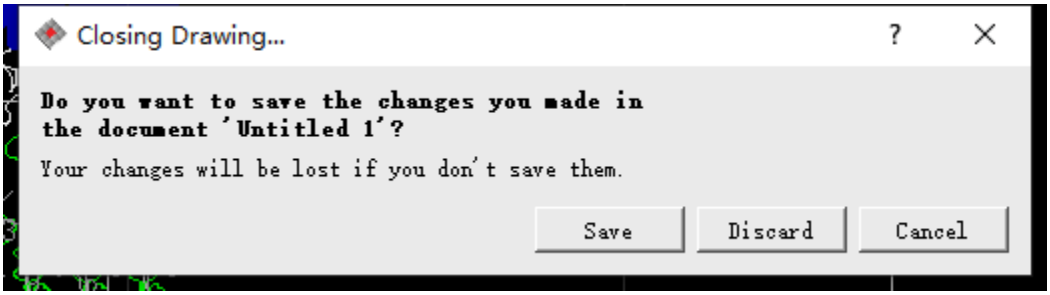


Figure 6.1-4

If you need to save, please click [Save] Save, Save it. The steps are the same as the save. The software will automatically exit after saving. If the file does not need to be saved, please click [Discard] Discard. The software will quit. If; you are delayed, you can click [Cancel] Cancel, Cancel the exit operation. The software will return and proceed.

6.1.2 Edit

Click [Edit] on the menu bar. 编辑(E) , The edit menu will pop up, as shown in Figure 6.1-5.:

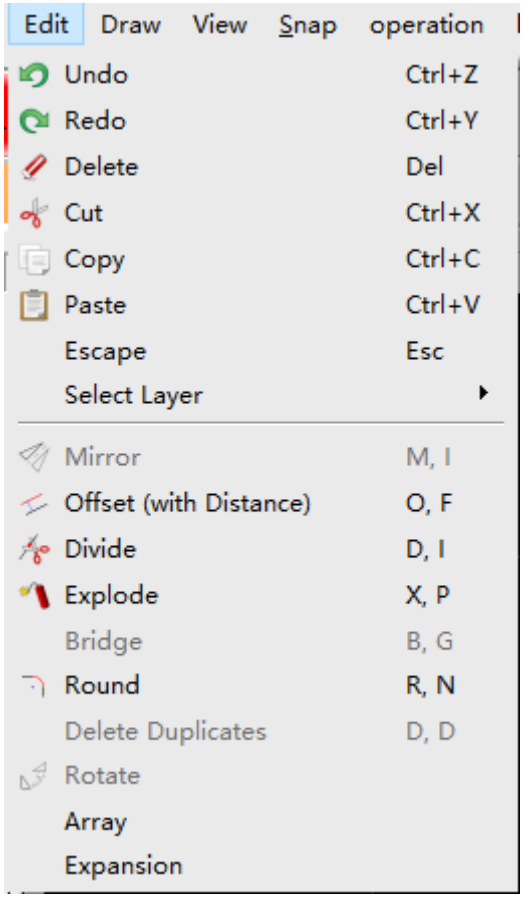


Figure6.1-5

6.1.2.1 Undo and redo

【Undo】 Undo Ctrl+Z Can cancel the previous step, make the file restore, you can use the shortcut key Ctrl+Z to achieve the function;

【Redo】 Redo Ctrl+Y Contrary to revocation, redoing is done once again with the steps of the previous revocation, so that after the error is revoked, the file can be returned to the

original state more quickly. You can also do this with the shortcut Ctrl+Y.

6.1.2.2 Delete

【Delete】 

Ability to remove unwanted parts.

When you want to remove some elements, you can use the mouse to click the selection or frame selection action to select the unwanted part, then click [Delete] in the Edit menu.

, That is, all selected elements can be deleted. You can also select the part you want to delete and use the shortcut button to delete the part. Can be revoked if the misoperation is deleted.

6.1.2.3 Cut

【Cut】 

Ability to crop selected features and move to the target location. You can also use the shortcut Ctrl+X to use this feature.

To use this feature, first select the image and click Cut in the Edit menu to crop the image. Therefore, the image just selected on the drawing will disappear. At this point, click [Paste] in the edit.

, Or use the shortcut Ctrl+V to paste the image. You can find that the primitive that has just been cropped is attached around the mouse, and the element will follow the mouse movement position. Move to the location you want to place and click the left mouse button to place the element. You can place as many object files as you want after cutting. After the cut operation is

completed, click the right mouse button or ESC to exit the cut operation and proceed to the next work.

It is worth noting that the cut operation can be performed in two different files. For example, in the file A, select the element and perform the cut operation. Close file A, open file B, and paste the file cut from the A file into file B.

6.1.2.4 Copy

[Copy] 

The selected images can be copied and placed in different locations as needed. This function can be used with the shortcut key Ctrl+C.

To use this function the same as cutting, you need to first select the file you want to copy, then click Copy in the Edit menu, and the image copy is complete. Click [Paste] in the Edit menu. 

Or use the quick key Ctrl+V to paste the image. The image that you want to copy appears in the display area and attaches to the mouse. The display element will follow the mouse movement, move to the position you want to place, and click the left mouse button to place the image. When placing the image, it is the same as the cut. We can place multiple groups until we no longer need to place it. Click the right mouse button or press the keyboard ESC to exit the copy operation and continue working. Copy and cut, can operate between two files.

6.1.2.5 Paste

[Paste] 

Used to transfer copied, cut images to the current work interface. Ability to use the shortcut Ctrl+V for quick operation.

6.1.2.6 Cancel

[Cancel] Escape Esc It is used when you click the function button again and you need to exit the function. The shortcut button is Esc.

Click to exit the current function, mouse recovery select initial function.

6.1.2.7 Layer selection

[Layer selection] Select Layer Used to select all the lines of the target layer. After clicking, the layer selection window will pop up, as shown in Figure 6.1-6:

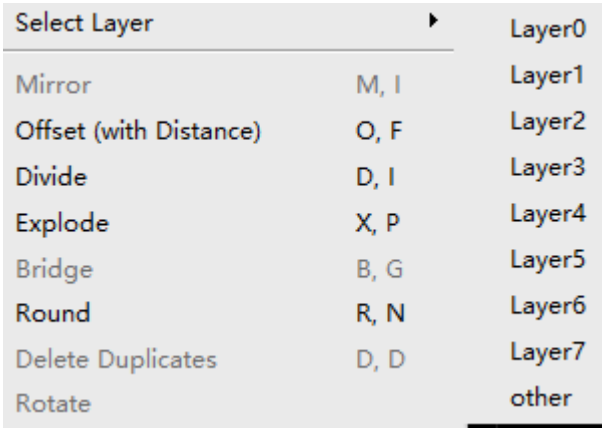


Figure6.1-6

Select the layer according to your needs. After clicking, all the lines of the layer are selected, you can continue to operate.

6.1.2.8 Mirror

[Mirror] Mirror M, I Ability to mirror selected images.

Select the graphic that needs to be mirrored, then click [Mirror] in the Edit menu. Mirror M, I . The mouse becomes a cross selection mode. Used to select a point. Because the processed

primitives are two-dimensional graphics, the mirroring operations of the primitives are determined relative to a straight line. Two points to determine a line, now click the left mouse button to complete the first point position setting, then there will be a line segment connected to the mouse from the first point, and a new image after mirroring according to this line segment. Mirror processing is based on this line segment, moving the mouse, you can change the angle and length of the line segment, and then the graphics after the mirror will change in the process of moving. After getting the desired graphic, click the left mouse button again, and the dialog box shown in Figure 6.1-7 will pop up:

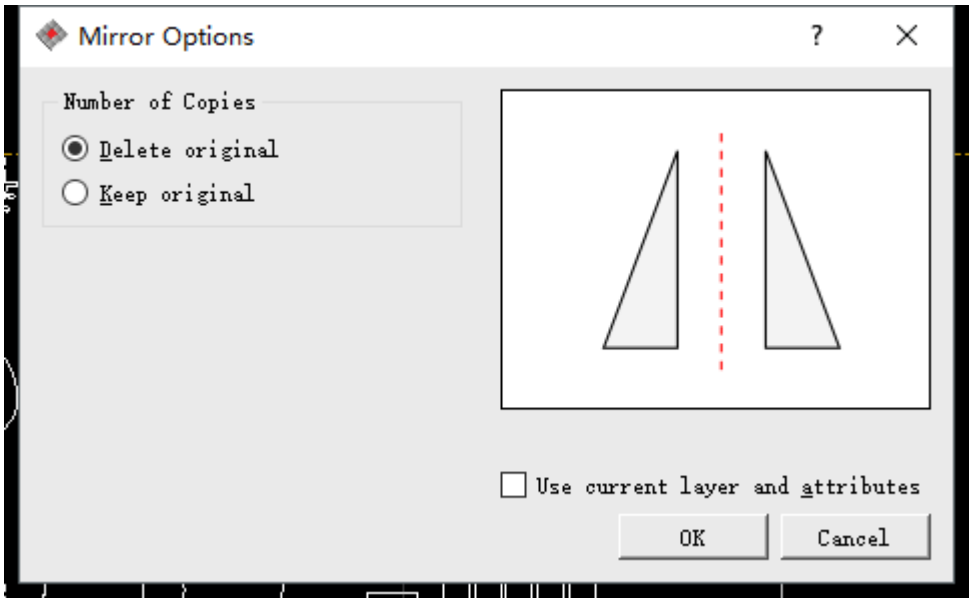


Figure6.1-7

In the left selection, you can choose whether to delete the original image.[Delete original]Indicates that the original image is deleted, and the original image is deleted, that is, only the image after the image is retained.[Keep origianl] Indicates that the original image is retained, and the two primitives before and after the image are retained.

There is a checkbox option: [Use current layer and attributes]. If this option is checked, the file attributes after mirroring will use the file attributes of the layer where the original graphic is located.

After checking the settings, click [OK] OK , The mirror operation is completed and the editing interface is returned. If you are not satisfied with the mirror result, click [Cancel] Cancel , Cancel this mirroring operation and return to the editing interface.

6.1.2.9 Isometric copy

Click [Isometric Copy] Offset (with Distance) O, F Ability to turn on isometric copying. Ability to perform isometric copying of direct and circular lines, etc.

Click to enter the isometric copy function. At this time, two data frames will be added to the upper part of the display area, as shown in Figure 6.1-8.:

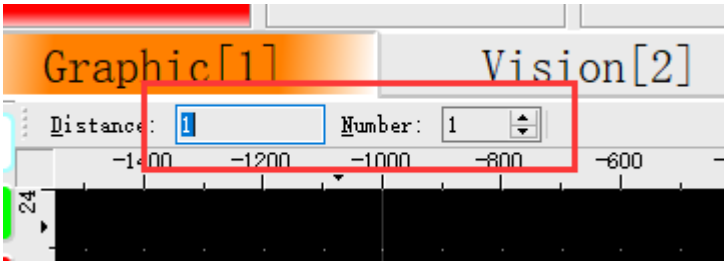


Figure6.1-8

The first data frame Distance represents the distance between the target and the secondary target; the second data frame Number represents the number of copies that need to be copied at a time. Adjust these two parameters to control the distance and copy number of the isometric copy, place the mouse on the target to be copied, and the software will display the copied target. Click the left mouse

button to confirm the copy, and the equidistance assist is completed. Click Esc to return to the mouse initial state.

6.1.2.10 Interrupt

[Interrupt] 

Can be used to break straight and curved segments. When the user needs to separate a part of an element, the interrupt function can be used.

Click [Interrupt] and the mouse becomes a cross. Use the mouse to select the line segment you want to break. After clicking, if it is a straight line segment, a dotted line perpendicular to the straight line segment will appear. The position of the dotted line can be controlled by the mouse. The intersection of the dotted line and the line segment is the breaking point. If the curve segment needs to be broken, the dashed line is the perpendicular of the tangent to the point. The broken breakpoint is the intersection of the dotted line and the line segment.

6.1.2.11 Break up

[Break up] 

Click to break the target graphic. When there is a large overall figure, and the user only needs a part of the graphic, the break function can be used.

First use the mouse to select the file, then click on the break in the Edit menu, the graph is broken up. After breaking up, you can finish editing some of the lines.

6.1.2.12 Bridge position

[Bridge] 

Ability to set the bridge function for selected line segments and curve segments.

Click [Bridge] and the dialog box for the bridge setting will pop up. As shown in Figure 6.1-9:

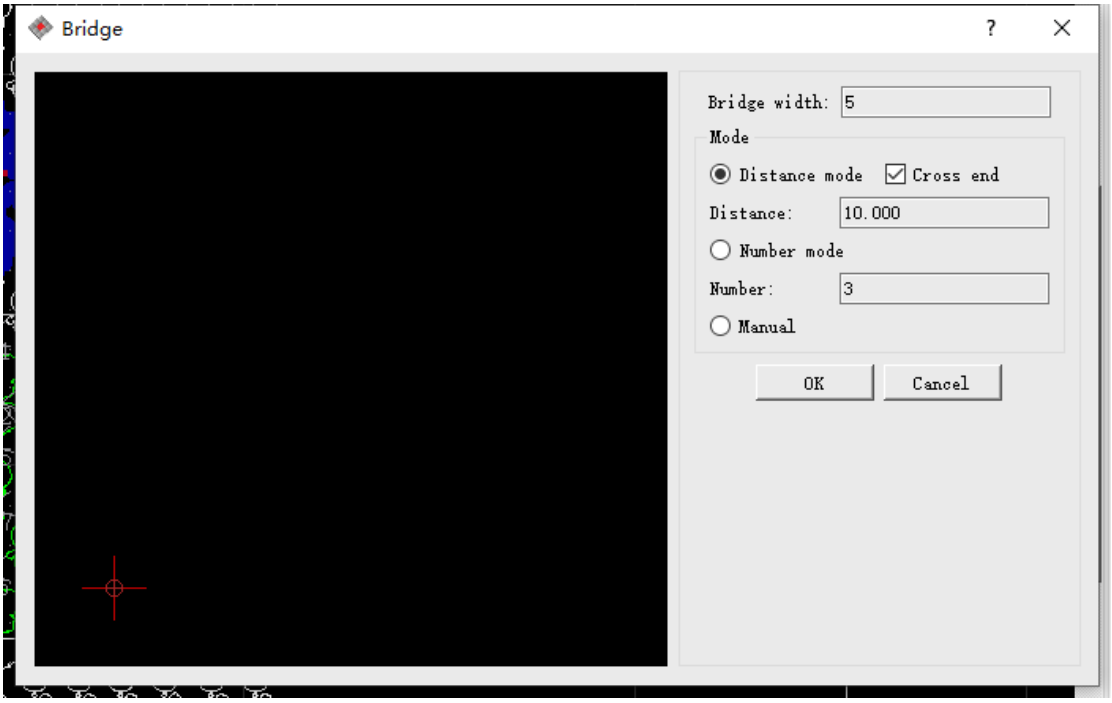


Figure6.1-9

The user can set the bridge position according to the parameters of the dialog by modifying it.

The bridge width indicates the gap width after the break. After setting the bridge width, you can select the interrupt mode. There are 3 modes for the bridge setting to choose from: distance mode, quantity mode, manual mode. After the setup is complete, the user can preview the effect in the view box on the left.

