NcStudio V10 Waterjet Cutting CNC System

Users' Manual

1st Edition

Weihong Electronic Technology Co., Ltd.

The information contained in this manual is constantly being updated. You can login to the official website of Weihong Company www.en.weihong.com.cn to download the latest PDF edition for free.

Preface

About This manual

This manual is mainly targeted to machine users. If you use Weihong CNC system for the first time, it is suggested to read through this manual. If not, however, you can search for the desired information via the contents.

With 5 parts, this manual can be divided into 8 chapters, as follows:

1) Part 1: Preface, introducing the precautions about transportation and storage, installation, wiring, debugging, usage and so on. You need to read them first carefully to ensure safe operations.
2) Part 2: Installation, referring to chapter 1. This chapter gives description of NcStudio V10 waterjet cutting CNC system from the perspective of system configuration, installation and wiring, etc.
3) Part 3: Referring to chapter 2 to 6, introduction to software operation. This part gives introduction of software interface's functions and manipulation, lists related parameters and setting instructions. It aims to help users to be familiar with manipulation units and commands.
4) Part 4: Maintenance, referring to chapter 7 which mainly introduces common problems during usage, helps users with corresponding measure to solve them.
5) Part 5: Appendix, referring to chapter 8, which includes the introduction of quick start guide, basic conception of NcStudio and software license agreement for users.

Applicable Product Model

This manual is applicable to NcStudio V10 waterjet cutting CNC system (for 2-5 axis machines). Refer to the table below for details.

Product Model	Remarks
NcStudio V10 waterjet cutting CNC system	1) Herein the system matches control card PM85A/PM95A, controller Lambda 4S/5S/5E, and software NcStudio V10 waterjet cutting CNC system. 2) Lambda 4S/5S/5E can be shortened as Lambda controller or controller.

Contact Us

You can contact us by the following info for technical support and pre-sales / after-sales service:

Company Name: Weihong Electronic Technology Co., Ltd.

Headquarters Address: No.1590, Huhang Rd., Fengxian, Shanghai, PRC 201400

Tel: +86-21-33587550

Fax: +86-21-33587519

Website: http://en.weihong.com.cn

Revision History

You can refer to the following table for the revision records of each edition.

Date	Edition	Revision
2016.08	R1	Released for the first time.

Precautions

Precautions can be divided into caution and warning according to the degree of possible loss or injury in case of negligence or omission of precautions stipulated in this manual.

CAUTION: general info, mainly for informing, such as supplementary instructions and conditions to enable a function. In case of negligence or omission of this kind of precautions, you may not activate a function. Note that in some circumstances, negligence or omission of this kind of precautions could cause physical injury or machine damage.

WARNING: warning info requiring special attention. In case of negligence or omission of this kind of precautions, you may suffer physical injury, or even death, machine damage or other losses.

WARNING

1) Precautions Related to Storage and Transportation

- The products should be transported properly in terms of the weight;
- An excess of specified quantity of stacking products is prohibited;
- Climbing, standing or placing heavy loads on the products is prohibited;
- Dragging or carrying the products via cables or devices connected to them is prohibited;
- The products should be protected from moisture during storage and transportation.

2) Precautions Related to Installation

- Only when this equipment installed in the qualified electricity cabinet can it be used. The construction of the cabinet must reach IP54 grade of protection;
- Paste sealing strips on the joint of the cabinet to seal all the cracks;
- Cable entry should be sealed while easy-to-open on the spot;
- A fan or heat exchanger should be adopted for the heat dissipation and air convection of the

WARNING

cabinet;

- If a fan is adopted, air strainer is a must in air inlet or air outlet;
- Dust or cutting fluids may have access to the CNC device via the tiny cracks and air outlets. Therefore it is necessary to pay attention to the surroundings and air flow direction of the air vent to make sure that the outflow gas is towards pollution source;
- 100 mm space should be preserved between the back of the CNC device and the cabinet wall for plugging cable connected with the device and the ventilation & heat dissipation in the cabinet;
- Space between this device and other equipment should also be preserved according to the requirements;
- The product should be installed firmly and without vibration. During installing, casting, knocking, striking, or loading on the product is forbidden;
- To reduce electromagnetic interference, power-supply components used should be above AC or DC 50V and the space between cable and CNC device should be preserved above 100mm;
- It will be better if CNC device is installed at a position facilitating debugging and maintenance.

3) Precautions Related to Wiring

- Only qualified people are allowed to participate in the wiring and checking;
- The CNC device should be grounded reliably and grounding resistance should be less than 4 ohm. Neutral line is absolutely not allowed to replace earth wire. Otherwise, it may result in malfunction of the device due to the interference;
- Wiring should be firm and steady, or misoperation may occur;
- Voltage values and positive & negative polarity of any connection plug should be in accordance with specifications set forth in the manual, or it may result in breakdowns such as short circuit and permanent damage to the device;
- To guard against electric shock or CNC device damage, fingers should keep dry before plugging or touching switch;
- The connecting wire should not be damaged and squeezed, or the leakage or short circuit may occur;
- It is prohibited to plug or open the chassis of CNC device when power on.

4) Precautions Related to Running & Debugging

- Parameters setting should be checked before running, since wrong setting may lead to accidental movements;

WARNING

- Modification to parameters should be within the allowable range, or such breakdowns as unsteady running and machine damage will occur.

5) Precautions in Use

- Before power-on, please make sure that the switch is on blackout to avoid occasional start-up;
- Please check the electromagnetic compatibility during electrical design in order to avoid or reduce electromagnetic interference to the CNC device. A low pass filter should be employed to reduce electromagnetic interference if there are other electrical devices nearby;
- It is not allowed to frequently power on and power off. It is recommended to power up the machine again at least one minute later after power failure or blackout.

CAUTION

1) Precautions Related to Product and Manual

- Matters related to restrictions and functions available stipulated in the manuals issued by the machine manufacturer are prior to those in this manual;
- This manual assumes all the optional functions are available, which you must confirm through manuals issued by the machine manufacturer;
- Please refer to manuals issued by the machine manufacturer for the instructions of machine tools;
- Functions, and software interfaces vary with the system and the version of software. Before using the system, you must confirm the specifications.

2) Precautions When Opening the Package

- Please make sure that the products are what you have ordered;
- Check if the products are damaged in transit;
- Check if the components and accessories are damaged or missing in terms of the detailed list;
- Please contact us promptly if product discrepancy, accessory missing or transit damage occurs.

Contents

1 System Installation

1.1 Host Computer Configurations

CPU:	Main frequency 1G or above
Memory:	512M or above
Hard disk:	20G or above
Graphics card:	supports resolution of 1024*768 at least
Display:	14" VGA or above
CD-ROM:	4x CD-ROM driver or above (optional)
Mainboard expansion slot:	1 PCI/PCIE slot or above (PCI for PM85A and PCIE for PM95A)

1.2 Installation of NcStudio

NcStudio system includes two parts: the software and motion control card. As a result, the setup of the system is also divided into two stages: the software setup and the motion control card setup.

Before installing a new version of NcStudio, please uninstall the old version. Refer to section 1.3 for uninstallation. You can also install a new version of software directly, as shown in section 1.2.1.

1.2.1 Software Setup

1) Power on the computer to enter the operating system automatically. Close other running applications, if any.
2) Put the setup CD of NcStudio system into the CD-ROM. Double click the “My computer" icon, then double click the CD-ROM icon. Find out the SETUP.EXE file and double click its icon  .The first dialog box comes the [Language of the theme] dialog. If you click [English], the system will run in English as default. Likewise, if you click [选择中文界面], the system will run in Chinese. Besides, switchover between languages can be done in the system.
3) In order to avoid disturbance, the system will ask you to save parameters and delete the old version before going on further installation.
4) By clicking [OK], installation begins. A dialog box will pop up and ask “if users want to save the former parameters of software”, where you can transfer parameter settings of last version of the software to the new version, saving troubles of repeated setting. Make your decision according to your own needs.
5) NcStudio will be installed to the default directory, C:\Program Files\Naiky. Installation progress bar will show the installation progress.

6) During the installation process, it is suggested to choose [No] when the system prompts “whether to place generate files and executive files separately”.
7) After installation, it is suggested to shut down the computer when the system prompts “shutdown your computer now?” Then the software setup is finished.

CAUTION

If it is the first time for installation, step 3) and 4) can be skipped, for your information.

1.2.2 Installation and Connection of Control Card

Here are steps of motion control card installation:

1) Power off host computer’s power supply, open the cover, and insert the motion control card into an empty and matched PCI/PCIE slot. While installing the motion control card, gently press the two flanks of motion control card, to make sure that the motion control card is firmly inserted into the slot, and well connected with the computer baseboard, dependable without fluttering. Then, tighten the locknut of the card and finally lid the cover. The installation of motion control card is completed.
2) Similar to the installation of control card, insert peripheral components into the related slots or ports and make sure they are firmly fixed without fluttering.
3) The installation is completed. Restart the computer.

Wiring and connection of communication card, Lambda controller and other devices will be shown in Fig. 1-1:

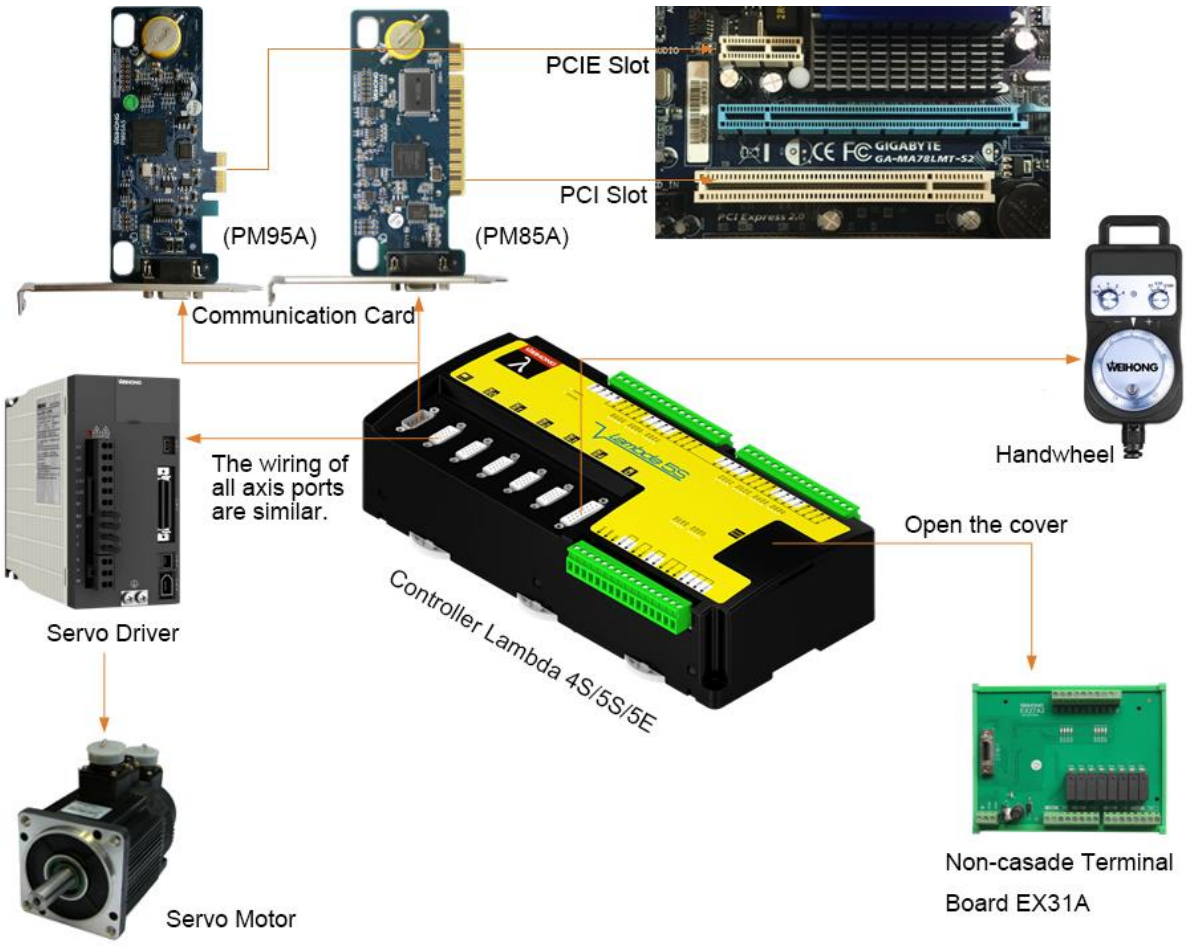


Fig. 1-1 Overall wiring diagram

1.2.3 Update the Hardware Driver Manually

After installing motion control card and other devices, you need to manually update hardware driver; or the software will not run properly. Steps are shown as follows:

1) Right click [Computer], select [Properties]→[Device Manager]. In [Device Manager] window, find [运动控制器] item and click the ▷ icon, then right click the item under [维宏运动控制器(PM 系列)] item. After that, select [Update Driver Software…] to start updating the hardware driver as shown in Fig. 1-2.

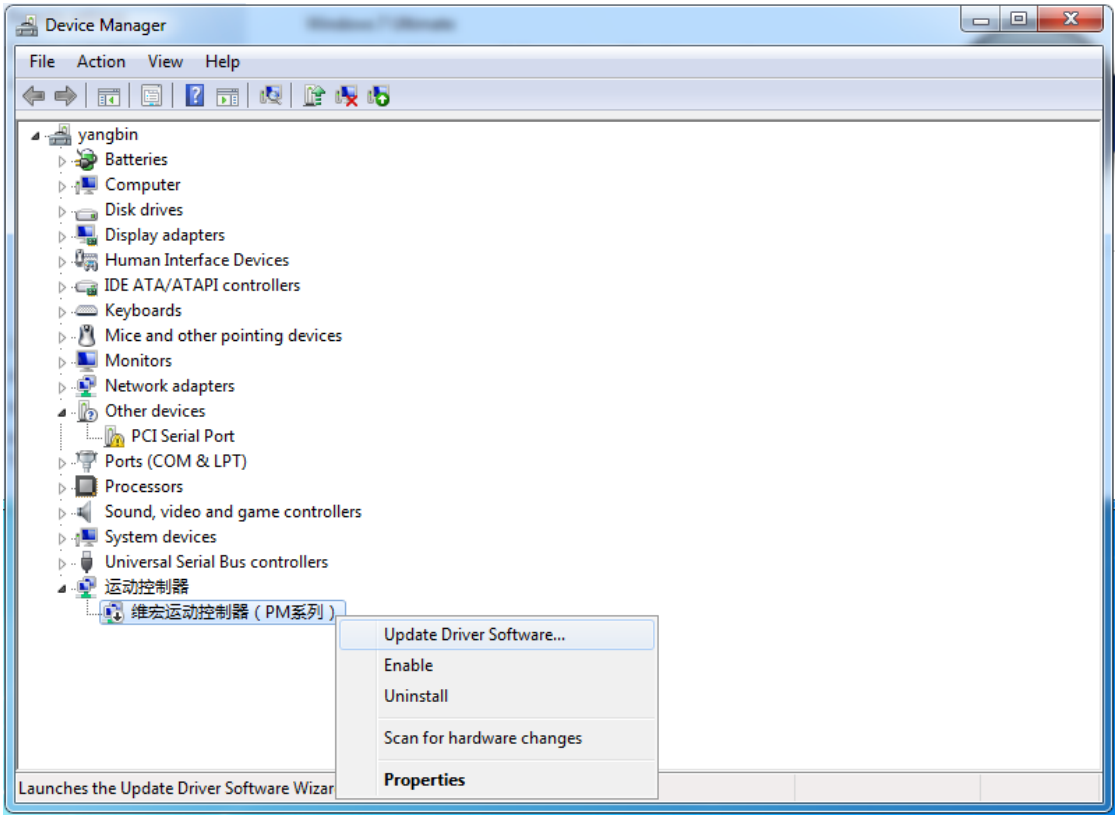


Fig. 1-2 Update driver software

2) In [Update Driver Software] window, click “Browse my computer for driver software”. See Fig. 1-3:

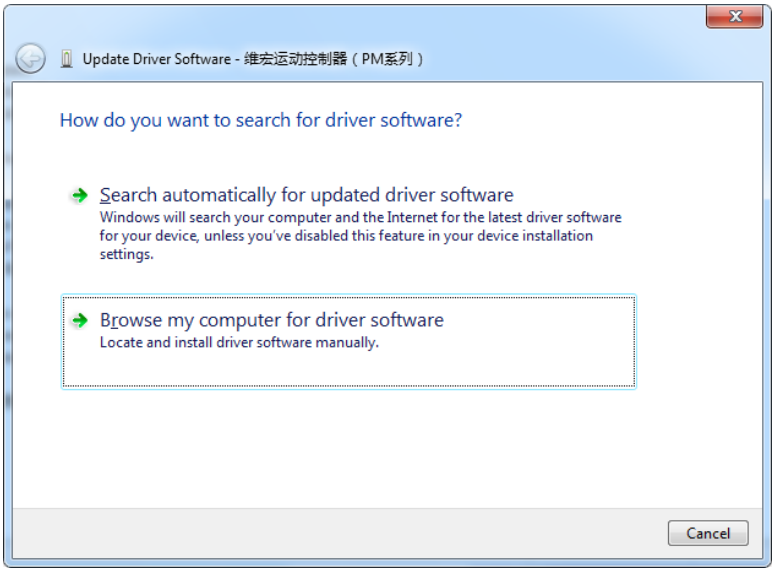


Fig. 1-3 Hardware update wizard

3) Select “Let me pick from a list of device drivers on my computer” and then click “Next”. See Fig. 1-4.

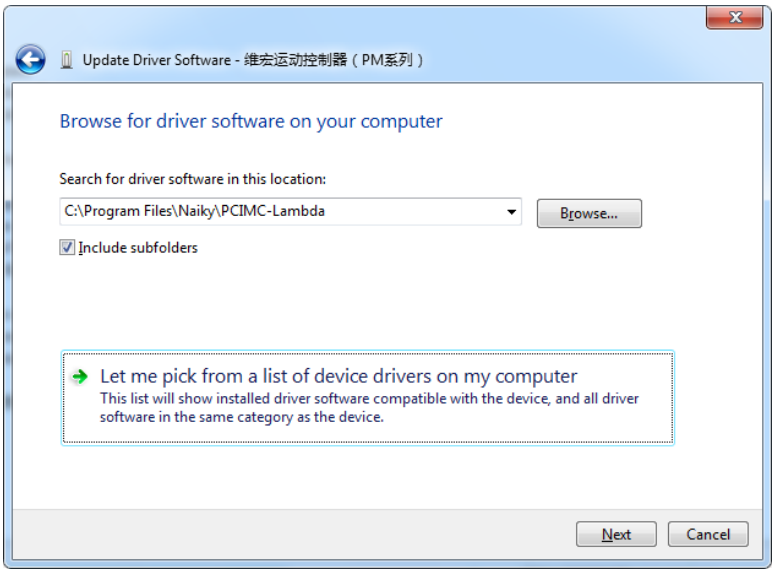


Fig. 1-4 Driver installation options

4) As shown in Fig. 1-5, click [Have Disk…], and then a new window pops up as shown in Fig. 1-6.

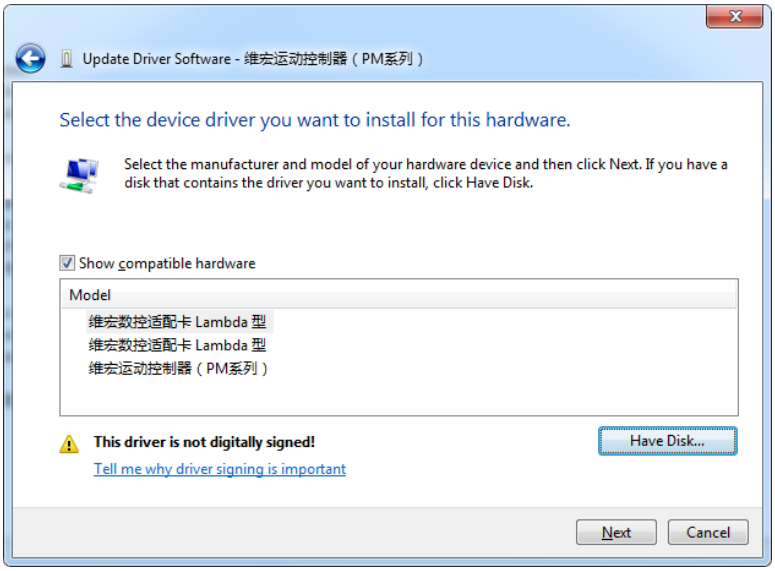


Fig. 1-5 Find driver location-1

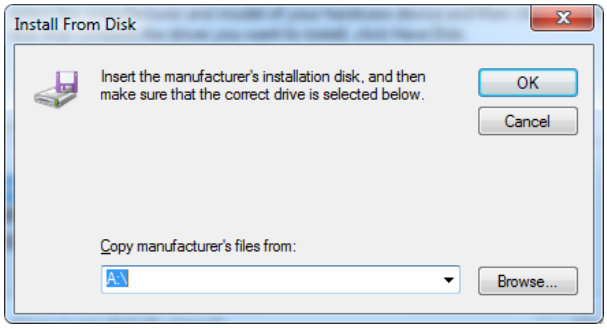


Fig. 1-6 Find driver location-2

5) Click the [Browse] button in dialog box “Locate File”, and select the target .inf file, shown as Fig. 1-7.

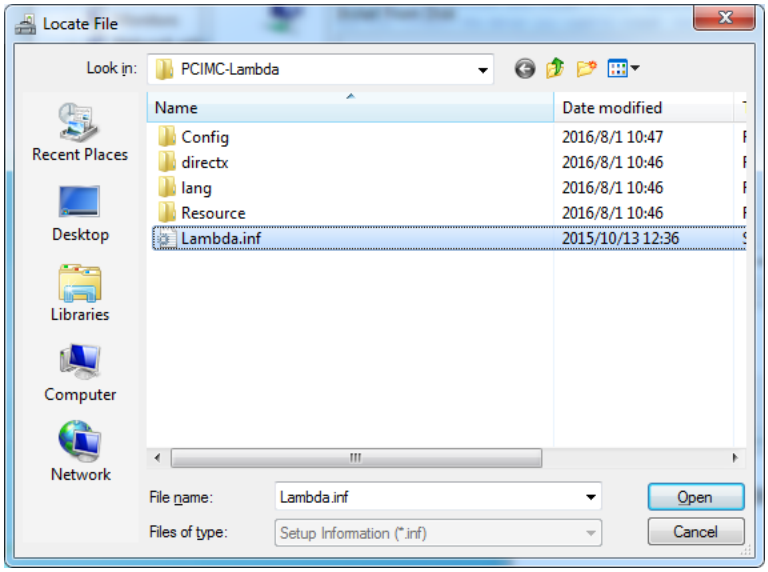


Fig. 1-7 Find driver location-3

6) After selecting the corresponding driver file, click [Open] to return to the previous page, showing the file directory under “Copy manufacturer’s files from”, see Fig. 1-8.

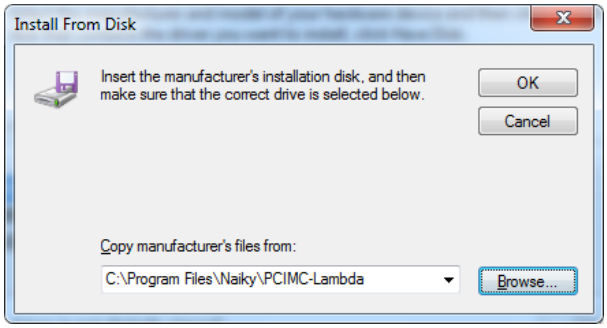


Fig. 1-8 Confirm the file directory

7) Click [OK], and then click [Next] to start updating the hardware driver.
8) After updating is completed, click [Close] to complete the update of the hardware driver.

1.3 Uninstallation of NcStudio

NcStudio is a green software. To uninstall NcStudio, you only need to delete its folder under directory of *C:\Program Files\Naiky*, and delete the NcStudio shortcut icon on the desktop as well as in the Start menu.

2 Software Overview

The NcStudio waterjet cutting CNC system is independently developed and owned by Weihong Electronic Technology Co., Ltd. It can directly support G code, DXF format, PLT code format and ENG code of JDPaint processing files that are created by CAD/CAM applications such as UG, MasterCAM, CASMate, ArtCAM, AutoCAD, and CorelDraw, etc. Based on PC operating system, the interface of NcStudio is simple, reliable and user-friendly.

2.1 Software Features

- Supports auto machining. Completely supports G code of ISO standard, HP PLT format, JDPaint ENG format and DXF format.
- Supports manual machining. You are allowed to control the machine tool not only by handheld device but also by input with computer devices, such as keyboard and mouse.
- Supports incremental feeding for your convenience to set accurate feed amount, and step-size can be adjusted.
- Supports MDI (manual data input) function. You can directly input G code online and immediately execute it.
- Supports single block mode. You can set the program to be executed as [Single Block] (also available in the menu), which is helpful for diagnosis and failure recovery.
- Supports advanced auto function of breakpoint memory and program block skip execution.
- Supports saving / restoring workpiece origin.
- Supports returning to the reference point accurately.
- Supports feedrate override on-line adjustment. During the machining process, you can adjust the feedrate override at any time. The default minimal value is “0”, indicating the processing is suspended and the maximal value is 120%. You can change the minimal or maximal feedrate override by modifying the related parameters.
- Supports high-speed smooth connection algorithm. In a general CNC control system, connection speed between two G codes is usually fixed (e.g., it may be 0 or a small value). However, the new system adopts a particular adaptive prediction algorithm of processing speed. The algorithm adaptively decides the connection speed between the current command and the next command according to speed, direction and the maximal acceleration, and look-ahead function. It has greatly increased the machining efficiency from 30% to 300% and improved the machining performance by eliminating the speed chatter marks on workpiece surface.