Distance mode: Click the circle on the left side of the distance mode to select the distance mode. In this mode, you can choose whether to cross the endpoint. When selected, if there is a breakpoint setting on the endpoint according to the distance, it will automatically skip. On the endpoint. The user can set the distance between two points in the distance data reading box according to the requirements.

Quantity mode: Click the circle to the left of the quantity mode to select the quantity mode. This mode means to set a specified number of breakpoints for the selected straight curve. The distance of each breakpoint is equidistant. The number of users can be in the lower frame. Settings.

Manual mode: Click the circle on the left side of the manual mode to select the manual mode. After selecting the bridge width, you can edit it in the view box on the left. After moving the mouse over the line, a circular box icon will appear, with the circle contained in it indicating that this is the path that will be eliminated. It can be eliminated with a mouse click, and the user can use the mouse wheel to zoom in the view. You can also use the shortcut key Ctrl+mouse wheel to move the interface up and down; the shortcut key Shift+mouse wheel to move the interface left and right.

After completing the settings, click [OK] below 

The processed graphics will be applied to the drawing; if you need to cancel the bridge operation, click [Cancel] 

.

6.1.2.13 Chamfer

Click [Chamfer] 

, Ability to enter the chamfer function. After clicking, the mouse will become a cross.

Click on an edge that requires chamfering, then move the mouse to the other side of the corner, and you can preview the image after the corner chamfer in the view box. The radius of the chamfer can be set at the top of the re-view frame, as shown in Figure 6.1-10:

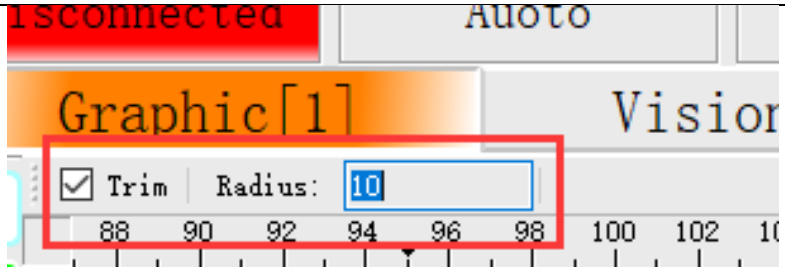


Figure6.1-10

Checking the Trim option means deleting the original corner, and unchecking it will not delete it. The user can select according to the requirements; the radius of the chamfered arc can be set in the Radius option box, and the filled data is the radius of the arc.

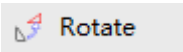
After the chamfering is completed, click the right mouse button to exit the chamfer function and perform another operation.

6.1.2.14 Delete repeat line

Click [Delete Repeat Line] 

Enter the delete repeat line function. When using the user, you must first select the area where you want to perform the deduplication operation. Click [Delete Repeat Line], and the repeat line is deleted.

6.1.2.15 Rotate

Click [Rotate] 

Enter the rotation function. The user needs to select the graphic that needs to be rotated first, then click the rotation of the editing menu. At this point the mouse becomes a cross and can select a point, which is the reference point for the rotation. Click the mouse to determine the position of the point. After the mouse click, the rotation setting window will pop up, as shown in Figure 6.1-11:

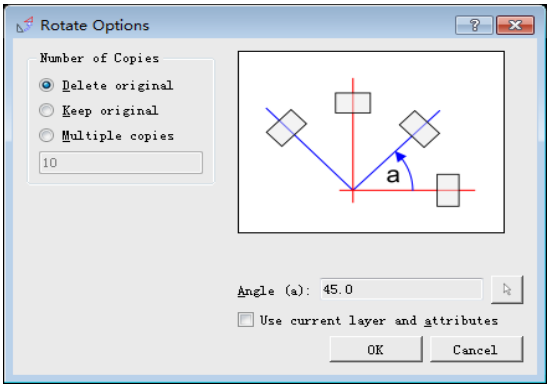


Figure6.1-10

The user can select the copy mode according to the number selection box on the left side: check [Delete original] to delete the graphic before the rotation and only retain the graphic after the rotation; check [Keep original] to retain Two graphics before and after rotation; [Multiple copies] means that the number of copies of the user settings is copied and all files including the source files are retained.

The user can set the angle of rotation in the Angle Settings box. The rotation direction defaults to counterclockwise. Checking "Use current layer properties" will make the graphic properties after rotation the same as before rotation. Users can choose the rotation angle and use layer properties according to their needs.

6.1.2.16 Array

Click [Array] on the Edit menu Array, Users can perform array processing on graphics. Select the graphics that need to be processed by the array, click to pop up the array setup window, as shown in Figure 6.1-11:

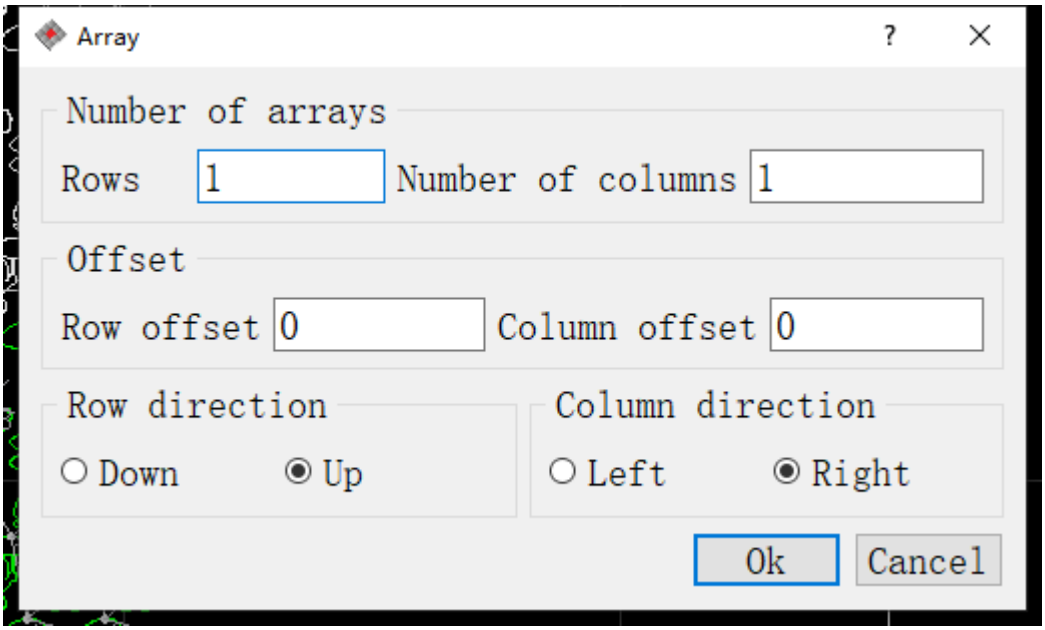


Figure6.1-11

The number of rows and columns represents the total number of rows and columns after array processing; the row offset represents the distance between two rows after array processing, and the column offset represents the distance between the two columns after array processing. When the value is 0, the graphics will be connected end to end; the row direction and column direction settings indicate that the array processed by the array is placed in the opposite position of the original graphics.

6.1.2.17 External expansion

Click [Extended Expansion] on the Edit menu Expansion Can enlarge or reduce the graphics. Tool interpolation can be implemented using this function.

Select the graphic to be operated, click the expansion and expansion in the edit menu, and the setting window will pop up, as shown in Figure 6.1-12.：

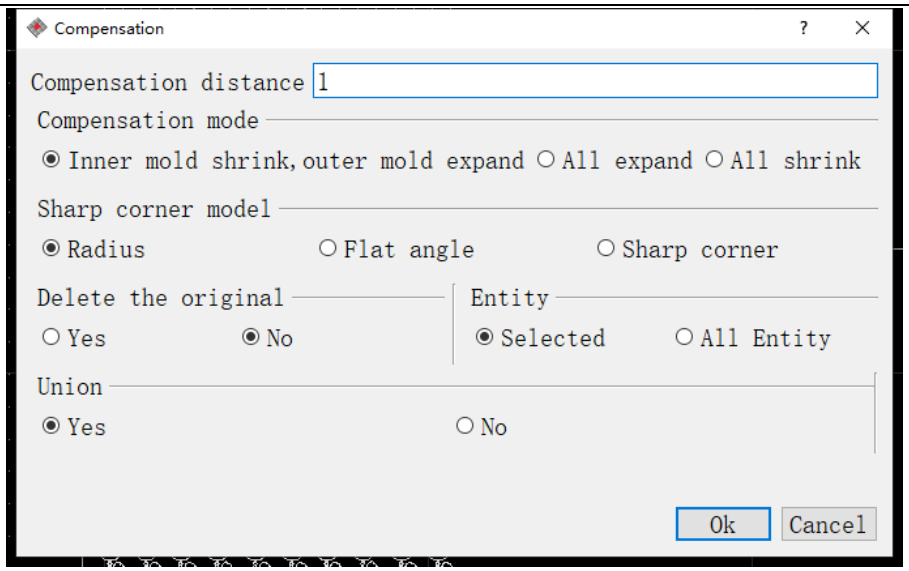


Figure 6.1-12

The offset distance indicates the distance between the graphic and the original graphic after the expansion and contraction; the expansion in the offset mode indicates that the original graphic is expanded to the outside, and the indentation indicates that the original graphic is reduced to the inside; when the graphic is scaled, if the graphic is With focus, the graphics angle is the same as the original. If there is no focus, the sharp angle mode of the graphic after processing can be specified according to the selection of the user's sharp corner mode; the user can select whether to delete the original image according to the demand by whether to delete the original file tab.

Users choose whether to check the compensation fusion according to their needs. The fusion is very large in the actual use of the user. The gap between the graphics drawn by the user may be less than twice the diameter of the tool. In this case, if the expansion without fusion is used, the processing road will overlap. The product will be destroyed during processing. Therefore, in this case, the fusion function should be enabled.

6.1.3 Draw

Click [Draw] on the menu bar Draw, The drawing menu will pop up, as shown in Figure 6.1-13:

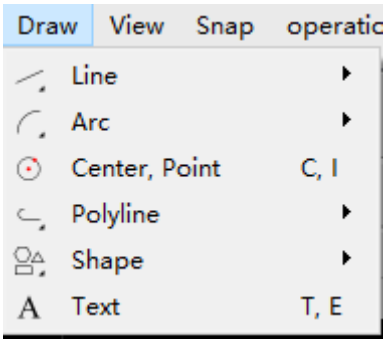


Figure 6.1-13

6.1.3.1 Straight line

Click [Line] The line drawing mode selection bar will pop up, including [two-point drawing line]、[angle drawing line]、[horizontal line]、[vertical line].

Click [two-dot line] Line from 2 Points L, I, Perform two-point line drawing operation, The mouse will change to a cross. Click on the first line to confirm the starting point of the line segment and click again to confirm the end point of the line segment.

Click [angle drawing line], When the angle is drawn, the mouse will change to a cross. Ability to preview the rendering effect and click the mouse to confirm placement in that location. At the top, you can set the angle, length and reference point of the line segment.

Click [horizontal line], With a horizontal line drawing operation, the mouse will change to a cross. Ability to preview the rendering effect and click the mouse to confirm placement in that

location. The line length and reference point can be set at the top. As shown in Figure 6.1-15:

Click [vertical line], With a vertical line drawing operation, the mouse will change to a cross. Ability to preview the rendering effect and click the mouse to confirm placement in that location. Same as horizontal line drawing, you can set the length of the line segment and the reference point above.

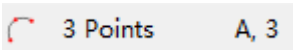
6.1.3.2 Arc

Click [Arc] 

The arc drawing mode selection box will pop up, including [two points and radius] and [three points].

Click [two points and radius] 

With two points and a radius for the arc, the mouse will change to a cross. The radius and direction of the arc can be set at the top of the drawing. After the arc radius is set, the mouse moves to the target point and clicks. This point is the starting point of the arc. Move the mouse, the arc will rotate, click to confirm the second point.

Click [three points] 

Perform a three-point arc operation and the mouse will change to a cross. Click the mouse to determine the starting point of the arc. The second click determines the middle point through which the arc passes, and the third click confirms the end point of the arc. During the drawing process, move the mouse to preview the drawing result, and then use the left mouse button to place the user's needs.

6.1.3.3 Circle

Click [circle] 

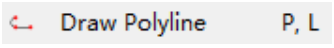
When you draw a circle, the mouse will change to a cross. At this time, the center coordinates

are selected, and the center position can be selected by mouse movement. The X and Y coordinates of the center of the circle can also be input through the coordinate input box above the drawing to complete the precise positioning. After setting the center of the circle, you can confirm the radius by the movement of the mouse. You can also enter the radius of the target circle through the radius input box above the drawing.

6.1.3.4 Multi-lin

Click [Multi-line] 

Multi-line drawing menu will pop up, including [Draw multi-line segments].

Click [Draw on the segment line] 

The mouse becomes a cross, starts drawing multiple lines, and clicks the mouse to determine each point. Ability to set multi-line on top of the drawing, as shown in Figure 6.1-16:

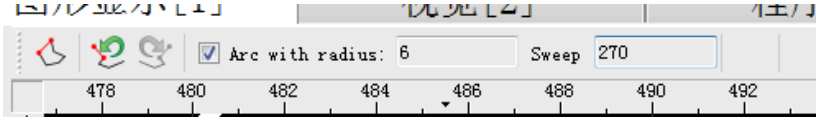


Figure 6.1-16

Icon  Indicates that the graphic is closed. Clicking on the function will directly lead a line segment from the end point to connect with the starting point to complete the graphic closure;  Indicates that the multi-line operation is revoked, the icon  Indicates to redo a multi-line operation; check [Arc with radius] to draw from a straight line to an arc, the radius of the arc is the value filled in by the user; the user can Use the mouse to confirm the center angle of the arc segment. You can also set the center angle of the arc by [Sweep] and fill in the data at the back. With

this operation, the user can perform accurate arc drawing. It is worth noting that the graph drawn using the multi-line function is a whole.

6.1.3.5 Shape

Click [shape] 

Ability to draw common graphics, pop-up lists include [Rectangle], [Rounded Rectangle].

Click [Rectangle] 

Draw a rectangle and the mouse becomes a cross, and you can draw a rectangle at this time. The user can check [Group] above the drawing, and the graphic drawn by this option will be a whole; uncheck the drawn graphic is composed of four straight lines, and can edit each straight line independently.

Click [Rounded Rectangular] 

Draw a rounded rectangle and the mouse becomes a cross, in which case you can draw a rounded rectangle. The mouse can be used to determine the starting point, then the preview rectangle clicks to confirm the diagonal point. When precise positioning is required, the coordinates of the X and Y axes can be entered in the text box above the illustration. When determining the second point, there will be [Group], [Wide], [High], and [Chamfer Radius] options above the drawing. The [Group] option works in the same way as the rectangle; [Wide] and [High] are the width and height of the rectangle; [Chamfer Radius] is the radius of the four corner arcs of the rounded rectangle, and the user adjusts according to his own needs.

6.1.3.6 Text

Click [Text] A Text T, E Can enter the text editing interface, as shown in Figure 6.1-17:

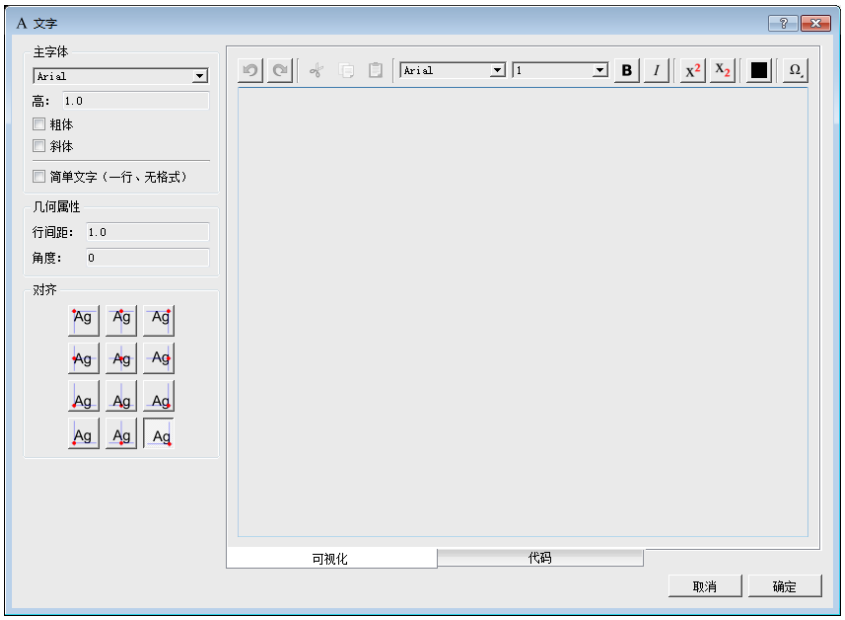


Figure6.1-17

The user can preview the text editing effect in the right view box..

In the [Main Font] selection box, you can select the font; [High] text edit box input can set the size of the text; [Bold], [Italic] is used for text bolding and tilting, check to set; Text] is used to set the profile editing mode, in which the font and font size cannot be set.

In the [Geometry Attributes] column, the line spacing of the edited text and the tilt angle of the text can be set in [Line Spacing] and [Angle].

[Alignment] The selection box provides the user with a variety of alignment methods, which can be selected according to their own.

There is a row of logos at the top of the text edit box: [Retract], [Redo], [Cut], [Copy], [Paste], [Font Selection Bar],

[Font Size Selection Bar], [Bold 】, [Tilt], [Reduce Up Aligned Writing], [Zoom Down Aligned Writing], [Font Color], [Mathematical Symbol]. Click the icon and the user can perform the corresponding operation.

If you need to abandon the text editor, click [Cancel]; if you want to complete the editing, you need to click [OK]. The drawing will be returned and the edited text will be placed using the mouse..

6.1.4 View

Click [View] in the menu bar, the view tab appears, as shown in Figure 6.1-18:

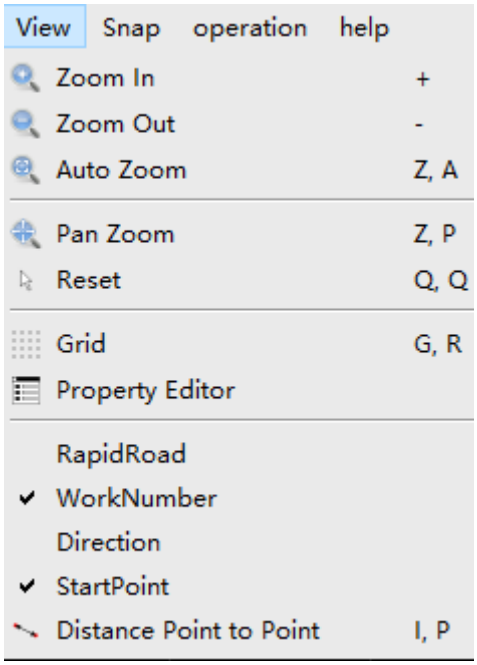


Figure6.1-18

6.1.4.1 Drawing view

By [zoom in] 

, [Zoom out] 

, 【automatic】 

Zooming on the drawing, you can also use the mouse wheel or "+", "-" for quick operation. [Automatic] It is judged by the software to adjust the zoom level of the drawing. By [drag the canvas] 

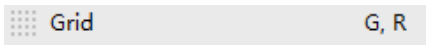
Function to drag and drop drawings, view other positions of the drawing.

6.1.4.2 Reset

[Reset] 

Used to cancel the mouse to select the function to return to the initial cursor state. The right mouse button and Esc can also do this.

6.1.4.3 Grid

[Grid] 

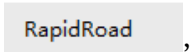
Used to select whether to display the grid, click to switch.

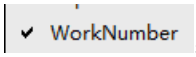
6.1.4.4 Attribute editor

[Property Editor] 

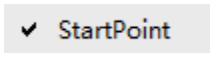
Clicking will open the Attribute Editor, which is located on the right side of the drawing. In this editor, users can view graphical properties and modify some properties.

6.1.4.5 Drawing display selection

[Processing empty path] 

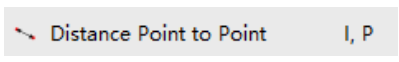
, [processing serial number] 

, [machining direction] 

, 【starting point】 

Four options by clicking to check whether to display machining empty, machining serial number, machining direction, starting point.

6.1.4.6 Two points distance

[two points distance] 

Ability to measure the distance between two specified points. Click the mouse to change to a cross shape, move the mouse to the point to be measured, click, the distance between the point and the position of the mouse

will be displayed dynamically. Click another point, the distance information will be fixed.

6.1.5 Snap

Click [Snap] on the menu bar Snap, The capture menu will pop up, as shown in Figure 5.1-19:

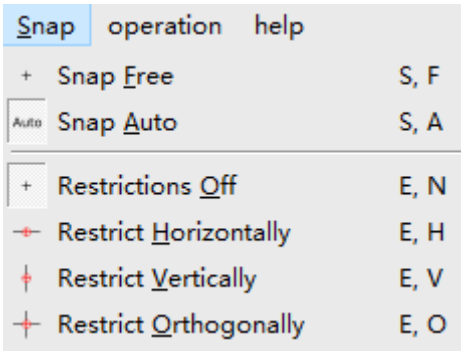


Figure6.1-19

The user will be able to select the capture method. Capture mouse-assisted positioning for drawing. Users can choose whether to enable capture and capture.

6.1.5.1 Snap Freed

[Snap freed] Snap Free S, F Click to select the release mode. After selecting this mode, the mouse will no longer automatically capture the anchor point.

6.1.5.2 Snap Auto

[Snap Auto] Snap Auto S, A Clicking will turn on automatic capture, in which the capture function is turned on and the drawing coordinates can be captured. Turn on assisted positioning for easy positioning when drawing graphics.

6.1.5.3 Restrictions off

[Restrictions off] Restrictions Off E, N Click to turn off strict capture, this feature is used for normal movement without

mouse restriction when horizontal, vertical, or quadrature capture is not required.

6.1.5.4 Restrict Horizontally

[Restrict Horizontally] 

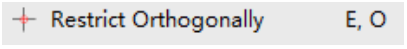
Click to turn on the horizontal capture function. After turning on this function, the user first confirms a point, and the subsequent points are all limited to the same horizontal position as the point.

6.1.5.5 Restrict Vertically

[Restrict Vertically] 

Click to turn on the vertical capture feature. After turning this feature on, the user first confirms a point, and the subsequent points are all limited to the same vertical position as the point.

6.1.5.6 Restrict Orthogonally

[Restrict Orthogonally] 

Click to open the quadrature capture function. After turning this feature on, the user first confirms a point, and the subsequent points are all limited to the same horizontal or vertical position as the point.

6.1.6 Operating

Click [Operation] on the menu bar operation The operation menu can be opened, as shown in Figure 5.1-20:

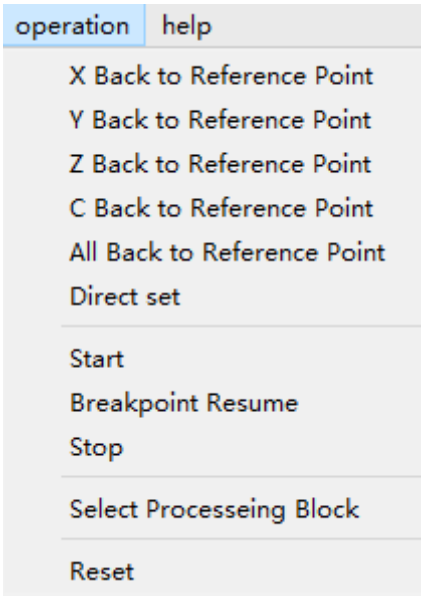


Figure6.1-20

6.1.6.1 Zero return operation

Click [X Back to Reference Point] X Back to Reference Point, [Y Back to Reference Pint] Y Back to Reference Point, [Z Back to Reference Point] Z Back to Reference Point, [All Back to Reference Point] All Back to Reference Point It is possible to complete the mechanical zero return of the corresponding axis and determine the mechanical origin. You can also perform the same action by clicking [X Zero], [Y Zero], [Z Zero], [All Zeros F9] in the [Reference Point] column on the right machine control bar.

6.1.6.2 Processing operation

After the mechanical return to zero, processing is possible. Click [Start] Start, The machine will start machining. During the processing, click [Stop] Stop Can stop processing. If the last processing is not completed, click [Breakpoint Resume] Breakpoint Resume It is possible to continue processing from the last processing breakpoint. If the processing is completed, it will prompt that the last processing is completed normally.

The user can select a section of the file for processing. Click [Select Processing Block] Select Processeing Block, The software pops up the advanced processing selection interface, as shown in Figure 5.1-21:

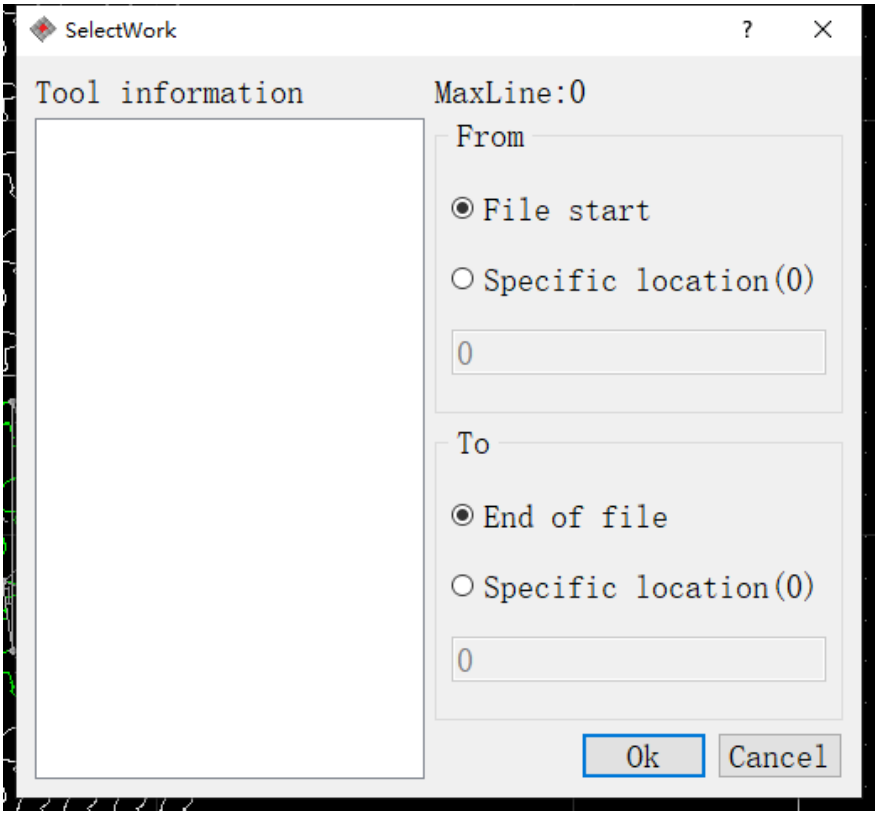


Figure 6.1-21

The user can select the start line of the file, enter the start line number in the text box, or select the end line of the process and enter it in the text dialog box. Click OK to complete the setup.

6.1.6.3 Reset

The software will report an error after the user makes a mistake, and will keep the error status. After the user determines to correct the error, click [Reset] 

Will clear the error status and return to the initial state

6.1.7 Hhelp

6.1.7.1 About

There are options under the Help menu, click to view company information and software information. As shown in Figure 5.1-22:

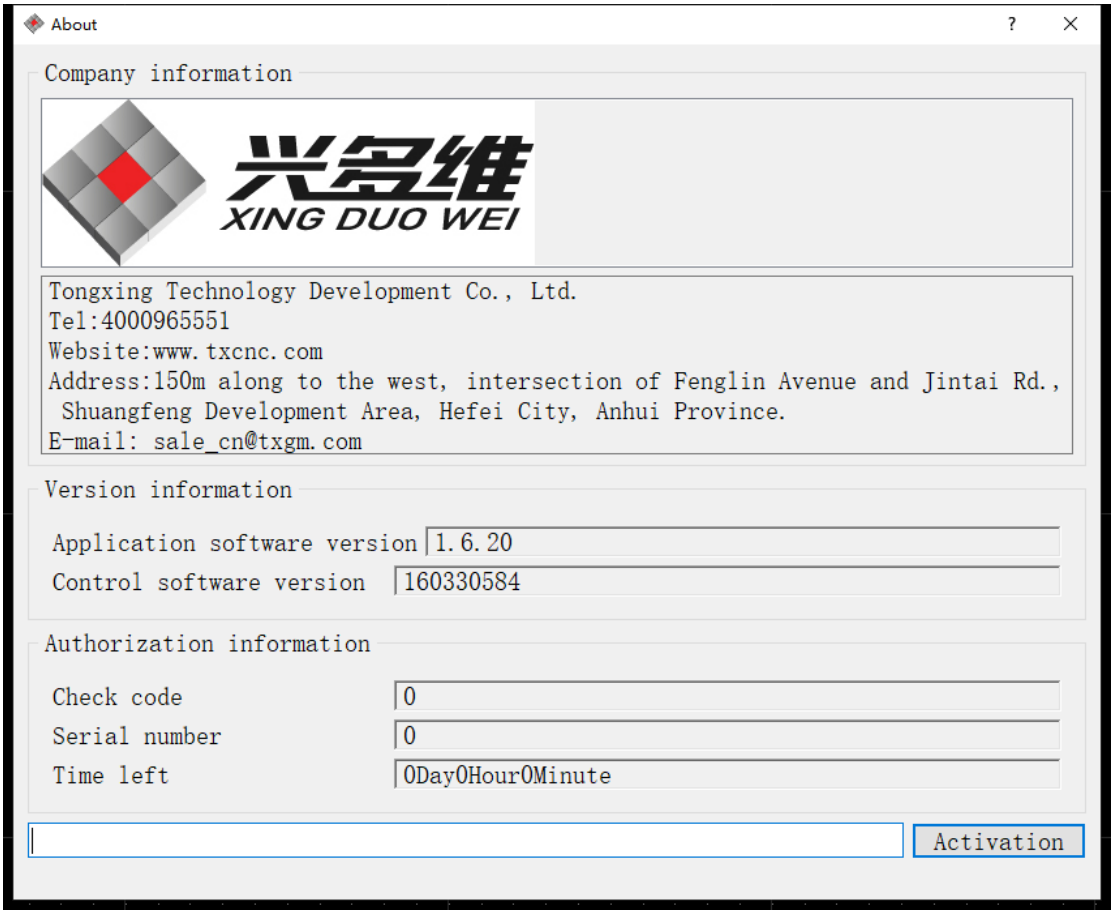


Figure6.1-22

The user can perform software activation operations on this interface. When authorization is required, the authorization code and serial number are sent to the supplier, and the activation key is obtained from the supplier. After inputting, click [Activate].