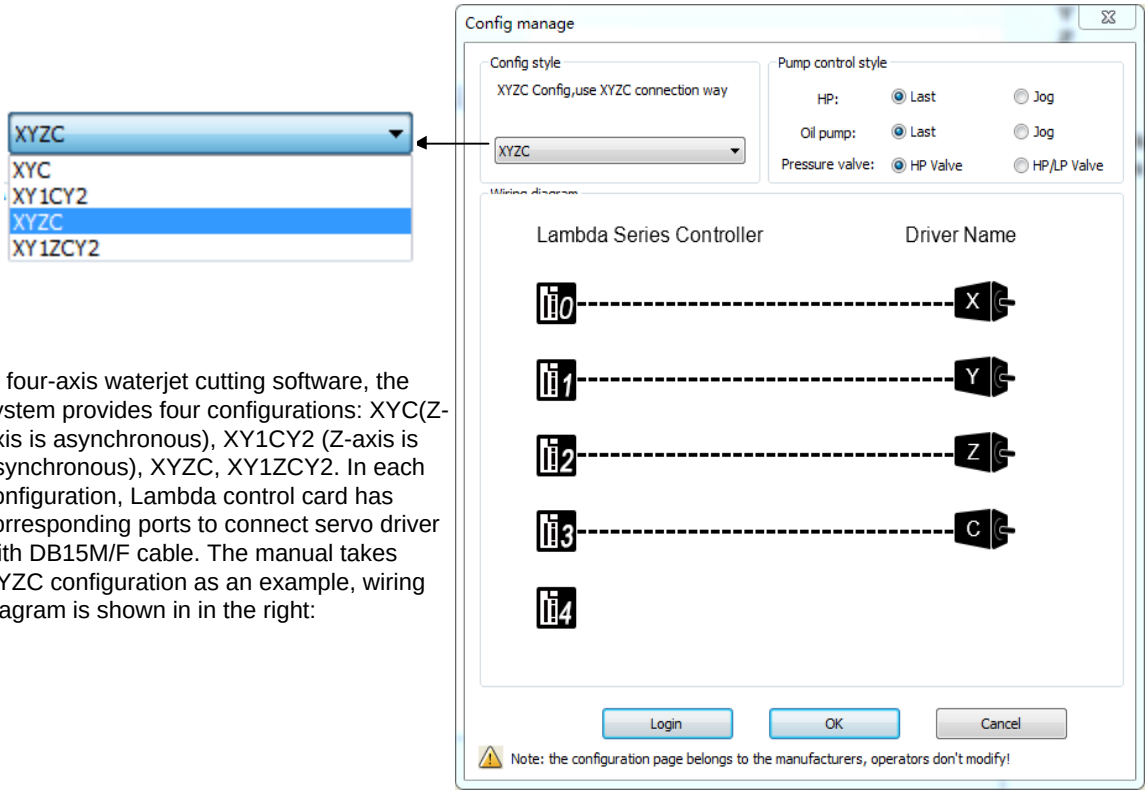
- Supports simulation display. You can observe the machining result from different angles to understand the processing result more accurately and intuitionally with simple operation.
- Supports pre-loading machining path. After the processing file is loaded, the system will automatically analyze the processing paths. You can directly view the paths.
- Supports simulation function. The system can simulate machining process and will finish it in short time. Meanwhile, it can check if there is any mistake in the program and if the processing result is satisfactory. Moreover, it can calculate the actual processing time precisely.
- Supports strong log function, which is very helpful for you to view the detailed processing information and system diagnoses.
- Supports built-in machining file manager. You only need to save files into a specified directory. And NcStudio will administrate the files in the built-in file manager.
- Supports built-in machining file editor. You can edit and modify machining files in the file editor at any time.
- Supports displaying machining information. By simulation or actual machining, machining information window can help you count some important information, such as machining time, machining range, finished workpiece count, etc.
- Supports parameter auto backup function. The system can automatically save parameter settings, which saves you from setting parameters each time.
- Supports backup of 8 groups of parameters. You can save different parameter settings for different processing files, and can read them any time after you save them.
- Supports mapping simulation, test and polarity modification of I/O ports. Through this function, you can realize better hardware detection, processing supervision, and error diagnoses.
- Supports PLC module, with which I/O ports are controlled.
- Supports intelligent look-ahead function. The system will analyze in a distance ahead or behind the current point, and decide an interpolation strategy to improve the smoothness of whole workpiece.
- The maximal length of machining file is 700M byte. Recommended configuration is PC with 32-bit operating system and 2G RAM.
- Supports control of sand amount override. The range is 0-100%.
- Supports auto control valve on/off sequentially, and valve on/off delay.
- Supports MPG (manual pulse generator). You can determine whether to use MPG or not according to your actual needs.
- Supports multi-languages. Currently, English and Chinese are embedded in the software. Other languages will be added in the future.
- Supports parameters transfer between different versions of software.
- Supports height measurement.

2.2 Configuration Management

NcStudio waterjet cutting CNC system supports motion control over machines varying from 2~5 axes.

As name implies, 2 axes system supports motion control of 2 axes at most, namely, XY type of kinematics; 3 axes system supports motion control of 3 axes at most, XYZ type of kinematics; 4 axes system supports motion control of 4 axes at most, XYZC type of kinematics; 5 axis system supports motion control of 5 axis at most, that is, XYZAB type of kinematics and XYZAC type of kinematics. See section 3.1.1 for details.

Following is an example of 4 axes control system. Click [Config Manage] under [Machine Tool] menu in accordance with machine structure, as shown in Fig. 2-1.



In four-axis waterjet cutting software, the system provides four configurations: XYC(Z-axis is asynchronous), XY1CY2 (Z-axis is asynchronous), XYZC, XY1ZCY2. In each configuration, Lambda control card has corresponding ports to connect servo driver with DB15M/F cable. The manual takes XYZC configuration as an example, wiring diagram is shown in in the right:

Fig. 2-1 Configuration management

In addition, Series or Jog mode can be selected in line with control method of high pressure and oil pump, with series mode by default. Only ports of “HP On” and “Oil Pump On” need to be connected in wiring; high pressure or high/low pressure control method can be designated for pressure valve.

CAUTION

1) If you need to change configurations, click “Login” button. Manufacturer’s previlege is required. Note that unauthorized change is prohibited.

2) Without special explaination, screenshots provided in chapter 2 and 3 are examples of 4 axes system.

2.3 Interface of NcStudio

NcStudio and NcEditor interfaces will pop up at the same time when you open the software NcStudio. NcStudio is mainly used for manipulation during machining process; NcEditor provides basic editing functions, mainly used for editing of tool path or CAM post-processing. The default interface of NcStudio is shown in Fig. 2-2:

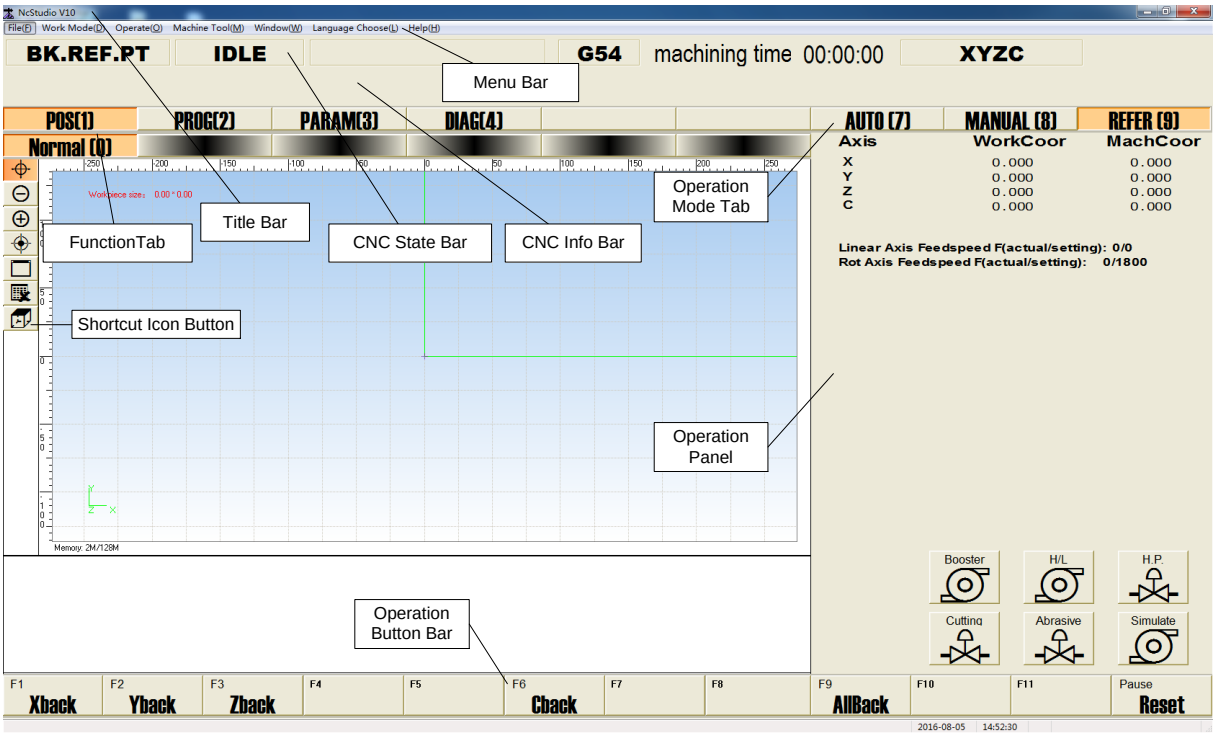


Fig. 2-2 Interface of NcStudio

As shown above, there are title bar, menu bar, CNC state bar, CNC info bar from the top to the bottom. The largest area in the center of the software is the function window. Above the function window are function buttons and below the function window are operation buttons. On the right of function window is the operation panel. Above the operation panel are operation mode buttons.

◆ **Title bar**

It is used to show the name of the application. The color of title bar indicates whether the window is activated.

◆ **CNC state bar**

The embedded prompt boxes shown on the CNC state bar range from operation mode, operation state and other appended information, to configuration information. In the middle-right, there is the processing time. It offers an intuitional way to learn the current machine state and the processing time, in order to decide the next operation.

◆ Menu bar

The menu bar includes several concealed drop-down menus; each drop-down menu is composed of several menu items which can realize a special function, action, or state that is relative to a certain procedure. Menus are different in different modes. Below are the menus in auto mode.

File	Work Mode	Operate	Machine Tool	Window	Language Choose	Help
Open and Load...	Auto Mode	Single Block	Open Valves Order ▶	Normal	Chinese	About NcStudio
Unload	Manu Mode	Handwheel Guide	Close Valves Order ▶	Program	English	Manufactory Customize
New	Jog	WorkCoor	Open/Close Booster	Information		
Open and Edit...	Handwheel	Offset	Open/Close H/L	Setting		
Edit Current Machining Program	Stepping*0.01	Set Workpiece Origin	Open/Close H.P.	User Command		
Load Objects	Stepping*0.1	Save Workpiece Origin	Open/Close Cut	Harddisk list		
Stop Loading Objects	Stepping*1	Load Workpiece Origin	Open/Close Abrasive	USB disk list		
Clear Objects	Stepping*10	Set Current Origin	Hipressure Time Statistic	History		
Recent Loaded Files	Custom Stepping	Start	Statistics Information	General Parameters		
Generate Installation(M)	Back to Reference Point Mode	Pause	Deadline Manage	Waterjet Parameters		
Generate Emute Installation		Stop	Config Manage	Parameters Backup		
Restart Software		Simulation Mode		Parameters Auto Backup		
Shutdown System		NearPoint Processing Block...		Log		
Reboot System		Breakpoint Resume		IO Port		
Show Desktop		Circumrotate Mirror Process		PLC		
Exit		Back to Workpiece Origin		NcEditor		
		Back to Fixed Point				
		All Back to Reference Point				
		Reset				

Fig. 2-3 Menus of NcStudio

◆ **CNC Info bar**

Three types of message will be displayed in this area, that is, normal prompt, warning prompt and error prompt information.

- Normal Prompt: it refers to information related to the normal running of the system, with bar shown in current background color and texts in black.
- Warning Prompt: It refers to information that needs to be well aware by the user, with bar shown in yellow and texts in black.
- Error Prompt: It shows system errors which may lead to operation discontinuance, such as ESTOP alarm, limit alarm, file error, software error, and so on. The bar will be shown in red and texts in white.

◆ **Operation mode tab**

Three modes including auto mode, manual mode and reference point mode are offered in the system. You can switch between them by clicking corresponding buttons. In different modes, function selection buttons, operation panels and manipulation button bars differs.

In auto mode, the system can automatically analyze the loaded program to control the machining of target workpiece. In manual mode, you can control the machine by manual operation. REFER mode (reference point mode) is a process to synchronize the internal coordinates and the actual machine coordinates. Each time the system starts, it turns to REFER mode by default, which is convenient for the user to conduct homing operation.

2.4 Interface of NcEditor

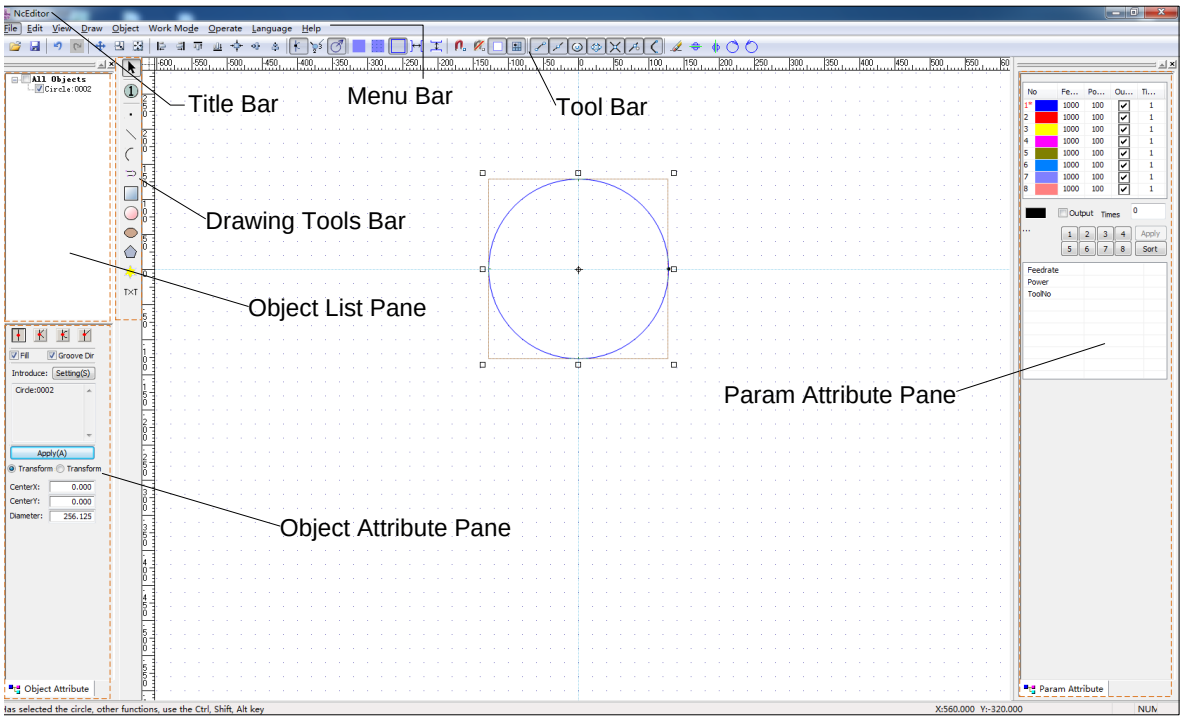


Fig. 2-4 Interface of NcEditor

NcEditor interface is as shown above. You can turn to parameter window [General]-[Manufacturer] and set parameter [N87000] “Interface Show Type” to decide separate or simultaneous opening of NcStudio and NcEditor.

In NcEditor, from the top to the bottom, there is title bar, menu bar and tool bar. The white central part of interface is object editing space. Drawing tool bar on the left is correspondent with submenus of [Draw] menu. Objects in object list pane are correspondent with objects in objects editing space.

◆ **Object attribute pane**

Object attribute pane is mainly used to set the attributes of currently selected objects, including setting of lead line, fill or not, the precise position of object in the coordinate system, and machining direction, etc.

◆ Menu bar

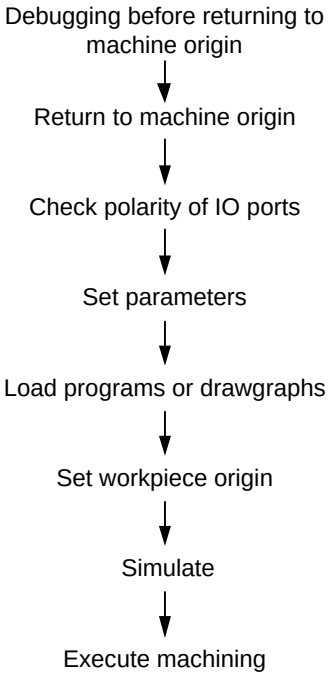
The figure of menu is shown in the below for your reference.

File	Edit	View	Draw	Object	Work Mode	Operate	Language	Help
New	Undo	Toolbar	Draw Point	Delete Object	Auto Mode	Single Block	Chinese	About
Open	Redo	Statusbar	Draw Line	Combine Object	Manu Mode	Handwheel Guide	English	Manufactory Customize
Open Text File	Select All	Object List	Draw Arc	Combine Object Tolerance Setting	Jog	AutoBack		
Import G Code File	Insert NCE Code File...	Draw Toolbar	Draw Polyline	Align ►	Handwheel	Set Workpiece Origin		
Import DXF Code File	Insert G Code File...	Parameters bar	Draw Rectangle	Auto Enter Set Machining Order	Stepping*0.01	Save Workpiece Origin		
Import ENG Code File	Insert DXF Code File	Machine control bar	Draw Circle	Machining Order List	Stepping*0.1	Load Workpiece Origin		
Import PLT Code File	Insert ENG Code File	NcStudio	Draw Ellipse	Machining Order List	Stepping*1	Start		
Save	Insert PLT Code File	System Parameters	Draw Polygon	Set Introduce	Stepping*10	Pause		
Save as...		Normal Parameters	Draw Star	Group	Custom Stepping	Stop		
Save Selected Objects as ...		View Translate	Draw Text	Break Group	Back to Reference	Simulation Mode		
Recent Loaded NCE Files		View Zoom	Measure distance	Rect Array		NearPoint Processing Block...		
Recent Loaded Text Files		Fit to Window	Set Origin	Circle Array		Breakpoint Resume		
Generate Installation		Catch Option		Delete Same Object		Back to Workpiece Origin		
Generate Emute Installation		View Introduce		Add Bridge		Back to Fixed Point		
Restart Software		View Order		Chamfer		Back to Reference Point		
Shutdown System		View Direction		Tool Path		Reset		
Reboot System		Ordinary		Delete Path		Mach. Selected Objects		
Show Desktop		Translucent		Clockwise		Mach. By Color		
Exit		Line Rim Dodel		Counter Clockwise		Mach. Current Group		
		Set Machining Order		Single Block		Mach. From Selected Objects		
		View Machining File Info.		Series Machines		Locate Work Coor...		
				Mach. Selected Objects				
				Clear Process Trace				
				Text Translate to Graphic				
				Dispart CadPath				
				Ellipse Translate to Polyline				
				Show Groove Direction				
				Option				

Fig. 2-5 Menus of NcEditor

3 Software Operation

After the motion control card and software are installed properly according to section 1.2, the system is ready for use. Software operation process is shown below:



3.1 Debug before returning to machine origin

3.1.1 Adjust of Axis Direction

Firstly, to decide each axis's positive direction according to the right-hand rule. The right-hand coordinate system is shown in the Fig. 3-1.

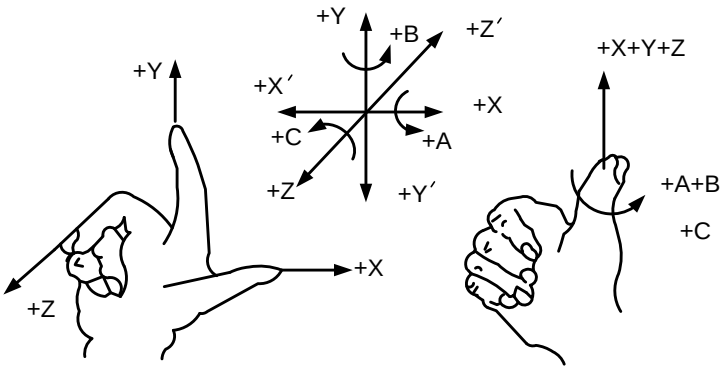


Fig. 3-1 Right-hand rule coordinate system

For cutting machine, the direction of machine tool axes is decided by both the type of machine tool and the layout of each component. The basic coordinate axes of a cutting machine are X-axis, Y-axis, and Z-axis:

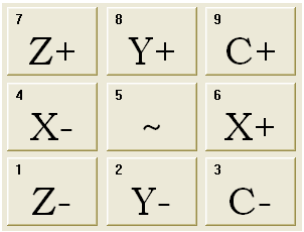
——the direction of Z-axis moving away from workpiece is the positive direction of Z-axis (+Z);

——X-axis is perpendicular to Z-axis and parallel to the clamped surface of workpiece. For a single column machine, if you face the waterjet nozzle and look in the column direction, the direction of right moving will be the positive direction of X-axis (+X);

——X-axis, Y-axis and Z-axis together constitute a right-hand rule coordinate system;

Please refer to chapter 5 for definitions of A-axis, B-axis and C-axis.

After determining the axes direction you must adjust coordinate axes. In manual mode, users can move the machine tool through pressing the operation buttons in the operation panel or the numeric keyboard. And then check whether the actual moving direction of the machine tool is consistent with the direction shown on operation buttons. Below is an illustration of the operation panel in manual mode.



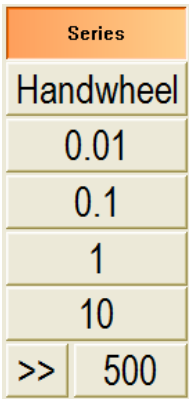
The figure on the left is numeric direction buttons. Numbers appear on the upper left stand for shortcut keys, and "~" key stands for rapid, whose shortcut key is number 5 (Here, it refers to numeric keypad area, hereinafter referred briefly as "num 5"). Note that it should be used with other buttons. For example, to make X-axis moving in negative direction at low speed (also called normal jog speed on some occasions), you can click [X-] or press num 4 directly; while to make X-axis moving in negative direction at high speed (namely, the rapid speed), you need to press num 4 and num 5 at the same time.

Manual mode is subdivided into jog mode, handwheel mode, and stepping mode (or incremental mode).

1) Handwheel mode: the [Handwheel] button is added to support handwheel (MPG—Manual Pulse Generator). In manual mode, after connecting a handwheel, you can click the button to manipulate the machine tool with a handwheel. When the button is pressed, axis direction buttons are disabled.

2) Jog mode: when the current active window is in MANU.JOG mode, click an axis button on the panel and hold, or press down the numeric key on the keyboard. When the button or numeric key is clicked or pressed down, the machine tool starts moving; when the button is not clicked or the key is not pressed anymore, the machine tool stops.

3) Stepping mode (0.01, 0.1, 1, 10, customized step): different from jog mode, stepping*(0.01, 0.1, 1, 10, or custom stepping length) mode can accurately control the feed distance of motion axes. You can select a stepping length level and then click the direction button or push the numeric key once. The machine tool will move the stepping length in the direction. For example, if you select level “0.1”, the stepping length will be 0.1mm. If you click button **>>**, an input box “Please Input Custom Stepping Length” will pop up. You can input a length in the box. The custom stepping length you have set will show on the right of button **>>**. For example, if you enter “500” in the input box, the custom stepping length 500 will show automatically on the right of button **>>**. You just need to click button [500] to trigger the machine tool to move 500mm in the axis you select.



In jog mode, you can press down several keys at the same time to achieve multi-axis motion. However, the multi-axis motion is limited by conflict of keys. In stepping mode, only single-axis motion is supported.

☞ **Related parameters**

Parameter		Range	Meaning
N10000~N10005	AXIS_DIR (X/Y/Z/A/B/C)	1; -1	The moving directions of X/Y/Z/A/B/C axis.

3.1.2 Adjust Pulse Equivalent

Pulse equivalent (P): It refers to the moving distance of workbench or the rotation degree of rotary axis with one pulse sent by CNC device, and it is the minimum available distance controlled by CNC system as well. The pulse equivalent negatively correlates with the machine precision and surface quality of workpiece; while it positively correlates with the feed speed. Under the condition of meeting the demand of feeding speed, lower pulse equivalent should be set. The relationship between maximal feedrate and pulse equivalent is as follows:

Max. feedrate = pulse equivalent × 60 × frequency

For example, the hardware frequency of Lambda 4S is 1MHz, assuming the pulse equivalent is 0.001mm/p, then:

Max. feedrate =0.001 × 60 ×1000000= 60m/min

Mechanical deceleration (m/n): the ratio of reducer input speed to output speed, equal to the ratio of the teeth number of driven wheel to that of driving wheel. When applied in CNC machines, it specifies the

ratio of motor speed to screw speed.

$$\text{Mechanical deceleration ratio} = \frac{\text{Reducer input speed}}{\text{Reducer output speed}} = \frac{\text{Teeth No. of driven wheel}}{\text{Teeth No. of driving wheel}} = \frac{\text{Motor speed}}{\text{Screw speed}}$$

Pitch (d): The axial distance between the corresponding points of two adjacent teeth on the threads.

The calculation of pulse equivalent varies with different motor systems.

◆ **Stepping motor**

In general, firstly set the subdivision and then calculate the pulse equivalent. You can set the pulse equivalent before calculating subdivision. Their relationship can be shown as:

$$\frac{d}{p} = \frac{360}{\theta} \times x \times \frac{m}{n}$$

Hereinto, p stands for pulse equivalent, x represents subdivision of stepping motor, while θ refers to stepping angle. Therefore,

$$\text{Pulse equivalent} = \frac{\text{Screw pitch}}{\frac{360}{\text{Stepping angle}} \times \text{Subdivision} \times \text{Mechanical deceleration ratio}}$$

For instance, the selected screw lead of X-axis for a certain type of machine tool is 5mm, and the stepping angle of stepping motor is 1.8 degree, with “10” subdivision and motor directly connected with screw by coupling. Thus, the pulse equivalent of X-axis is:

$$\text{Pulse equivalent} = \frac{5\text{mm}}{\frac{360}{1.8} \times 10 \times 1} = 0.0025\text{mm/p}$$

◆ **Servo motor**

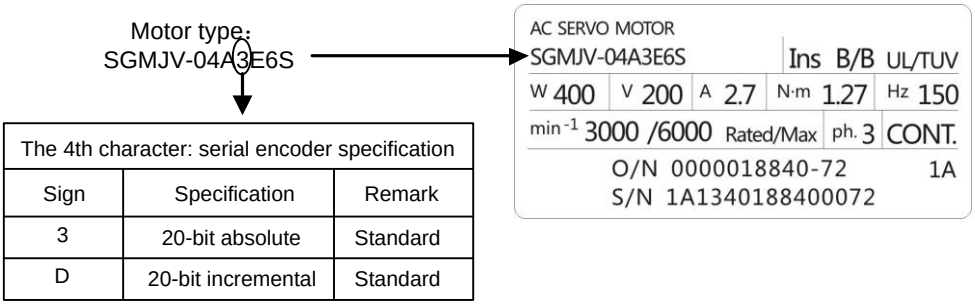
In general, set the default of pulse equivalent as 0.001mm/p and calculate electronic gear ratio (B/A). Their relationship can be shown as:

$$\text{Electronic gear ratio } \frac{\text{B}}{\text{A}} = \frac{\text{Encoder resolution}}{\frac{\text{Screw pitch}}{\text{Pulse equivalent}}} \times \text{Mechanical deceleration ratio}$$

Namely, $\frac{B}{A} = \frac{F \times p}{d} \times \frac{m}{n}$

Electronic gear ratio (B/A): the parameter of servo driver (take YASKWA driver as an example, B is PN202, while A is PN203), referring to the increase or decrease of pulse frequency received by the servo from CNC system. If the value of B/A is greater than 1, it means to magnify or scale up; otherwise, to minify or scale down. For example, aussming the pulse frequency sent by CNC system is 100HZ, the numerator of electronic gear ratio (B) is 1 and the denominator is 2, the actual running speed of the servo is 50HZ. On the contrary, if the numerator is set as 2 while denominator is 1, the actual running speed turns to 200HZ.

Encoder Resolution (F): needed pulse number for one circle of servo motor Please see the servo motor label plate and then refer to the corresponding manual to confirm its encoder resolution. YASKAWA SGMJV type motor is shown as follow, and the 4th character in motor type is the serial encoder specification, so the resolution of this motor is 2^{20}, i.e. 1048576.



The 4th character: serial encoder specification		
Sign	Specification	Remark
3	20-bit absolute	Standard
D	20-bit incremental	Standard

Fig. 3-2 Label of YASKAWA servo motor-encoder

For instance: (an example of YASKAWA servo) screw pitch of a certain type of machine is 5mm, with 17 bit encoder resolution, “0.0001mm/p” pulse equivalent and “1:1” deceleration ratio.

$$\text{Electronic gear ratio} = \frac{\text{PN202}}{\text{PN203}} = \frac{2^{17}}{5/0.0001} \times 1 = \frac{8192}{3125}$$

◆ **Rotary axis**

The pulse equivalent of rotary axis refers to the pulse unit drives the rotary angle of the axis who undertakes workpiece clamping. The difference between rotary axis movement and linear axis movement is that the screw pitch of rotary axis is 360°. Therefore, you need to replace screw pitch with 360 during calculating rotary axis pulse equivalent.

That turns out:

1) For stepping motor:

$$\text{Pulse equivalent} = \frac{360}{\frac{360}{\text{Stepping angle}} \times \text{Subdivision} \times \text{Mechanical deceleration ratio}}$$

2) For servo motor:

$$\text{Electronic gear ratio } \frac{B}{A} = \frac{\text{Encoder resolution} \times \text{ Pulse equivalent}}{360} \times \text{Mechanical deceleration ratio}$$

☞ **Related parameters**

Parameter		Range	Meaning
N10010~N10015	AXIS_PULSE_FACTOR (X/Y/Z/A/B/C)	1e-009~999	It refers to the displacement or angle generated on the relative feed axis per control pulse.

CAUTION

1) The setting value of pulse equivalent must match with that of the electronic gear ratio of servo driver or that of subdivision of stepping driver.

2) Generally, pulse equivalent of rotary axis is recommended to be 3~5 times that of linear axis.

3.1.3 Set Upper & Lower Limits of Workbench Stroke

Workbench stroke refers to the valid machining stroke range of machine tool in X/Y/Z/A/B/C direction. Through setting the workbench stroke limits in the software, the machine tool can be protected.

You can set the limits according to the actual size of machine tool.

☞ **Related parameters**

Parameter		Range	Meaning
N10020~N10022	WORKBENCH_LOWER_LIMIT (X/Y/Z)	-99999~99999 mm	The machining coordinates of the lower travel limit in condition that the workbench travel range is checked.
N10023~N10025	WORKBENCH_LOWER_LIMIT (A/B/C)	-99999~99999 deg	
N10030~N10032	WORKBENCH_UPPER_LIMIT (X/Y/Z)	-99999~99999 mm	The machining coordinates of the upper travel limit in condition that the workbench travel range is checked.
N10033~N10035	WORKBENCH_UPPER_LIMIT (A/B/C)	-99999~99999 deg	
N10040~N10045	CHECK_WORKBENCH_RANGE (X/Y/Z/A/B/C)	Yes; No	Whether to check workbench travel range.

CAUTION

In case of accidents, please check the actual effective motion range of the machine tool in the first time to set the travel limit values of the workbench.

3.2 Return to Machine Origin

After operating [All Back to Reference Point], the machine will return to machine origin and adjust the coordinate system.

When you open the software, the mode is REFER by default. In this mode, you can either press F9 to operate the function of [All Back to Reference Point], or press F1~F6 to execute the same command. Also, you can choose the menu of [Operate]- [All Back to Reference Point] to operate the command of all axes go back to machine origin.

After the process of returning to machine origin is completed, axes button on the operation panel will appear the icon ⊕ which indicates the corresponding axis is in machine origin.

CAUTION

1) It is strongly suggested to return to reference point firstly before machining.
2) These functions will be available only after returning to the machine origin: software limit, fixed point setting, and tool change, etc.

3.3 Check Polarity of I/O Ports

In [IOPort] window under tab [DIAG], all input and output ports information are displayed, which is very helpful for system monitoring and troubleshooting.

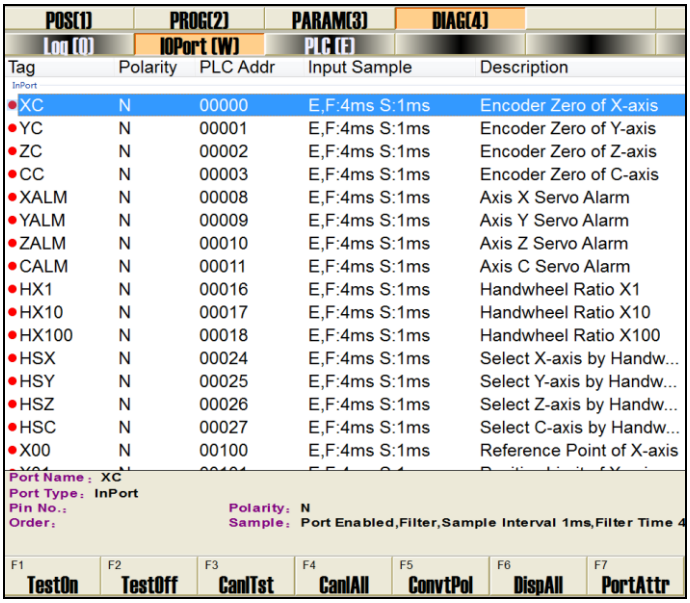


Fig. 3-3 I/O Ports

As shown above, tag, polarity, PLC address and functional description of each port are displayed in the window. You can set the polarity of I/O port according to the switch type, namely, [P] stands for a normally close switch (NC switch); while [N] stands for a normally open switch (NO switch). Polarity setting fault should be modified timely, otherwise, alarms or other warning information may appear, even cause failure.

◆ **Invert polarities of I/O ports**

Firstly, select the target port in the [IOPort] window, click [ConvtPol] button or press F5, and input

manufacturer's password. The modification of port polarity takes effect after the software is re-started.

◆ **Set attributes of I/O ports**

Select the target port in the [IOPort] window, click [PortAttr] button or press F7 to open a dialog box named [Inport Sampling Setting], where sampling interval, filter function, port enabled, port name and description can be set.

3.4 Set Parameters

3.4.1 Set Speed Parameters

If you only have to set common parameters, you can set them in ATUO or manual mode according to section 6.1.

If you need to set speed parameters fully, you can refer to the tables as follows. Parameters in this part can be divided into 5 categories: velocity, acceleration, reference circle & circular speed limit, interpolation algorithm and smoothness setting.

☞ **Related parameters (velocity)**

<table>
<tr><th colspan="2">Parameter</th><th>Meaning</th><th>Range</th></tr>
<tr><td>N64000</td><td>AX_STARTUP_VOL</td><td>The max. achievable speed of stepping motor in start-up process without acceleration.</td><td>0~100000 mm/min</td></tr>
<tr><td>N71000</td><td>JOG_VOL</td><td rowspan="2">There are two speed modes to choose under manual mode: jog speed and rapid jog speed, which can be switched by pressing the acceleration key on the operation panel. The default running speed mode is jog speed.</td><td>0~Rapid jog speed</td></tr>
<tr><td>N71001</td><td>RAPID_JOG_VOL</td><td>Setting value of manual low speed~ Max speed of each axis (Note: The maximal federate supported by hardware is 60000mm/min.)</td></tr>
<tr><td colspan="4">The parameter N64000 [AX_STARTUP_VOL] matches along with [startup frequency] of stepping & servo driver (this parameter should be set zero in driver).
The startup frequency refers to the maximal achievable frequency of motor in startup without acceleration. Reasonable setting of this parameter will improve the machining efficiency, and avoid the low speed segment when the motion performance of motor is abnormal. "Startup frequency" is generally included in ex-factory parameters, but the value changes when it is installed, especially in loading motion. Thus, it should be set based on the actual measurement of motor power and inertia of machine tool.
Parameter confirmation method: set a lower value at first, and repeatedly make the machine executes typical motion & multi-axis synchronized motion. Then increase the value gradually until</td></tr>
</table>

Parameter	Meaning	Range
fixing the maximal startup speed. The actual setting value of this parameter is half of the maximal startup speed, with general setting range “300~400”.		

☞ **Related parameters (acceleration)**

Parameter		Meaning	Range
N64101	AX_LINEAR_POST_ACC	The maximal linear acceleration of each axis when positioning.	0.001~100000mm/s^2
N64102	Z_LINEAR_ACC	The maximal linear acceleration of Z-axis.	0.001~100000mm/s^2
N64120	AX_CON_ACC	The maximal resultant feed acceleration of adjacent two axes.	0.001~100000mm/s^2
N64150	AX_ACC_AC	The change rate of acceleration of a single axis (acceleration’s acceleration).	0.001~100000mm/s^3
[AX_CON_ACC]: the maximal resultant feed acceleration of adjacent two axes. Recommended setting is 1~2 times of the single axis acceleration. For higher speed, recommended setting is 2~4 times of the single axis acceleration. Usually, the setting range is between 1200~1500. [AX_ACC_ACC]: the growth rate of acceleration, namely, the increment of acceleration in unit time. The unit is mm/s^3. It is available for S_type acceleration & deceleration, used to mitigate the bad effect caused by abrupt acceleration & deceleration of a machine.			

☞ **Related parameters (circular speed limit)**

Parameter		Range
N64207	LIMIT_ARC_VELO	Yes: valid; No: invalid
N64236	ARC_FORWARD_LENGTH	0~100mm
After installation of a machine completed, you can make the machine process a circle, in which vibration will occur due to centrifugal force. The higher the speed is, the stronger the vibration will be. Gradually increase the feed speed to see the state of vibration of the machine tool until the maximal circular speed is achieved, i.e. the maximal allowable speed of the machine tool without strong vibration. This circle is regarded as the reference circle, and its maximal allowable speed is the maximal speed of reference circle. Encountering other circles in machining, the system will calculate their maximal centripetal acceleration in terms of the reference circle and its max speed to ensure the centrifugal force is within the debugging value, i.e. the vibration will not be stronger than that during ex-factory debugging.		

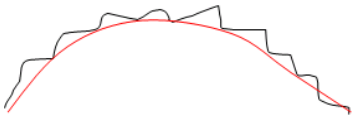
☞ **Related parameters (interpolation algorithm)**

Parameter		Meaning	Range
N64203	INTERP_ALGO	Select the most suitable interpolation algorithm to reduce error after debugging.	0: trapezoid; 1: S-type

☞ **Related parameters (smoothness setting)**

Parameter		Meaning	Range
N63000	CONN_ANGLE _DIST	The parameter is used for velocity limitation of processing short arcs in LEP algorithm. The system will analyze the distance before or after the present point, and then decide the interpolation strategy.	0~999 mm
N63006	SMOOTH_TIME	The parameter value positively correlates with the surface of the workpiece. But some details may be weakened due to the value is too large. Usually, it is suggested that the value on molding machine tool should be no more than 0.01, and the wood working machine tool should be no more than 0.03.	0.0~0.064 s
N64027	Min Smooth angle during lines connect	When the included angle is smaller than the value of the parameter, smooth the angle.	0~180 deg
N64240	SLIDE_TIME _GENE	Ratio of smoothing speed interval to a control cycle.	0.01~10
N64241	Slide speed for small lines	Eliminating velocity fluctuation when machining short segments.	Yes: valid; No: invalid.
N64242	Reference length of slide speed for small lines	Segments shorter than the value of this parameter will be executed speed smoothing.	0.001~10 mm
N64243	SPEED_SLIDE _LENGTH	During corner cutting, the waterjet swing at the look-ahead distance.	0~300 mm

[SLIDE_TIME_GENE]: The parameter value negatively correlates with blur details of workpiece. That means the workpiece will be smoother if the parameter value is large. But when in arc machining process, it will bring about radius decrease. Likewise, in the process of ripple machining on a workpiece, the larger value of slide time factor will make the wave crest shorter, as shown below. The recommended value should be no more than 0.05 second.

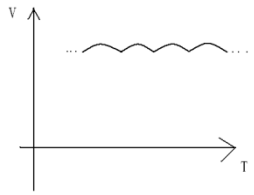


[Reference length of slide speed for small lines]: In machining an arc (or other curves) composed of short segments, velocity fluctuation, like frequent acceleration and deceleration obvious in our S-type algorithm, will occur at places where curvature is relatively large, as follows:

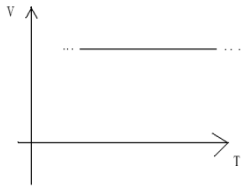
Assuming each segment is rather short, and the curvature between point A to point B is large, the

Parameter	Meaning	Range

actual speed of machining process is possibly shown as follows:



This velocity diagram (acceleration→deceleration→acceleration) will lead to oscillation of a machine tool. At this time, the parameter “Slide speed for small lines” should be set to “Yes”, and the value of the parameter “Reference length of slide speed for small lines” should be set larger than the length of short segments in the tool path. When the short segments in the tool path are shorter than the reference length, the velocity will be executed smooth treatment. Otherwise, there is no treatment. The velocity after treatment is as follows:



In this way, frequent acceleration and deceleration is avoided, oscillation of a machine tool is eliminated, and machining quality is improved.

3.4.2 Parameters of Delayed Actions

Different hardware in high pressure systems and different control methods, the system provides various combinations of delay for high pressure valve, water valve and sand valve. You can set the combinations according to your needs. Also you can set the ON/OFF delay of individual valves.

☞ **Related parameters**

Parameter		Range
N90010	OPEN_HP_DELAYTIME	0~600000 ms
N90011	CLOSE_HP_DELAY_TIME	0~600000 ms
N90020	OPEN_CUTTING_DELAY_TIME	0~600000 ms
N90021	CLOSE_CUTTING_DELAY_TIME	0~600000 ms
N90030	OPEN_ABRASIVE_DELAY_TIME	0~600000 ms
N90031	CLOSE_ABRASIVE_DELAY_TIME	0~600000 ms
N90032	PAUSE_BEFORE_CUT	0~600000 ms
N90033	PRESSURE_DISCHARGE_DELAY_TIME	0~600000 ms
N90034	PAUSE_AFTER_CUT_DELAY_TIME	0~600000 ms
N90045	After Stop Close Abrasive And Hv Delay time	0~600000 ms
N90046	Close Booster Delay Time	30~900000 ms

[OPEN_HP_DELAYTIME]: High pressure valve controls the operation of high pressure devices.

Parameter	Range
When the high pressure valve is open, it needs a period of time to reach the stable high pressure. Then, water valve and sand valve can be opened. When you need to cut and penetrate thick materials, you should keep the high pressure system for a while in order to cut the material through. [CLOSE_HP_DELAY_TIME]: Close delay time can be set differently according to material types. In normal conditions, you should set close delay time a small value. [OPEN_CUTTING_DELAY_TIME]: when water valve is opened, high pressure system needs a period of time to reach a stable status, and then the high pressure water is stable. After high pressure water is stable, open the sand valve, which can ensure abrasive materials are fully mixed in the waterjet and ejected from the waterjet. [CLOSE_CUTTING_DELAY_TIME]: When sand valve is closed, you should ensure that there isn't any remaining abrasive in the waterjet, and delay a period of time before closing the water valve. Water valve close delay is used when there is no pressure on delay function, the water valve close delay function can substitute it. You should close the high pressure valve before water valve close delay. [OPEN_ABRASIVE_DELAY_TIME]: The parameter is mainly used for ensuring the abrasive material and high pressure water are fully mixed in the chamber, so that the waterjet gets an effective grinding force. [CLOSE_ABRASIVE_DELAY_TIME:] The parameter is mainly used for ensuring the workpiece can be cut completely. If the delay time is set too long, a hole will be cut on the workpiece. [PAUSE_BEFORE_CUT]: Since G00 positioning speed is greater than machining speed, the machine may not stop completely after positioning. A better cutting effect will be obtained through setting the parameter. The function of piercing before cutting can be realized by setting the parameter as well. [PAUSE_AFTER_CUT_DELAY_TIME]: When a complete tool path is completed, the machine needs to delay for a period of time to reach stable status for executing the next step. When the whole machining process is finished, the machine also needs to delay for a period of time to make sure the material can be cut completely. [PRESSURE_DISCHARGE_DELAY_TIME]: Due to the high pressure, the water in the chamber will flow back and moisten the sand after machining ends. Wet sand agglomerates and blocks the waterjet. Due to the waterjet lacks the force, when it reaches workpiece surface, the power of waterjet cannot be focused, and water spray will be formed. To solve the problem, open water valve for a period of time after all valves are closed to eliminate the remaining pressure in high pressure system. Then the internal and external pressure of the chamber can be balanced and water in the chamber won't flow back.	

3.4.3 Deceleration Distance of Lead Line

The fuction of lead line is to leave the unqualified part out of the cutting graphic. Due to trailing or lag of waterjet cutting, the cutting effect may be affected when cutting speed is too high.

To set deceleration distance of lead line is to draw a line segment at the start of lead line. And the machining speed of the segment will be slow, which can eliminate the poor cutting effect at the joint point of lead line and tool path.

☞ **Related parameters**

Parameter		Range
N63007	Slow Distance of intro line	0~2
N63008	Whether the Slow Distance of intro line is valid or not.	Yes: valid; No: invalid
N63009	The Slow Distance of intro line is near to the start of intro line or to the figure.	Yes: near to the figure; No: near to the start of intro line
N64022	Speed Percentage of Slow Distance of intro line	0~99%
[Slow Distance of intro line]: draw a line segment at the start of lead line which is used to set a low speed. [Speed Percentage of Slow Distance of intro line]: the ratio of speed in slow distance to actual machining speed.		

3.5 Load Programs or Draw Graphics

After setting the parameters, you can load the machining programs in NcStudio. Click on the sub-menu [Open and Load] under [File] menu, and choose the machining file in the dialog box popping up. Then click [Open] to load the file.

Besides, you can also load or draw machining graphs in NcEditor. You can click sub-menu [Open] to load NCE format file, or [Open Text File] under [File] menu in NcEditor to load the files in supported format. “File failed to open” warning box will pop up if you force to open any file in the format NcEditor does not support. In addition, you can draw graphs using the [Draw] menu or drawing tool bar in NcEditor.

CAUTION

1) In NcEditor, the difference between sub-menu [Import G Code File] under [File] menu and sub-menu [Insert G Code File] under [Edit] menu in NcEditor: If there is already a loaded machining file in NCEditor, the file loaded by clicking [Import G Code File] will replace the existing file while the file loaded by clicking [Insert G Code File] will not replace the exsiting file but coexists with the exsiting file.

2) If the track you are loading is too large, the system will run very slowly. Then you should select sub-menu [Stop Loading Tracks] under [File] menu in NcStudio to stop loading the track timely. You can also choose sub-menu [Clear Tracks] to clear the loaded tracks and forbid the system to load any of the tracks.

3.6 Set Workpiece Origin

Workpiece origin is the coordinate zero on X/Y/Z/A/B/C axis in machining programs. Before machining starts, you need to set workpiece origin. The system provides various methods to set workpiece origin. Below are two methods of setting workpiece origin by directly resetting coordinates. Please refer to section 4.2.3 for more information about setting workpiece origin.

1) Manually move X/Y/Z axis to the position you need to set as workpiece origin. In manual mode, in [POS] function tab, press F1/F2/F3/F6 or separately click [XClear], [YClear], [ZClear] and [XYClear] button to reset the coordinates of current position. Thus, the position will be taken as the workpiece origin during machining process.
2) Manually move X/Y/Z axis to the position you need to set as workpiece origin. Select [Set Workpiece Origin] under menu [Operate], reset the coordinates of current position. Thus, the position will be taken as the workpiece origin during machining process.

In addition, you can set workpiece origin in NcEditor. Two methods of setting are shown as follows:

1) Click [Set Origin] under [Draw], and dialog box [Set Origin] as shown in Fig. 3-4 will pop out. You can set workpiece origin by particular positions, set the X/Y coordinate of workpiece origin directly, or by mouse.

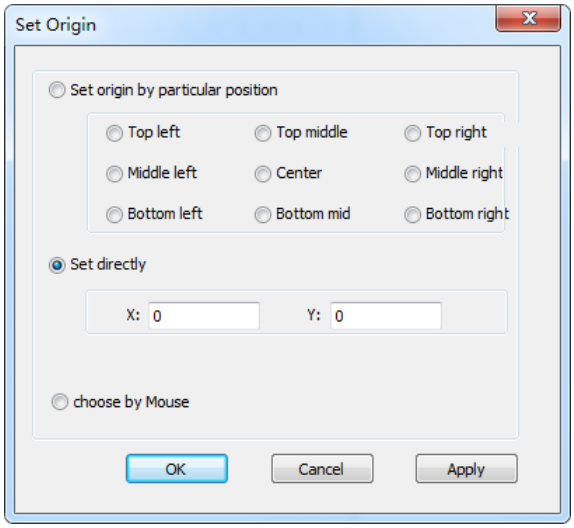


Fig. 3-4 Set Origin

2) Object editing space is subject to the workpiece coordinate, and the origin of object editing space is also the workpiece origin. There is an icon at the intersection point of horizontal scale and vertical scale. Click and hold the crossing of the icon and drag it to the point you need to set as workpiece origin.

3.7 Simulate

After setting the workpiece origin, you can learn the machine tool motion in advance by executing simulation function, which is used to prevent the damage of machine tools caused by programming errors. . You can also learn other additional information.

Select [Simulation Mode] under [Operation] menu to enter simulation mode. [Normal] window under [POS] will show Simulation. Click [Start] button, and the system will run simulation automatically.

Once the simulation process starts, the sub-menus [Start], [Pause], [Stop] and [Breakpoint Resume] will turns into [Simulation Start], [Simulation Pause], [Simulation Stop] and [Simulation Breakpoint Resume]. Sub-menu [Simulation Mode] will turn into Stop and [Exit Simulation Mode], and the simulation process will stop once you click on this sub-menu.

3.8 Execute Machining

Start machining

Machining can be started by the following three ways:

1) Select [Start] item under [Operate] menu list.
2) In auto mode, click [Start] button on the operation button bar.
3) In auto mode, press shortcut key F9.

Pause

During auto machining, there are three optional ways to suspend machining.

1) Select [Pause] item under [Operate] menu list.
2) In auto mode, click [Pause] button on the operation button bar.
3) In auto mode, press shortcut key F10.

Stop

During auto machining, there are three ways to stop machining and turn the system into [IDLE] state:

1) Select [Stop] item under [Operate] menu list.
2) In auto mode, click button [Stop] on the operation button bar.
3) In auto mode, press shortcut key F11.

4 Software Functions

4.1 Functions in Auto Mode

4.1.1 NearPoint/Selective Machining

During machining process, if you only need to machine a segment, you can use the nearpoint/Selective machining.

In auto mode, in [POS] function tab, by clicking [NearPoint] button on the operation bar, or pressing F5, you can enable the function. It can be used in simulation as well.

◆ **Nearpoint machining**

When machining Nce files, you can enable nearpoint machining function. After stopping the machining process, if you want to re-process from any point, you can manually move the waterjet to near the workpiece. Then click [Nearpoint] button on the operation bar. The waterjet will continue to work from the nearest point. This function will help you to move the waterjet to a processing point accurately and intuitively.

◆ **Selective machining**

When machining non-Nce files, you can enable selective machining function. Click the [NearPoint] button, and a dialog box will pop out as shown in Fig. 4-1. You can execute any section of machining program by inputting the start and end line number in [Specific Position]. When you select [File Start] or [File End] item, [Specific Position] items will be disabled.

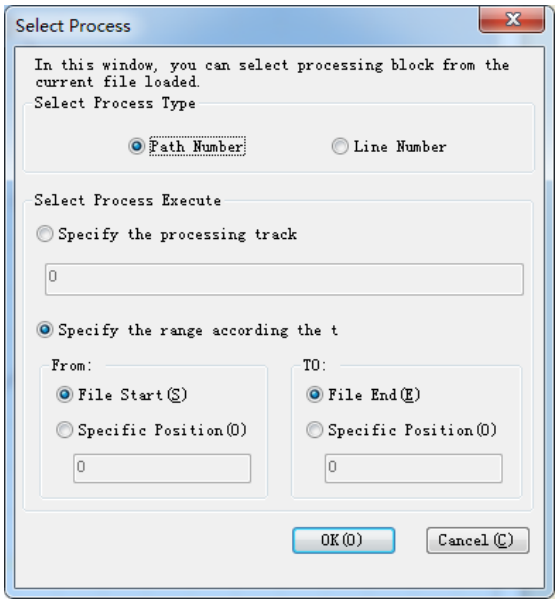


Fig. 4-1 Near Point

☞ **Related parameters**

Parameter		Meaning	Range
N79006	NEARPOINT_RANG	Adjacent point processing function can be enabled in near point range.	0~500 mm
During adjacent point processing, the distance from waterjet to the nearest point must be less than the set value of the parameter. Otherwise, the system will pop up a warning as follows: 			

4.1.2 Auto Back and Breakpoint Resume

Auto Back: when you need to process in the reverse direction, namely, to process in opposite direction from the current point (for instance, when some parts of workpiece are not cut completely), you can use the auto back function.

Breakpoint Resume: The function is used for coordinating with auto back function. After auto back process is finished, you can use the function to continue machining at the breakpoint without returning to machine origin, which can improve machining efficiency.

During processing, auto back function and breakpoint resume function can be used cooperatively. Auto back function is used differently during processing of Nce files and non-Nce files.

◆ **When processing Nce files**

If you need to use auto back function since machining effect of some segments is not satisfying, or abrasive is not enough, you should stop the current processing task and click [AUTOBACK] button on operation button bar or press F7. The system will enable the function automatically. Machining information will be shown on the CNC information bar. After auto back is finished, the system will enter idle state. You can click [B.P.Resume] button to continue machining.

◆ **When processing non-Nce files**

In auto mode, click [AUTOBACK] button on the operation button bar or press F7, and a dialog box will pop up as shown in Fig. 4-2. You can manipulate the system through button [Auto Back], [Auto Forward] and [Stop]. After auto back is finished, click [B.P.Resume] button or press F3 to continue machining from the breakpoint.

During processing a non-Nce file with auto back function, [Stop] button is disabled when the program is out of auto back buffer range. When the dialog box is closed, the system will execute auto forward automatically.

☞ **Related parameters**

Parameter		Meaning	Range
N53100	Turn Off Ports When Tracking Back	Whether to turn off ports or not when auto back.	Yes: turn off ports if they are open originally; No: keep the original status.

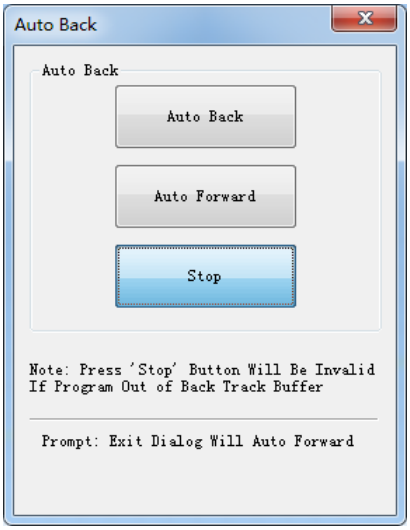


Fig. 4-2 Auto Back

CAUTION

For the convenience of operation, it is suggested to save the machining file as Nce format in NcEditor before using auto back function.

4.1.3 Jiggle

During the machining process, you can find the best cutting height by adjusting the height of waterjet with jiggle function according.

◆ **Operation Steps**

1) You can set jiggle related parameters, including [N71021] "Jiggle Feedrate" and [N71022] "JIGGLE_STEP_LENGTH".

2) During auto machining, you can click button Z+ and Z-, or shortcut keys num 1 and num 7 to adjust Z-axis, according to your experience and needs. The operation is valid during machining process or when the machine is suspended.