6.2 Graphic display

On the center black block paper of the graphic display interface, the user will be opened to get the file for display.

In this interface, the graphic after the extracted graphic file outline can be displayed and edited. Process setting and process management are available below. The graphical interface is shown in Figure 5.2-1:

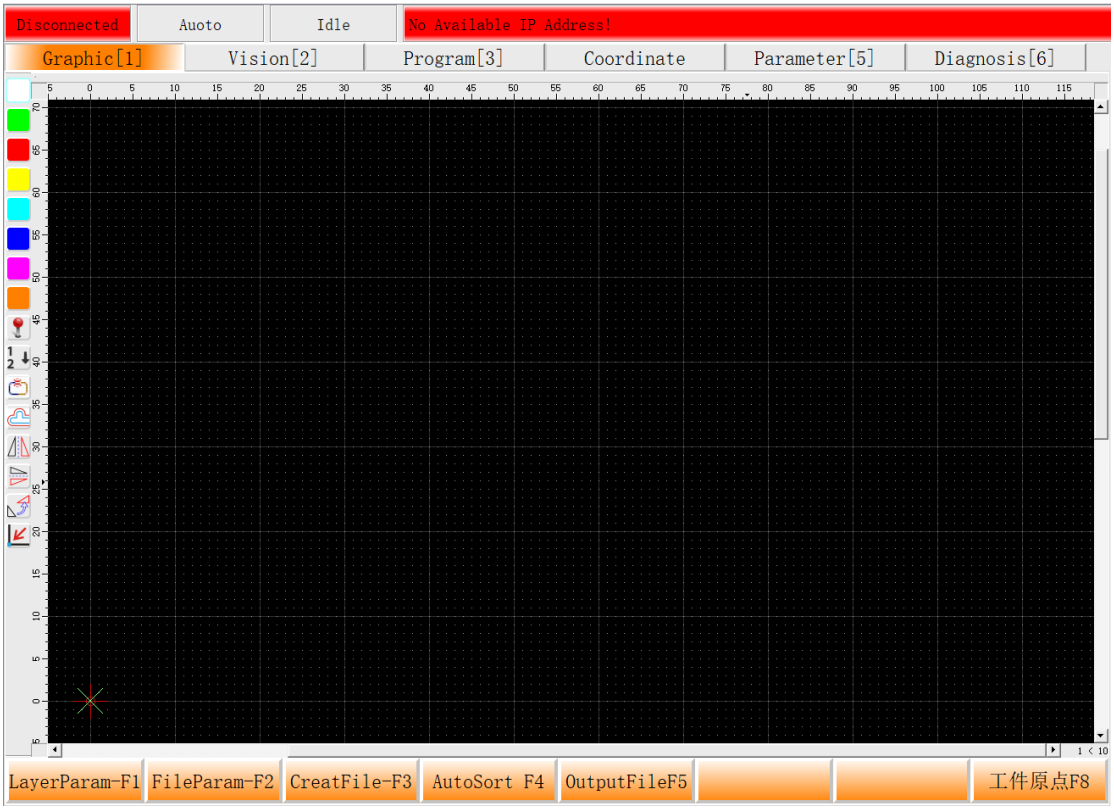


Figure 6.2-1

6.2.1 Processing options and editing

Processing options: Users can make simple processing settings in this column.

[layer selection]: Select the graphic and modify the layer where the graphic is located by the layer selection bar on the left.

[sort]: When the machining graphics are selected, the software that needs to be processed is automatically sorted. If the user needs to modify the processing order, you can click [Sort] The open function modifies the machining sequence. Click to start sorting, the user clicks the graph in order of processing to complete the sorting. During the sorting process, you can view the serial number of this time next to the mouse. Users can exit the sort in the middle.

[Processing start point]: The software will automatically define the machining starting point when the machining file is selected, and the user can customize the machining starting point according to the requirements. Select the graph and click [Start Point Settings] Use this function to select a point to start the graph. Setup completed.

[External expansion and contraction]: Select the graphics to expand and shrink the graphics. See 5.1.2.17 [Extended Expansion].

Edit: Simple editing of the elements, mirror rotation, etc.

[Horizontal Mirror] and [Vertical Mirror]: Mirror the selected primitives. The axisymmetric line is the centerline of the primitive. After mirroring, the location of the primitive is not changed.

[Rotate]: Rotate the selected element to the reference point in the center of the element. The rotation angle provided by this shortcut is three common angles of 90, 180, and 270, and the position after rotation does not change. If you need a more advanced rotation, you can use the rotation in the menu bar to select a more appropriate angle.

[Move]: Move all selected elements to the lower left corner, and do not change the relative position between each element after moving.。

5.2.2 File option

Below the view box, you have [Layer Parameters], [File Parameters], [Generate Load], [Auto Sort], [Output File], [CCD Processing].

[Layer parameters]: Click [Layer parameters] LayerParam-F1 The parameters of each layer can be set, and the attribute box shown in Figure 6.2-2 pops up:

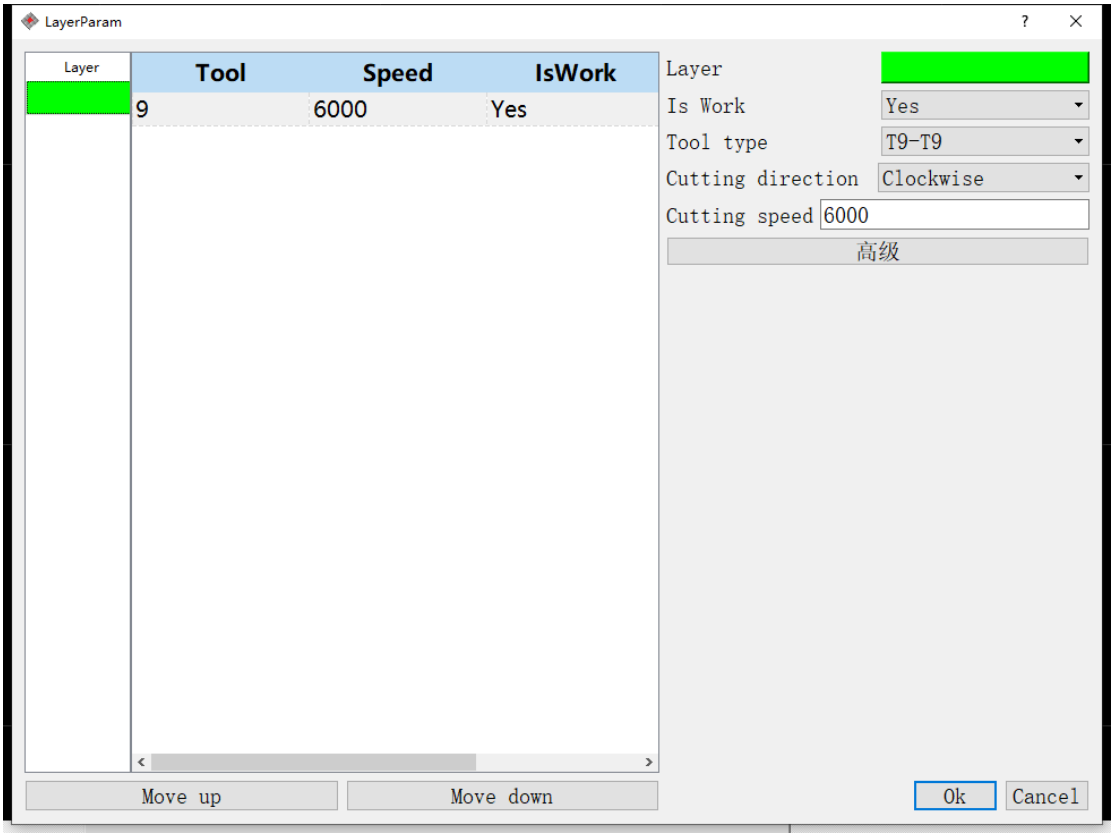


Figure 6.2-2

Click on the parameter in the middle of each layer to switch to the parameter settings for that layer. By setting the interface parameters, it is possible to satisfy a variety of processing

techniques and processing sequences. [Layer]: used to view the currently selected layer; [Processing]: In this interface, it is possible to set whether the software processes the layer. When No is selected, the layer will not be processed; when the workpiece requires multiple processing When different processes select different layers, change the tool type of each layer to achieve multi-process machining setting; [Cutting direction]: adjust the cutting direction according to the workpiece and the tool, and select the appropriate machining direction to make the machining effect more Good; [cutting speed]: used to set the relative moving speed between the tool and the workpiece during the cutting process. The user sets the cutting speed according to the actual material type and tool condition, and the cutting speed; [cutting thickness] and [in The number of times is set according to the process requirements. The cutting depth and the number of feeds are the depth of each feed. After selecting the layer, use the [Up] and [Down] settings below to click. [Confirm] save the current settings, if you need to abandon the editing, click Cancel.

[File parameters]: as shown in Figure 6.2-3:

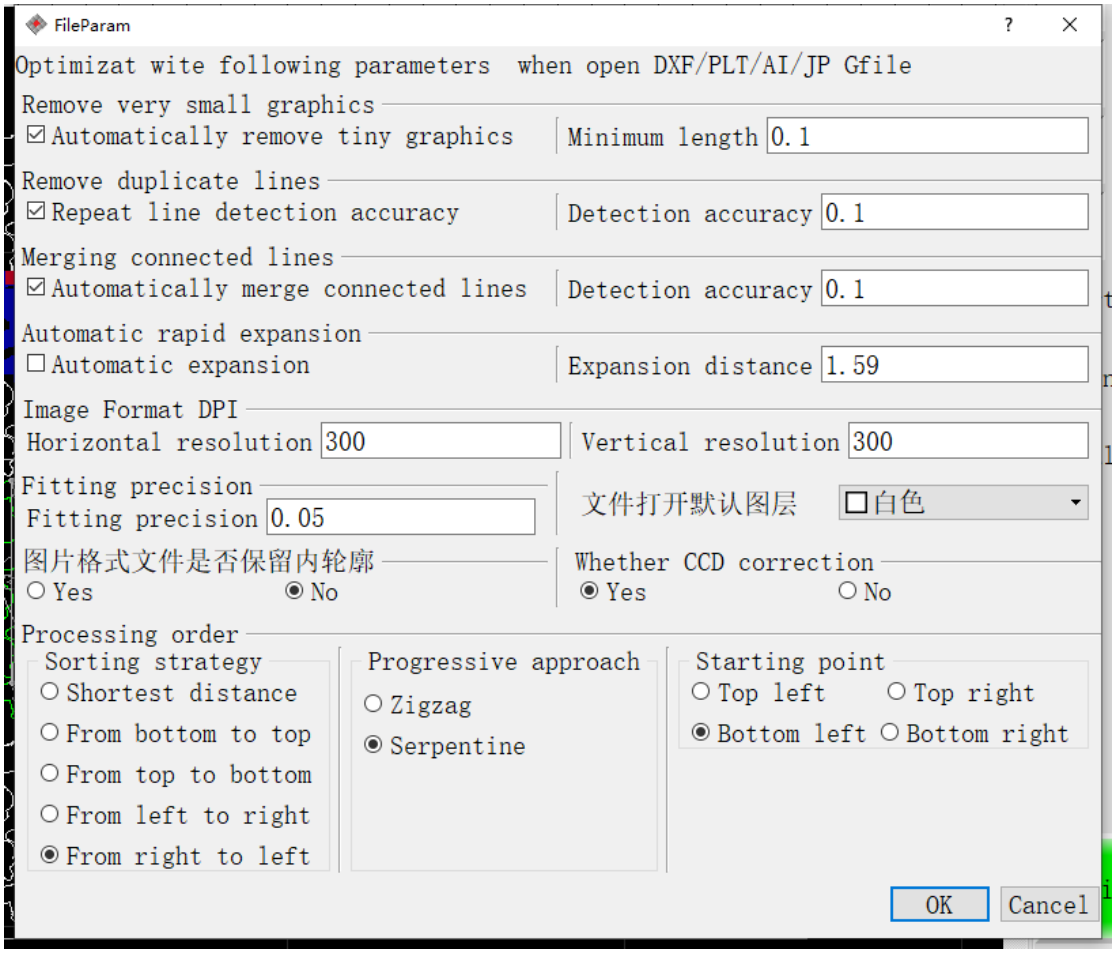


Figure6.2-3

[Remove very small graphics]: There may be noise when performing file recognition. The graphics noise is generally a very small graphic, which is turned on by ticking. The removal threshold can be filled in, and the small graphics smaller than this value will be removed during the loading process.

[Remove Repeat Line]: When the file is loaded, the software can delete the duplicated line segments and enable this function by checking the box. The threshold can be filled in at the rear, and the repeat length is greater than the value. During the loading process, the repeating line segment will be removed.

[Merge connected line]: When the file is loaded, the software can automatically connect the line segments whose two endpoints are close to each other, and enable this function by checking the check box.

The removal threshold can be filled in. The distance between the endpoints of the two segments is less than the value. During the loading process, the second two lines are connected.

[Automatic expansion]: When the file is loaded, the elements in the file are expanded, and the user does not need to perform manual expansion again. Enable this function by checking the box. The external expansion data can be filled in, and the external expansion distance will be the data.

[Picture Format DPI]: This column fills in the DPI value of the graphic file. Open the file attribute box to view it. You need to fill in the correct DPI parameter value to make the loaded file size normal. Ensure that the processing is carried out normally.

[Whether the image format file retains the internal line]: When the image file is loaded, the graphic internal line will also be recognized, and the user selects whether to keep it according to the demand.

[Whether CCD processing]: The user selects according to the processing requirements. When CCD patrol processing is required, please check it. Otherwise, the patrol positioning will not be performed during the processing.

[Sorting Strategy]: This strategy is used to sort the selected processing elements, and the user can choose how to sort according to their own needs. Click on the desired sorting method.

[Forward mode]: Select the starting position of the next line of elements after processing one line of primitives. The Z type always starts processing from the first direction set by the user; the snake type indicates starting from the lower element of the last element of the line. Processing the next line of workpieces。

[Start Position]: Select the position where each element starts machining. Selecting before the option can be processed in the corresponding way.

[Generate Load]: The G code can be generated for the file currently being edited to be processed, and can be viewed in the [Program] column of the function selection bar.

[Auto Sort]: The graphics to be processed are sorted by software according to file parameters.

[Output file]: Generate the G code file of the selected generation processing graphic and export it. Click to pop up the file saving dialog box, select the save path, and change the file name. Click OK and the file output is complete. If the file output fails, the software will prompt the output to fail.

[CCD processing]: After selecting the graphic to be processed as the processing layer, click [CCD Processing] and the machine will start processing. Users need to be aware that the camera needs to be moved to the first Mark point before performing this operation, otherwise it will not be processed and warned because the Mark point is not recognized.

6.3 Vision

The visual page user can perform camera control, camera offset setting, camera focus height setting, recognition point template management, image display, and cross test function.

The camera projects the ingested image to the left window, where the user views the camera's shooting position and moves the camera to the Mark point through the machine console.

The visual interface is shown in Figure 5.3-1:

Figure 6.3-1

6.3.1 Camera recognition template and template settings

On the right side of the camera view is the camera recognition template setting area where you can add, delete, and select camera recognition templates.

At the top of the area, you can view the template style. The default recognition template of the software is circular, and the

processing effect is best when the Mark point is circular. Therefore, the user should use the circular Mark point as much as possible to improve the processing quality of the product during the production and use process.

Below is the [Template List]. The user uses the mouse to select the template he needs in the list, and browses through the template display area above to confirm whether it is the target Mark point. When the target template is no longer needed, click [Delete Template] to delete the selected template.

In order to be compatible with the case where the camera is mounted on the spindle, the parameter "Focus height" is set. Before machining, adjust the Z-axis to move the camera to the appropriate position to make the image clear. Click [Load Current Z-Axis Coordinates] and the software sets the height to the camera height each time the Mark point is recognized. Double-click the number on the right to modify it manually. Click [OK] to save the data after the modification. In order to facilitate the user to view the effect, click [to focus height] to enable the camera to reach the position.

Click [Template Add] to enter the template add interface interface. Click [Frame Selection Template] to enter the box selection function. Move the camera to the Mark point and use the mouse to select the frame in the image display box. After the frame is selected, click [Test] to check whether the template can be recognized normally. If there is a problem with the identification, repeat the appeal operation until it can be recognized normally. In the [Template Settings] below, you can modify the name of the template and select the corresponding shape. Click [Save] above to

save the new template. Now you can select and use the template in the [Template List].

The camera can be set in the [Camera Settings] column below. The page has 4 rotation angles for the user to select. When using it for the first time, the screen is rotated according to the requirements. When the screen is normal, the software will save the setting when exiting, and do not need to modify it multiple times. .

When the actual motion direction of the motion does not match the direction of motion of the image display, check [Flip Horizontal] below, the view will be flipped horizontally; check [Vertical Flip] to try to flip vertically.

Slide the [Gray Value] slider to adjust the brightness of the image. The camera itself can adjust the brightness. This option is not recommended for modification. Sliding [Proximity] slider will adjust the accuracy of identifying Mark points. The larger the value, the larger the recognition probability but the more likely to be misidentified. The smaller the value, the lower the probability of misrecognition and the more accurate the recognition.

6.3.2 Camera switch and processing

At the bottom of the camera page, you can control the CCD. Click [CCD] to process and the software will recognize it and then process it. Make sure the camera recognizes the first Mark point before processing. Click [CCD] to simulate the wake-up simulation and view the simulation results. [Turn on the camera] to turn the camera on; [Turn off the camera] to turn the camera off.

6.3.3 Camera offset and cross test

On the right side of the view are the [Load Current Position] and [Back to Recognition Start] buttons, click [Load Current Coordinate], the software sets the position to identify the starting point of the Mark point; and click [Back to Recognition Point], Will return to this position. When the machine is parked above this point, you can view the coordinates of the recognition starting point. This point does not require user setting under normal circumstances. When the Mark point is recognized for the first time, the software automatically sets the position of the first Mark point as the starting point of the recognition point.

The user can set the camera offset in the [Camera Offset Test] column. Both the spindle and vibrating knife camera offset settings can be measured using a cross test. The cross test is now briefly explained. Mainly divided into the following steps:

1. Prepare a piece of material and fix it. Move the knife to the surface of the material, that is, move to the target position. Click “Z Zero” to ensure that the surface can be traced.。
2. Click [Cross Test] and the machine starts to move. Will draw a cross on the material.
3. Switch the machine control interface to the manual interface to move the camera position. Align the center of the view with the center of the cross and click [Calculate Offset]. The software will calculate and set the camera offset. This step requires entering the vendor password, which is the same as the installation password.

Spindle and Camera Offset Setup In addition to using the cross test method, the software provides an automatic measurement method. The user needs to have a black plate with a very thin white coating on the surface to ensure that the milling position changes from white to black. Specific steps are as follows:

1. Fix the spray white blackboard on the table top, and tool the tool to the surface of the material, click [Z Clear] Z->0, [X, Y clear] XY->0 Complete the workpiece origin setting. Use manual control to raise the Z axis to a safe height.

2. Switch the interface to the [Blinding] interface under the [Program] page, set the circular milling bottom, set the [Working Diameter] to 6, [Tool Diameter] to the actual tool diameter; [Tool Distance] is less than the tool diameter. Click [Generate Load], the software will automatically call the 6mm round milling file.

3. Click [Start], the machine will start processing, and a 6mm circle will be milled on the whiteboard, the circle is black, and the surrounding is white.

4. Switch interface to [Vision], click [Open Camera] OpenCameraF3 Open the camera, manually move the machine to align the camera to the circle, make sure that the circle just milled out can be observed in the display interface, click [Offset Measurement] Offset MeasurementF7, The test machine starts moving and automatically places the circle at the center of the image. When the green cross completely coincides with the center of the circle in the display window, the offset is automatically set.

6.3.4 Marker Assignment

In some extreme cases, the coordinates of each Mark point can be entered manually when the camera recognition condition cannot be met. The software will omit the process of recognizing the Mark point by the patrol edge, and perform the offset calculation based on the manually input Mark point coordinates.

For marking convenience, the user is allowed to move the lens to the target and record the current position. Manually control the moving head through the machine control interface to ensure that the green cross in the view coincides with the center of the circle. If the error is large during this operation, it will cause the problem of inaccurate cutting position during processing.

After the center of the green cross is coincident with the center of the circle, click [Load] below to try to record the position of the measuring point. On the left side, you can view the X and Y axis coordinates of the marked point. Check the rear [Specify], the system will use this coordinate as the position coordinate of the corresponding point; if it is not checked, it will not be used.

After setting each point position, click [CCD Processing], the machine will process at this moment, and the machine will not process the identification directly. Subsequent processing control is exactly the same as CCD patrol processing.

6.4 Program

This page allows you to view, edit, simulate, and customize the G code file. File management and milling can also be performed. Users can use different functions according to the actual situation to make processing more convenient and higher quality.

The [Load File] below can load the G code file, click the pop-up file selection dialog box, select the file, and double-click to load the file.

Click [Uninstall File] to release the currently loaded file. After clicking, the software returns to the state of the unloaded file.

The [Simulation] button simulates the loaded file and displays the visual view after the simulation in the view interface.

[Select Processing] Click to open the selection processing interface, as shown in Figure 5.4-1:

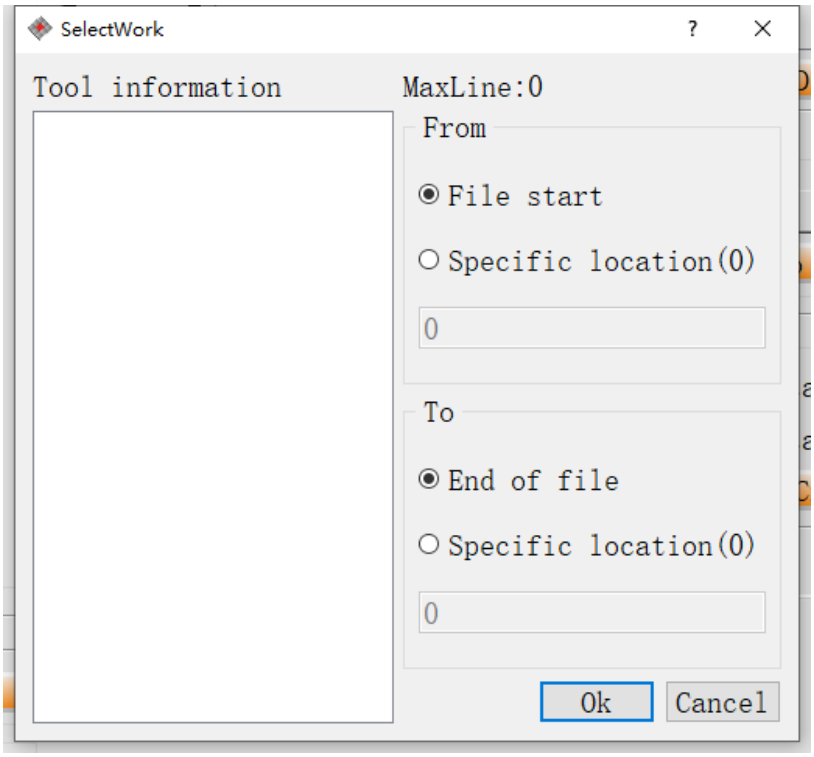


Figure 6.4-1

Click [Specify Line Number] and you will be able to set the start and end line numbers separately. Complete the processing of the specified block once.

[Edit file] After clicking, the text editing window will pop up, and the G code file can be edited and saved again. It can be modified here to make the user's operation more convenient.

6.4.1 3D

The user can simulate the loaded G code file in this interface and view the simulation results. After the G code file is loaded, click [Simulation] below, and the text will be simulated. The G code will be visualized and can be viewed in the window.

On the left side of the view box, there is a list of view option buttons, which are used to view the simulation view from different angles to meet various needs.

The first four are interface zoom and clear buttons:  [Zoom out view];  [Zoom in view];  [magnified view];  [Clean up simulation graphics].

The following six are six-sided perspectives that will show a plan view. The upper view; the lower view; the left view; the right view; the front view; the back view.

The next four buttons are stereoscopic viewing angles. The lower left angle of view; the lower right angle; the upper right angle; the upper left perspective.

 [Information] is the file information button, click to switch whether to display the file information. Customers can choose whether to display file information according to their needs. This column can

display the total number of lines of the file, the total length of the stroke, the length of the cut, the minimum maximum of the X-axis, the Y-axis, and the Z-axis.

6.4.2 G code

This interface can view the loaded G code file. If you need to edit, click [Edit File] below to edit. When processing, the ongoing line number will be placed on the interface.

6.4.3 Processing settings

The interface user can set the dry running speed, machining speed, spindle speed, speed specification mode, whether to ignore the spindle speed in the program, the number of cycle machining, and the cycle processing interval according to their own needs. Click on the back number and change to change the way.

Users can also set quality-efficiency adjustments on this page. To change this parameter, you need to enter the manufacturer password.

6.4.4 Document management

Users can perform batch file management on this interface, click [File Path] FileDir, Popup folder selection box. The user selects the target folder, and Soft Drop will automatically recognize all the software supported in the folder and display it in the file list below. On the right are [Load], [Edit], [New], [Delete], [Refresh].

After selecting the file, click [Load] to open the selected file; if the selected file is a G code file, click [Edit] to pop up a convenient interface for user editing; [New] will create a new NC

file; select the file and click [Delete], the target file will be deleted, please be careful; if a new file is added to the folder, click [Refresh] to refresh the file in the folder.

6.4.5 Milling bottom

The user of the milling bottom interface can perform cutting and milling operations, and can perform square machining and circular machining. Users can select according to their needs.。

When using the cutting function, the user can select the outer frame cut or the inner frame cut. For square cutting, change the length and width of the rectangle by clicking the number box next to the height and width; the lower X, Y starting coordinates are offset from the workpiece origin; the tool diameter is the tool offset that the user needs to set. Each feed is the depth of the machining once; the engraving depth is the machining depth after machining. For example, if the feed amount is 1 and the engraving depth is 5, the workpiece is processed 5 times, and the machining depth is 1 each time.

When milling the bottom, you can set the tool nose distance, click the rear number box to change, the number of settings should be smaller than the tool diameter; switch the horizontal milling bottom and the vertical milling bottom by clicking above.

After the parameters are set, click [Load File] to load the file. After the loading is completed, you can simulate in the 3D interface.

6.5 Coordinate System

The user can set the coordinate system on this page; save and select the workpiece coordinate system; set the common offset; also can perform the tool setting operation.

6.5.1 Workpiece coordinates

At the top of the interface, you can view the coordinate information of the workpiece origin currently used. Click the loading current coordinate on the right side to save the coordinates of the workpiece origin currently in use to the selected workpiece coordinates.

The common offset of the X, Y, and Z axes can be set separately in the "Common Offset Setting". With this option, offsets during machining can be made for customer convenience.

There are 6 groups below, each group of 10, a total of 60 workpiece origins for users to choose. Users can set each workpiece origin separately.

6.5.2 Access

There are ten workpiece origins for the user to set on this interface. Click [Save] to save the workpiece coordinate system to the computer. After reopening the software, you can also select the workpiece origin in this page. Click [Read] to set the selected target coordinate system to the workpiece coordinate system used.

6.5.3 Tool Setting

The CNCRouter system supports two tool setting modes: floating tool setting and fixed tool setting. The user can change the parameter 30201 tool setting mode and select the corresponding mode.

Before setting the tool, you need to set the thickness of the tool 30202. The correct calculation of the parameter software can calculate the workpiece origin normally. In the floating tool setting mode, only the tool setting position Z is required; in the fixed tool setting mode, the tool setting position X, Y is also required. The tool setting position Z is the position where the positioning of the tool is started.

The interface user can set the workpiece software below, move the tool to the workpiece origin, and click [X Zero], [Y Clear], [Z Clear] to complete the workpiece origin setting. Click the [Floating Tool Set] / [Fixed Tool Set], [First Time Pair], [Tool Change After Tool Change] to perform the corresponding tool setting operation.

6.5.3.1 Floating tool

The floating tool setting process is shown in Figure 5.4-1. Move it manually to the top of the tool block and click [Load Current Z Coordinate]. This position is the Z-axis coordinate at which the tool is fine-tuned. Click on the floating tool setting. At this point, the tool is started. When the spindle is lowered to this position, the tool touches the tool setting tool at a slow speed. When the tool is touched, the software gets a signal. The software automatically calculates the Z-axis coordinate of the workpiece surface and sets it as the workpiece origin coordinate. The Z-axis is pulled up to complete the tool setting.

After the user finishes measuring the surface of the workpiece, he needs to click [Point once to set the knife] and record the Z-axis coordinates to facilitate subsequent tool change. Click [First Time Tooling] to start the tool setting. The tool moves down to touch the tool setting device, and the software records the current Z axis coordinate.

In the process of machining, when the tool needs to be changed, the user replaces the tool and clicks [Tool setting after tool change]. After the tool is completed, the software resets the Z coordinate of the workpiece origin, and the user can continue processing.