☞ **Related parameters**

Parameter		Meaning	Range
N71021	Jiggle Feedrate	The feedrate during jiggle operation.	0~100000 mm/min
N71022	JIGGLE_STEP_LENGTH	The moving distance of Z-axis during each jiggle operation.	0.001~2 mm

CAUTION

1) In idle state, jiggle function is disabled.
2) In auto mode, [Z+] and [Z-] buttons in the software are used for jiggle of Z-axis. In manual mode, [Z+] and [Z-] buttons are used for manually moving Z-axis in positive or negative direction. Please note the difference.

4.2 Functions in Manual Mode

4.2.1 Edge Finding

1) When you cannot locate cutting material during loading since the area of material is too large, or the weight of material is too heavy, you can build a new coordinate system with edge finding function.
2) According to actual cutting situation, you can find the edge according to graphic layout of a cutting file to maximize material usage.

In manual mode, after an Nce file is loaded, [LocateCoor] button on the operation button bar in [POS] window will be activated. Click [LocateCoor] button, and a dialog box will pop up as shown in Fig. 4-3. Then select [Use the located-workcoor] item to enable the function. Both X-axis and Y-axis are locating axes. Locating modes include one-point locating, two-point locating and three-point locating.

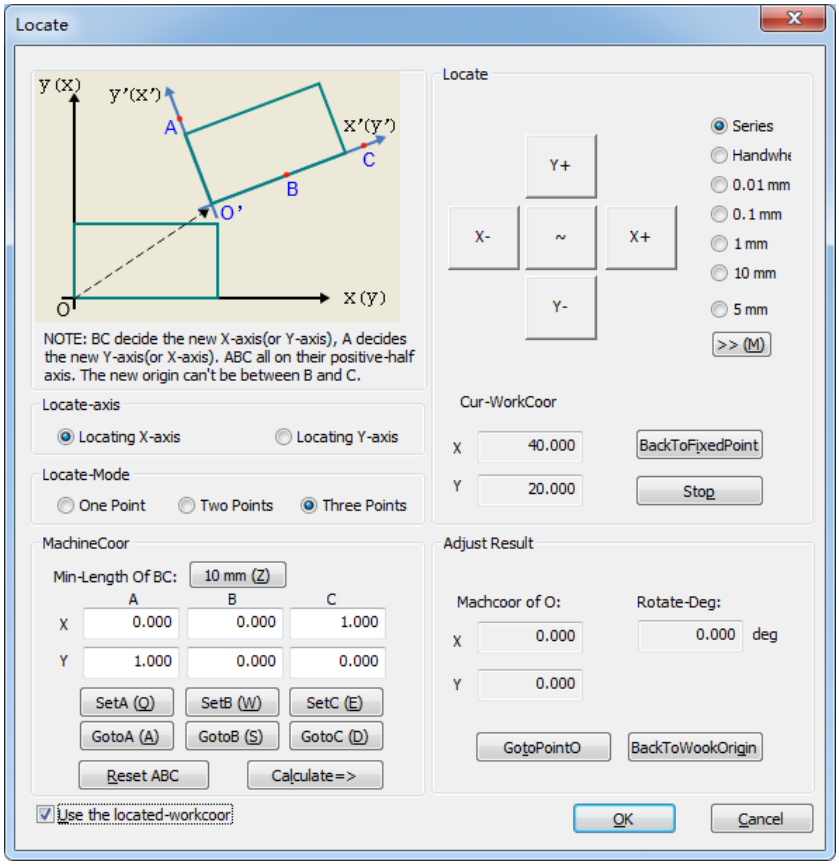


Fig. 4-3 Locate

CAUTION

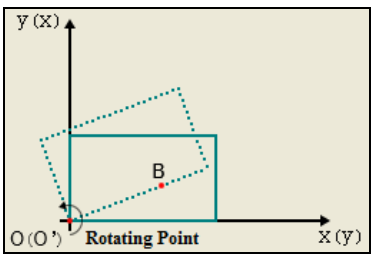
1) Edge finding function is only enabled for processing of Nce files. For non-Nce files, you should open it in NcEditor, and save as Nce format. Then you can process it with edge finding function.

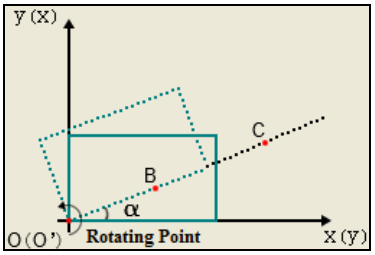
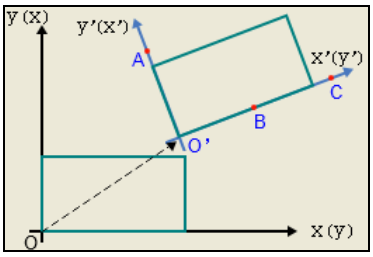
2) Edge finding function is only valid for current machining.

◆ **Locating axes**

You can determine locating axes according to actual situation of the machine tool. If the travel limit of X/Y-axis is too large, you can set a new X/Y-axis as the locating axis firstly. Below is an example of edge finding with X-axis as locating axis.

◆ **Locating modes**

One-point	Set point B on one side of the material by manually moving waterjet, and set the line through workpiece origin O and point B as the new X-axis. Point O is also the rotating center. The line vertical to line OB at point O will be set as new Y-axis automatically, as shown right.	

Two-point	Set point B and C on one side of the material by manually moving waterjet, and the rotating angle of X-axis can be determined. The new X-axis goes through workpiece origin and is parallel with BC. Point O is also the rotating center. The line vertical to line BC and through point O will be set as new Y-axis automatically, as shown right.	
Three-point	Set point B and C on one side of the material by manually moving waterjet, and set point A on the other side. Set line BC as the new X-axis, and set the line vertical to line BC and through point A as the new Y-axis. The intersection of the two lines will be the origin of the new coordinate system, as shown right.	

CAUTION

Both point B and C should be set at positive half of axes. During two-point locating and three-point locating, point B must be determined before point C. Otherwise directions of axes will be reversed. The new origin cannot be set between point B and point C.

◆ **Operation steps**

When you have decided the locating mode, you can start edge finding. Below is an example of two-point locating operation.

1) After placing the material on the workbench, manually move the waterjet to make it near the edge of workpiece until to point B on the edge. Then click [SetB] to set the coordinate of current point.
2) Move the waterjet along the positive direction of X-axis, and then set point C with the same method as in step 1). Then click [SetC]. The new X-axis is parallel to line BC, and goes through workpiece origin O.
3) The software will build a new coordinate system according to point B and C automatically. Click [OK], and the software will calculate the rotating angle of the processing material from machine coordinate system. During machining, the software will rotates the workpiece coordinate system in tool path file for corresponding angle. If the angle is greater than 45°, it indicates the offset of material is too much. And you need to place the material and edge finding again.

If the current coordinates of point A, B and C is inaccurate or you have to re-select them, click [Reset ABC] button to restore the coordinates of point A, B and C to default.

If the distance between point B and C is less than [Min-Length of BC], the system will pop up a dialog box to remind you that point B and C is too close.

CAUTION

1) When machining process is normally finished, the system will clear location results. If you don't locate the material again before processing, the processing will still be normal with zero rotating angle. Locating data will not be cleared when location results are cleared. If you still need the location result for processing, check [Use the located-workcoor] and click [OK] in dialog box "Locate".
2) If you do not need edge finding function before machining is normally finished, you can uncheck [Use the located-workcoor] and click [OK] button in dialog box "Locate".

4.2.2 Height Measurement

Height measurement function is used to ensure the distance between waterjet and workpiece surface is fixed, thus avoiding cutting effect influenced by cutting height change.

In normal conditions, height measurement function is disabled. If you want to enable the function, set operator parameter [N79201] as "1": enable height measurement. After setting the parameter, in manual mode, click [Altimetry] button on the operation button bar or press F10 in [POS] window to use the function. A dialog box will pop up as shown in Fig. 4-4.

◆ **Height measurement steps**

1) Load an Nce file in NcStudio; and prepare the workpiece to be processed on the workbench.
2) Set height measurement parameter.
3) Select one of the height measurement modes which include manual mode, automatic mode and three-point mode.
4) Measure height and start machining.

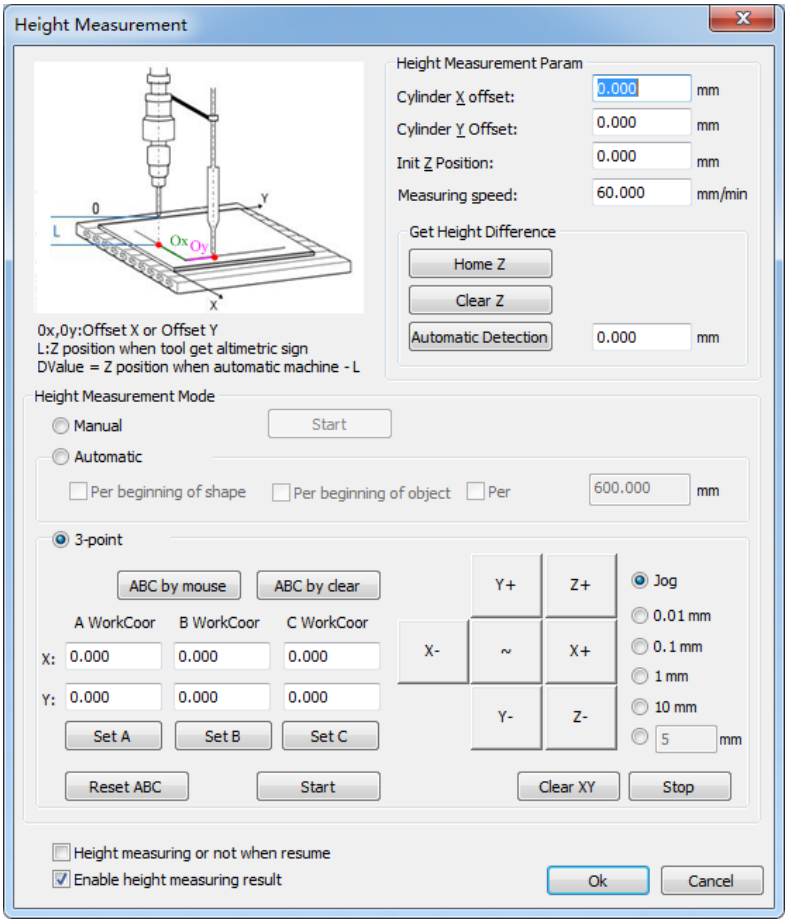


Fig. 4-4 Height Measurement

◆ **Height measurement parameters**

Cylinder X Offset	The offset of height indicator to waterjet in the direction of X-axis.
Cylinder Y Offset	The offset of height indicator to waterjet in the direction of Y-axis.
Init Z Position	The coordinate of Z-axis before height measurement.
Measuring Speed	The measuring speed when height measurement signal is detected.

CAUTION

After height indicator is installed, you need to manually measure and input the offset value of height measurement cylinder X/Y.

◆ **Acquiring height difference**

After setting height measurement parameters, you need to acquire the height difference. There are two ways to acquire the height difference: manually calculate and input the value in the dialog box; or automatic detection.

- Manually calculate and input the value: the distance between waterjet and workpiece surface during auto machining is fixed. The distance between waterjet and workpiece when height measurement signal is detected is also fixed. Thus, you can calculate the result through: Height difference=

distance between waterjet and workpiece during auto machining – distance between waterjet and workpiece when height measurement signal is detected. Then input the value of height difference into the dialog box.

- Automatic detection: Click [Home Z] button first, and then manually move waterjet to the best cutting height from the surface of workpiece during auto machining. Click [Clear Z] button and then [Automatic Detection] button to execute height measurement. When the measurement is finished, the height difference will be updated automatically in the dialog box.

◆ **Height measurement modes and steps**

Manual mode:

You can choose any point in the range of workbench for manually height measurement, and then record the height value. The processing file will adjust coordinate of Z-axis according to the height value. Steps are shown as follows:

1) Ensure that Z-axis has returned to machine origin.
2) Set workpiece origins of X-axis and Y-axis.
3) Move waterjet to the manually height measurement point, and then click [Start] button. When the height measurement is finished, click [OK].
4) Start to machine the Nce file.

Automatic mode:

Automatic height measurement consists of [per beginning of shape], [per beginning of object] and [per…]. In [per beginning of shape], the system measures the height at the start point of Nce file; In [per beginning of object], the system measures the height at the start point of each graphic (a file maybe include many graphics). In [per…], that the system will measure the height based on the measuring space. When machining distance reaches the measuring space, the system will measure height automatically during the whole machining process. Steps are shown as follow:

1) Click [Home Z] button;
2) Set workpiece origins of X-axis and Y-axis.
3) After selecting [Automatic] item, then select from items of [per beginning of shape], [per beginning of object] and [per…], and click [OK].
4) Start machining the Nce file. During the machining process, if you choose the item [per beginning of shape] or [per beginning of object], the system will measure the height and adjust Z-axis's position before machining the first graphic. If the item [per…] is selected, the system will measure the height each time the machining distance reaches the measuring space. The height measurement operation will be synchronized with machining process.

Three-point mode:

Generally, this function is applied to height measuring of workpiece inclined to the workbench and with smooth surface. You can set 3 points randomly on the workpiece, and then measure heights from waterjet to these points; so that the angle between the workpiece and workbench can be determined. During machining process, you can dynamically adjust Z-axis' position in every processing point. Steps are shown as follows:

1) Make sure that Z-axis has returned to machine origin.

2) Set workpiece origins of X-axis and Y-axis.

3) Acquire 3 points in the range of workbench. There are three ways. By manually inputting, by manually moving X-axis and Y-axis and setting workpiece coordinates of X-axis and Y-axis as the three points, or by clicking [ABC by mouse] button and then selecting 3 points in the [Pos]-[Normal] window randomly. If you are not satisfied with points you selected, click [ABC by clear] to reset and select again.

4) Click [Start] button to begin three-point height measurement. When it is finished, click [OK].

5) Start to process the Nce file. During machining process, you should adjust Z-axis's position in real time.

The button Stop in the dialog box is valid for all height measurement modes. You can click the button to stop height measurement.

If you click button Clear XY in the dialog box, all workpiece coordinates of X-axis and Y-axis will be cleared.

If you select the item [Height measuring or not when resume], the system will continue measuring under the function of [Break Point Resume].

Height measurement results in manual mode and three-point mode are always valid after Nce files are reloaded or software is restarted. If you do not need the height measurement result to be invalid, please manually cancel the item [Height measuring or not when resume]. In automatic mode, height measurement will be executed in every machining process. If you didn't want to use the function, please manually cancel the item [Height measuring or not when resume].

☞ **Related parameters**

Parameter		Range
N79201	SOFTWARE_CONFIGURATION	0: normal; 1: enable height measurement

CAUTION

1) Before using the height measurement function, you need to execute returning to machine origin. Otherwise it may cause tool crash.

2) You must measure the height again after changing to a material with different thickness. Otherwise tool crash may occur since the height measured last time is incorrect. Since the software cannot detect whether you have changed the material, it only prompts tips. How to operate is up to you.

4.2.3 Setting Workpiece Origin

In manual mode, click [Set Current Origin] under [Operate] menu, and a dialog box will pop up as shown in Fig. 4-5. There are 4 methods to set workpiece origin.

Method 1: Before loading a file, manually move the waterjet to the program origin of workpiece, namely, upper left, through manual panel in the dialog box or in interface of manual mode. Then set offset by

clicking [Zero] button.

Method 2: When you manually input the workpiece coordinate, you should select item [Whether the manual input] firstly. Then, input workpiece coordinate in the activated box. And click [Apply] or [Ok] button to finish the operation. Other methods are invalid at the moment. Additionally, you can directly click [WorkCoor] under menu [Operate] to input the coordinate value.

Method 3: Set offset through the function [Set the current point for application edge points] (top left, bottom left, top right, bottom right). If a tool path file is blank or the program origin is not in the four corners, these buttons will be disabled in grey.

Method 4: Read the saved workpiece offset. Select [Read the origin of the workpiece] firstly to activate the drop-down menu, and then read the saved offset data. Furthermore, you can also select [Save workpiece origin] and [Read the origin of the workpiece] under [Operate] menu to operate.

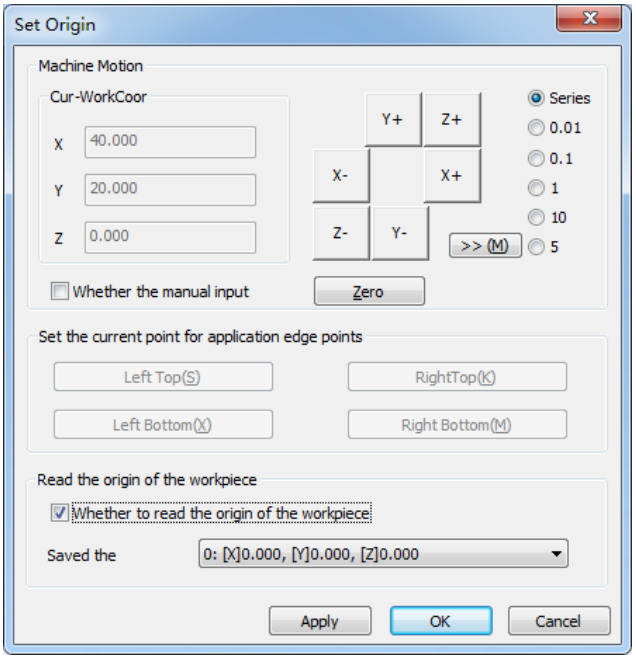


Fig. 4-5 Set Origin

4.2.4 Handwheel Operation

◆ **Handwheel mode**

The system supports three operation modes: auto mode, manual mode and reference point mode. Manual mode is subdivided into jog mode, stepping mode and handwheel mode. You can select handwheel mode by clicking [handwheel] button in operation panel of manual mode, or by selecting [Handwheel] under [Work Mode] menu.

The handwheel is shown in the right. You can control the movement of machine tool when it is activated, steps are as follows:

1) Turn the “Axis Selection Knob” to select an axis.
2) Turn the “Gear Selection Knob” to decide the displacement of moving component (linear axis or rotary axis) against each gear of the handwheel. Parameter N52003 ~N52005 are used to decide the displacement.
3) Turn the “Handwheel Control Wheel” to control the movement of a tool in selected gear.

Fig. 4-6 Actual sample of a handwheel

◆ **Handwheel guide**

In auto mode, when handwheel guide is activated, the system will execute processing programs with the handwheel swing. Machining programs end when the handwheel swing stops. Machining speed changes with the handwheel swing speed. Before machining starts, you can use the function to check if the program runs correctly. In reference point mode, you can also use the function to execute returning to machine origin.

There are two methods to enable the function:

1) In auto mode, turn to [POS] functional area, and click [HWheel].
2) Select [Handwheel Guide] item under [Operate] menu.

CAUTION

Handwheel is optional.

4.2.5 User Command

The system offers the function of user command with which you can input and execute simple commands to achieve quick respond of system.

In manual mode, select [UserCom] under menu [POS], as shown in Fig. 4-7. In this window, you can enter 10 commands, and you can repeatedly enter command at the same line.

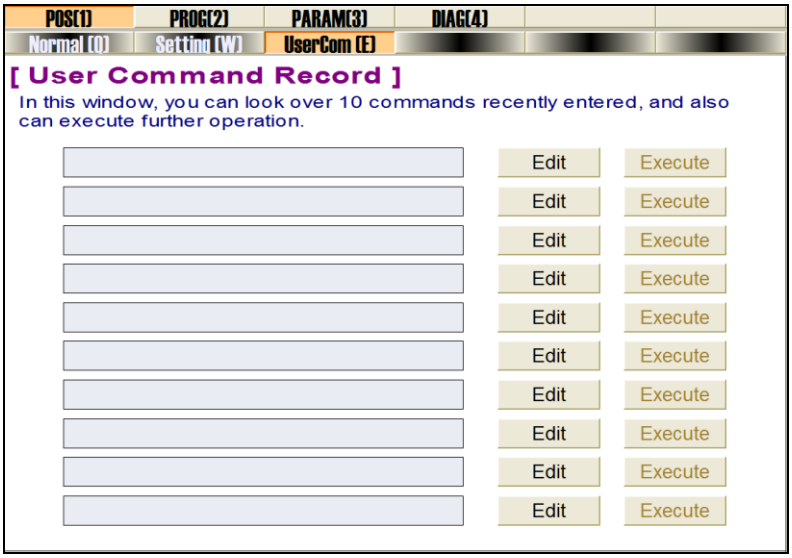


Fig. 4-7 User Command

1) When no command is entered in an entry, the button Execute on the right of the entry is disabled.
2) Click button [Edit], an input box will pop up. You can enter a command in the input box and click [OK].
3) After entering a command, the [Execute] button in the same line is enabled. Click button [Execute], the system will execute the command.

After entering a G command, the system will save the command automatically as the current command for your convenience to look up it.

4.3 Functions in Reference Point Mode

4.3.1 Modes of Returning to Machine Origin

The system provides 2 modes of returning to machine origin: stepping mode and servo mode.

In stepping mode, since there is no encoder in stepping motor, the system does not detect zero signal. In servo mode, there are encoders in servo motor, and the system detects zero signal.

CAUTION

Mode of returning to machine origin has been set and fixed by the manufacturer. Operators are limited to modify.

4.3.2 Modes of Single Axis Returning to Machine Origin

In practice, not all axes are installed with origin switches. For the convenience of all users, a group of parameters have been added to control whether a single axis returns to machine origin.

☞ **Related parameters**

Parameter		Range
N74091~N74096	Axis Type of Bkref (X/Y/Z/A/B/C)	Yes: The parameter will execute [Back to REFER Point] for single axis, and it was added in [ALLBACK] operation. No: The single axis doesn't need to return to machine origin.
For machine tools whose axes are installed with origin switches, you should set all the parameters to "Yes". For machine tools whose X-axis and Y-axis are not installed with origin switches, you need to set the parameter of X-axis and Y-axis to "No" and the rest parameters to "Yes".		

CAUTION

1) After other axes return to machine origin, axes who are set to "No" will show icon automatically.
2) If C-aixs is set not to return machine origin, when you click [Cback], CNC information bar will show No need to move to Ref. point along C axes.

4.3.3 Double Y-axis Returning to Machine Origin

In double Y-axis system, if the beam is unparalleled with X-axis due to motion of one Y-axis, you should make the Y-axis return to machine origin. The system will automatically correct the position of beam and make it parallel with X-axis according to origin switches of Y1 and Y2.

In double Y-axis configuration, the modes of X-axis and Z-axis returning to machine origin are same in standard configuration, while mode of Y axes returning to machine origin is different. The system provides 3 modes of Y axes returning to machine origin: independent double Y-axis mode, one driving two mode and pulse forbidden mode.

◆ **Independent double Y-axis**

Independent double Y-axis mode needs two axis ports and two origin switches. The driven axis returns to machine origin after driving. In this mode, Y-axis origin detection must be executed. Before returning to machine origin, you need to check the distance error between the two Y-axis by clicking [Detect Y. REF].

In double Y-axis machine tool, an encoder is installed on each axis. When installing the two encoders, make sure the line between the encoders is paralleled with X-axis so that the line between two machine origins is parallel with X-axis after returning to machine origin is completed. In actual installation, the line between two encoders is not always parallel with X-axis due to inherent errors and installation mistakes. Thus, you need to detect the origin and measure the position error between two encoders by clicking [Detect Y. REF. When Y-axis returns to machine origin, the system will compensate the error to make the line between machine origins paralleled with X-axis.

Sketch of Y-axis origin detection is shown as follows:

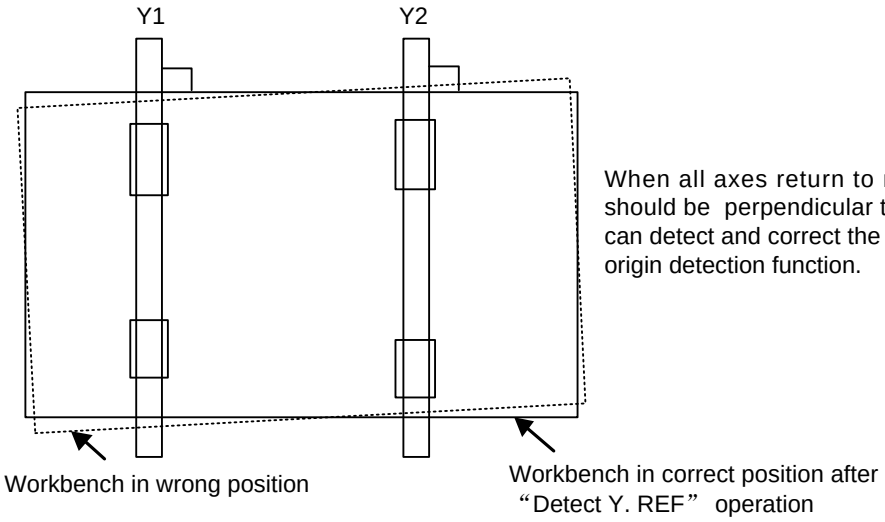


Fig. 4-8 Y-axis origin detection

Requirements of setting

- Set driver parameter of Y2-axis [Pn50A] to 8101, and [Pn50D] to 8848. Keep the driver parameters of Y1-axis unchanged, namely, set [Pn50A] as 8100, [Pn50D] as 8888.
- Install an origin switch on Y2-axis at port GX32.
- Set software configuration to [StdA with 2Y].

Debug

1) Power off the machine, and then manually adjust the perpendicularity of the machine tool.
2) When the perpendicularity reaches the requirement, click [Detect Y. REF] or press F10 in reference point mode. Then the system will detect the position error between origin switches of the two Y-axes automatically and shows it in parameter [N74102] "Double Y Encoder Origin Error".
3) After executing the steps above, every time you click [Yback] button, the system will correct Y1-axis and Y2-axis to make the beam paralleled with X-axis.
4) If error still exists in step 3), you can adjust parameter [N74102], and then click [Yback] again. When modifying parameter [N74102], the original value plus positive number can make Y1-axis move in positive direction while the value minus a positive number can make Y1-axis in negative direction.

◆ **One driving 2**

One driving two mode needs only one axis port, and one origin switch. For Y-axis, two drivers are controlled by only one Y-axis servo port with an exclusive servo cable. Since the motions of the two drivers are the same, there is only one Y-axis origin signal.

◆ **Pulse forbidden**

Pulse forbidden mode needs only one axis port and two origin switches. You have to connect servo driver when you use pulse forbidden function. There is a pin in the port of servo driver used for pulse forbidden function. After pulse forbidden is enabled, servo drivers are still in enable state, but they cannot receive command pulses. Therefore, the whole process is: the driving axis returns to the machine origin first, receives pulses but does not move. Then the driven axis returns to machine origin.

☞ **Related parameters**

Parameter		Range
N74102	Double Y Encoder Origin Error	-10000~10000 mm
N74104	Enable Value of Double Y Encoder Origin Error	0~1e+006 mm

4.4 Auxiliary Functions

4.4.1 Axes Types

To adapt different types of machine tools, the system provides 3 types of axes: linear axes, rotary axes and rotary axes with shortest path.

◆ **Linear axes**

X/Y/Z axes, for linear cutting.

◆ **Rotary axes**

A/B/C axes, which are used to achieve higher cutting technic requirements such as cutting of irregular tool path and slope.

◆ **Rotary axes with shortest path**

Since there are two direction CW or CCW for the waterjet to move along during positioning, the system calculates the minimal rotating degree automatically, i.e., the system selects the shortest tool path and makes the waterjet move along with the direction, which can improve machining efficiency.

☞ **Related parameters**

Parameter		Range
N10060~N10065	AxisType (X/Y/Z/A/B/C)	1: Line. 2: Revolving. 3: Revolving with the shortest path.

4.4.2 Drilling Function

If you want to drill a hole with small radius, you can install a drill head on the machine.

◆ **Operation steps**

1) In operator's parameter, you can set drilling related parameter [N93220], [N93221], and [N93223] to decide whether to drill at the cutting point, drilling option and the delay time of drill.
2) After setting the parameters, click drill button to start machining.

☞ **Related parameters**

Parameter		Range
N93220	Use drill not	Yes: use; No: not use
N93221	Drilling Option	1: First drill one hole, then cut the path; 2: First drill all holes, then cut all paths
N93223	Delay time of drill	0~1e+007
For parameter [Drilling Option], setting value "0" represents drill and cut one point when one start point is located during multi-graphics machining, and "1" represents all start points will be located before drilling and cutting them during multi-graphics machining.		

CAUTION

In the software, the drill is marked as tool 1, while the waterjet is marked as tool 2. There is a positional offset in the direction of X-axis or Y-axis between the 2 tools. The system switches to tool 1 for drilling motion. When drilling operation is finished and the machine is about to process, the system switches to tool 2 from tool 1.

4.4.3 Program Management

In [PROG] function window, you can press Q to enter [HD List] screen, or press W to enter [USB Disk List].

[HD List] window shows all machining programs and folders under directory of D:\NCFILES. You can click buttons in manipulation button bar to [Load], [Edit], [Delete], [New] and [Rename] the processing program. By double clicking, you can enter a folder to select the processing program you need, and then press F1~F5 to operate.

[USB List] window shows files in USB disk. For instance, if a file is under the directory "naiky" in the flash disk, you can click buttons in the operation button bar to [Load], [Copy to HD], [Safely Remove]. In addition, If there is more than 1 USB disk, you can click F3 [Sel Drive] to select among them.

CAUTION

1) You can put machining files under the directory of D:\NCFILES, for the conveience of finding and manipulating the file in hard disk directly.

2) Besides in [HD/USE List] window, you can also edit programs by clicking [Open and Edit] or [Edit Current Machining Program] under [File] menu. The submenu [Edit Current Machining Program] is valid when the loaded file is a G code file.

4.4.4 Offset Setting

You can click [Offset] under [Operate] menu. Then, public offset and workpiece offset will appear in the dialog box as follows:

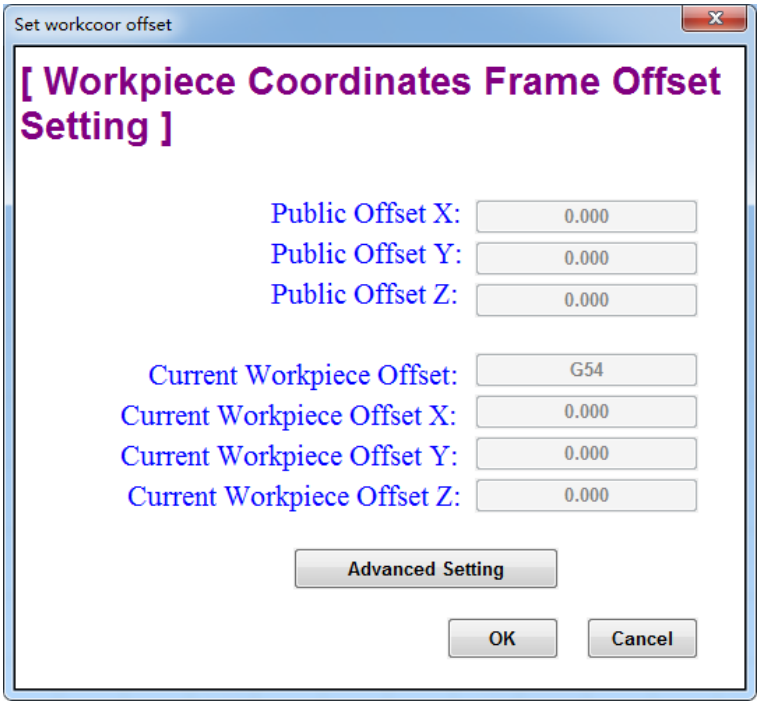


Fig. 4-9 Workpiece coordinates offset setting

Workpiece coordinate = machine coordinate – public offset – workpiece offset

You can click [Advanced Setting] button to enter [Offset Setting] dialog box.

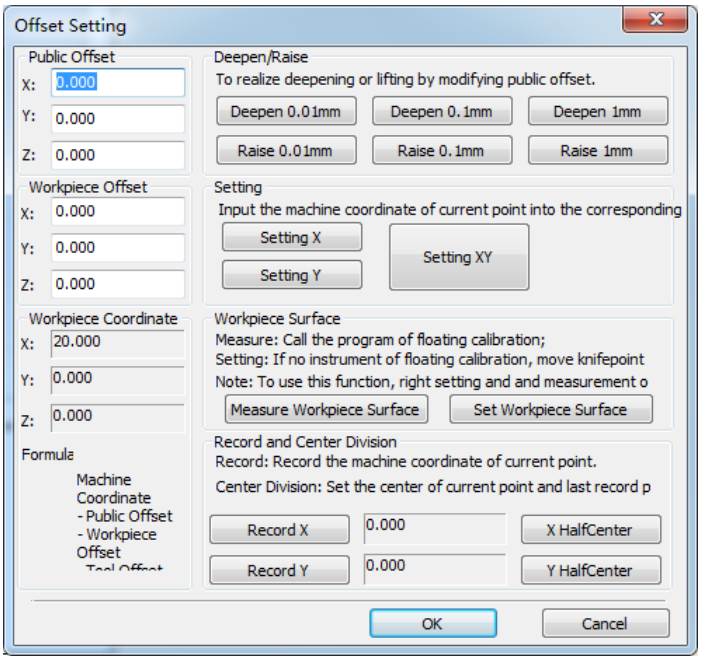
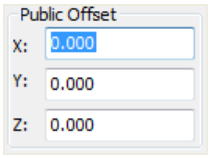


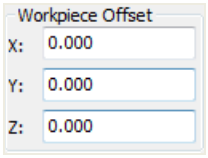
Fig. 4-10 Advanced offset setting

- Public offset



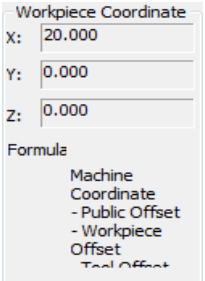
It is also called external offset which is used for recording the temporarily adjusting value of workpiece origin. The value can only be modified manually, and no automatic functions can change the value.

- Workpiece offset



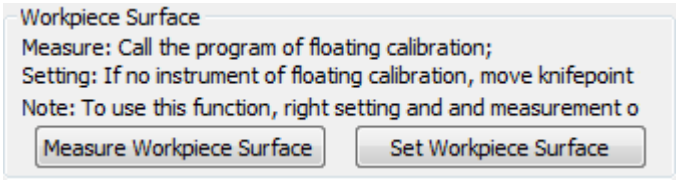
The offset of workpiece to machine origin, namely, the machine coordinates of workpiece origin. You can input manually the mechanic machine coordinates of workpiece origin (not recommended).

- Workpiece coordinate



The workpiece coordinates of current point. Please refer to section 3.6 for how to clear the workpiece coordinate of each axis.

- Workpiece surface



The function can be used for clearing workpiece coordinate of Z-axis.

[Measure Workpiece Surface] refers to calling out mobile calibration function, inputting the machine coordinates of position where the tool touches workpiece surface and clearing the workpiece coordinates after the machine coordinate is set. The function is used when there is a tool sensor.

[Set Workpiece Surface] refers to manually moving the tool to workpiece surface and setting the workpiece surface, and then clearing the workpiece coordinates after the surface is set. The function is used when there is no tool sensor.

4.4.5 Circumrotate Mirror Processing

The function is used for circumrotate mirror processing of a machining program.

You can click [Circumrotate Mirror Process] under menu [Operate]. The dialog box will pop up as follows:

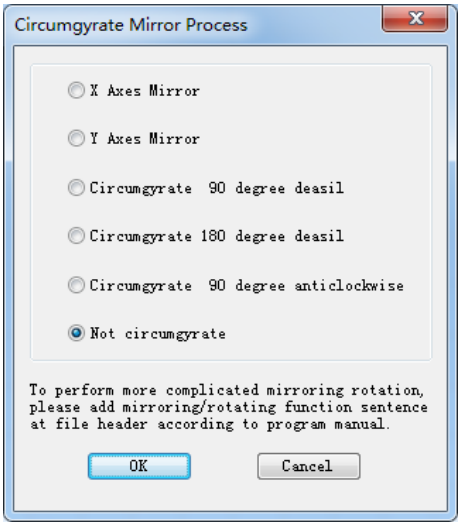


Fig. 4-11 Circumrotate Mirror Process

Please select items according to your needs, and then click [OK]. The system will execute the corresponding operations for the machining file loaded.

4.4.6 Single Block

You can set the program task to be executed as [Single Block], which supports error diagnoses and failure recovery well. In [Single Block] mode, machining process stops when the resultant velocity of all axes is zero.

Click [Single Block] under menu [Operate]. Namely, when ✔ appears before the item, each time you

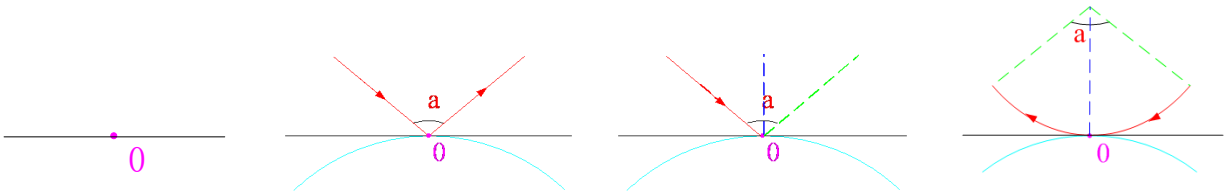
click [Start] button, the program will be executed once. Then, the system will suspend until you click [Start] button to execute the next line of the program. The machine stops after the whole program is executed.

4.5 NcEditor Operation

4.5.1 Set Lead Lines

Types of Lead Line

Lead line function can only be used for closed graphics, and it can be divided into four types: no lead line, tine lead line, arc lead line and kerf lead line.



No Lead Line	Tine Lead Line	Arc Lead Line	Kerf Lead Line
It means no lead line. You can decide whether to set it or not.	Lead-in edge and lead-out edge are all straight lines. The tool cut in along one edge of a closed angle and cut out along the other. For instance, lead line on the vertexes of rectangle, or on other vertexes.	Lead-in edge and lead-out edge are all arcs. The tool cut in along one arc and cut out along the other arc. For instance, lead line on circles, ellipse sand arcs, etc.	Principle: Set lead-in edge and no lead-out edge. The tool will stop after overcut a distance according to different wrap length along the original path in the end of cutting.

Fig. 4-12 Types of lead lines

CAUTION

1) There is no definite difference in the usage of tine lead line, arc lead line and kerf lead line, and they are interchangeable in practice. For instance, during setting an arc lead line on a rectangle vertex, the system will calculate the tine lead line automatically. Lead line types are selected depending on cutting technics. Lead line is used for precise machining. You can choose to set no lead line when lead lines are unnecessary.

2) Red lines are lead lines.

Lead lines can be divided into internal lead line and external lead line depending on the reserved parts of workpiece after machining completes.

If the object you selected is to be filled, the system will set internal lead line; if the object is not to be filled, the system will set external lead line. Details are as follows.

1) External lead line: after machining is finished, the inner part of object will be reserved, while the outer part will be abandoned.

2) Internal lead line: after machining is finished, the outer part of object will be reserved, while the inner one will be milled.

In object editing space, select a closed object. And there are two ways to set lead line on the object. Right click on the object and select “Set Introduce” in the shortcut menu, or select [Set Introduce] under menu [Object]. Dialog box “Set Introduce” is as shown in Fig. 4-13.

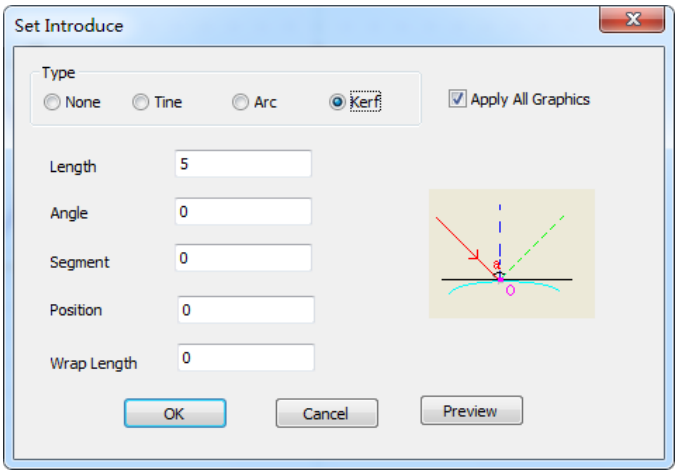


Fig. 4-13 Dialog box of setting lead line

In the dialog box above, you can set length, angle, segment number and position ratio of lead lines, the concrete definitions are as follows.

- Length: the length of cutting-in edge or cutting-out edge of a lead line;
- Angle: the angle of the two sides of lead line, namely, the value of angle “a”;
- Segment number: there are probable N lines (N is a natural number) on the selected object. The numeric order of the lead line among all the N lines is called ‘Segment number’’, which is used to fix the location of lead-in line.
- Position ratio: the ratio of the distance from the lead line to the start point of the closed segment divided by the total length of the segment, and the range for position ratio is (0, 1);
- Wrap length: compared with tine lead line and arc lead line, kerf lead line needs to set warp length. After the cutting is finished and lead-in point is reached, the tool continues to move a certain distance along the original path before it stops. Wrap length is the distance from lead-in point to stop point.

CAUTION

1) Segment order: select a closed object, or the combination of multiple objects, and there will be a mark on the object. Select [View Direction] under [View] menu, and the direction can be modified in [Object Attribute] window. The segement order starts from the mark point snf scoording to the machining order. And the segement number of the first segemnt is 0.You can set the segment number on of lead line.

2) Lead line is only valid for closed objects. You can set lead line on polylines which are completed with key C pressed.

4.5.2 Combine Object

Below is an example of executing [Combine Object] on two lines.

Firstly, draw the first line, and then click icon button [Catch Object] to catch the end points of the first line. Draw a second line from one end of the first line. Select [Combine Object] under menu [Object], or right click and select [Combine Object]. The two lines will be combined into one line.

During combining between lines, arcs and multi-lines, you can turn on catch object to catch the end points of the graphs for better use of function [Combine Object].

1) When end points of open path objects are not connected precisely because of improper operation or other reasons, you can use function [Combine Object] to combine the open objects into one object, i.e. multi-line or path if the distance between the end points of two objects is less than 0.1mm. It is suggested to turn on catch object before using function [Combine Object].
2) When you combine two or more arcs with function [Combine Object], the arcs will be combined from the original arc and in the direction of counterclockwise.

Tolerance Setting for Combining

In combining object, a certain standard about how far two objects are from each other to achieve combination need to be set. And the standard is combine object tolerance. Select [Combine Object Tolerance Setting] and you can set a value in range 0.001~5 in the dialog box popping up and click OK to confirm your setting.

4.5.3 Catch Options

Catch object is a function which can precisely catch character points of graphs in drawing. When you move the cursor near a character point and you can catch it easily. Select [Catch Option] under menu [View], or click on icon  in toolbar, a dialog box as shown in Fig. 4-14 will pop up.

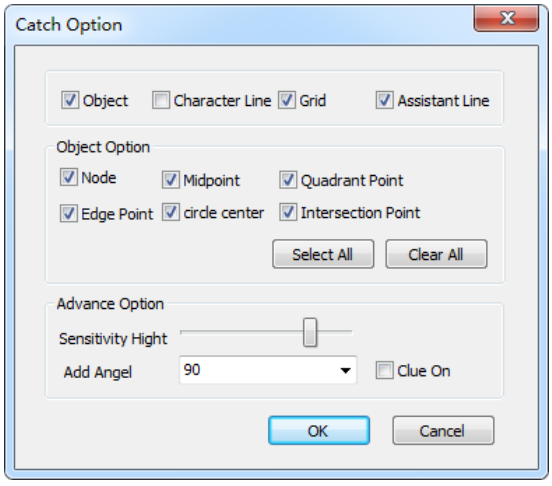


Fig. 4-14 Dialog box “Catch Option”

Character points of different graphics

1) Character point of a point: point;

2) Character point of a straight line: two end points, and a midpoint;

3) Character point of a multi-line: nodes and a circle center

4) Character point of a rectangle: 4 edge points;

5) Character point of a circle: 4 quadrant points and a circle center;

6) Character point of an ellipse: 4 quadrant points and a circle center;

7) Character point of an arc: a starting point and an end point, a midpoint, and a circle center.

[Sensitivity], [Add Angle], [Character Line] and [Assistant Line]

[Sensitivity] refers to the catching response intensity when the cursor approaches the character point.

[Add Angle] refers to the angle of catching operation. For example, if you set angle as [45], click on a starting point, and move the cursor. When the angle between the line you are drawing and the horizontal line is 45°, the character line appears in the editing space.

[Character Line] is used to prompt a specific position.

[Assistant Line] refers to the line expanded from the scale to the editing space. It can assist you to limit the position to draw a machining object.

4.5.4 Group and Array

◆ **Group**

To edit several objects, object groups or both object and object groups at the same time, you need to set them as a group. In [Object List] window, select all objects or object groups that need to be grouped, and then select [Group] under menu [Object], or right click and select option [Group].

◆ **Break groups**

To edit part of the objects in a group, you need to break the group.

In [Object List] window, select all object groups that need to be broken. Click on the group name and it will turn into dark blue. Then select [Break Group] under menu [Object], or right click and select option [Break Group]. Objects in the group will go back to the upper group.

◆ **Rectangular array**

In [Object List] window, select one or more objects or object groups, then click [Rect Array] item, and input row number, column number, row space and column space in the dialog box popped up. Click [OK] to confirm the setting.

◆ **Circular array**

In [Object List] window, select one or more objects or object groups, then click [Circle Array], input radius, original, and units on circle in the dialog box popping up. Click [OK] to confirm the setting.

4.5.5 Chamfer

NcEditor supports two types of chamfer, [Chemfer] and [Fillet]. Both of them have two modes: [The two sides] and [Internal graphics].

1) [The Two Sides] mode

[The Two Sides] mode refers to the chamfer angle between two edges.

For [Chemfer] in two-side mode, there are two kinds of size setting, distance setting and angle setting. To draw a chamfer with distance setting, select [Distance], and enter the distances for the first and second chamfer. Click [OK] and select two adjacent edges. To draw a chamfer with angle setting, select Angle, and enter the relevant value into input boxes. Click OK. Then select two adjacent two edges.

For [Fillet] in two-side mode, There is only radius setting. Input a value and click [OK]. Then select two adjacent on two edges.

2) [Internal Graphics] mode

In internal graphics mode, chamfer refers to the chamfer of all angles that meet the requirements in a graphic. Select Internal Graphics, enter the relevant data, and select any edge of the graphic, the system will automatically process all the angles that meet requirements. There are also two kinds of size setting, distance setting and angle setting in this mode. Detailed setting is similar to that in two-side mode.

4.6 Management of Waterjet Cutting

4.6.1 The Order of Switch Valves

You can choose the order of switch valve on your own.

By clicking [Open Valves Order] or [Close Valves Order] under menu [Machine Tool], you can set the open and close orders of valves and pumps, and look up items in their submenus where all on & off combination of valves and pumps are listed. Select one pair, and the system will automatically operate in the order, which is very convenient.

Additionally, Under [Machine Tool] menu, there are items of [Open/Close Booster], [Open/Close H/L], [Open/Close H.P.], [Open/Close Cutting], and [Open/Close Abrasive]. These items correspond to buttons on the operation panel. When these items are selected, buttons on the operation panel will change accordingly. The shortcut key is Shift+｜F1~F5｜.

4.6.2 High Pressure Time Statistics

By clicking [Hipressure Time Statistics] under menu [Machine Tool], the system will start to count the operation time of high pressure valve. In waterjet system, high pressure device is not only expensive, but also vulnerable. The manufacturer maintains the device on schedule. Click the item, and the system will pop up dialog box “High Pressure Time Statistics”. See Fig. 4-15. [Edit] button is used for resetting the time, which needs password to enter. Common users only have the privilege to read.

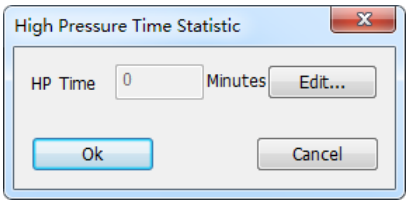


Fig. 4-15 High Pressure Time Statistics

4.6.3 Machining Information Statistics

After a file is processed, you can click [Statistics Information] under menu [Machine Tools], and a dialog box will pop up shown as Fig. 4-16. Machining information includes file name, process time, GXX L.(M), total L.(m), GXX time, total time and count.

If you need to calculate the charges of some files, in the charging section, set unit price, and select charging type from Gxx L.(m), Total L.(m), Gxx Time and Total time, and then calculate the charges.

By clicking [Print] button, you can save the charged machining information as txt file.

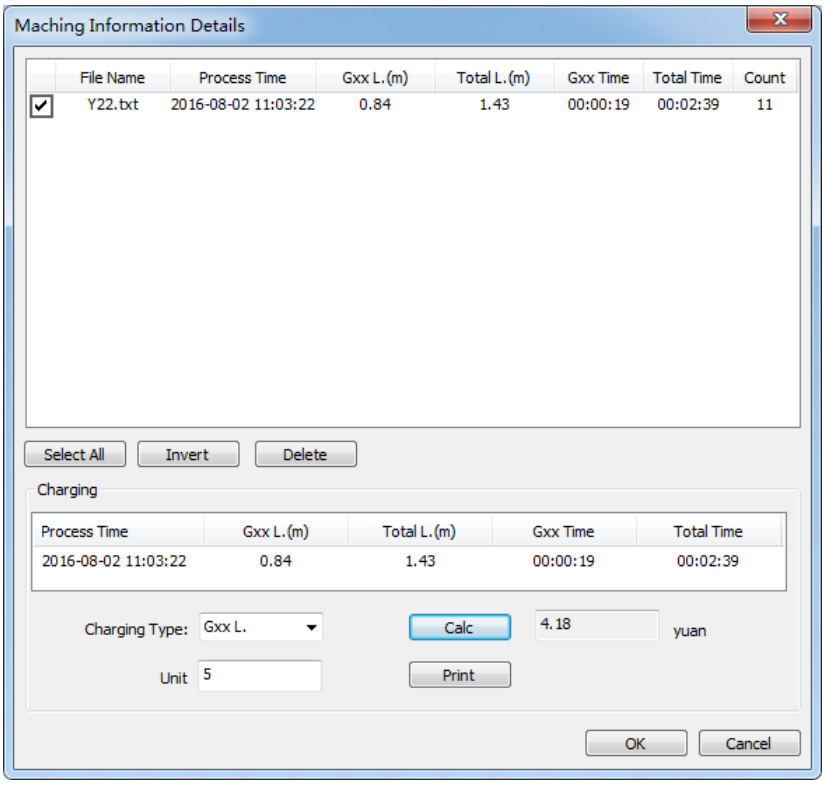


Fig. 4-16 Machining Information Details

4.6.4 Management of Vulnerable Devices

This function is used for the management of vulnerable devices. Click [Deadline Management] under menu [Machine Tools], and a dialog box will pop up as follows.

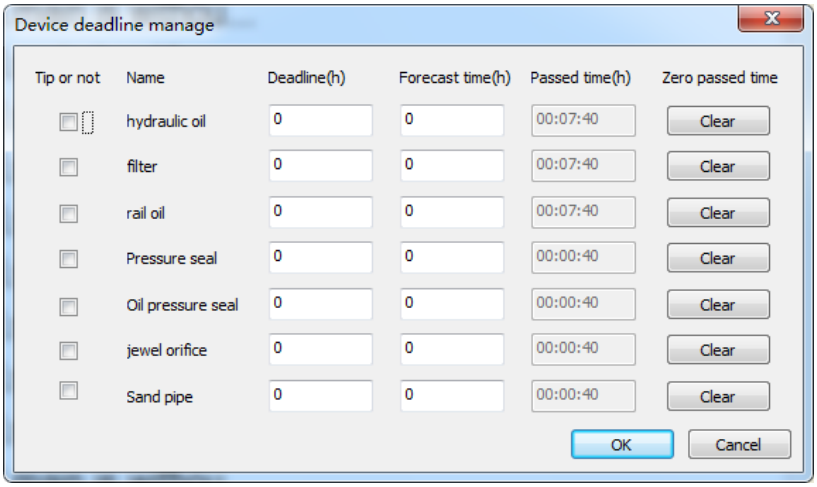


Fig. 4-17 Device Deadline Management

As shown above, you can input in [Deadline] box the life of a device, or in [Forecast time] box after how much time the system will prompt. After selecting [Tip or not] item, for example selecting [rail oil], the system will prompt the warning as follows when the [Forecast time] is reached. [Passed time] column shows the used time of devices. You can click [Clear] button to reset the [Passed time].

Fig. 4-18 Rail oil prompt

4.7 System Management

4.7.1 Language Selection

The software supports Chinese and English at present. You can select software language during installation, or switch the language over when the software is running.

By clicking [Language Choose] menu, you can choose languages between English and Chinese.

4.7.2 View System Information

In the software, you can view system information, such as version number, control card information and manufacturer information.

By clicking submenu [AboutNcStudio] under menu [Help], and you can view system information in dialog box “About NcStudio”.

4.7.3 Register

[Register] function is used for limiting the usage time of the system.

Click [About NcStudio] under menu [Help] to open [About NcStudio] dialog box as shown in Fig. 4-19, where you can obtain information, such as software version, the manufacturer, system information, as

well remaining usage time. If remaining time runs out, please send the “Device No.” to manufacturer to get a registration code. By clicking [Register] button, dialog box [Register] will pop up as shown in Fig. 4-20. Input the registration code and confirm to register usage time.