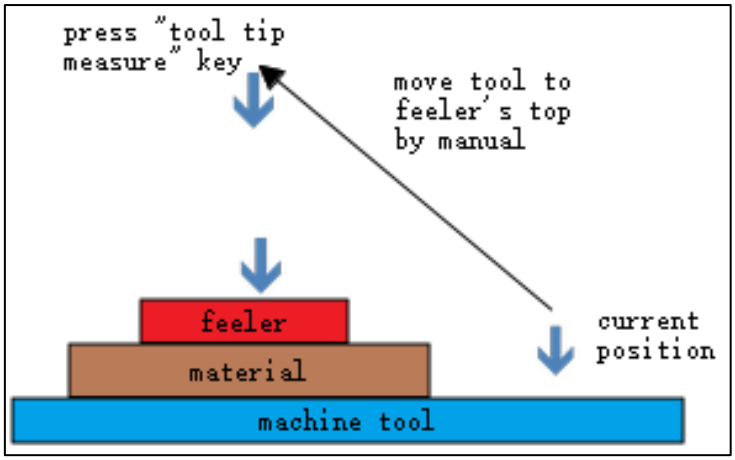


Figure 6.5-1

6.5.3.2 Fixed tool setting

The fixed tool setting process is shown in Figure 5.4-2. Manually move the tool to the top of the tool setting tool and click to load [current X axis coordinate] and [load current Y axis coordinate]. Move the Z axis, move the tool to a point above the tool setting tool, and click [Load Current Z Coordinate] to complete the tool setting position. Then click on the [fixed tool setting] below and the system starts to set the tool. When the tool tip touches the tool

setting tool, the Z axis coordinate of the workpiece surface is automatically calculated and set as the workpiece origin coordinate, and the Z axis is pulled up to complete the tool setting.

After the user finishes measuring the surface of the workpiece, he needs to click [Point once to set the knife] and record the Z-axis coordinates to facilitate subsequent tool change. Click [First Time Tooling] to start the tool setting. The tool moves down to touch the tool setting device, and the software records the current Z axis coordinate.

In the process of machining, when the tool needs to be changed, the user replaces the tool and clicks [Tool setting after tool change]. After the tool is completed, the software resets the Z coordinate of the workpiece origin, and the user can continue processing.

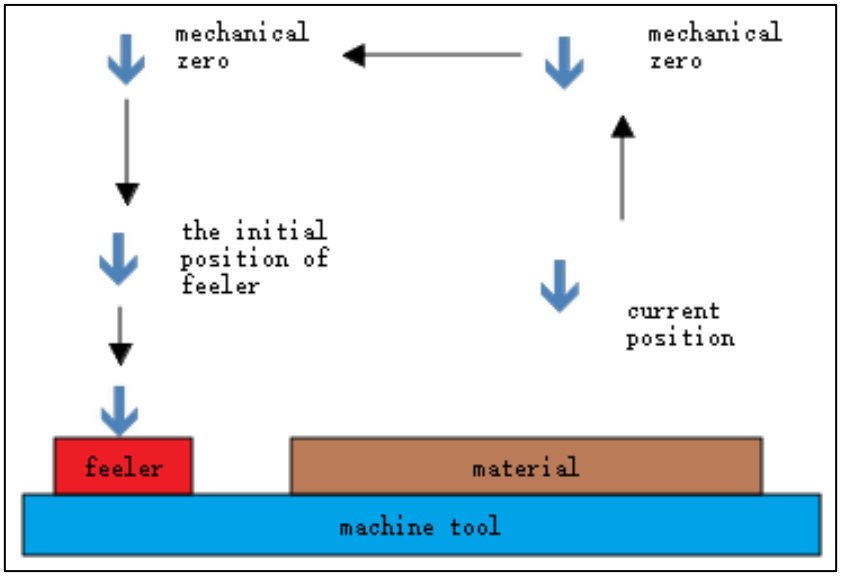


Figure 6.5-2

6.5.3.3 Two points and three points

The user automatically calculates the square blank center through the "two points in the middle" function. Process is as follows:

1. Manually move the tool to the left of the workpiece and click [Record X];

2. Move the tool to the right side of the workpiece and click [Zone X]. The software will calculate the X coordinate of the workpiece origin according to the recorded X coordinate of the previous point and the X coordinate of the current point, and record;

The workpiece origin Y-axis coordinate is set to be similar to the X-axis:

1. Move the tool to touch the upper side of the workpiece and click [Record Y];

2. Then move the tool to touch the lower side of the workpiece and click [Zone Y]. The software calculates the original Y-axis coordinate of the workpiece based on the recorded Y-axis coordinate of the previous point and the Y-axis coordinate of the current point, and records it. Find the center point and set it as the workpiece origin;

The circular blank center coordinates can be automatically calculated by recording the three-point coordinates around the circular blank.

1. Manually move the tool to a point on the circumference of the workpiece, click [Record Point 1], and the software records the current machine coordinate as the first set of coordinates;
2. Move the tool to another point on the circumference of the workpiece, click on record point 2, and the software records the current machine coordinate as the second set of coordinates;

3. Move the tool to the third point on the circumference of the workpiece, click [3 points] software to calculate the center of the circle and set the workpiece origin according to the recorded two sets of coordinates and the current machine coordinates.

6.5.4 Tool magazine

The CNC_Router system supports the tool change action. According to the number of tools supported by the actual mechanical structure, fill in the parameter 30001: Tool magazine capacity for matching. In this interface, it is possible to complete the tool change, measure the tool length, the tool magazine position setting, and change the tool.

Note: Please do not modify this interface parameters at will. Please modify it under the guidance of a special person!

The tool library interface is shown in Figure 5.4-3:

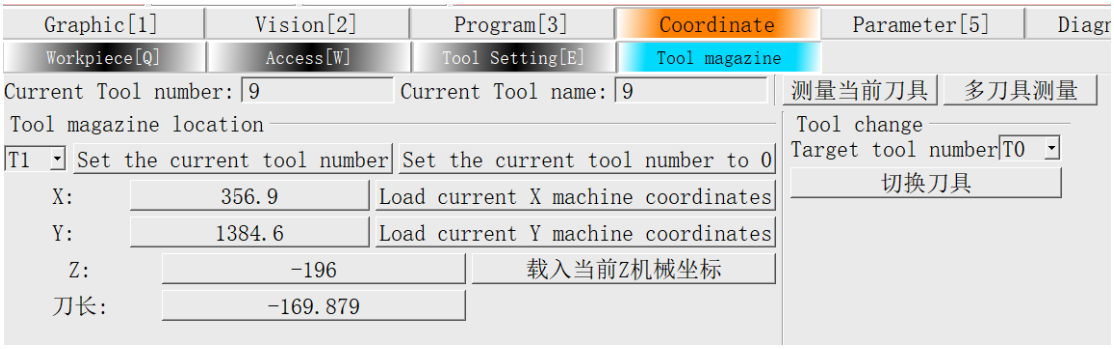


Figure5.5-3

[Current tool number]: Used to display the tool number of the current spindle clamped to the tool holder. The current tool number can also be viewed in the machine control bar.

[Current tool name]: Display the name of the current tool. The user can change the tool installed in each tool position. In the

parameter list, change the tool name of the corresponding tool number.

The tool length measurement is required each time the tool is replaced. Supports single tool measurement and multi-tool measurement modes; [Measure current tool]: Only measure the tool length of the current tool. [Multi-tool measurement]: Measure the tool length of the tool number selected by the user. A dialog box will pop up for the user to select the measurement item number; as shown in Figure 5.4-4:

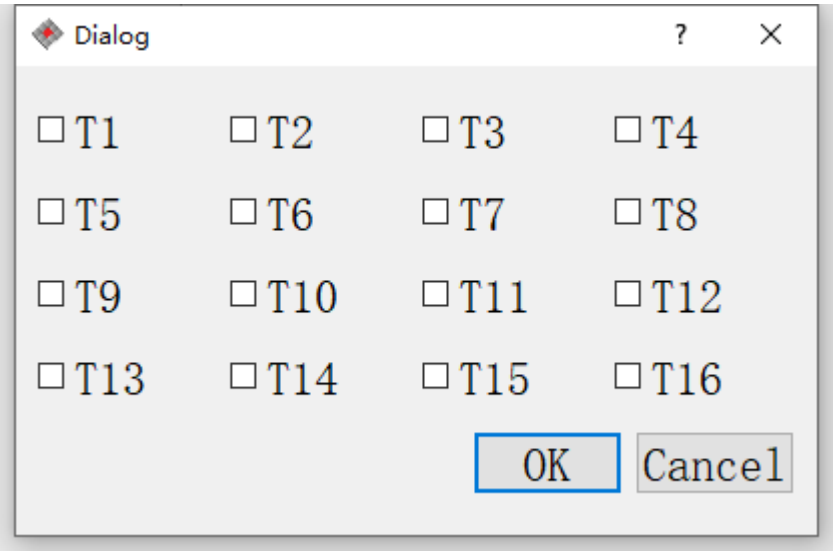


Figure 5.5-3

The user will need to measure the length of the tool according to his own needs, and then click [OK]; the machine will start measuring the length of the selected tool.

[tool magazine location]:

Here, the tool number of the current clamping knife can be changed. When an extreme situation occurs, such as a power interruption in the tool changing process, the spindle actually clamps the tool number and does not match the current tool number of the software. The user needs to manually set the current tool number as follows: 1. Open the tool number drop-down list and select the actual tool number on the current spindle; 2. Click [Set as current tool number]; if the current spindle is not clamped, it will be zero. No., click [Set current tool number to 0] to complete the current tool number correction.

Note: Please ensure that the tool number recognized by the software matches the actual tool number before using the machine, otherwise the tool magazine will be damaged!

Before using the machine tool, it is necessary to set the position of each tool holder of the tool magazine to ensure the safe and correct completion of the tool change. Please refer to the machine commissioning chapter for detailed commissioning procedures.

Here, the user can view and modify the location of the magazine, and the user clicks on the number to modify it; or click [Load Current Machine Coordinates] to set the machine coordinate of the current spindle position to the machine coordinate of the magazine location corresponding to the tool number.

Note: Do not change the location of the magazine at will, otherwise the tool magazine will be damaged!

[Tool length]: This parameter does not need to be manually filled in by the user. After using automatic measurement, the tool length data software will be automatically set.

Note: Users are not allowed to modify it at will, otherwise it will lead to inaccurate processing and other issues!

[Tool change]: When the user needs to manually change the tool, select the target tool number to be switched in the [Tool change] field, click [Switch tool], then the machine will start the tool change action. After the tool change is completed, the machine will stop.

6.6 Parameter

This page allows you to view and modify all parameters, import and export parameters, and back up parameters. When there are multiple machines, the model number of the model can be selected again.

6.6.1 Parameter overview

This page can be used to view and modify all parameters. When you need to view the vendor parameters, you need to output the vendor password. The content of this section is closely related to machine debugging, so the sixth chapter will be explained in detail.

5.6.2 IO Parameter

The user can modify the name of the input and output ports, IO number, trigger level, and second speed on this page. The limits of customization are large, making it easy for users to use personalized usage habits.

Double-click the parameter that needs to be modified, and the IO parameter modification dialog box will pop up. The user can modify the port name, function description, port number, and trigger level in the secondary dialog box. Change completed Click OK to take effect immediately.

6.6.3 Backup

When the parameter adjustment is completed, click [Save], enter the name in the pop-up dialog box, and click OK to complete the backup. It can be found that the parameter just named in the parameter list on the left has been added. Click [Restore] after the

parameter needs to be restored, and the software will restore the secondary parameters.

This parameter is backed up in another storage space of the computer and will not be deleted with the software update and uninstall. Save the parameters with the user.

6.6.4 Configuration

If the user needs to switch the model, click [Select] at the back of the corresponding model to switch, and the software language can be selected below.

6.7 Diagnosis

The user of the interface can view the status of each input and output port in real time, so as to diagnose whether each port can be used normally.

When the input signal changes, the corresponding input signal will change color; when the output button is clicked, the output port signal of the control board will change.

In the box below, you can view the historical status of the software running, so that users can find problems.

6.8 Machine control bar

The user of the interface can view the mechanical coordinates of the current position and the workpiece coordinates. During the movement of the machine, the real-time speed of the machine as well as the spindle speed and the current tool number can be viewed.

In re-machining, the machining speed can be adjusted in real time by sliding the slider next to the feedrate override. Clicking on the rear digital display block can modify the default machining speed.

Adjusting the G00 magnification slider can adjust the speed of the machine's dry running; the spindle can also control the speed in real time through the slider.

This interface can manually turn the spindle and vacuum pump on or off. Click [Spindle] to switch the spindle status; click [Vacuum Pump] to switch the vacuum pump status. Green is in standby mode and red is in working state.

When the file is loaded, click [Start] and the machine will start processing. During the machining process, click [Pause], the machining will be paused, click [Breakpoint Continue], and the machining will continue from the stop. Click [Stop], the software will stop the current processing.

After the error occurs, the software will automatically report an error and the status bar will turn red. After the user finds the problem and cuts it, click [Reset] to release the alarm status of the software.

The user can manually control the machine to move in all directions. Click the button, the phase axis will move in the corresponding direction. Click the center [H/L] button to switch the manual motion mode, H is the height motion, and L is the low speed motion mode. User can adjust manual parameters [manual high speed] and [manual low speed] custom speed.

The user next to the machine control bar can click on the sport mode for the user to select the mode: continuous, handwheel, motion

0.01, motion 0.1, etc. The user can click the button to customize the step size.

In addition to this, the user can enter the target position to move the machine to here. Click [Target Move] Target move Then pop up a dialog box, enter the coordinates of the target location, and click OK. The machine moves to the target position. Click [Stop] and the machine will stop the current action.

When you need the handwheel to guide, click [Handwheel] to enter the handwheel mode. At this point the user can control the machine via the handwheel.

Various errors may occur during use, the software will automatically report an error, and the status bar will turn red. Troubleshoot according to the error message displayed in the status bar above. When the user finds the problem and finishes the processing, click [Reset] to release the alarm status of the software. The status bar will also change from red to white.

Click [Z Clear] Z->0 , [X, Y clear] XY->0 Set the workpiece coordinates to determine the machining position.

Chapter Seven Parameter and Equipment Debugging

The user can view and modify all parameters in the parameter interface, including axis parameters, manual parameters, machining parameters, file parameters, tool parameters, spindle parameters, CCD parameters. Some parameters are vendor parameters. You need to enter the manufacturer password to view and modify this type of password.

The user can view and modify all parameters in the parameter interface, including axis parameters, manual parameters, machining parameters, file parameters, tool parameters, spindle parameters, CCD parameters. Some parameters are vendor parameters. You need to enter the manufacturer password to view and modify this type of password.

7.1 Axis parameter

In this column, set the three axis parameters of X, Y and Z, including the setting of parameters such as speed, direction and pulse equivalent.

10001：X-axis direction; used to control the direction of machine tool movement, 1 is positive and -1 is reverse. When the actual direction of motion is found to be opposite to the command issued, the axis motion reversal can be done by changing this parameter. The Y axis and Z axis settings are the same. For the direction of the coordinate system, please refer to the 3.1 coordinate system.

10101：X pulse equivalent; displacement produced by each control pulse, in mm/P. This parameter is provided by the manufacturer. The wrong singular number will cause the machine tool to move less than the specified position, and generally does not need to be modified. The Y axis and Z axis settings are the same.