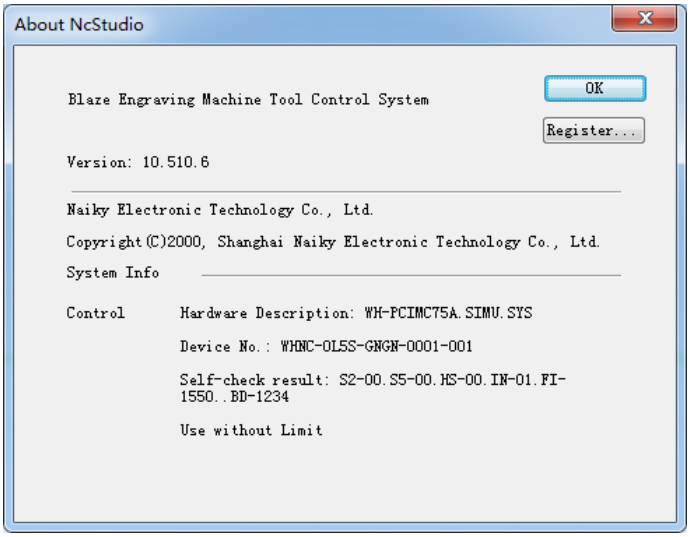


Fig. 4-19 About NcStudio

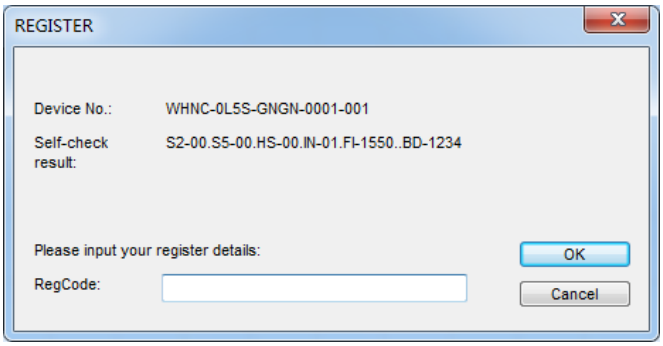


Fig. 4-20 Register

4.7.4 Manufactory Customize

Select [Manufactory Customize] under menu [help], and a dialog box will pop up. You can modify the splash of English or Chinese Interface by selecting [Chinese] or [English] in this dialog box. You will need to login to modify the title and add your company logo on the software. The picture imported for splash must be BMP format. Restart the software after you confirm the import to complete setting of splash. After setting a title, the software title will be updated immediately. The structure of the title will be [NcStudio icon + NcStudio + version + the customized title].

4.7.5 Generate Installation Pack

By clicking [Generate Installation] under menu [File], you can generate a complete installation based on current system data, which benefits system backup and distribution of stable version software.

4.8 Diagnosis

4.8.1 Logs

In [Log] window under [DIAG], important operation and system events can be recorded and displayed. You can browse the log info of start-up this time and history log information.

The log list contains items including: initialization and shutdown of the system, starting and end of auto machining, changes of workpiece coordinates, system alarms and other information. The manipulation button bar at the lower part of the window is shown below:

Fig. 4-21 Operation button bar in [Log] window

[Refresh]: shortcut key F1, which is used to refresh the current interface, and update logs.

[Clear]: shortcut key F2, which is used to clear all current logs.

[Show Info/Warning/Error/System/Nowadays]: corresponding shortcut keys are F3~F7. When you click these buttons, corresponding information of the button will be displayed in the log list; otherwise, the information will be hidden. In default setting, buttons except F6 [System] have been pressed down, namely, information, warning and error info since start-up in this time will be displayed by default. By clicking buttons, you can hide corresponding information and cancel the orange highlighted status.

CAUTION

1) Please note the difference of info types. Information log is marked with the icon , system log with icon , warning with icon and error log with icon .
2) When highlight button [Nowadays] is cancelled, all logs instead of logs since this time start-up will be shown in the list.
3) Please clear logs regularly. When the record file becomes too large in size, it may lag down the system perfomance and responsive time.

4.8.2 I/O Ports

Please see section 3.3.

4.8.3 PLC

With built-in PLC module of the software, I/O ports can be controlled by PLC operation.

The [PLC] window under [DIAG] is used for debugging the PLC. Refer to *NcStudio Programming Manual* for details. [PLC] window is shown as Fig. 4-22.

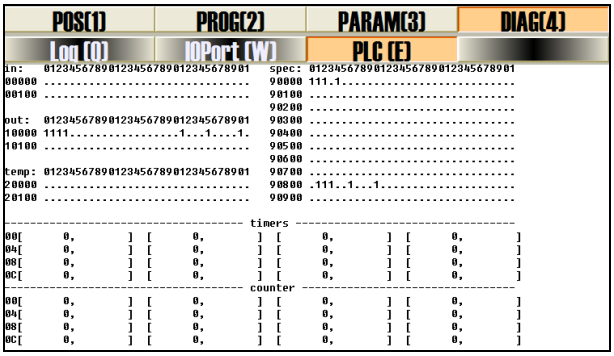


Fig. 4-22 PLC window

Buttons on manipulation bar are shown as Fig. 4-23. When buttons are clicked, corresponding dialog boxes will pop up, where you can input the channel No., timer No. and counter No. which need to be displayed. After it is confirmed, information of ports will be displayed in [PLC] window.

Fig. 4-23 Operation buttons in [PLC] window

4.9 Shortcuts

Shortcuts include shortcut icons, shortcut menus and shortcut keys list.

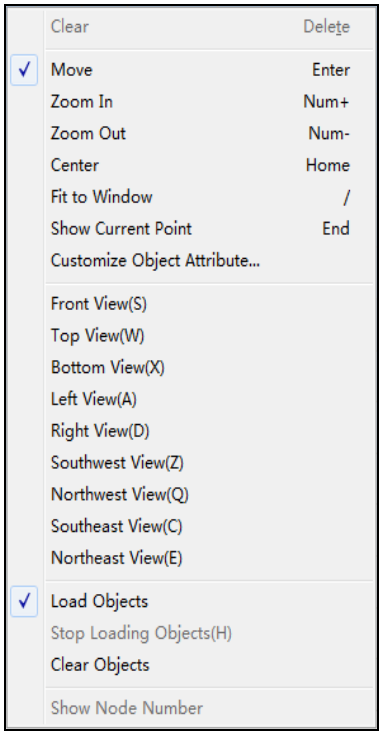
4.9.1 Shortcut Icon Buttons

In any operation mode, access [Normal] window under [POS]. Shortcut icons are shown on the left of the window.

	[Move]. The function can be achieved by keyboard manipulation. When the icon is clicked down, press and hold key Enter and press direction key, the track view will move in the direction you selected. When the numeric keys are unlocked, they can also be used as direction keys.
	[Zoom In] and [Zoom Out]. The shortcut keys are respectively key －and key + on numeric keypad.
	[Show Current Processing Point] .The shortcut key for the icon is key End.
	[Fit to Window Size]. The shortcut key for the icon button is key /.
	[Clear]. You can clear the actual machining track with this button. The shortcut key for this icon is key Delete.
	[Top View]. The shortcut key for the icon is key W.

4.9.2 Shortcut Menus

You can also right click on [Normal] window at random area to call out shortcut list.



➢ Clear

Function [Clear] in the menu is as same as the icon in the left of the [POS] window. By clicking this function, you can clear the actual machining track.

➢ Customize Object Attribute

By clicking [Customize Object Attribute], and dialog box [Custom Parameter] will pop up.

Click one of color buttons, and a color selection box will pop up. A new function of preloading track is added into NcStudio V10. Command colors are divided into simulation track colors and actual track colors.

In the dialog box [Custom Parameter], check [Show travel range], and workbench surface will be shown in the track window. Check [Draw the gridding], and gridding will be shown in the window. Check [Delete the Tracking View automatically after New Files Opened], the auto-clear function will be enabled.

When you load machining track and start machining, both the actual machining track and the stimulation track will be shown in the window. Function [Center] and [Fit to Window] will only be applied to the current machining track. Check [give current track priority to this file track] to give current track priority to simulation track. Otherwise, current track priority will be given to the actual track.

4.9.3 Shortcut Keys

Global Shortcut Keys	1 / 2 / 3 / 4	POS / PROG/ PARAM/ DIAG
	7 / 8 / 9	AUTO/ MANUAL/ REFER
	Q / W / E / R	1st/ 2nd/ 3rd/4th sub-tab of each ribbon tab
	F1~F8	Operation buttons
	Pause	Reset
	Ctrl+ O	Open and Load
	Ctrl+ U	Unload
	Ctrl+ N	New
	Ctrl+ E	Open and Edit
	Ctrl+ P	Edit Current Machine Program
	Ctrl+ M	Generate Installation

	Ctrl+ Shift+ R	Restart Software
	Ctrl+ I	Shutdown System
	Ctrl+ B	Reboot System
	Ctrl+ K	Show Desktop
	Alt+ F4	Exit
Shortcut Keys in Auto Mode	Ctrl+ F9	Select/Nearpoint Processing Block
	Shift+ F9	Breakpoint Resume
	F9	Start
	F10	Pause
	F11	Stop
Shortcut Keys in Interface Normal	Home	Center
	End	Show Current Processing Point
	+	Zoom In
	-	Zoom Out
	/	Fit to Window Size
	Delete	Clear
	S	Front View
	W	Top View
	X	Bottom View
	A	Left View
	D	Right View
	Z	Southwest View
	Q	Northwest View
	C	Southeast View
	E	Northeast View

5 Configuration of Five-axis Waterjet Cutting

5.1 Tool Path Generation for Five-axis Waterjet Cutting

1) Set five-axis machining parameters.
   Set the parameters in [PARAM]->[Waterjet], like [The lean direction of tool] (left or right), [The lean angle of plane cut], [Fitting accuracy tolerance], [Arc fitting accuracy] and [The height of workpiece], etc. Then set other related machining parameters, such as [RAPID_TRAVEL_FEEDRATE], delay time of high pressure, cutting and abrasive valve, and relative parameters of machining file, etc.
2) Import a machining file generated by CAD/CAM via NcEditor; take DXF file as an example:
   Open NcEditor. Select [Menu]-> [File]-> [Import DXF Code File]. Before importing a DXF file, set the related parameters of [6.5File translation] in NcStudio-> [PARAM] ->[Program].
3) Preprocess the objects in NcEditor. For example, combine the objects within the combine tolerance, uniform the direction of CAD objects according to the lean direction set in step 1), set lead-in/out line and tool compensation data. Please check whether the direction of machining objects is consistent with the previously set lean direction before combining objects.
4) CAD objects should be saved as Nce-formatted file. After all operations are finished, save the machining file again.
5) Make sure to home all axes. Guarantee that the workpiece origin of rotary axes is coincidental with the machine origin and on the workpiece surface. Be noted that workpiece origin of A axis and B axis cannot be directly set by resetting to prevent the misalignment of their workpiece origin and machine origin.
6) Start machining.

5.2 Five-axis Standard AB Type Configuration

AB type configuration of five-axis waterjet cutting CNC system consists of X, Y, Z, A and B axes, among which A-axis rotates around X-axis, B-axis rotates around Y-axis.

Fig. 5-1 shows five-axis AB type machine model:

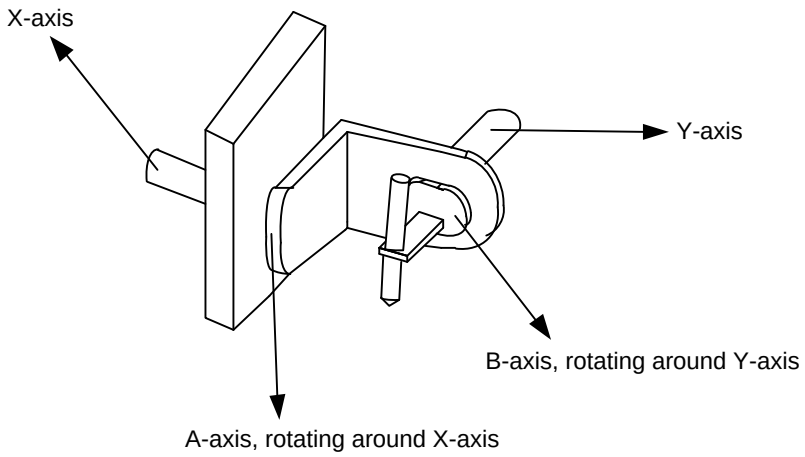


Fig. 5-1 Motion model of five-axis machine tool under AB type

Fig. 5-2 shows abstract model of five-axis AB type machine tool.

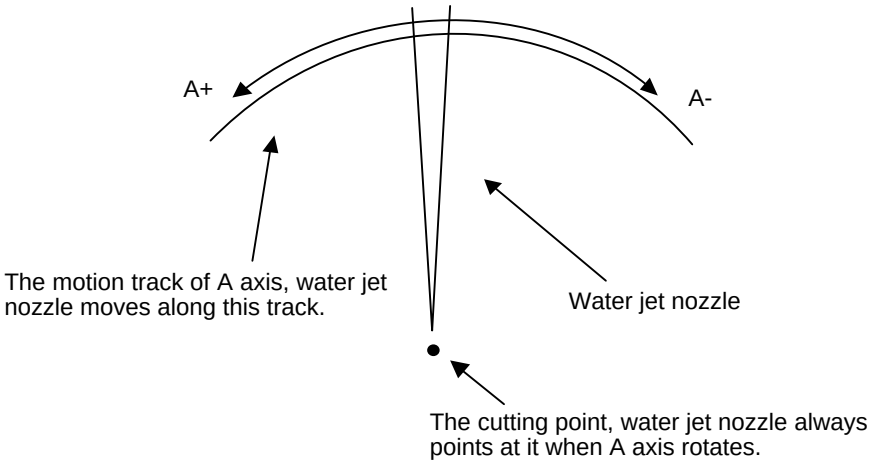


Fig. 5-2 Motion model of A-axis under AB type

5.2.1 Set “Back Distance”

Five-axis AB type waterjet demands that when A-axis and B-axis returns to machine origin, the direction of waterjet should be definitely vertical both to the X-axis and Y-axis, or it will impact on the process accuracy. For instance, due to the different groove angle in various places on processed products, it even gets influence on sizes of products.

For the convenience of users’ operation, after A-axis and B-axis returns to machine origin, the system will automatically and separately adjust axes’ angles in order to keep the waterjet vertical to the surface of X-Y. It needs to set parameter of [Back Distance]. Setting details are shown as follows.

Setting of A-axis “Back Distance”

After A-axis return to the machine origin, manually move A-axis and B-axis so that the waterjet is vertical to the XY surface. Then, set parameter [N74083] as the sum of original parameter value and coordinate of A-axis.

Setting of B-axis “Back Distance”

Similar to A-axis’ setting, when B-axis returns to the machine origin, manually move A-axis and B-axis so that the waterjet is vertical to the XY surface. Then, set parameter [N74084].

☞ **Related parameters**

Parameter		Range
N74083	BACK_DISTANCE A	-1000~1000 deg
N74084	BACK_DISTANCE B	

5.2.2 Sett “Show cutter”

Parameter [N87001] is the unique parameter of five-axis AB type waterjet cutting, because the motion track of rotary axis control point is different from that of cutting point (namely, the actual shape of processed products), the parameter is used for setting whether the motion track of control point or cutting point is shown on the system screen.

When the parameter is set to “Yes”, the track shown on the screen will be the cutting point’s motion track; while the parameter is set to “No”, the track will be the motion track of rotary axis control point.

☞ **Related parameters**

Parameter		Range
N87001	Show cutter	Yes: Show cutter; No: Not show cutter.

5.2.3 The Distance between Control Point and Cutting Point

For the five-axis waterjet machine with AB structure, especially the structure of the waterjet, cutting point and control point is different. Control point refers to the joint point of central lines of A-axis motor and B-axis motor; cutting line refers to when the waterjet is vertical to the surface of workpiece, the line through the projective point on the workpiece. It is shown in Fig. 5-3. During the processing, the joint point of A-axis and B-axis (control point) puts the cutting point in motion.

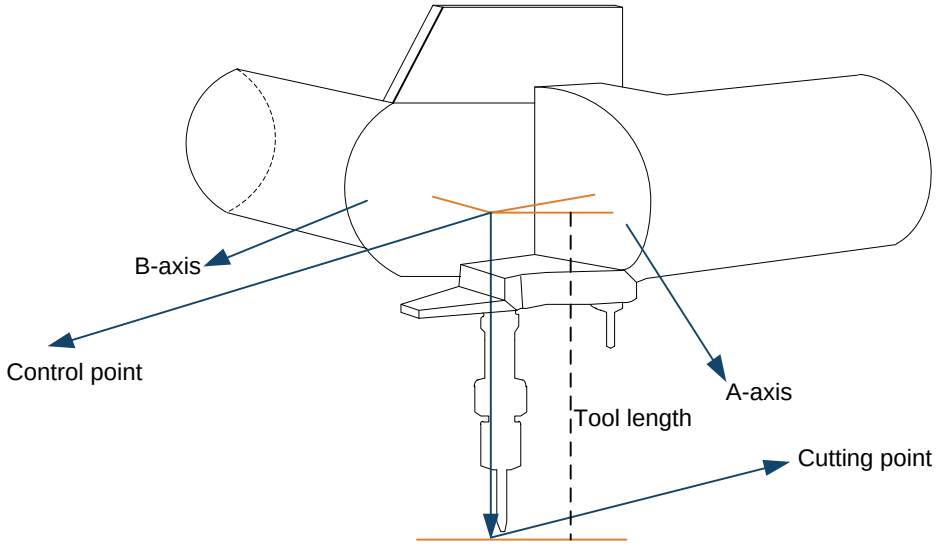


Fig. 5-3 Sketch map of the distance from control point to cutting point

Related parameters

Parameter		Range
N91034	Distance between control point and cut point	0~10000 mm

5.3 Five-axis Standard AC Type Configuration

Five-axis AC type waterjet cutting CNC system is consisted of X, Y, Z, A and C axes, among which A-axis is a self-rotating axis, C-axis rotates around Z-axis. As is shown in Fig. 5-4:

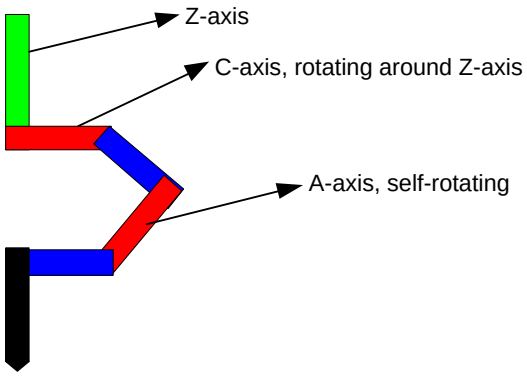


Fig. 5-4 Abstract model of AC type machine

5.3.1 A-axis and C-axis Rotation Model

According to mechanic structure, it is abstracted the motion's model between A-axis and C-axis which is shown below. As shown in the figure, A-axis rotates around the pivot line of itself, while C-axis's motion track is a circle in the horizontal level; and there is no linkage between them. In other words, A-axis will not put C-axis in motion or vice versa. Both their motion is independent, and there is no relationship of subordination. However, the angle of A-axis will influence C-axis's rotation angle.

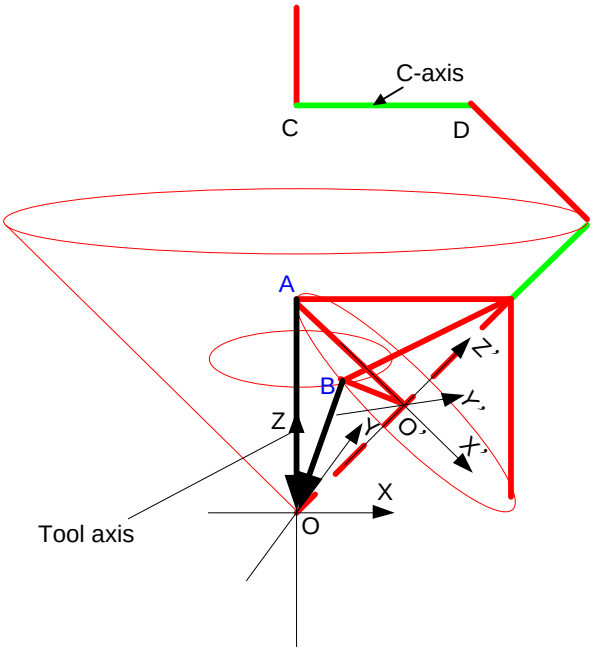


Fig. 5-5 Motion model of rotary axis track

In the above figure, AO stands for the waterjet, A the jet, O cutting point, O' the fixed rotating point, OO' self-spin axis, and the green horizontal line CD C-axis.

Through the figure, it is observed that the motion track of C-axis is a circle on XY surface with the cutting point as its center and C-axis rotating radius as its radius. Assume the circle as C1. A-axis is self-rotating axis. When A-axis is rotating, it will lead the water-jet-axis inclines to some degree. The motion track of waterjet axis A is a circle with O' as its center and OA as its radius. Assume the circle as C2. It is a circle in a space. Thus, you may use coordinate translation and rotation while calculating it.

Now assuming the parameter of waterjet cutting is set, and the waterjet tool path is generated, the whole motion process can be divided into 2 steps:

Step 1: AO is the initial position of waterjet axis. First, waterjet axis moves from point A to B along C2 under the force of self-rotation (it can be calculated according to inclined angle and direction). Then it moves to point B. In this phase, only A-axis rotates, the vector of waterjet is BO.

Step 2: after step 1, the waterjet-axis moved to position BO, according to the actual position of tool path, waterjet-axis BO moves to specified location along with C1, only C-axis rotates in this process. Any processes can be divided into 2 steps, but there is no relationship of between A-axis and C-axis, you should set the speed of A-axis and C-axis (through interpolation) according to waterjet-axis' accessible location. Through correct setting, these axes will arrive at the same location at the same time.

According to the figure above, the relationship between inclined angle and A-axis' angle can be calculated:

Angle of A-axis= $2\times\arcsin(\sin(\theta/2) / \sin(\phi))$

θ: Inclination angle of cutting plane;

ϕ: Included angle of self-rotating axis and tool axis initial position, 35 deg currently

5.3.2 Set "Back Distance"

Five-axis waterjet AC type machine demands that keep waterjet vertically straight down when A-axis and C-axis return to the zero position, the motor of A-axis should be in the straight right of the machine.

For the convenience of operation, after A-axis and C-axis return to the machine origin, the system will automatically and separately adjust axes' angles in order to keep the waterjet vertical to the XY surface. It needs to set parameter of [Back Distance]. Setting details are shown as follows.

Setting of A-axis "Back Distance"

Similar to A-axis' setting, when B-axis returns to the machine origin, manually move A-axis so that the waterjet is vertical to XY surface. Then, set parameter [N74083] as the sum of original parameter value and coordinate of A-axis.

Setting of C-axis "Back Distance"

After A-axis returns to the machine origin, manually move C-axis so that the spin axis of A-axis is parallel to X-axis. Then, set parameter [N74085] as the sum of original parameter value and coordinate of A-axis.

☞ **Related parameters**

Parameter		Range
N74083	BACK_DISTANCE A	-1000~1000 deg
N74085	BACK_DISTANCE C	

5.3.3 Introduction to Lambda 5E Absolute Encoder

This section is the introduction to waterjet with Lambda card AC structure which support YASKAWA absolute encoder.

5.3.3.1 Background and Principles

In AC structure waterjet cutting machine tool, A-axis and C-axis are rotary axes. A-axis is the self-rotating axis, rotating from -60° to 60°, while the range of C-axis is from -360° to 360°. Most of them adopt hollow motor, the cable connecting the motor of A-axis goes through the hollow part of C axis motor.

Due to the improperness to install travel limit switch on C-axis' structure, coiling situation of A-axis easily happens, which usually occurs when the machine stops emergently or return to machine origin repeatedly.

To solve the coiling problem with A-axis cable, absolute encoder can be used to record the actual physical position of machine, and it is unnecessary to return to machine origin repeatedly. Meanwhile, you can set parameter "The max rotating angle of C-axis" to coordinate with the machine during the process. Thus, the coiling can be solved.

In-built encoder in the servo motor is used for the feedback of position information. The category of encoder can be divided into: incremental encoder and absolute encoder.

Incremental encoder is different from absolute encoder in the aspect of structure which makes the servo motor with incremental encoder feeds back a pulse after it rotates to an angle, then the servo driver will rotates to an angle according to the feedback pulse. Servo motor with absolute encoder has the definite pulse code in the scope of 360°. Through the feedback pulse code of servo motor, the angle that servo motor has rotated can be determined.

Therefore, compare with incremental encoder servo system, absolute encoder servo system can record machine origin. The machine tool only needs to be set machine origin one time during the process of debugging. And in later machining, there is no need return to return to machine origin, which shortens preparation time, and improves production efficiency, especially in the situation of multi-axis.

In the perspective of hardware structure, the machine can leave out the signals of positive travel limit, negative travel limit and the origin, which saves the cost and decreases the complexity of wiring.

5.3.3.2 Setting Instruction and Application Steps

◆ **Parameter setting**

Set driver parameter Pn002=X0XX to the default of absolute encoder.

◆ **Application steps**

When you use encoder for the first time or after you reset the encoder, you should set the relationship between absolute encoder and actual machine position, that is, to determine the machine zero. Once set, actual machine position can be directly read after software startup. Detailed steps are as follows.

1. In sub function section [PosDiag] under [DIAG], click [RefSet] button and then enter password "ncstudio".
2. Move the machine to machine zero position manually of with handwheel.
3. Set the datum of each axis.
4. Restart the software following the prompt.

When getting actual machine position, and reading encoder position is succeeded, the position read is the correct position and origin mark will show in the corresponding axis. Otherwise, yellow alarm instead of origin mark will display. Please diagnosis and find the cause in this case.

After E-stop is canceled, and "Wait Time OnEstop" time, the system will get the actual machine position automatically.

You can read the actual machine position in sub function section [PosDiag]. Please be noted that 1 to 2 pulse difference between actual machine position and position shown in the software is normal.

5.3.3.3 Export and Import Datum

Steps of exporting datum are as follows.

1) In sub function section [PosDiag], click [ExportDatum] or press F1, and a dialog box will pop out.

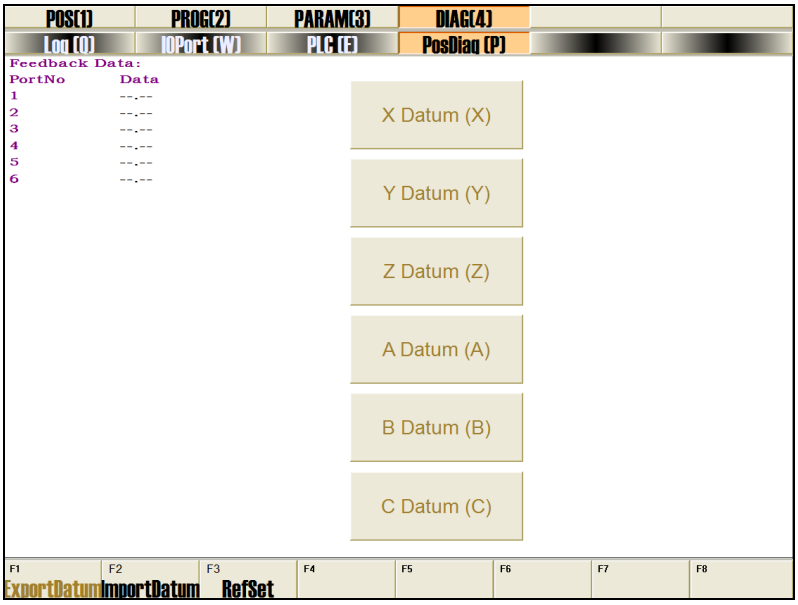


Fig. 5-6 Export Datum

2) Select the path to save the file and enter the file name. Click [Save], and exporting is completed.

After you update the software, you can import the datum as follows.

1) In sub function section [PosDiag], click [ImportDatum] or press F2, and a dialog box will pop out.
2) Find the datum file and click [Open].
3) Restart the software, the system will find the datum you set and get the current position.