10201：The lower limit of the X-axis stroke; the minimum value of the X-axis coordinate allowed by the X-axis, in mm. This parameter is set according to the size of the machine. When the X-axis coordinate is close to this value, the software will stop sending commands to the motion board, so this function is the software limit. The user can ensure that the machine movement does not go over during use, and protect the safety of personnel and machines. The Y axis and Z axis settings are the same.

10301：The lower limit of the X-axis stroke; the minimum value of the X-axis coordinate allowed by the X-axis, in mm. This parameter is set according to the size of the machine. When the X-axis coordinate is close to this value, the software will stop sending commands to the motion board, so this function is the software limit. The user can ensure that the machine movement does not go over during use, and protect the safety of personnel and machines. The Y axis and Z axis settings are the same.

10401：The X soft limit is enabled; the user can choose whether to use the set X-axis travel minimum and maximum values. 1 enable, use settings; 0 disable, do not use settings. When the machine is being debugged, the function can be disabled. After the debugging is completed, please keep the limit function enabled to ensure safety. The Y axis and Z axis settings are the same.

10501：X-axis type; select according to the axis type. 1 is a linear axis and 2 is a rotary axis. Choose according to the actual type. The Y axis and Z axis settings are the same.

10601：X-axis address; the same as the IO parameters, the CNCRouter system supports user-defined editing of the axis signal address to make the user's wiring and wiring more flexible. After the

change is completed, the axis signal wiring on the board also needs to be modified to make the system operate normally. The Y axis and Z axis settings are the same.

11002：Whether it must return to zero before processing; 1 is yes, 0 is no. The system defaults to zero to ensure normal use, and the user can modify the tool's own requirements.

11003：Motor type; 1 is stepping and 2 is servo. Set according to the motor used by the machine.

11005：Whether the connection is automatically zeroed; whether the automatic zero return option is prompted after the computer and the control board are successfully connected. 1 is yes, 0 is no.

11101：X-axis origin; the mechanical coordinate of the trigger point of the X-axis mechanical zero return limit signal, in mm. This parameter is set according to the machine limit switch position and is provided by the manufacturer. The settings for the Y axis and the Z axis are the same.

11201：Initial orientation of the X axis; 1 positive, -1 negative. Set according to the position of the machine limit switch. If it is installed on the stroke minimum side, select the negative direction; if it is installed on the stroke maximum side, select the forward direction. The Y axis and Z axis settings are the same.

11301：X coarse positioning speed; unit mm / min. In the process of zero return, the machine first triggers the limit signal at a faster speed, which is the coarse positioning speed. The speed is not too fast, and the speed is too fast, which will cause the machine to pass through during the zero return process. The settings for the Y axis and the Z axis are the same.

11401：X fine positioning speed; unit mm / min. During the zero return process, after the coarse positioning is completed, it will enter the fine positioning phase. Fine positioning will touch the limit switch at a slower speed, which is the fine positioning speed, which should also slow down as much as possible. The settings for the Y axis and the Z axis are the same.

11501：X retraction distance; unit mm. During the zero return process, after touching the limit signal, it will move backwards by a small distance. This distance is called the retreat distance. The retraction distance is not easy to set too large. If the setting is too large, the return to zero time will be greatly extended, which will affect the efficiency of use. The settings for the Y axis and the Z axis are the same.

12001：X-axis speed limit; the maximum speed allowed for X-axis motion, which will not be exceeded at any time. Y axis, Z axis are the same

7.2 Manual parameter

This column can view and modify the parameters of the manual moving machine, including the setting of take-off speed, manual high speed and manual low speed.

60005：Linear axis manual take-off speed; unit mm/min. The speed is the starting speed when the machine is manually moved. The speed is not too large. If it is too large, the machine will shake greatly at the moment of starting. If the setting speed is too high, the motor will be blocked at the start.

60007：Linear axis manual acceleration; unit mmm/s^2. The acceleration is set according to the motor parameters and the mechanical structure. The maximum speed is provided by the machine tool builder. To ensure normal use, please set it less than this value.

60101：X-axis manual low speed; unit mm/min. Low speed and high speed motion can be selected for manual control. Changing this parameter will change the speed of movement in low speed mode. The Y axis and Z axis settings are the same.

60102：X-axis manual high speed; unit mm/min. Low speed and high speed motion can be selected for manual control. Change this parameter award to change the speed of movement in high-speed motion mode. The Y axis and Z axis settings are the same.

7.3 Processing parameters

This column lists the parameters in the process for users to view and modify, including processing speed settings, effect settings, spindle vacuum pump settings, etc.

50001：Processing take-off speed; unit mm / min. That is, the initial speed at the start of machining. Similar to the manual take-off speed, the speed should not be set too large, otherwise the motor will block and affect normal use.

50002：Dry running speed; unit mm/min. During machining, when machining a workpiece, moving the Z axis to the next position is called empty operation. This parameter can be modified after the

parameter is modified, and the appropriate dry running speed can improve the machining efficiency of the machine tool.

50003：Processing speed; unit mm/min. The speed at which the reference point on the tool moves relative to the workpiece along the tool path during machining. Modifying this parameter will affect this speed. Changing this speed depends on various types of materials, material thickness, machining depth, and machining process. The slow processing speed makes the processing time longer, which affects the processing efficiency; too fast processing speed will affect the product effect, and the risk of breaking the tool may endanger personal safety. Therefore, it is not allowed to increase the processing speed blindly, and should be adjusted according to the actual situation.

50004：Maximum processing speed; unit mm/min. When using this software to generate a machining file, set the maximum processing speed of the file at this speed.

50007：Processing acceleration; unit mm/S^2. The acceleration of the machining speed during machining. Change this parameter to change the change in the speed of the axis movement. This parameter should be set according to the motor performance and mechanical mechanism.

50008：Free-range acceleration; unit mm/S^2. Similar to the machining acceleration, this acceleration is the acceleration during the lost motion.

50010：Smoothing time; the larger the unit ms value. The smoother the processing effect, but it will also increase the error.

50012：Falling speed; unit mm/min. The speed at which the tool descends from the safe height to the Z coordinate of the workpiece origin.

50013：Arc speed limit; unit mm/min. The speed is the machining speed of the arc with a radius of 5mm. The arc of other radius and the arc of 5mm are used as the reference to adjust the speed of the limit. The smaller the arc, the slower the speed, the faster the arc is.

50014：Short line segment reference length; unit mm. When the length of the small line segment is less than the value of this parameter, it will be treated as a small line segment. When machining the machine, the parameters of the small line segment will be used for machining.

50015：Short-term acceleration; unit mm/min. When processing small line segments, processing with this value as acceleration.

50016：Arc interpolation control accuracy; unit mm. Accuracy selection when performing arc machining, the smaller the value, the higher the arc accuracy.

50017：Z-axis safety height; unit mm. After the machining is completed, the Z axis will be pulled up, and the height of the pull will be the safe height.

50018：Z-axis falling speed selection; CNCRouter will provide three kinds of falling knives: no processing; only Z-axis is effective downward; it contains Z-axis lower component. 1. No processing: no limitation on the falling speed; 2. Only the Z axis is effective downwards only: the speed limit is only performed when the Z axis is lowered alone; 3. The Z axis is the lower component: in motion, if Z The shaft will move downwards and speed limit will be applied.

50100：Whether to use the file processing speed; when processing with a G code file, the file contains processing speed information.

Change this parameter to choose to use file processing speed or use software processing speed.

50101：Whether to use the file spindle speed; when machining with a G code file, the file contains the spindle speed information. Changing this parameter allows you to choose to use the file spindle speed or use the software spindle speed.

50102：Whether the spindle starts automatically when machining starts; changing this parameter can select whether the spindle is automatically turned on when machining starts.

50103：Whether the vacuum pump is automatically turned on at the beginning of processing; if this parameter is changed, it is possible to select whether the vacuum pump is automatically turned on at the beginning of processing.

50110：The processing end parking mode; the software provides three parking modes: current position, specified machine coordinates, and end position offset. 1. Current position: After the machining is finished, the spindle will stop at the end position, and after the Z axis is pulled up to the safe height, no movement will be performed. 2. Specify the machine coordinates: After the machining is finished, pull up the Z axis and move the spindle to the specified machine coordinate. 3. End position offset: After the machining is finished, pull up the Z axis, and then move the spindle to the position where the specified XY coordinates are offset from the end point.

50111：Parking position X-axis coordinates. This parameter is used to specify the X coordinate of the parking position. When the parking mode parameter is selected as the current position, the parameter is useless; when the parking mode parameter is selected as the specified machine coordinate, the parameter represents the X-axis

coordinate of the stop position; when the parking mode parameter is selected as the end position offset, the parameter Indicates the X-axis offset of the stop position relative to the machining end point.

50112：Parking position Y-axis coordinate. This parameter is used to specify the Y position of the parking position. When the parking mode parameter is selected as the current position, the parameter is useless; when the parking mode parameter is selected as the specified machine coordinate, the parameter represents the Y-axis coordinate of the stop position; when the parking mode parameter is selected as the end position offset, the parameter Indicates the Y-axis offset of the stop position relative to the machining end point.

50201：Current quality strategy group. The software provides 5 sets of strategy groups for public account selection and use. Fill in the corresponding group number will use the group parameter.

50211：Group 0 Turning speed adjustment factor: The processing quality and processing efficiency are coordinated by adjusting the turning speed. The smaller the value, the better the effect. The turning speed adjustment factor function of the remaining group is the same.

50212：Group 0 acceleration adjustment factor: coordinate the machining quality and machining efficiency by adjusting the acceleration. The smaller the value, the better the effect. The remaining group's acceleration adjustment factor function is the same.

50213：Group 0 arc speed limit adjustment factor: coordinate the machining speed and machining efficiency by adjusting the arc speed limit. The smaller the value, the better the effect. The arc limit rate adjustment factor of the remaining group is the same.

7.4 File parameter

The column is more convenient to view and adjust the parameters related to the file, including contour extraction, processing depth, picture information, automatic expansion, processing of connected lines and tiny graphics.

52001：The height of the tool file lift height; the height of the tool lift when the air travel is used when machining with a graphic file. The height is offset from the Z coordinate of the origin of the workpiece.

52002：The depth of the graphic file file processing; this parameter will be used as the machining depth when machining with graphic files. This parameter is the offset to the left of the Z axis relative to the origin of the workpiece.

52003：Automatically delete duplicate lines; use this parameter to adjust whether to delete duplicate lines.

52004：Automatically removes the repeat line accuracy; in mm. The length of the overlapping line is greater than the value of this parameter and will be recognized as a repeating part. When the file is opened, if the automatic repeat line function is turned on, the duplicate part will be deleted.

52005：Automatically merge connected lines; use this parameter to adjust whether to merge connected line segments.

52006：Automatically merge connected line accuracy; unit mm. If the distance between the lines is less than the value of the

parameter, it will be recognized as a connected line. Automatically connected when the file is opened.

52007：Automatically remove tiny graphics; use this parameter to adjust whether to automatically remove tiny graphics.

52008：Automatically removes small graphics accuracy; in mm. If the length of the graphic is less than the value of the parameter, it will be treated as a small graphic. If the automatic micro-graphics function is turned on, these graphics will be automatically removed when the file is loaded..

52009：Automatic sorting; use this parameter to select whether to automatically sort when opening a file.

52010：Automatic sorting method; the software provides 5 automatic sorting strategies. Use the numbers to correspond to the corresponding functions. 1. Shortest distance, 2. From bottom to top, 3. From top to bottom, 4. From left to right, 5. Right to left.

52011：Automatic expansion; whether to automatically expand when opening a file by changing this parameter.

52012：Automatic expansion distance; if the automatic expansion function is turned on, the parameter value is the distance that is automatically expanded when the file is opened.

52013：Horizontal resolution; the horizontal resolution of the image format file, you can get the data by viewing the loading image property.

52014：Vertical resolution; the vertical resolution of the image format file, you can get the data by viewing the loading image property.

52016：Whether the spline is fitted; this parameter is used to select whether to perform spline fitting when opening the image

format file. Using spline fitting can smooth the smooth processing of the line segment, but it will bring certain errors.

52017：Z-axis workpiece origin setting; when using the graphic file to generate the machining file, select the Z-axis workpiece origin based on the material surface or based on the material bottom. 1. Material surface, 2. Material bottom surface.

52020：The smoothing accuracy of the processing file is generated; smoothing is performed when the processing file is generated, and the larger the value, the smoother the value. But too large a value will be wrong.

52021：Whether JPG retains the inner contour; whether the inner contour is retained when importing the JPG file, if the processing file only needs to be processed by the outer contour, it is not reserved; if the processing file has a hollow operation, it needs to be retained, such as a window flower. 0 means not reserved, 1 means reserved.

52022：Contour extraction threshold; the extraction threshold of the image file when extracting the contour. This parameter value is a gray value. The black gradation value is 0, and the white gradation value is 255. If the graphic gray value exceeds the parameter value, it will be judged as the contour. If the graphic is inaccurate in the extraction contour stage, the parameter can be improved by changing the parameter.

52024：Outline extraction smoothing level; smoothing contour extraction when importing image format files, the higher the number of levels, the smoother the extracted lines.

7.5 Tool parameter

This column can set the parameters related to the machining tool, including the magazine capacity, tool number, tool setting mode, tool setting position, etc.

30001：Tool magazine capacity; indicates the number of tools that the machine magazine can hold.

30002：Current tool number; the number of the tool currently being used.

30003：The offset of the magazine in position Y is forward; the unit is mm. The distance offset before reaching the magazine position when changing the tool.

30201：Tool setting mode; select the mode of automatic tool setting. 1. Floating tool setting, 2. Fixed tool setting.

30202：Tool set thickness; unit mm. The thickness of the tool, the distance between the origin of the workpiece and the upper surface of the tool.

30203：The number of times the knife is aligned; the number of times the tool is accurately matched will be averaged for multiple times to improve the accuracy of the data.

30204: The positioning speed of the knife is fine; in the automatic tool setting project, when the tool setting position is reached, the parameter value will be used as the speed to touch the tool setting tool.

30205：The tool is quickly in place; during the tool setting, the tool first reaches the tool setting position at a faster speed, which is called the tool setting speed. Change this parameter to change the speed.

30210：Fixed tool position X; when the tool is fixed, the tool position X coordinate.

30211：Fixed tool setting position Y; when fixing the tool setting, the tool position Y coordinate.

30212：Fixed tool setting position Z; when fixed tool setting, the tool position Z coordinate.

31001：No. 1 knife name; the user enters the name of a good knife here, double-click the parameter bar to customize the input. The remaining group tool names are the same.

31002：No. 1 knife type; changing this parameter changes the tool type, the software presets three tools. 1. Straight barrel knife, 2. Vibration to, 3. Drilling knife. Users make more choices. The remaining sets of tool types are the same.

31010：No. 1 knife offset type; this parameter is divided into 1. The position is programmed as the magazine, and 2. The offset is the offset from the workpiece origin. Set this parameter according to the machine type. When the machine is a multi-tool change tool, it is type 1; when the machine is multi-spindle, it is type 2. The zone group tool offset type is the same.

31011：No. 1 knife X position offset; if the tool change is for the tool magazine, the parameter value is the X axis coordinate of the position of the tool magazine; if it is multi-spindle, this parameter is the X coordinate offset of the current tool relative to the workpiece origin. The remaining set of tool X-axis position offsets are the same.

31012：No. 1 knife Y position offset No. 1 knife Y position offset; if the tool change is for the tool magazine, the parameter value is the Y axis coordinate of the position of the tool magazine;

if it is multi-spindle, this parameter is the current tool relative to the workpiece origin Y-axis coordinate offset. The remaining set of tool Y-axis position offsets are the same.

31013：No. 1 knife Z position offset No. 1 knife X position offset; if it is a tool change, the parameter value is the Z axis coordinate of the position of the tool magazine; if it is multi-spindle, this parameter is the current tool relative to the workpiece origin Z-axis coordinate offset. The remaining group tool Z-axis position offset is the same.

7.6 Spindle parameter

This column parameter can set the spindle speed, control mode, spindle open delay and so on.

20001：Maximum spindle speed; maximum speed allowed by the spindle.

20002：The default speed of the spindle; the default speed of the spindle must be the same as the inverter setting.

20003：Spindle start delay; after the spindle receives the start signal, it takes a while to reach the specified speed. During this period, it is necessary to wait for a period of time, which is the spindle start delay. Change the software wait time by changing this parameter.

20004：Spindle off delay; after the spindle receives the shutdown signal, it takes a period of time to completely stop. This period of time is called the spindle off delay. Change the software to wait for the spindle to close by changing this parameter.

20006：Analog minimum voltage value; unit V. When using analog mode, changing this parameter modifies the board output analog to get the minimum value.

20007：Analog maximum voltage value; unit V. When using analog mode, changing this parameter modifies the board output analog to get the maximum value.

20008：Spindle speed control mode; change this parameter to select the spindle control mode. 1. Analog quantity; 2. Multi-speed.

20010：The 0th step speed; the multi-speed speed 0th step speed should be the same as the 0th step speed setting of the inverter.

20011：The 1st step speed; the multi-speed speed 1st speed should be the same as the 1st speed setting of the inverter.

20012：The 2nd step speed; the multi-speed second step speed should be the same as the second step speed setting of the inverter.

20013：The 3rd step speed; the multi-speed third step speed should be the same as the third step speed setting of the inverter.

20014：The 4th step speed; the fourth speed of the multi-speed, should be the same as the fourth step of the inverter.

20015：The 5th step speed; the multi-speed 5th step speed should be the same as the 5th step speed setting of the inverter.

20016：The 6th step speed; the 6th step speed of the multi-speed, should be the same as the 6th speed setting of the inverter.

20017：The 7th step speed; the 7th step speed of the multi-speed, should be the same as the 7th speed setting of the inverter.

7.7 CCD Parameter

This column can display and modify visual related parameters, including pixel equivalent, camera offset, number of recognition points, camera recognition height, camera flip and so on.

53001：X pixel equivalent; shows the X-axis length of the actual physical size represented by the X-axis length of one pixel in the image.

53002：Y pixel equivalent; shows the Y-axis length of the actual physical size represented by the Y-axis length of one pixel in the image.

53003：Camera X offset; the X-axis offset of the camera relative to the spindle, used to determine the spindle position, this parameter needs to be set accurately to ensure the cutting accuracy of the processed product.

53004：Camera Y offset; the Y-axis offset of the camera relative to the spindle, used to determine the spindle position, this parameter needs to be set accurately to ensure the cutting accuracy of the processed product.

53005：Cross test length; when setting the camera offset, the cross test is often used to make the initial settings. The machine will draw a cross on the surface of the material. Modifying this parameter will adjust the length of the cross to suit the situation.

53006：Cross test depth; the depth of the groove milled on the material during the cross test, which is the distance from the Z coordinate of the workpiece origin.

53007：The number of CCD markers; the software supports three-point patrol at four points. The number of patrol points is modified

by this parameter. The user needs to customize the number of patrol points.

53008：The CCD corrects the first Mark point orientation; determines the position of the first point, and the machine patrol sequence rotates counterclockwise. Ensure that the orientation of the first point is consistent with this parameter setting for the machine to work properly.

53009：CCD single point correction accuracy; the pixel value of the parameter value is used as the error allowed in the recognition during the recognition process. The smaller the value, the more accurate the correction, but the more steps it takes to move to the center of the Mark point.

53010：CCD grayscale expectation value; grayscale value of camera exposure.

53011：CCD accuracy; the accuracy of the recognition point, the smaller the parameter, the higher the recognition rate, but it is easy to cause misidentification; the larger the parameter, the lower the recognition rate, but it is not easy to misidentify.

53012：The camera recognizes the Z-axis height; if the camera of the machine is mounted on the spindle, it will change position as the spindle moves up and down. In this case, you need to set this parameter, which is the Z-axis height when the camera recognizes it. Be sure to see the image at this height.

530013：Camera type; software support for two brands of cameras. 1. Daheng, 2. Maidweiwei.

530014：The camera calculates the angle of rotation; the camera may rotate when the camera is rotated during installation. Modify this parameter at this point to make the graphic erect.

530015：Whether the camera image is flipped horizontally; when the camera image is upright, this parameter needs to be changed when the left and right moving images show the opposite direction of motion. 0. No; 1. Yes.

530016：Whether the camera image is flipped horizontally; when the camera image is upright, this parameter needs to be changed when the image is moved up and down. 0. No; 1. Yes.

Appendix

Shortcut list

Hot key	Features	Hot key	Features
Ctrl + N	create new file	R、R	Rounded Rectangle
Ctrl + O	Open DXF file	T、E	Text
Ctrl + Shift +I	Import DXF files	+	Zoom in
Ctrl + Q	Exit	-	Zoom out
Ctrl + Z	Undo	Z、A	Adapt to the window
Ctrl + Y	Redo	Z、P	Drag
Delete	Delete	Q、Q	Select
Ctrl + X	Cut	G、R	Fence
Ctrl + C	Copy	I、P	Two point distance
Ctrl + V	Paste	S、F	Freed
Esc	End operation	S、A	Automatic capture
M、I	Mirror	E、N	Close strict capture
O、F	Offset	E、H	Horizontal capture
D、I	Interrupt	E、V	Vertical capture

X、P	Blast	E、O	Orthogonal capture
B、G	Bridge	PageUP	Anticlockwise rotation
R、N	Chamfer	PageDown	clockwise rotation
D、D	Remove duplicate lines	Numeric keypad 6	Manual X+ mobile
L、I	Two-point line	Numeric keypad 4	Manual X-Mobile
A、3	Three-point arc	Numeric keypad 8	Manual Y+ mobile
C、I	Center circle	Numeric keypad 2	Manual Y-Mobile
P、L	Polyline	Numeric keypad 9	Manual Z+ mobile
R、E	Rectangle	Numeric keypad 1	Manual Z-Mobile

Support G code list

G code number	Features	G code number	Features
G00	Rapid positioning	G54	Set the workpiece coordinates
G01	Linear interpolation	G55	Set the workpiece coordinate two
G02	Clockwise circular interpolation	G56	Set the workpiece coordinate three

G03	Counterclockwise circular interpolation	G57	Set the workpiece coordinate four
G04	Timed pause	G58	Set the workpiece coordinate five
G17	Machining XY plane	G59	Set the workpiece coordinate six
G18	Processing XZ plane	G68	Tool offset inner angle
G19	Processing YZ plane	G69	Tool offset outer angle
G28	Automatically return to reference point	G90	Absolute size
G51	Tool offset	G91	Relative size

Parameter List

Axis parameter

Numbering	Features	Numbering	Features
10001	X-axis direction	11002	Whether it must be returned to zero before processing
10002	Y-axis direction	11005	Whether online must be returned to zero
10003	Z-axis direction	11101	X-axis origin
10101	X-axis pulse equivalent	11102	Y-axis origin
10102	Y-axis pulse equivalent	11103	Z-axis origin
10103	Z-axis pulse equivalent	11201	X coarse positioning direction
10201	X stroke lower limit	11202	Y coarse positioning direction
10202	Y stroke lower limit	11203	Z coarse positioning direction
10203	Z stroke lower limit	11301	X coarse positioning speed
10301	X stroke limit	11302	Y coarse positioning speed
10302	Y stroke limit	11303	Z coarse positioning speed

10303	Z stroke limit	11401	X fine positioning speed
10401	X fine positioning speed	11402	Y fine positioning speed
10402	Y soft limit enable	11403	Z fine positioning speed
10403	Z soft limit enable	11501	X retreat distance
10501	X axis type	11502	Y retreat distance
10502	Y axis type	11503	Z retreat distance
10503	Z axis type	12001	X-axis speed limit
10601	X axis port address	12002	Y-axis speed limit
10602	Y-axis address	12003	Z-axis speed limit
10603	Z axis address		

Manual parameter

Numbering	Features	Numbering	Features
60005	Linear axis manual take-off speed	60013	Y-axis manual low speed
60007	Linear axis manual acceleration	60014	Y-axis manual high speed
60011	X-axis manual low speed	60015	Z-axis manual low speed
60012	X-axis manual high speed	60016	Z-axis manual high speed

Processing parameters

Numbering	Features	Numbering	Features
50001	Processing takeoff speed	50110	Processing end parking mode
50002	Dry running speed	50111	Parking position X-axis coordinates
50003	Processing speed	50112	Parking position Y-axis coordinate
50004	Maximum processing speed	50201	Current quality strategy group
50007	Processing acceleration	50211	Group 0 turning speed adjustment factor
50008	Free motion acceleration	50212	Group 0 acceleration adjustment factor
50009	Turning speed	50213	Group 0 arc speed limit adjustment factor
50010	Smoothing time	50221	Group 1 turning speed adjustment factor
50012	Falling speed	50222	Group 1 acceleration adjustment factor
50013	Arc limit	50223	Group 1 arc speed limit adjustment factor
50014	Short segment reference length	50231	Group 2 turning speed adjustment factor

50015	Short-term acceleration	50232	Group 2 acceleration adjustment factor
50016	Arc difference does not control accuracy	5233	Group 2 arc speed limit adjustment factor
50017	Z safety height	50241	Group 3 turning speed adjustment factor
50018	Z axis falling speed selection	50242	Group 3 acceleration adjustment factor
50100	Whether to use file processing speed	50243	Group 3 arc rate limiting factor
50101	Whether to use the file spindle speed	50251	Group 4 turning speed adjustment factor
50102	Whether the spindle starts automatically when machining starts	50252	Group 4 acceleration adjustment factor
50103	Whether to start the vacuum pump at the beginning of processing	50253	Group 4 arc speed limit adjustment factor

File parameter

Numbering	Features	Numbering	Features
52001	Graphic file lift height	52011	Automatic expansion

52002	Graphics file processing depth	52012	Automatic expansion distance
52003	Automatically delete duplicate lines	52013	Horizontal resolution
52004	Automatically remove repeat line precision	52014	Vertical resolution
52005	Automatically merge connected segments	52016	Whether spline fitting
52006	Automatically merge connected line segments with precision	52017	Z axis workpiece origin setting
52007	Automatically remove tiny graphics	52020	Processing file generation smoothing accuracy
52008	Automatically removes small graphics accuracy	52021	Does JPG retain the inner contour?
52009	Automatic sorting	52022	Contour extraction threshold
52010	Contour extraction threshold	52024	Contour extraction smoothing series

Tool parameter

Numbering	Features	Numbering	Features

30001	Tool magazine capacity	31311	No. 4 knife X position offset
30002	Current tool number	31312	No. 4 knife Y position offset
30003	Tool magazine in place before Y offset	31313	No. 4 knife Z position offset
30201	Tool setting mode	31401	No. 5 knife name
30202	Tool setting thickness	31402	No. 5 knife type
30203	Number of times the knife is positioned	31410	No. 5 tool offset type
30204	Tool positioning speed	31411	No. 5 knife X position offset
30205	Fast setting speed	31412	No. 5 knife Y position offset
30210	Fixed tool setting position X	31413	No. 5 knife Z position offset
30211	Fixed tool setting position Y	31501	No. 6 knife name
30212	Fixed tool setting position Y	31502	No. 6 knife type
31001	No. 1 knife name	31510	No. 6 knife tool offset type
31002	No. 1 knife type	31511	No. 6 knife X position offset
31010	No. 1 tool offset type	31512	No. 6 knife Y position offset
31011	No. 1 knife X position offset	31513	No. 6 knife Z position offset

31012	No. 1 knife Y position offset	31601	No. 7 knife name
31013	No. 1 knife Z position offset	31602	No. 7 knife type
31101	No. 2 knife name	31610	No. 7 tool offset type
31102	No. 2 knife type	31611	No. 7 knife X position offset
31110	No. 2 tool offset type	31612	No. 7 knife Y position offset
31111	No. 2 knife X position offset	31613	No. 7 knife Z position offset
31112	No. 2 knife Y position offset	31701	No. 8 knife name
31113	No. 2 knife Z position offset	31702	No. 8 knife type
31201	No. 3 knife name	31710	No. 8 tool offset type
31202	No. 3 knife type	31711	No. 8 knife X position offset
31210	No. 3 tool offset type	31712	No. 8 knife Y position offset
31211	No. 3 knife X position offset	31713	No. 8 knife Z position offset
31212	No. 3 knife Y position offset	31801	No. 9 knife name
31213	No. 3 knife Z position offset	31802	No. 9 knife type

31301	No. 4 knife name	31810	No. 9 knife tool offset type
31302	No. 4 knife type	31811	No. 9 knife X position offset
31310	No. 4 tool offset type	31812	No. 9 knife Y position offset

Spindle Parameter

Numbering	Features	Numbering	Features
20001	Maximum spindle speed	20010	Multi-speed 0th speed
20002	Spindle default speed	20011	Multi-speed 1th speed
20003	Spindle start delay	20012	Multi-speed 2th speed
20004	Spindle off delay	20013	Multi-speed 3th speed
20005	Spindle off timeout	20014	Multi-speed 4th speed
20006	Analog minimum voltage value	20015	Multi-speed 5th speed
20007	Analog maximum voltage value	20016	Multi-speed 6th speed
20008	Spindle speed control mode	20017	Multi-speed 7th speed

CCD Parameter

Numbering	Features	Numbering	Features

53001	X pixel equivalent	53009	CCD Single point correction accuracy
53002	Y pixel equivalent	53010	CCD Grayscale expectation
53003	Camera X offset	53011	CCD Accuracy
53004	Camera Y offset	53012	The camera recognizes the Z-axis height
53005	Cross test length	53013	Camera type
53006	Cross test depth	53014	Camera image rotation angle
53007	Number of CCD markers	53015	Whether the camera image is flipped horizintally
53008	CCD corrects the first Mark pount orientation	53016	Whether the camera image is flipped vertically

Contact us

You can get our services and technical support in the following ways:

Company Name: Chengdu multidimensional robotization technologycompany

Company Address: No. 406, 4th Floor, Unit 4, Building 1, No. 333, Jiqing 3rd Road, Chengdu High-tech Zone, China (Sichuan) Pilot Free Trade Zone

Phone: 028-64844144

Zip code: 610041

Official website: http://www.mdrobotization.com

Mailbox: 365800795@qq.com

Wechat public account: 成都兴多维科技有限公司