◆ **Equations related to absolute encoder**

$$\text{Encoder position} = (\text{Current revolution} - \text{Revolution in machine origin}) \times \text{Screw pitch} + \text{Pulse in 1 revolution} \times \text{Screw pitch} \div \text{PG frequency dividing ratio}$$

$$\text{Current position} = \text{Machine origin position} + \text{Axis encoder direction} \times \text{Encoder position}$$

$$\text{PG frequency diving ratio} = \text{Encoder divider pulse No.} \times 4$$

☞ **Related Parameters**

Parameter		Range
N11001~ N11006	ABSOLUTE_ENCODE (X/Y/Z/A/B/C)	Yes: use; No: do not use
N11110~ N11115	AX_ENCODER_DIR (X/Y/Z/A/B/C)	1: positive direction; -1: negative direction
Encoder direction=axis direction ×logic control direction Driver uses negative logic. Set driver parameter Pn200 to 0005. When you debug with absolute encoder device, please set axis encoder direction according to the actual conditions.		
N11160~N11162	Leadscrew Pitch (X/Y/Z)	0.001~99999 mm
N11163~N11165	Leadscrew Pitch (A/B/C)	0.001~99999 deg
In case of directly-connected rail, set screw travel to the value of pitch; in case of gear, set screw travel to the result of "$\frac{\text{Mechanical deceleration ratio}}{\text{Screw pitch}}$".		
N11170~N11175	Frequency Division Pulses of PG (×4) (X/Y/Z /A/B/C)	1~999999
Encoder feedback pulse No. via frequency division of servo per revolution of motor.		
N11180~N11182	POWER_ERR (X/Y/Z)	0~99999 mm
N11183~N11185	POWER_ERR (A/B/C)	0~99999 deg
If the position difference between power and reboot is greater than this value, alarm.		
N11303	Wait Time On Estop	0.001~10 s

6 System Parameters

6.1 Common Parameters

Select [POS]-[Setting] to enter common parameter setting screen. Below are introductions to common parameters in different operation modes.

◆ **Common parameters settings in auto mode**

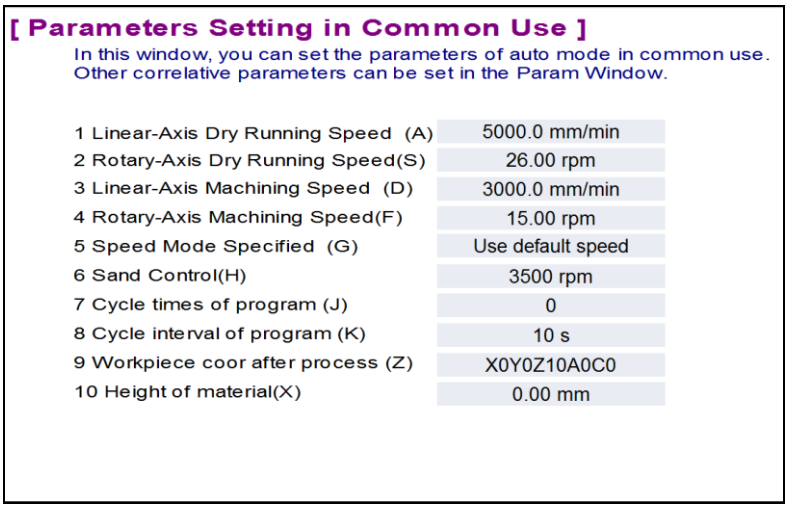


Fig. 6-1 Parameters Setting in Common Use in auto mode

Dry Running Speed	The speed of G00, namely, the speed of machine tool when positioning, parameter number is [N64020] “RAPID_TRAVEL_FEEDRATE”.
Machining Speed	Feedrate, referring to the speed of machine tool when machining. The corresponding parameter is [N92003]-[DEFAULT_FEEDRATE]. The machining speed should not be greater than dry running speed.
Speed Mode Specified	The parameter is [N72001] “SPEED_ASSIGNTYPE”. If it is set to “Use default speed”, the system will operate with “Machining Speed”, even if the program had set the speed. If it is set to “Use the speed in File”, the system will operate with the speed which is set as F command, otherwise it will operate with [Machining Speed].

◆ **Common parameter setting in manual mode**

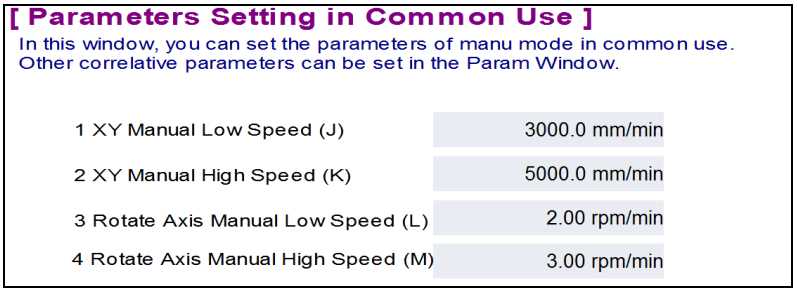


Fig. 6-2 Parameters Setting in Common Use in manual mode

Manual Low Speed	Including “XY Manual Low Speed” and “Rotate Axis Manual Low Speed”. Parameter numbers are respectively [N71000] and [N710002].
Manual High Speed	Including “XY Manual High Speed” and “Rotate Axis Manual High Speed”. Parameter numbers respectively are [N71001] and [N710004].

6.2 Parameter Modification

Enter [Param] function screen, and double click the parameter you need to modify, the system will pop up a dialog box. Then you can modify the parameter according to your needs.

All parameters can only be modified when the system is in idle state.

6.3 List of Common Parameters

According to privileges, parameters can be divided into operator parameters, manufacturer parameters and developer parameters.

Manufacturer parameters need developer password to visit. Developer parameters need developer password, and can only be accessed by developers. This section lists manufacturer parameters.

Param	Name	Range	Default	Effecting
1.0 Axes general				
N10000	AXIS_DIR X	1:positive direction; 2: negative direction.	1	After restart
N10001	AXIS_DIR Y		-1	After restart
N10002	AXIS_DIR Z		-1	After restart
N10003	AXIS_DIR A		1	After restart
N10004	AXIS_DIR B		1	After restart
N10005	AXIS_DIR C		1	After restart
N10010	AXIS_PULSE_FACTOR X	9e-007~999 mm/p	0.001	After restart
N10011	AXIS_PULSE_FACTOR Y		0.001	After restart
N10012	AXIS_PULSE_FACTOR Z		0.002	After restart
N10013	AXIS_PULSE_FACTOR A	9e-007~999 deg/p	0.01	After restart

Param	Name	Range	Default	Effecting
N10014	AXIS_PULSE_FACTOR B		0.01	After restart
N10015	AXIS_PULSE_FACTOR C		0.01	After restart
N10020	WORKBENCH_LOWER_LIMIT X	-99999~99999 (mm)	0	After restart
N10021	WORKBENCH_LOWER_LIMIT Y		0	After restart
N10022	WORKBENCH_LOWER_LIMIT Z		-300	After restart
N10023	WORKBENCH_UPPER_LIMIT A	-99999~Workbench Upper Limit (deg)	-10	After restart
N10024	WORKBENCH_UPPER_LIMIT B		-8	After restart
N10025	WORKBENCH_UPPER_LIMIT C		-10000	After restart
N10030	WORKBENCH_UPPER_LIMIT X	-99999~99999 mm	10000	After restart
N10031	WORKBENCH_UPPER_LIMIT Y		10000	After restart
N10032	WORKBENCH_UPPER_LIMIT Z		0	After restart
N10033	WORKBENCH_UPPER_LIMIT A	-99999~99999 deg	10	After restart
N10034	WORKBENCH_UPPER_LIMIT B		8	After restart
N10035	WORKBENCH_UPPER_LIMIT C		10000	After restart
N10040~ N10045	CHECK_WORKBENCH_RANGE (X/Y/Z/A/B/C)	Yes: check; No: do not check	Yes	After restart
N10060	AxisType X	1: Linear axes; 2: Rotating axes; 3: Rotating axes with shortest path.	1	After restart
N10061	AxisType Y		1	After restart
N10062	AxisType Z		1	After restart
N10063	AxisType A		2	After restart
N10064	AxisType B		2	After restart
N10065	AxisType C		2	After restart
N10070	Rotation Axis Group Type	0:Only transitional axes rotate in the process of machining; 1: Only A-axis rotates in the process of machining; 2: Only B-axis rotates in the process of machining; 3: Only C-axis rotates in the process of machining; 4:Only A-axis and B-axis rotate in the process of machining; 5:Only B-axis and C-axis rotate in the process of machining; 6: Only A-axis and C-axis rotate in the process of machining;	Configuration of 4 axes: 3; Configuration of 5 axes AC type: 6; Configuration of 5 axes AB type: 4;	After Save Program

Param	Name	Range	Default	Effecting
		7: A-axis ,B-axis and C-axis all rotate in the process of machining; 8: A-axis rotates a setting value Automatically before process of machining, and only C-axis rotates in the process of machining; 9: A-axis and C-axis both rotate with tool length.		
1.2 Compensation				
N12000	ENABLE-LEADSCREW_CMPN	Yes: enable; No: disable	Yes	After restart
N12001	ONLY_BACKLASH_CMPN	Yes: the system will only read data from BACKLASH parameters to do compensation. No: it will read the data of backlash and pitch errors from error data file and take comprehensive compensation.	Yes	After restart
N12010	BACKLASH X	0~1 mm	0	After restart
N12011	BACKLASH Y		0	After restart
N12012	BACKLASH Z		0	After restart
N12013	BACKLASH A	0~1 deg	0	After restart
N12014	BACKLASH B		0	After restart
N12015	BACKLASH C		0	After restart
N12020	ENABLE_QE_CMPN	Yes: enable; No: disable	No	After restart
Parameters of group 0 are listed below. Other groups' parameters will not be introduced here.				
N12030	TIME	0~10 sec	0	After restart
N12031	DISTANCE	0~10 mm	0	After restart
N12032	DELAY	0~10 sec	0	After restart
N12033	INTENSITY	0.01~0.99	0.75	After restart
1.3 Velo/Acc limits				
N13000	AX_MAX_VOL X	0.001~100000 mm/min	48000	Immediately

Param	Name	Range	Default	Effecting
N13001	AX_MAX_VOL Y	0.001~100000 mm/min	48000	Immediately
N13002	AX_MAX_VOL Z	0.001~100000 mm/min	48000	Immediately
N13003	AX_MAX_VOL A	0.001~277.228 rpm	133.3336	Immediately
N13004	AX_MAX_VOL B	0.001~277.228 rpm	133.3336	Immediately
N13005	AX_MAX_VOL C	0.001~277.228 rpm	133.3336	Immediately
4.1 Lube				
N41000	AUTO_LUB	Yes: Automatically open lubrication pump; No: Do not open lubrication pump.	No	Immediately
N41001	LUB_INTERVAL	3.6~3.6e+006 sec	18000	Immediately
N41002	LUB_DURATION	1~100 sec	5	Immediately
4.2 Cylinder				
N42000	RED_LIGHT_WHEN _FINISH	Yes: prompt; No: do not prompt.	No	After restart
5.2 Handwheel				
N52001	HW_COUNT_STRICTLY	Yes: Adopt; No: do not adopt.	Yes	After restart
N52002	HW_DIRECTION	1: positive direction; -1: negative direction	-1	After restart
N52003	HW_MULTIPLE_1	0.001~10 mm	0.001	After restart
N52004	HW_MULTIPLE_10	0.001~10 mm	0.01	After restart
N52005	HW_MULTIPLE_100	0.001~10 mm	0.1	After restart
N52006	HW_GEAR_FACTOR_numerator	1~1000	1	After restart
N52007	HW_GEAR_FACTOR_ denominator	1~1000	1	After restart
N52010	ForbidenHandwheelGearTrack BackFunction	Yes: forbidden No: do not forbidden.	No	After restart
N52012	HW_ACC	1~1000 mm/s^2	200	After restart
N52020	SupportNKExKeyset	0: none; 1: Mini keyboard; 2: Naiky extended keyboard.	0	After restart
N52021	InstallExtendedKeyboardHook	Yes: install; No: uninstall	Yes	After restart
5.3 Operation panel				
N53100	TurnOffPortsWhenTrackingBack	Yes--if on, turn off; No--no change	Yes	Immediately
6.2 G code options				
N62000	RT_DEC_LEN	0~999 mm	5	Immediately

Param	Name	Range	Default	Effecting
N62001	SPEED WHILE APPROACHING TO WORKPIECE	0.001~99999 mm/min	120	Immediately
N62020	IJK_INC	Yes: enable; No: disable	Yes	Immediately
N62021	RADIUS_TOL	0~9999 mm	1	Immediately
N62090	G09_TOL X	0~99 mm	0.001	Immediately
N62091	G09_TOL Y	0~99 mm	0.001	Immediately
N62092	G09_TOL Z	0~99 mm	0.001	Immediately
N62093	G09_TOL A	0~99 deg	0.001	Immediately
N62094	G09_TOL B	0~99 deg	0.001	Immediately
N62095	G09_TOL C	0~99 deg	0.001	Immediately
6.3 Track control				
N63000	CONN_ANGLE_DIST	0~999 mm	0	Immediately
N63001	TRACK_CORNER_TOL	0~0.1 mm	0	Immediately
N36006	SMOOTH_TIME	0~1 s	0	Immediately
N63007	Slow Distance of intro line	0~2	1.5	Immediately
N63008	Whether the Slow Distance of intro line is valid	0: invalid; 1: valid	No	Immediately
N63009	The Slow Distance of intro line is near to the start of intro line or to the figure	Yes: near to the figure; No: near to the start of intro line.	No	Immediately
6.4 Speed/Acc				
N64000	AX_STARTUP_VOL	0~100000 mm/min	0	Immediately
N64001	Whether use StartUpSpeed	Yes: to use; No: do not use.	No	Immediately
N64011	Reverse Link Speed Dif	0~100000 mm/min	600	Immediately
N64020	RAPID_TRAVEL_FEEDRATE	0~100000 mm/min	5000	Immediately
N64022	SLOW Distance of intro line	0~99%	50	Immediately
N64027	Min smooth angle during lines connect	0~180 deg	20	Load program
N64031	REV_RAPID_TRAVEL_FEEDRATE	0~100000 rpm	26	Immediately
N64034	REV_DEFAULT_FEEDRATE	0~100000 rpm	15	Immediately
N64080	ANGLE_ACCELERATION	0.001~100000 deg/s^2	300	Immediately
N64101	AX-LINEAR_POST_ACC	0.001~100000 mm/s^2	400	Immediately
N64102	Z_LINEAR_ACC	0.001~100000 mm/s^2	400	Immediately
N64120	AX_CON_ACC	0.001~100000 mm/s^2	3000	Immediately
N64150	AX_ACC_ACC	0.001~100000 mm/s^3	5000	Immediately
N64200	ENABLE_SAO	Yes: valid; No: invalid	Yes	Immediately
N64201		0~90 deg	90	Immediately

Param	Name	Range	Default	Effecting
N64203	INTERP_ALGO	0: triangle; 1: S_Type	1	Immediately
N64207	LIMIT_ARC_VELO	Yes: valid; No: invalid	No	Immediately
N64236	ARC_FORWARD_LENGTH	0~100 mm	2	Immediately
N64240	SLIDE_TIME_GENE	0.01~10	1	Immediately
N64241	Slide speed for small lines	Yes: valid; No: invalid	No	Immediately
N64242	Reference length of slide speed for small lines	0.001~10 mm	2	Load Program
N64243	SPEED_SLIDE_LENGTH	0~300 mm	0	Immediately
6.5 File translation				
N65100	TOOL_UP_HEIGHT	0~99999 mm	10	Load Program
N65102	DEPTH_LAYER	-99999~0 mm	0	Load Program
N65103	FIRST_POINT_AS_ORIGIN	Yes: use; No: do not use	Yes	Load Program
N65105	ENABLE_MACHINE_BOTTOM	Yes: enable; No: disable	Yes	Load Program
N65106	Force To Recongnize Dxf File As Metric Sys	Yes: recognize; No: do not recognize	Yes	Load Program
N65107	IGNORE HEIGHT IN DXF	Yes: ignore; No: do not ignore	Yes	Load Program
7.1 Manu				
N71000	JOG_VOL	0.001~ the highest velocity of XY-axis mm/min	3000	Immediately
N71001	RAPID_JOG_VOL	the lowest velocity of XY-axis mm/min ~100000 mm/min	5000	Immediately
N71002	REV-JOG_VOL	0~100000 rpm	2	Immediately
N71003	REV_RAPID_JOG_VOL	0~100000 rpm	3	Immediately
N71020	LOCK_DIR_AFTER_LIMITED	Yes: lock; No: not lock	Yes	After restart
N71021	Jiggle Feedrate	0~100000 mm/min	400	Immediately
N71022	JIGGLE_STEP_LENGTH	0.001~2 mm	0.5	Immediately
N71023	JOG_VOL_Z	0.001~ the highest velocity of Z-axis mm/min	800	Immediately
N71024	RAPID_JOG_VOL_Z	the lowest velocity of	1000	Immediately

Param	Name	Range	Default	Effecting
		Z-axis mm/min ~100000 mm/min		
N71028	The step range of Z axis	0~10000 mm	10000	Immediately
7.2 Auto				
N72001	SPEED_ASSIGN_TYPE	0: use the speed in the file; 1: use default speed; 2: use the speed in the fixed proportion	1	Immediately
N72006	HEIGHT_Z_RAISE	-10000~10000 mm	5 mm	Immediately
N72010 ~N72015	ENABLE_WORKAREA_CHECK (X/Y/Z/A/B/C)	Yes: enable; No: disable	No	Immediately
N72020	WORKAREA_LOWER_LIMIT X	-99999~99999 mm	-10000	Immediately
N72021	WORKAREA_LOWER_LIMIT Y		-10000	Immediately
N72022	WORKAREA_LOWER_LIMIT Z		-10000	Immediately
N72023	WORKAREA_LOWER_LIMIT A	-99999~99999 deg	-10000	Immediately
N72024	WORKAREA_LOWER_LIMIT B		-10000	Immediately
N72025	WORKAREA_LOWER_LIMIT C		-10000	Immediately
N72030	WORKAREA_UPPER_LIMIT X	-99999~99999 mm	10000	Immediately
N72301	WORKAREA_UPPER_LIMIT Y		10000	Immediately
N72032	WORKAREA_UPPER_LIMIT Z		10000	Immediately
N72033	WORKAREA_UPPER_LIMIT_A	-99999~99999 deg	10000	Immediately
N72034	WORKAREA_UPPER_LIMIT_B		10000	Immediately
N72035	WORKAREA_UPPER_LIMIT_C		10000	Immediately
7.3 Pause				
N73000	PAUSE_DOWN_VOL	0~100000 mm/min	600	Immediately
N73001	PAUSE_UP_VOL	0~100000 mm/min	600	Immediately
N73002	PAUSE_OPTION	0: lift up a specified value; 1: go to a specified position in workpiece coordinate.	0	Immediately
N73003	Z_WPCOOR_ON_PAUSE	0~9999 mm	10	Immediately
N73004	Z_OFFSET_ON_PAUSE	0~500 mm	0	Immediately
N73006	PAUSE_PORT_CLOSE_SYTLE	1: Keep opening; 2: Immediately close; 3: Delay close	2	Immediately
N73007	START_OPEN_PORT_SYTLE	1: Open; 2: Immediately open; 3: Delay open	3	Immediately
7.4 Bkref				

Param	Name	Range	Default	Effecting
N74000	CANCEL_REFPT_AFTER_ESTOP	Yes: cancel; No: not cancel	Yes	Immediately
N74010	REFPT_COOR X	0~10000 mm	0	After restart
N74011	REFPT_COOR Y	0~10000 mm	0	After restart
N74012	REFPT_COOR Z	-300~0 mm	0	After restart
N74013	REFPT_COOR A	-10~10 deg	0	After restart
N74014	REFPT_COOR B	-8~8 deg	0	After restart
N74015	REFPT_COOR C	-10000~10000 deg	0	After restart
N74020	COARSE_LOCATING_DIR X	1: positive direction; -1: negative direction	-1	Immediately
N74021	COARSE_LOCATING_DIR Y		-1	Immediately
N74022	COARSE_LOCATING_DIR Z		1	Immediately
N74023	COARSE_LOCATING_DIR A		1	Immediately
N74024	COARSE_LOCATING_DIR B		1	Immediately
N74025	COARSE_LOCATING_DIR C		-1	Immediately
N74030	COARSE_LOCATING_VELO X	0.001~10000 mm/min	1800	Immediately
N74031	COARSE_LOCATING_VELO Y		1800	Immediately
N74032	COARSE_LOCATING_VELO Z		1800	Immediately
N74033	COARSE_LOCATING_VELO A	0.001~10000 rpm	5	Immediately
N74034	COARSE_LOCATING_VELO B		5	Immediately
N74035	COARSE_LOCATING_VELO C		5	Immediately
N74050	FINE_LOCATING_DIR X	1: positive direction; -1: negative direction	1	Immediately
N74051	FINE_LOCATING_DIR Y		1	Immediately
N74052	FINE_LOCATING_DIR Z		-1	Immediately
N74053	FINE_LOCATING_DIR A		-1	Immediately
N74054	FINE_LOCATING_DIR B		1	Immediately
N74055	FINE_LOCATING_DIR C		1	Immediately
N74060	FINE_LOCATING_VELO X	0.01~10000 mm/min	60	Immediately
N74061	FINE_LOCATING_VELO Y		60	Immediately
N74062	FINE_LOCATING_VELO Z		60	Immediately
N74063	FINE_LOCATING_VELO A	0.01~10000 rpm	0.166667	Immediately
N74064	FINE_LOCATING_VELO B		0.166667	Immediately
N74065	FINE_LOCATING_VELO C		0.166667	Immediately
N74080	BACK_DISTANCE X	-1000~1000 mm	2	Immediately
N74081	BACK_DISTANCE Y		2	Immediately
N74082	BACK_DISTANCE Z		-2	Immediately
N74083	BACK_DISTANCE A	-1000~1000 deg	10	Immediately
N74084	BACK_DISTANCE B		8	Immediately
N74085	BACK_DISTANCE C		8	Immediately
N74090	Location Times	-	1	Immediately
N74091	Axis Type of Bkref X	Yes: back to the	Yes	After restart

Param	Name	Range	Default	Effecting
N74092	Axis Type of Bkref Y	reference point before programming; No: not back to the reference point before programming		After restart
N74093	Axis Type of Bkref Z			After restart
N74094	Axis Type of Bkref A			After restart
N74095	Axis Type of Bkref B			After restart
N74096	Axis Type of Bkref C			After restart
N74102	Double Y Encoder Origin Error	-10000~10000 mm	0	Immediately
N74104	Enable Value of Double Y Encoder Origin Error	0~1e+006 mm	10000	Immediately
7.9 Operation others				
N79003	SAFE_HEIGHT	-10000~10000 mm	10	Immediately
N79006	NEARPOINT_RANG	0~500	100	Immediately
N79007	INTERVAL_DISTANCE	0~10000	10	Immediately
N79008	GROOVE_LENGTH	0~100	3	Immediately
N79009	Use Break-down Height or not	Yes: to use; No: not to use	No	Immediately
N79010	Use special control of Z axis or not	Yes: to use; No: not to use	No	Immediately
N79100	Action After Programming	0: Do not move; 1:Back to fix point; 2:Back to workpiece origin; 3:Back to set point (workpiece coordinates)	3	Immediately
N79101	SETPT_POS X	-99999~99999 mm	0	Immediately
N79102	SETPT_POS Y		0	Immediately
N79103	SETPT_POS Z		10	Immediately
N79104	SETPT_POS A	-99999~99999 deg	0	Immediately
N79105	SETPT_POS B		0	Immediately
N79106	SETPT_POS C		0	Immediately
N79110	FIXPT_POS X	-99999~99999 mm	0	Immediately
N79111	FIXPT_POS Y		0	Immediately
N79112	FIXPT_POS Z		0	Immediately
N79113	FIXPT_POS A	-99999~99999 deg	0	Immediately
N79114	FIXPT_POS B		0	Immediately
N79115	FIXPT_POS C		0	Immediately
N79200	DisplaySimuOutRange	Yes: prompt; No: not prompt	Yes	Immediately
N79201	SOFTWARE_CONFIGURATION	0: normal; 1: Enable height measurement	1	After restart

Param	Name	Range	Default	Effecting
8.1 Position view				
N81000	ATUO_LOAD_TRACK	Yes: load; No: not load	Yes	Immediately
N81001	AUTO_LOAD_TRACK_LIMIT	0~100000 KB	1000	Immediately
8.3 Door Open Parameter				
N83009	Y-AxisWorkRange	1: positive interval; 2: negative interval	1	Immediately
8.7 Show view				
N87004	FONT_SHOW_ADJUST_FACTOR	0~100	1	Immediately
9.0 Application general				
N90000	Use Metric	Yes: to adopt; No: do not adopt	Yes	After restart
9.3				
N93005	Axis of rotation is valid or not	1: valid; 2: invalid	1	Load program

6.4 List of Waterjet Cutting Parameters

The list of waterjet cutting parameters is closely related to waterjet cutting machine. All parameters under 2 axes, 3 axes, 4 axes and 5 axes (AB & AC structure) configurations are listed below.

Param	Name	Range	Default	Effecting
9.0 Application general				
N90010	OPEN_HP_DELAYTIME	0~600000 ms	1000	Immediately
N90011	CLOSE_HP_DELAY_TIME	0~600000 ms	1000	Immediately
N90020	OPEN_CUTTING_DELAY_TIME	0~600000 ms	1000	Immediately
N90021	CLOSE_CUTTING_DELAY_TIME	0~600000 ms	1000	Immediately
N90030	OPEN_ABRASIVE_DELAY_TIME	0~600000 ms	1000	Immediately
N90031	CLOSE_ABRASIVE_DELAY_TIME	0~600000 ms	1000	Immediately
N90032	PAUSE_BEFORE_CUT	0~600000 ms	5000	Immediately
N90033	PRESSURE_DISCHARGE_DELAY_TIME	0~600000 ms	0	Immediately
N90034	PAUSE_AFTER_CUT_DELAY_TIME	0~600000 ms	0	Immediately
N90045	After Stop close Abrasive And Hv Delay time	0~600000 ms	0	Immediately
N90046	Close Booster Delay Time	30~900000 ms	30000	Immediately
9.1 Operator Parameter				
N91000	The lean angle of plane cutted	180~60 deg	2	Load program

Param	Name	Range	Default	Effecting
N91002	The lean direction of tool	0: machine plane lean right of machining direction; 1: machine plane lean left of machining direction	1	Load program
N91006	Arc fitting accuracy	0.0001~1000 mm	0.01	Load program
N91008	Angle between spin axis and tool vector when tool vector is on zero position	-90~90 deg	45	Load program
N91025	Max rotating angle of C-axis	360~108000 deg	0	Load program
N91027	Max rotating angle of A-axis	0.01~90 deg	20	Load program
N91034	Distance between control point and cut point	0~10000 mm	0	Load program
9.2 Program Parameter				
N92000	AX_LINEAR_MACH_ACC	0.001~100000 mm/s^2	400	Immediately
N92001	REF_CIRCLE_MAX_VELO	0.001~100000 mm/min	1800	Immediately
N92002	CIRCLE_MIN_VELO	0.001~100000 mm/min	100	Immediately
N92003	DEFAULT_FEEDRATE	0~100000 mm/min	3000	Immediately
N92004	EXACT_STOP_TIME	0~999 s	0	Immediately
N92006	Break-down Height	0~1000 mm	0	Immediately
N92008	AX_STARTUP_VOL	0~100000 mm/min	0	Immediately
N92009	Whether use StartUpSpeed	Yes: use; No; not use	No	Immediately
N92011	WHETHER_USE_ACC	Yes: use; No; not use	No	Immediately
N92012	CONNER_SPEED_PERCENTAGE	0%~100%	100	Immediately
N92013	CORNER_ACC_DISTANCE	0~100 mm	2	Immediately
N92014	CORNER_DECEL_DISTANCE	0~100 mm	2	Immediately
N92015	CLOSED_OBJECT_DECEL_DISTANCE	0~100 mm	2	Immediately
N92018	CONNER_SLOW_ANGLE	0~180 deg	60	Immediately

6.5 Parameter Backup and Auto Backup

NcStudio system is entitled with parameter backup and auto backup function. With the backup function，you can manually back up parameter settings and restore them as you need, saving you from setting again. If you forget to save the parameters after setting, you can use auto backup function to restore the parameters set as early as the factory date and as late as last time closing software.

Parameter backup

In [Backup] window under [PARAM], you can back up the parameter settings. After all parameters have been well set, click “Save” button to save the parameter setting. A dialog box where you can input the name will pop up. Up to 8 groups of parameter settings can be saved. When any one of the parameter settings group is needed to be restored, you can click “Load” button to read the settings and load them into the system.

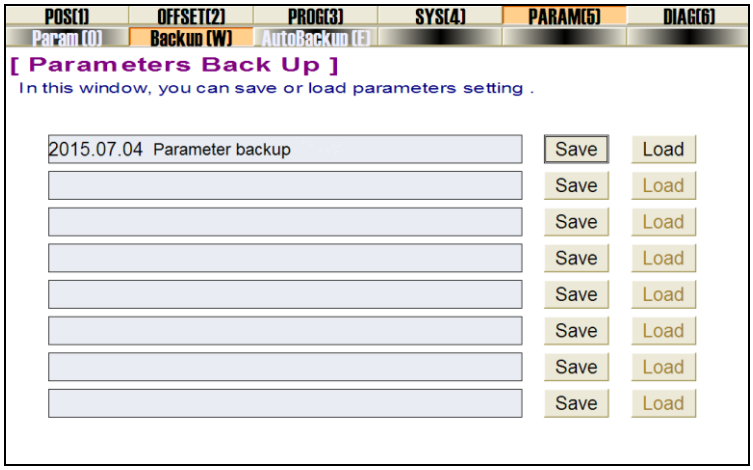


Fig. 6-3 Parameter Backup

◆ **Parameters Auto Backup**

In [AutoBackup] window under [PARAM], you can restore parameter settings which have been saved automatically. Several options are offered, in which you can select and load them into the system, as shown below.

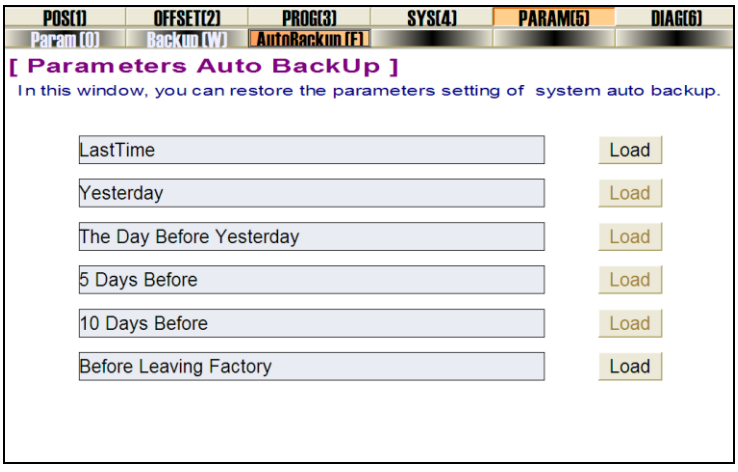


Fig. 6-4 Parameter Auto Backup

7 Common Problems

7.1 Common Problems in Returning to Machine Origin

◆ **The origin signal cannot be detected during returning to machine origin. It is usually caused by origin switch failure. The detection steps are shown as follows：**

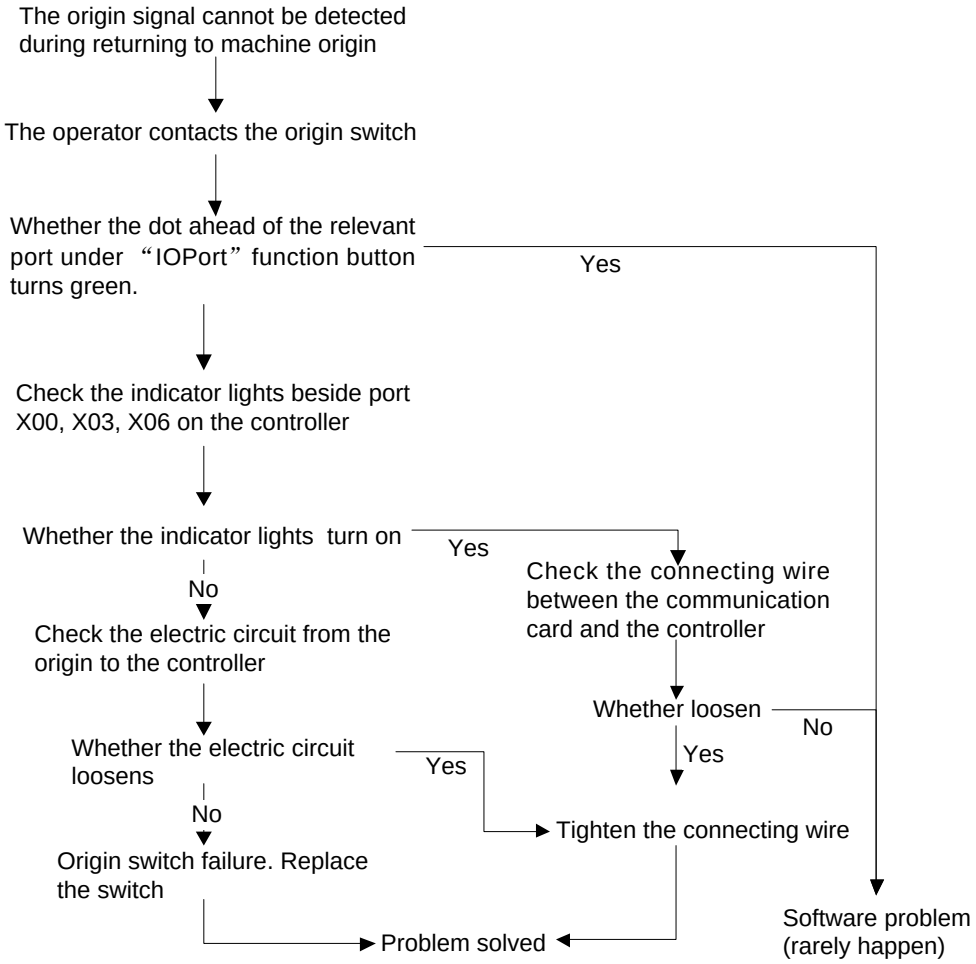


Fig. 7-1 Detection steps

◆ **The possible reasons for wrong direction of the machine tool during returning to machine origin are as follows.**

1) The polarity of the origin signal is wrong. The polarity of the origin signal is N when the origin switch is normally open; otherwise it is P;
2) Wrong setting of related parameters. Check parameter “Coarse Locating Direction”, and change the value of the parameter accordingly.

◆ **The possible reasons for low-speed coarse locating during returning to machine are as follows.**

1) The setting value of parameter “Coarse Locating Speed” is too small;
2) The polarity of the origin signal in the software does not match the types of the origin switch. If the origin switch is normally close, and the polarity of the origin signal is N, the machine tool will move away from the origin at the fine positioning speed during returning to machine origin.

7.2 Warning Information

Type	Content	Cause	Solution
Warning message	Simulation results show that program range exceeds the machine travel limit. By clicking [Info] -> [POS] to check details.	Tool path of the program file exceeds the upper/lower limit of workbench travel, which are decided by settings of parameter N10020 and N10030 separately.	Check if the WCS zero is reasonable. Check the tool path program file. Modify parameter settings of N10020 and N10030 to enlarge the workbench travel limit. (see section 3.1.3)
	The system has not returned to the machine origin, failed to execute the operation!	The system has not returned to machine origin. Whether the system has to return to the machine origin is decided by parameter N74001.	Use this function after returning to the machine origin.
	The system is busy, this operation can’t be executed.	Some illegal operations are performed under machining state.	Stop machining, and execute some operations under idle state.
Limit alarm	Positive (negative) limit of X (Y/Z/A/B/C) axis	The polarity of X axis positive limit port is wrong.	Enter [IOPort] window under tab [DIAG], and modify the port polarity.
		X axis runs into limit switch directly during motion.	Manually move X axis away from limit switch.
		There is an error in limit switch itself.	Check if limit switch works normally.

Type	Content	Cause	Solution
Servo alarm	Servo alarm of X (Y/Z/A/B/C) axis	The polarity of X axis servo alarm port is wrong.	Enter [IOPort] window under tab [DIAG], and modify the port polarity.
		There is an error in X axis servo driver itself.	Check if X axis servo driver works normally.
E-stop alarm	E-stop	The polarity of E-stop port is wrong.	Enter [IOPort] window under tab [DIAG], and modify the port polarity.
		The E-stop button is pressed.	Turn the E-stop button clockwise to make it bounced.
		Alarm occurs in transducer.	Find the cause according to transducer alarm type.
Machining alarm	Machining executive program failure	Software license expired	Register the software.
Terminal board not connected error alarm	Terminal board not connected.	Cable has not been firmly connected or there is error in Lambda controller.	Re-plug the cable and restart the software to observe the occurrence of the error; Polarity of port is wrong. Invert the port polarity and restart the software; Analyze possible causes according to the status of indicator “SYSTEM” on Lambda controller; Change another Lambda controller.

8 Appendix

8.1 Basic Concepts of NcStudio

NcStudio is a complicated system and involves many conceptions, such as workpiece coordinate system, machine coordinate system, operation mode, operation state and so on. It is inevitable that users will deal with these conceptions during machining process. Therefore, it is suggested that users should comprehend these conceptions before using the newest system.

8.1.1 Operation Mode

Under any circumstances, the machine tool is always in one of the following operation modes. A good understanding of them is vital for proper operation.

Reference point mode (homing)
Returning to the machine origin, also called homing, is the process to synchronize the internal coordinate and machine actual coordinate in CNC system. For this reason, homing is a necessary step during startup. And reference point mode is the default operation mode after system startup, in convenience of related operation execution.

Auto mode
In auto mode, the machine tool executes motion according to the pre-loaded machining program. Therefore, machining programs must be pre-loaded in auto mode.

Manual mode
In manual mode, you can control the motion of the machine tool manually. Manual mode can be further divided into jog mode and stepping mode (also called incremental mode).

1) In jog mode, when you press the manual button, the machine tool move consecutively until the button is released.
2) In stepping mode, from the time you press the manual button till you release it, the machine tool will move a specific distance, known as the stepping length. The stepping length has four stages, 0.01mm, 0.1mm, 1mm and 10mm. You can control the displacement of the machine tool precisely by customizing the step length in stepping mode.

8.1.2 Operation State

Each operation mode can be subdivided into several operation states according to the motion of machine tool. The operation modes and the operation states together decide the status of a machine tool.

IDLE
This is a normal state. In this state, the machine tool does not output, but is ready for any new task.

E-STOP

This is an abnormal state. In case of hardware breakdown or the E-STOP button pressed, the system will enter into this state and execute the pre-set protection measures, such as turning off the spindle motor and the coolant pump. In this state, the machine tool is locked and incapable of moving. When the hardware problem is resolved or E-STOP button is released, the system will automatically execute [Reset] and restore the machine tool to IDLE state.

RUNNING

When the machine tool is generating any motion, the system enters into this state.

PAUSE

When the machine tool is running, if you implement [Operate | Pause] order, or the system parses a M01 command (Wait Command), the system will enter into PAUSE state and wait for the next instruction. You can then implement [Operation| Start] to continue the operation or select [Stop] or [Reset] to stop the current operation and make the system enter into IDLE state.

LOCK

As an internal state, lock state is rarely seen under normal circumstances and only exists during state-switching.

8.1.3 Coordinate Systems in Machine Tool

◆ **Machine coordinate system (MCS)**

Machine coordinate system is a set of fixed coordinate system following right-hand rule. Its coordinate origin is always relative to a fixed point on the machine tool. Therefore, at any time, a certain point in space can be fixed exclusively by the machine coordinate system.

To completely support the machine coordinate system, the machine tool must have the corresponding function of backing to the machine origin. Otherwise, the concept of machine coordinate system only exists in the software.

◆ **Workpiece coordinate system (WCS)**

It is more often to adopt workpiece coordinate system at machining various kinds of workpiece. Generally speaking, the machining position refers to a position relative to a certain point on the workpiece, while the position where the workpiece is clamped is always variable relative to the machine origin. Thus, it is necessary to introduce a set of more convenient coordinate system for workpiece machining. That is workpiece coordinate system adhering to the right-hand rule as well. Its origin is fixed relative to a certain point on the workpiece, but can be floating to the machine origin.

8.2 Quick Start Guide

This section is applicable to Lambda hardware platform or in situation when you finished electrical wiring, and installation of control card and NcStudio system.

◆ **Check if the wiring is correct, and test whether signal is available.**

Connect communication card and Lambda controller's JP1 port with Db9M/F communication cable, and supply 24V power to the Lambda controller. Power on the signal system of machine tool(approach switch, etc.). Check the input signal light LED of Lambda controller. If the port of origin switch was NP, the signal lights of X00, Y03, Z06 are on. You can touch the origin switch through artificial simulation (If it is a travel limit switch, you can press on the switch to check whether you can receive the signal. If it is a photoelectric switch, you can shadow the light to check whether you can receive the signal. And if it is a metal proximity switch, you can touch it with a metal block). If the LED lights out when the origin switch is touched, it shows that the origin switch signal is sent to terminal board. If the switch's state is NC, and LED lights out at usual time, the LED turns on when the origin switch is artificially touched. It indicates that the origin signal can be delivered to terminal board. Check other input terminals with the same method to make sure that the wiring between terminal board and the machine tool is correct. The debugging time will be shortened greatly.

◆ **Debug signals**

Power on the computer, run NcStudio, and then switch to [I/O port] window under [Diag]. Mapped I/O ports will be shown in the window. Red light before the mapped ports means signal is invalid. Modify the polarity of input ports according to the types of input signals such as origin switch, emergency stop button. [N] means a normally open point; [P] means a normally closed point. When the polarities of all ports have been modified, shut down NcStudio and restart it to make the modification effective. Then power on the electrical cabinet. And the lights before input ports such as [Program Start], [Program Stop], [Encoder Zero of X\Y\Z-axis], [Emergency Stop] in the I/O port function window will be red, which means the signals are invalid. If the lights are green, you should check the wiring of electrical circuit and polarities of signals. If the electrical circuit is ok, the polarities of corresponding signals need to be modified so that the lights before the ports are red.

◆ **Set parameters**

Set manufacturer parameter [Axis Pulse Factor]. The smaller the [Axis Pulse Factor] is, the higher the control resolution will be. If [Axis Pulse Factor] is set too small, it will influence the maximal feedrate. After [Axis Pulse Factor] is determined, calculate electronic gear ratio of servo motor through the value of [Axis Pulse Factor].

$$\text{Electronic gear ratio} = \frac{\dfrac{\text{Encoder resolution}}{\text{Screw pitch}}}{\text{Pulse equivalent}} \times \text{Mechanical deceleration ratio}$$

$$\text{Mechanical deceleration ratio} = \frac{\text{Reducer input speed}}{\text{Reducer output speed}} = \frac{\text{Teeth No. of driven wheel}}{\text{Teeth No. of driving wheel}} = \frac{\text{Motor speed}}{\text{Screw speed}}$$

Manually move the machine tool to make sure the motion directions of each axis is correct. Please be noted that NcStudio system adopts right-hand coordinate system where X right and upward are the respective positive directions for X-axis and Z-axis, and the direction away from Y-axis is the positive direction, If the directions are incorrect, you can modify parameter [Axis_DIR] or servo driver parameters. Set manufacturer parameter [WORKBENCH_UPPER/LOWER_LIMIT] based on the actual size of machine tool so that software limitation function can be enabled. Set manufacturer parameter [REFPT_COOR] based on the position of zero sensor on each axis, and execute returning to machine origin. During the process, observe motion direction of each axis. If the motion direction is abnormal, modify the manufacturer parameter [Axis_DIR].

8.3 Related Dimension Drawings

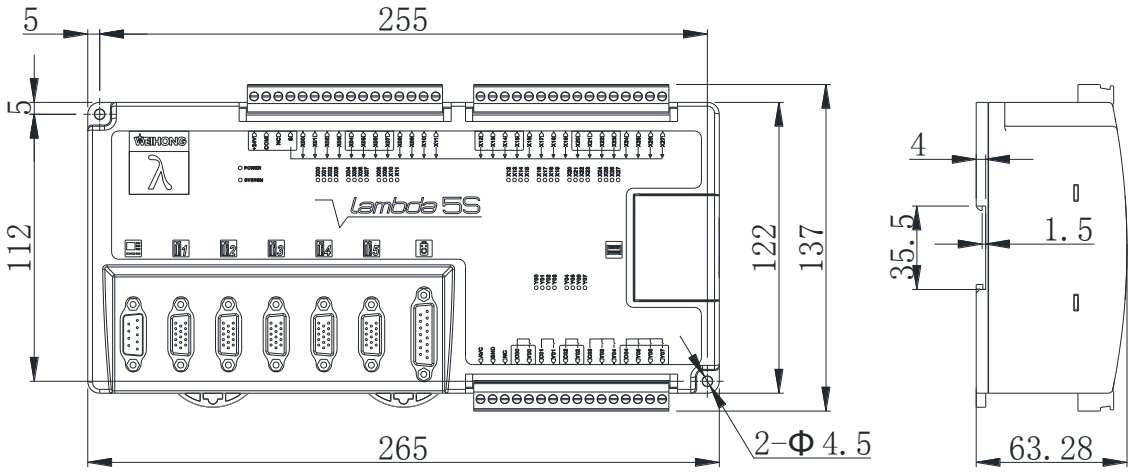


Fig. 8-1 Dimension drawing of Lambda 5S controller

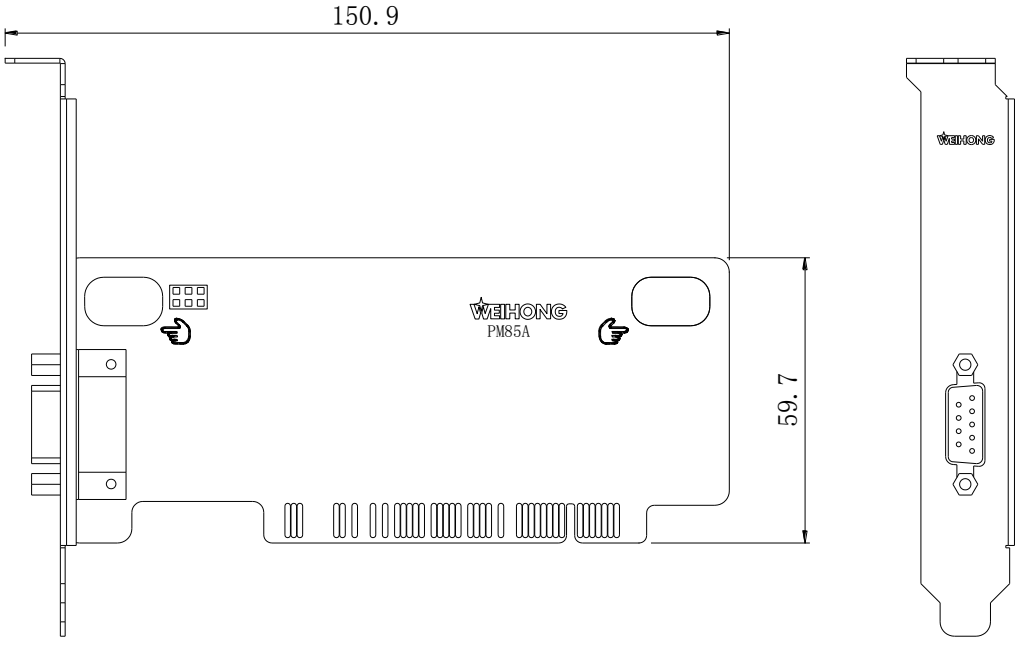


Fig. 8-2 Dimension drawing of communication card PM85A

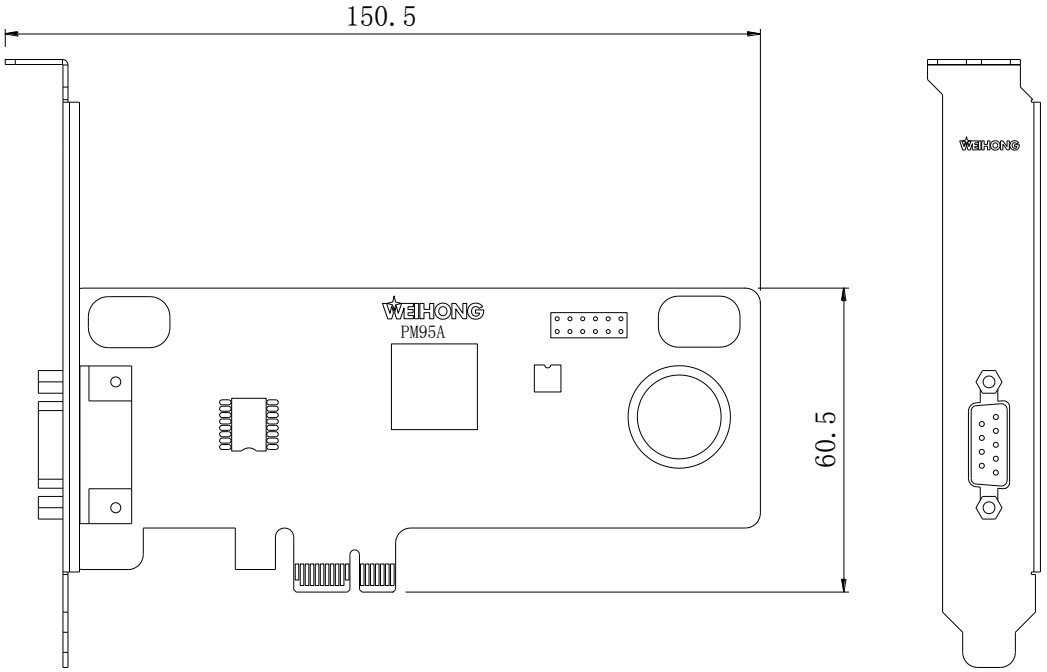


Fig. 8-3 Dimension drawing of communication card PM95A

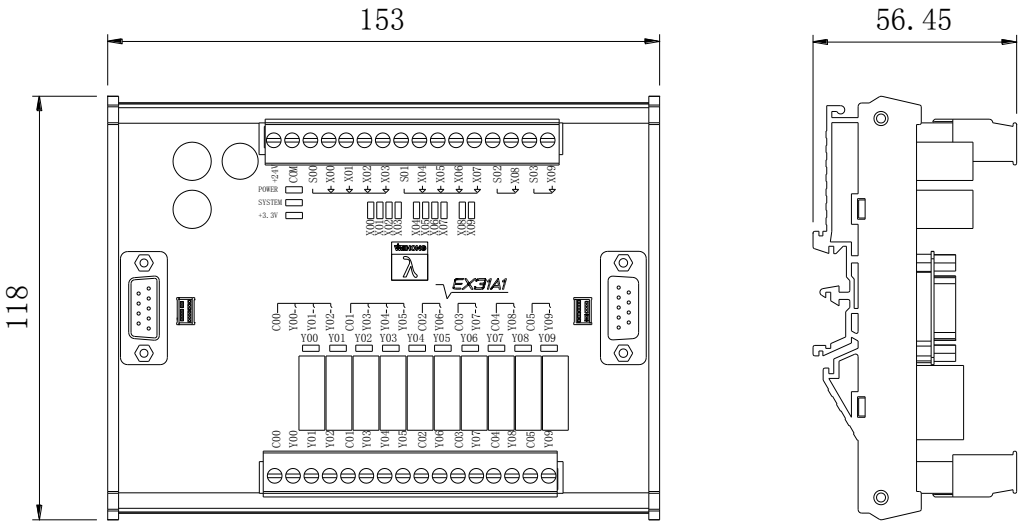


Fig. 8-4 Dimension drawing of terminal board EX31A

8.4 Software License Agreement

Important—Read Carefully before Using This Product:

The term “Software Product” includes all copies of the licensed software and its documentation. This license agreement is a legal agreement between You (either an individual, a legal entity or any affiliated companies or other entities) and Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). By installing, copying, or otherwise using the Software Product, you agree to be bound by the terms and conditions of this license. Unless otherwise stated in this agreement, you shall not use, copy, revise, rent, or transfer the Software product or any part of the Software Product for any other purposes.

Description of Further Rights and Restrictions:

1) You may install for use one copy of the Software Product on a single machine;
2) You may make a copy of the Software Product for archive or backup purposes and the copy is to be used on this machine only;
3) You may transfer the Software Product and the license agreement to a third party, provided that the third party accepts the terms and conditions stated in this agreement, with prior express permission from Weihong Company;
4) When transfer confirmed, you shall transfer all the copies of the original documents and the supplementary documents to the third party or destroy all the copies un-transferred.
5) You can use the Software Product on a network server or intranet server only if it is stipulated in explicit terms that you are allowed to use the Software Product on a network server or intranet server, or you have purchased license for each node and terminal using the Software Product;
6) You may NOT sublicense, assign or transfer the license agreement;
7) You may NOT or direct any third party to reverse engineer, decompile or disassemble the Software Product;
8) You may NOT copy or transfer the Software Product or any part of the Software Product unless otherwise expressly stated in this agreement;
9) The license agreement shall be terminated automatically upon you transfer the Software Product or copies of all or part of the Software Product to a third party.

After-sales Guarantee:

Weihong Company guarantees that for 90 days from the date of shipment the software carrier will be free from defects in materials and workmanship. When such a defect has been confirmed, our only responsibility is to replace the software carrier. This remedy is your exclusive remedy. This after-sales guarantee is invalid for any carrier defect caused by accidents, abuses or mal-operation. The replaced software carrier enjoys the remaining guarantee time of the original software carrier or of a 30-day guarantee time, whichever is longer.

Except as the after-sales guarantee made above, the Software Product does not enjoy any other form of

after-sale guarantee.

Limitation of Liability:

The above guarantee, whether made explicitly or by implication, constitutes the entire contents of the guarantee, including guarantee for the commerciality and applicability of special application aim. Whether you follow other terms in this agreement or not, Weihong Company, as well as its agents and sales staff, will not be responsible for any profits loss, availability loss, business break-off or any forms of indirect, special, accidental or inevitable damage or claim made by any third party, generated from the using of the Software Product, even if Weihong Company has been informed of the possible occurrence of such events in advance.

Termination:

This license may be terminated by Weihong Company at any time once you violate any terms or conditions made in this agreement. Once the license is terminated, you are obliged to destroy all the copies of the Software Product or return them to Weihong Company.

Applicable Law:

Copyright Law, Regulations for the Protection of Computer Software, Patent Law and other relevant laws and regulations.

Now, we affirm that you have already read through this agreement and understood it thoroughly and agreed to comply with all of the terms and conditions of this agreement strictly.

Weihong Electronic Technology Co., Ltd.